

# Simscape™ Electrical™

Reference (Electronics, Mechatronics, and Power Systems)



# MATLAB® & SIMULINK®

R2018b



## How to Contact MathWorks



Latest news: [www.mathworks.com](http://www.mathworks.com)  
Sales and services: [www.mathworks.com/sales\\_and\\_services](http://www.mathworks.com/sales_and_services)  
User community: [www.mathworks.com/matlabcentral](http://www.mathworks.com/matlabcentral)  
Technical support: [www.mathworks.com/support/contact\\_us](http://www.mathworks.com/support/contact_us)



Phone: 508-647-7000



The MathWorks, Inc.  
3 Apple Hill Drive  
Natick, MA 01760-2098

*Simscape™ Electrical™ Reference (Electronics, Mechatronics, and Power Systems)*

© COPYRIGHT 2013–2018 by The MathWorks, Inc.

The software described in this document is furnished under a license agreement. The software may be used or copied only under the terms of the license agreement. No part of this manual may be photocopied or reproduced in any form without prior written consent from The MathWorks, Inc.

FEDERAL ACQUISITION: This provision applies to all acquisitions of the Program and Documentation by, for, or through the federal government of the United States. By accepting delivery of the Program or Documentation, the government hereby agrees that this software or documentation qualifies as commercial computer software or commercial computer software documentation as such terms are used or defined in FAR 12.212, DFARS Part 227.72, and DFARS 252.227-7014. Accordingly, the terms and conditions of this Agreement and only those rights specified in this Agreement, shall pertain to and govern the use, modification, reproduction, release, performance, display, and disclosure of the Program and Documentation by the federal government (or other entity acquiring for or through the federal government) and shall supersede any conflicting contractual terms or conditions. If this License fails to meet the government's needs or is inconsistent in any respect with federal procurement law, the government agrees to return the Program and Documentation, unused, to The MathWorks, Inc.

### Trademarks

MATLAB and Simulink are registered trademarks of The MathWorks, Inc. See [www.mathworks.com/trademarks](http://www.mathworks.com/trademarks) for a list of additional trademarks. Other product or brand names may be trademarks or registered trademarks of their respective holders.

### Patents

MathWorks products are protected by one or more U.S. patents. Please see [www.mathworks.com/patents](http://www.mathworks.com/patents) for more information.

## Revision History

September 2013	Online only	New for Version 6.0 (Release 2013b)
March 2014	Online only	Revised for Version 6.1 (Release 2014a) (Renamed from <i>SimPowerSystems™ Reference (Third Generation)</i> )
October 2014	Online only	Revised for Version 6.2 (Release 2014b)
March 2015	Online only	Revised for Version 6.3 (Release 2015a)
September 2015	Online only	Revised for Version 6.4 (Release 2015b)
March 2016	Online only	Revised for Version 6.5 (Release 2016a) (Renamed from <i>SimPowerSystems™ Reference (Simscape™ Components)</i> )
September 2016	Online only	Revised for Version 6.6 (Release 2016b)
March 2017	Online only	Revised for Version 6.7 (Release 2017a)
September 2017	Online only	Revised for Version 6.8 (Release 2017b)
March 2018	Online only	Revised for Version 6.9 (Release 2018a)
September 2018	Online only	Revised for Version 7.0 (Release 2018b) (Renamed from <i>Simscape™ Power Systems™ Reference (Simscape™ Components)</i> and <i>Simscape™ Electronics™ Reference</i> )



## Blocks – Alphabetical List

**1**

## Functions – Alphabetical List

**2**

## Abbreviations and Naming Conventions in Simscape Electrical Power Systems Libraries

## Parameter Dependencies

**A**

<b>Parameter Dependencies</b> . . . . .	<b>A-2</b>
Parameter Dependency Tables . . . . .	<b>A-2</b>



# Blocks — Alphabetical List

---

## AC Cable

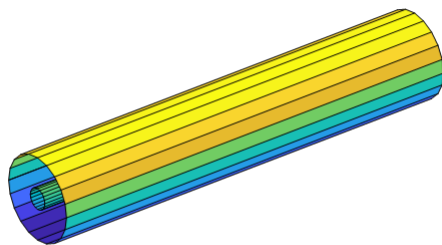
Model three-phase AC power cabling

**Library:** Simscape / Electrical / Power Systems / Passive Devices



## Description

The AC Cable block represents a three-phase AC power cable with a conducting sheath surrounding each phase. The figure shows a single-phase conductor inside a conducting sheath. The inner cylinder represents the main conductor for the phase, and the outer cylinder represents the conducting sheath.



The block has two variants:

- Composite three-phase variant (default) --- Contains three-phase connection ports for the sheaths and phases and a single-phase connection port for each electrical reference node.
- Expanded three-phase variant --- Contains single-phase connection ports for each sheath, phase, and electrical reference node.

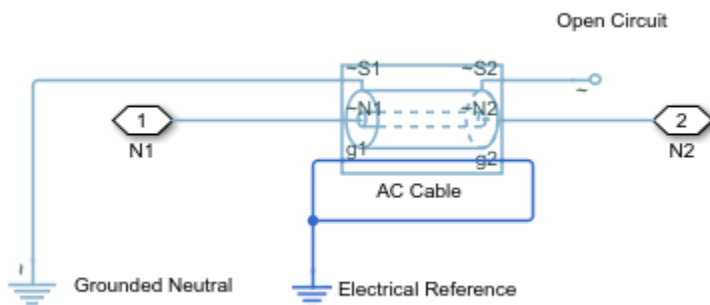


The AC Cable block includes inductances and mutual inductances between each phase, sheath, and return path. Therefore, you can connect an ideal electrical reference block to both return ports, **g1** and **g2**, while maintaining loss modeling in the Earth- or neutral-return line.

To facilitate simulation convergence when you connect the AC Cable block to a source block, include source impedance using one of these methods:

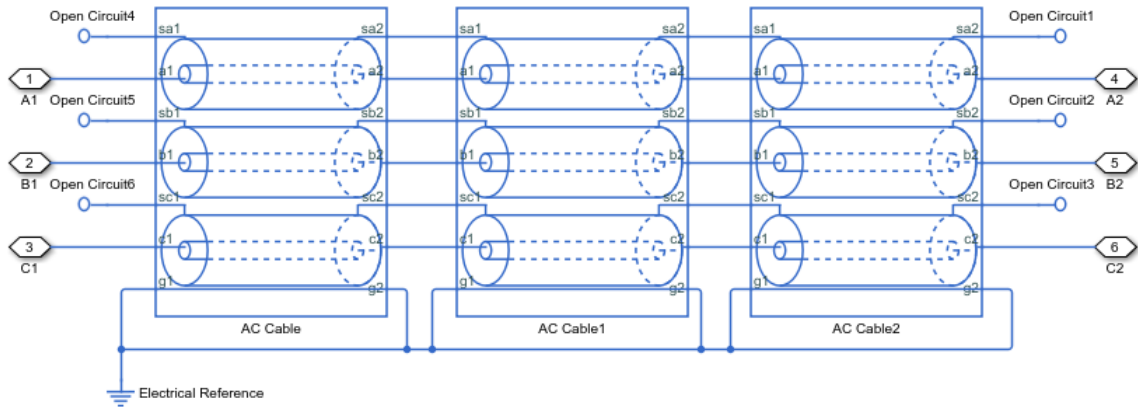
- Configure the source block to include impedance.
- Insert a block that models impedance between the source block and the AC Cable block.

To model unbonded sheaths, connect the unbonded sheaths to an Open Circuit block. The figure shows a model of single-point bonding using the composite three-phase variant of the block.

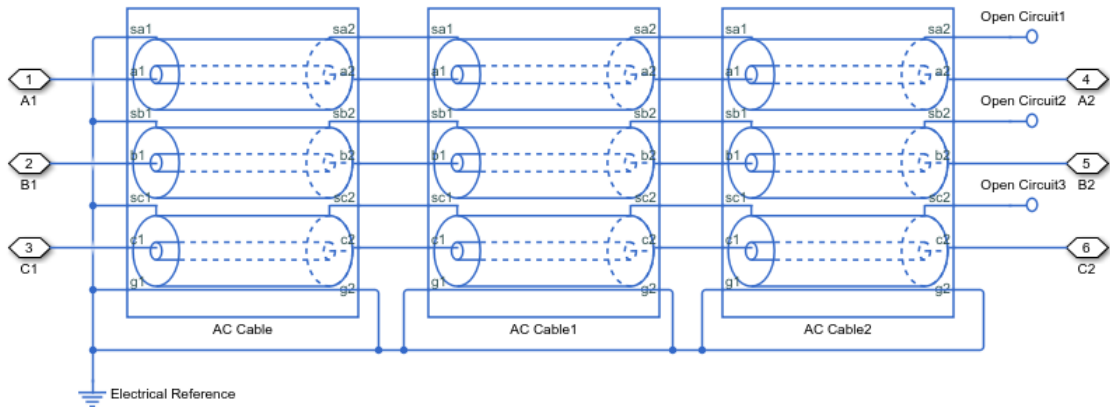


For high performance modeling, in terms of simulation speed, use a single AC Cable block. To improve model fidelity in terms of frequency behavior, connect several AC Cable blocks in series. For series-connected blocks, the sheaths and main conductors act as coupled transmission lines with perfect transposition of the phases. The number of AC Cable blocks that you use to model a particular physical length of cable must be less than the number of transpositions in the physical system that you are modeling. Types of continuous multi-segment cables that you can model include:

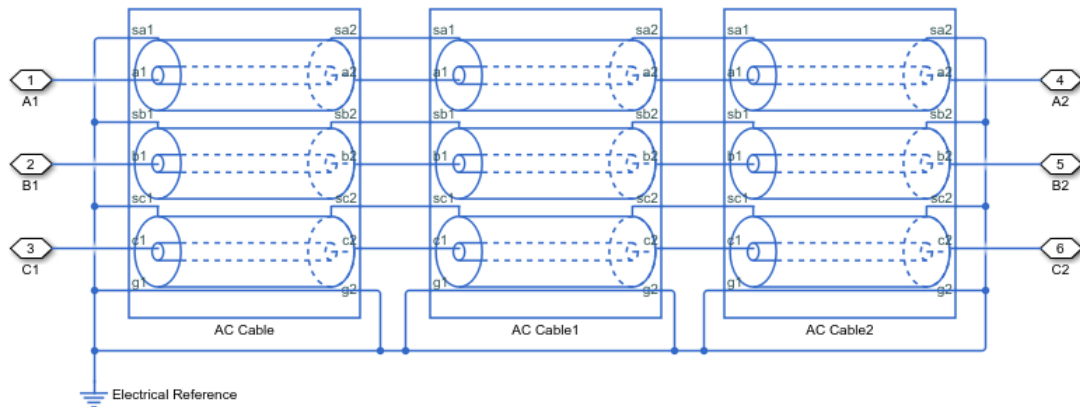
- Unbonded continuous cables



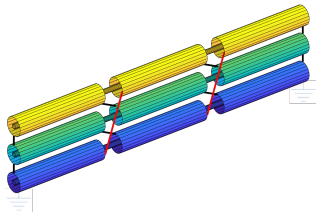
- Single-point bonded continuous cables



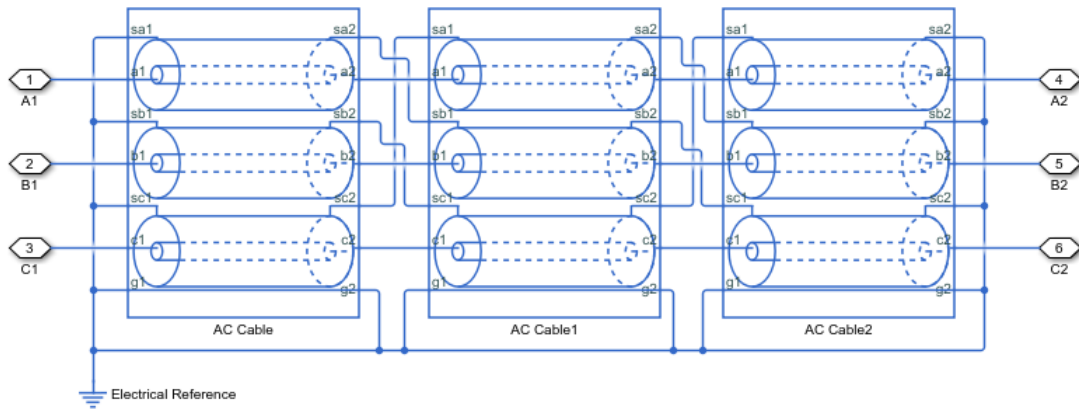
- Double-point bonded continuous cables



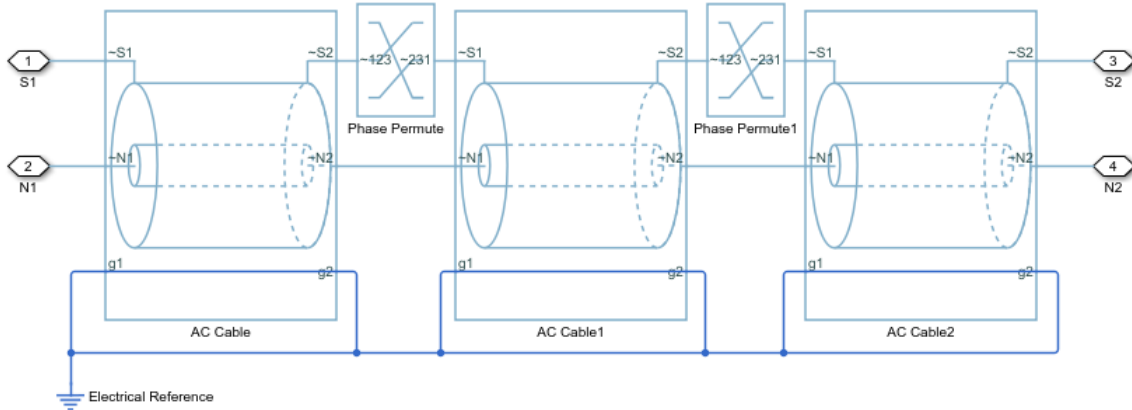
You can also model cross-bonded cables using the AC Cable block.



This three pi-segment cable model implements cross-bonding using expanded three-phase ports and single-phase connection lines. The sheath in the model is two-point bonded.



This model of blocks with composite three-phase-ports uses Phase Permute blocks to implement cross bonding. The sheath in the model is unbonded.



For an example that allows you to choose the number of segments and type of bonding, see “AC Cable with Bonded Sheaths”.

## AC Cable Model

The AC Cable block uses the concept of partial inductances to calculate the inductance values. These values include the partial self-inductance of each phase, sheath, and return path and the partial mutual inductances between each:

- Phase and each other phase
- Phase and the sheath of that phase
- Phase and the sheath of neighboring phases
- Phase and the return
- Sheath and each neighboring sheath
- Sheath and the return

For three equivalent phases, the matrix that defines the resistance relationships for the vector [phase A; sheath A; phase B; sheath B; phase C; sheath C] is

$$R = \begin{bmatrix} R_a + R_g & R_g & R_g & R_g & R_g & R_g \\ R_g & R_s + R_g & R_g & R_g & R_g & R_g \\ R_g & R_g & R_a + R_g & R_g & R_g & R_g \\ R_g & R_g & R_g & R_s + R_g & R_g & R_g \\ R_g & R_g & R_g & R_g & R_a + R_g & R_g \\ R_g & R_g & R_g & R_g & R_g & R_s + R_g \end{bmatrix}$$

$$R_a = R'_a l$$

$$R_g = R'_{\text{return}} l,$$

for which  $R'_{\text{return}}$  depends on the return parameterization method such that:

- For a return parameterization based on distance and resistance  $R'_{\text{return}} = R'_g$ .
- For a return parameterization based on frequency and Earth resistivity  $R'_{\text{return}} = \pi^2 10^{-7} f$

and

$$R_s = \dot{R}'_s l,$$

where:

- $R$  is the resistance matrix.
- $R_a$  is the resistance of a particular phase.
- $R_s$  is the resistance of a particular sheath.
- $R_g$  is the resistance of the Earth- or neutral-return.
- $R'_a$  is the resistance per unit length for the phase.
- $l$  is the cable length.
- $R'_s$  is the resistance per unit length for the sheath.
- $R'_{return}$  is the resistance per unit length of the return. The value of  $R'_{return}$  varies depending on the return parameterization method.
- $R'_g$  is the resistance per unit length for the Earth- or neutral return.
- $f$  is the frequency that the block uses to calculate Earth-return parameters if you parameterize the block using the frequency and Earth resistivity method.

The block uses standard expressions to calculate the capacitances between:

- Concentric or adjacent cylinders
- Each phase and its own sheath
- Each sheath and the return

The matrix that defines these capacitance relationships is

$$C = \begin{bmatrix} C_{as_a} & -C_{as_a} & 0 & 0 & 0 & 0 \\ -C_{as_a} & C_{as_a} + C_{s_a g} & 0 & 0 & 0 & 0 \\ 0 & 0 & C_{as_a} & -C_{as_a} & 0 & 0 \\ 0 & 0 & -C_{as_a} & C_{as_a} + C_{s_a g} & 0 & 0 \\ 0 & 0 & 0 & 0 & C_{as_a} & -C_{as_a} \\ 0 & 0 & 0 & 0 & -C_{as_a} & C_{as_a} + C_{s_a g} \end{bmatrix}$$

$$C_{as_a} = \frac{2\pi\epsilon_r\epsilon_0 l}{\ln\left(\frac{r_s}{r_a}\right)}$$

$$r_a = GMR \cdot e^{\frac{1}{4}}$$

$$C_{s_ag} = \frac{\pi\epsilon_{env}\epsilon_0 l}{\ln\left(\frac{D_{return}}{r_s}\right)},$$

for which  $D_{return}$  depends on the return parameterization method such that:

- For a return parameterization based on distance and resistance  $D_{return} = D_e$ .
- For a return parameterization based on frequency and Earth resistivity

$$D_{return} = 1650 \sqrt{\frac{\rho}{2\pi f}}$$

where:

- $C$  is the capacitance matrix.
- $C_{as_a}$  is the capacitance between each phase and the sheath of that phase.
- $C_{s_ag}$  is the capacitance between each sheath and return.
- $\epsilon_r$  is the permittivity of the dielectric.
- $\epsilon_0$  is the permittivity of free space.
- $r_s$  is the radius of the sheath.
- $r_a$  is the effective radius of the conductor. For a single-strand conductor,  $r_a$  is the radius of the strand.
- $GMR$  is the geometric mean radius of the conductor. For a single-strand conductor,  $GMR = r_{strand} e^{-\frac{1}{4}}$ , where  $r_{strand}$  is the radius of the strand.
- $\epsilon_{env}$  is the permittivity of the material between the sheathed lines and the return path.
- $D_{return}$  is the effective distance to the return. The value of  $D_{return}$  varies if you use the distance/return parameterization method.

- $D_e$  is the effective distance to the Earth- or neutral-return.
- $\rho$  is the effective Earth resistivity for an Earth-return.
- $f$  is the frequency that is used to determine the return path properties.

The block uses the concept of partial inductances to calculate inductance values. These values include the partial self-inductance of each phase, sheath, and return path and the partial mutual inductances between each:

- Phase and each other phase
- Phase and the sheath of that phase
- Phase and the sheath of neighboring phases
- Phase and the return
- Sheath and each neighboring sheath
- Sheath and the return

The equations that define these inductance relationships are:

$$L = \begin{bmatrix} D_a & \delta & A & \alpha & A & \alpha \\ \delta & D_s & \alpha & S & \alpha & S \\ A & \alpha & D_a & \delta & A & \alpha \\ \alpha & S & \delta & D_s & \alpha & S \\ A & \alpha & A & \alpha & D_a & \delta \\ \alpha & S & \alpha & S & \delta & D_s \end{bmatrix}$$

$$D_a = L_a + L_g - 2M_{ag}$$

$$L_a = 2 \times 10^{-7} l \left[ \ln \left( \frac{2l}{r_a} \right) - \frac{3}{4} \right]$$

$$L_g = 2 \times 10^{-7} l \left[ \ln \left( \frac{2l}{\sqrt{r_a r_s}} \right) - \frac{3}{4} \right]$$

$$M_{ag} = M_{sg} = 2 \times 10^{-7} l \left[ \ln \left( \frac{2l}{D_{\text{return}}} \right) - 1 \right]$$



$$D_s = L_s + L_g - 2M_{sg}$$

$$L_s = M_{as_a} = 2 \times 10^{-7} l \left[ \ln \left( \frac{2l}{r_s} \right) - \frac{3}{4} \right]$$

$$\delta = L_g + M_{as_a} - M_{ag} - M_{sg}$$

$$\alpha = L_g + M_{as_b} - M_{ag} - M_{sg}$$

$$M_{as_b} = M_{s_a s_b} = M_{ab} = 2 \times 10^{-7} l \left[ \ln \left( \frac{2l}{d_{ab}} \right) - 1 \right],$$

for which  $d_{ab}$  depends on the line formation parameterization method, such that:

- For a trefoil line formation parameterization  $d_{ab} = D_{ab}$ .
- For a flat line formation parameterization  $d_{ab} = D_{ab} \sqrt[3]{2}$ .

$$A = L_g + M_{ab} - 2M_{ag}$$

$$S = L_g + M_{s_a s_b} - 2M_{sg},$$

where:

- $L$  is the inductance matrix.
- $D_a$  is the self-inductance of a single phase through its entire path and return.
- $L_a$  is the partial self-inductance of each phase.
- $L_g$  is the partial self-inductance of the Earth- or neutral-return.
- $M_{ag}$  is the partial mutual inductance between each phase and the Earth- or neutral-return.
- $M_{sg}$  is the partial mutual inductance between each sheath and the Earth- or neutral-return.

- The factor,  $2 \times 10^{-7}$  is equal to  $\mu_0 / 2\pi$ , because permeability of free space,  $\mu_0$ , is equal to  $1.257 \times 10^{-6}$  or  $4\pi \times 10^{-7}$  H/m.
- $D_s$  is the self-inductance of a single sheath through its entire path and return.
- $L_s$  is the partial self-inductance of each sheath.
- $M_{as_a}$  is the partial mutual inductance between each phase and the sheath of that phase.
- $\delta$  is the effective mutual inductance between a phase and the sheath of that phase.
- $\alpha$  is the effective mutual inductance between a phase and a neighboring sheath.
- $M_{as_b}$  is the partial mutual inductance between each phase and the sheath of each neighboring phase.
- $M_{s_a s_b}$  is the partial mutual inductance between sheaths of different phases.
- $M_{ab}$  is the partial mutual inductance between each phase and each other phase.
- $d_{ab}$  is the effective distance between adjacent phases. The value of  $d_{ab}$  varies depending on the line parameterization method.
- $D_{ab}$  is the center-to-center distance between adjacent phases.
- $A$  is the effective mutual inductance between phases.
- $S$  is the effective mutual inductance between sheaths.

A modal transformation that is related to the Clarke transform simplifies the equivalent circuit. The six-by-six transformation,  $T$ , is

$$T = \frac{1}{\sqrt{3}} \begin{bmatrix} 1 & 0 & \sqrt{2} & 0 & 0 & 0 \\ 0 & 1 & 0 & \sqrt{2} & 0 & 0 \\ 1 & 0 & -\frac{1}{\sqrt{2}} & 0 & \sqrt{\frac{3}{2}} & 0 \\ 0 & 1 & 0 & -\frac{1}{\sqrt{2}} & 0 & \sqrt{\frac{3}{2}} \\ 1 & 0 & -\frac{1}{\sqrt{2}} & 0 & -\sqrt{\frac{3}{2}} & 0 \\ 0 & 1 & 0 & -\frac{1}{\sqrt{2}} & 0 & -\sqrt{\frac{3}{2}} \end{bmatrix}.$$

As  $T^\dagger = T^{-1}$ , applying the  $T$  transform yields the modal resistance matrix,  $R_m$ , the modal capacitance matrix,  $C_m$ , and the modal inductance matrix,  $L_m$ .

The transformed matrices are:

$$R_m = T^\dagger R T = \begin{bmatrix} R_a + 3R_g & 3R_g & 0 & 0 & 0 & 0 \\ 3R_g & R_s + 3R_g & 0 & 0 & 0 & 0 \\ 0 & 0 & R_a & 0 & 0 & 0 \\ 0 & 0 & 0 & R_s & 0 & 0 \\ 0 & 0 & 0 & 0 & R_a & 0 \\ 0 & 0 & 0 & 0 & 0 & R_s \end{bmatrix}$$

$$C_m = T^\dagger C T = \begin{bmatrix} C_{as_a} & -C_{as_a} & 0 & 0 & 0 & 0 \\ -C_{as_a} & C_{as_a} + C_{s_a g} & 0 & 0 & 0 & 0 \\ 0 & 0 & C_{as_a} & -C_{as_a} & 0 & 0 \\ 0 & 0 & -C_{as_a} & C_{as_a} + C_{s_a g} & 0 & 0 \\ 0 & 0 & 0 & 0 & C_{as_a} & -C_{as_a} \\ 0 & 0 & 0 & 0 & -C_{as_a} & C_{as_a} + C_{s_a g} \end{bmatrix} = C$$

$$L_m = T^\dagger L T = \begin{bmatrix} D_a + 2A & \delta + 2\alpha & 0 & 0 & 0 & 0 \\ \delta + 2\alpha & D_s + 2A & 0 & 0 & 0 & 0 \\ 0 & 0 & D_a - A & \delta - \alpha & 0 & 0 \\ 0 & 0 & \delta - \alpha & D_s - S & 0 & 0 \\ 0 & 0 & 0 & 0 & D_a - A & \delta - \alpha \\ 0 & 0 & 0 & 0 & \delta - \alpha & D_s - S \end{bmatrix}.$$

The transformation changes each six-by-six matrix into three uncoupled two-by-two matrices. The capacitance matrix is invariant under this transformation. The power is invariant in the transformed and untransformed domains because  $T$  is unitary.

## Assumptions and Limitations

- For resistance calculations, the phases are equivalent.
- Relative to the phase-to-sheath capacitance and the sheath-return capacitances all other capacitances, are negligible due to the shielding provided by the conducting sheaths.

## Ports

### Conserving

#### **~S1 — Sheath**

electrical

Expandable three-phase port associated with sheath 1.

#### **~N1 — Phase**

electrical

Expandable three-phase port associated with *a*, *b*, and *c* phases 1.

#### **g1 — Ground**

electrical

Electrical conserving port associated with ground 1.

#### **~S2 — Sheath**

electrical

Expandable three-phase port associated with sheath 2.

#### **~N2 — Phase**

electrical

Expandable three-phase port associated with *a*, *b*, and *c* phases 2.

#### **g2 — Ground**

electrical

Electrical conserving port associated with ground 2.

## Parameters

### **Cable length — Length**

120 km (default)

Length of the cable.

### **Geometric mean radius of conductor — Radius**

5 mm (default)

Geometric mean radius of the conductor, which is a function of the number and type of individual strands in the conductor of the AC cable.

### **Sheath radius — Radius**

10 mm (default)

Average radius of the sheath. To ensure that the sheath radius is greater than the physical radius of a single-stranded conductor with a particular GMR, the sheath radius must be

greater than  $GMR * e^{\frac{1}{4}}$ .

### **Line-line spacing (center-to-center) — Distance**

25 mm (default)

Distance between the line centers.

### **Line formation — Line configuration**

Trefoil (default) | Flat

Cable line formation.

### **Conductor resistance per length — Resistance**

1 Ohm/km (default)

Resistance per length of a conductor.

### **Sheath resistance per length — Resistance**

10 Ohm/km (default)

Resistance per length of a sheath.

### **Insulation relative permittivity — Permittivity**

2.4 (default)

Relative permittivity of the insulation.

**Relative permittivity between lines and return path — Relative permittivity**

1 (default)

Relative permittivity of the circuit.

**Return parameterization — Model**

Use frequency and Earth resistivity (default) | Use distance and resistance

Parameterization method.

**Dependencies**

Enabling either option enables other parameters.

**Frequency for Earth-return impedance — Frequency**

60 Hz (default)

Frequency at which the Earth-return impedance is calculated.

**Dependencies**

Selecting Use frequency and Earth resistivity for the **Return parameterization** parameter enables this parameter.

**Earth resistivity — Resistance**

100 m\*Ohm (default)

Earth-return resistivity.

**Dependencies**

Selecting Use frequency and Earth resistivity for the **Return parameterization** parameter enables this parameter.

**Effective distance to return path — Return path distance**

1 km (default)

Effective distance between the phases and the return path.

**Dependencies**

Selecting Use distance and resistance for the **Return parameterization** parameter enables this parameter.

**Return path resistance per length — Return path unit resistance**

0.1 Ohm/km (default)

Resistance per length of the return path.

**Dependencies**

Selecting Use distance and resistance for the **Return parameterization** parameter enables this parameter.

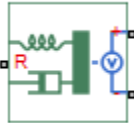
**See Also**

Phase Permute

**Introduced in R2017b**

# Accelerometer

Behavioral model of MEMS accelerometer



## Library

Sensors

## Description

The Accelerometer block implements a behavioral model of a MicroElectroMechanical Systems (MEMS) accelerometer. For the default output type **Voltage level**, the accelerometer provides an output voltage that is proportional to the acceleration rate presented at the mechanical translational physical port R. The output voltage is limited according to the values that you provide for maximum and minimum output voltage.

The block also has an alternative output type, **PWM duty cycle**. With this choice, the output of the block is a PWM signal with a duty cycle that is proportional to the measured acceleration. You can limit the variation in duty cycle to a specified range.

Optionally, you can model sensor dynamics by setting the **Dynamics** parameter to **Model sensor bandwidth**. Including dynamics adds a first-order lag between the angular rate presented at port R and the corresponding voltage applied to the electrical + and - ports.

If running your simulation with a fixed-step solver, or generating code for hardware-in-the-loop testing, MathWorks recommends that you set the **Dynamics** parameter to **No dynamics – Suitable for HIL**, because this avoids the need for a small simulation time step if the sensor bandwidth is high.



## Variables

Use the **Variables** section of the block interface to set the priority and initial target values for the block variables prior to simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

The **Measured acceleration** variable target specifies the initial output for the sensor.

## Parameters

### Output type

Select one of the following options to define the block output type:

- **Voltage level** — The amplitude of the output voltage is proportional to the measured acceleration. This is the default option.
- **PWM duty cycle** — The duty cycle (on time divided by the pulse total time) is proportional to the measured acceleration.

### Sensitivity

The change in output voltage level per unit change in acceleration when the output is not being limited. This parameter is only visible when you select **Voltage level** for the **Output type** parameter. The default value is 1000 mV/gee.

### Output voltage for zero acceleration

The output voltage from the sensor when the acceleration is zero. This parameter is only visible when you select **Voltage level** for the **Output type** parameter. The default value is 2.5 V.

### Maximum output voltage

The maximum output voltage from the sensor, which determines the sensor maximum measured positive acceleration. This parameter is only visible when you select **Voltage level** for the **Output type** parameter. The default value is 4 V.

### Minimum output voltage

The minimum output voltage from the sensor, which determines the sensor maximum measured negative acceleration. This parameter is only visible when you select **Voltage level** for the **Output type** parameter. The default value is 1 V.

## **Duty cycle sensitivity (percent per unit acceleration)**

The change in duty cycle per unit acceleration. Duty cycle is expressed as a percentage of the PWM period. This parameter is only visible when you select PWM duty cycle for the **Output type** parameter. The default value is 10 percent/gee.

## **Duty cycle for zero acceleration (percent)**

The duty cycle output by the sensor when the acceleration is zero. This parameter is only visible when you select PWM duty cycle for the **Output type** parameter. The default value is 50%.

## **Maximum duty cycle (percent)**

The maximum duty cycle output by the sensor. Increasing acceleration levels beyond this point will not register an increase in duty cycle. This parameter is only visible when you select PWM duty cycle for the **Output type** parameter. The default value is 75%.

## **Minimum duty cycle (percent)**

The minimum duty cycle output by the sensor. Decreasing acceleration levels beyond this point will not register a decrease in duty cycle. This parameter is only visible when you select PWM duty cycle for the **Output type** parameter. The default value is 25%.

## **PWM frequency**

The frequency of the output pulse train. This parameter is only visible when you select PWM duty cycle for the **Output type** parameter. The default value is 1 kHz.

## **Output voltage amplitude**

The amplitude of the output pulse train when high. This parameter is only visible when you select PWM duty cycle for the **Output type** parameter. The default value is 5 V.

## **Dynamics**

Select one of the following options for modeling sensor dynamics:

- **No dynamics – Suitable for HIL** — Do not model sensor dynamics. Use this option when running your simulation fixed step or generating code for hardware-in-the-loop testing, because this avoids the need for a small simulation time step if the sensor bandwidth is high. This is the default option.
- **Model sensor bandwidth** — Model sensor dynamics with a first-order lag approximation, based on the **Bandwidth** parameter value. You can control the initial condition for the lag by specifying the **Measured acceleration** variable target.

**Bandwidth**

Specifies the 3dB bandwidth for the measured acceleration assuming a first-order time constant. This parameter is only visible when you select **Model sensor** bandwidth for the **Dynamics** parameter. The default value is 3 kHz.

**Ports**

The block has the following ports:

R

Mechanical translational port

+

Positive electrical port

-

Negative electrical port

# ASM Current Controller

Discrete-time PI-based asynchronous machine current control

**Library:** Simscape / Electrical / Power Systems / Control / ASM Control



## Description

The ASM Current Controller implements discrete-time proportional-integral (PI) based asynchronous machine (ASM) current control in the rotor  $d$ - $q$  reference frame. You typically use the ASM Current Controller in a series of blocks that make up a control structure. For example, to convert the  $dq0$  reference frame output voltage to voltage in an  $abc$  reference frame, connect the ASM Current Controller to an Inverse Clarke Transform in the control structure.

## Equations

The block uses the backward Euler discretization method.

Two PI current controllers that are implemented in the rotor reference frame produce the reference voltage vector:

$$v_d^{ref} = \left( K_p_{id} + K_i_{id} \frac{T_s z}{z-1} \right) (i_d^{ref} - i_d) + v_{d\_FF},$$

and

$$v_q^{ref} = \left( K_p_{iq} + K_i_{iq} \frac{T_s z}{z-1} \right) (i_q^{ref} - i_q) + v_{q\_FF},$$

where

- $v_d^{ref}$ , and  $v_q^{ref}$  are the  $d$ -axis and  $q$ -axis reference voltages, respectively.
- $i_d^{ref}$ , and  $i_q^{ref}$  are the  $d$ -axis and  $q$ -axis reference currents, respectively.
- $i_d$  and  $i_q$  are the  $d$ -axis and  $q$ -axis currents, respectively.
- $K_{p\_id}$ , and  $K_{p\_iq}$  are the proportional gains for the  $d$ -axis and  $q$ -axis controllers, respectively.
- $K_{i\_id}$  and  $K_{i\_iq}$  are the integral gains for the  $d$ -axis and  $q$ -axis controllers, respectively.
- $v_{d\_FF}$ , and  $v_{q\_FF}$  are the feedforward voltages for the  $d$ -axis and  $q$ -axis, respectively. The feedforward voltages are obtained from the machine mathematical equations and provided as inputs.
- $T_s$ , is the sample time of the discrete controller.

## Voltage Saturation

Saturation is imposed when the stator voltage vector exceeds the voltage phase limit  $V_{ph\_max}$ :

$$\sqrt{v_d^2 + v_q^2} \leq V_{ph\_max},$$

where  $v_d$ , and  $v_q$  are the  $d$ -axis and  $q$ -axis voltages, respectively.

In the case of axis prioritization, the voltages  $v_1$  and  $v_2$  are introduced, where:

- For  $d$ -axis prioritization —  $v_1 = v_d$  and  $v_2 = v_q$ .
- For  $q$ -axis prioritization —  $v_1 = v_q$  and  $v_2 = v_d$ .

The constrained (saturated) voltages  $v_1^{sat}$  and  $v_2^{sat}$  are obtained as:

$$v_1^{sat} = \min\left(\max\left(v_1^{unsat}, -V_{ph\_max}\right), V_{ph\_max}\right)$$

and

$$v_2^{sat} = \min \left( \max \left( v_2^{unsat}, -V_{2\_max} \right), V_{2\_max} \right),$$

where:

- $v_1^{unsat}$  and  $v_2^{unsat}$  are the unconstrained (unsaturated) voltages.
- $v_{2\_max}$  is the maximum value of  $v_2$  that does not exceed the voltage phase limit. The

equation that define  $v_{2\_max}$  is 
$$v_{2\_max} = \sqrt{(V_{ph\_max})^2 - (v_1^{sat})^2}.$$

In the case of  $d$ - $q$  equivalence, the direct and quadrature axes have the same priority, and the constrained voltages are:

$$v_d^{sat} = \min \left( \max \left( v_d^{unsat}, -V_{d\_max} \right), V_{d\_max} \right)$$

and

$$v_q^{sat} = \min \left( \max \left( v_q^{unsat}, -V_{q\_max} \right), V_{q\_max} \right),$$

where:

$$V_{d\_max} = \frac{V_{ph\_max} |v_d^{unsat}|}{\sqrt{(v_d^{unsat})^2 + (v_q^{unsat})^2}}$$

and

$$V_{q\_max} = \frac{V_{ph\_max} |v_q^{unsat}|}{\sqrt{(v_d^{unsat})^2 + (v_q^{unsat})^2}}.$$

## Integral Anti-Windup

An anti-windup mechanism is employed to avoid the saturation of the integrator output. In such a situation, the integrator gains become:

$$K_{i\_id} + K_{aw\_id} \left( v_d^{sat} - v_d^{unsat} \right)$$

and

$$K_{i\_iq} + K_{aw\_iq} \left( v_q^{sat} - v_q^{unsat} \right),$$

where  $K_{aw\_id}$ ,  $K_{aw\_iq}$ , and  $K_{aw\_if}$  are the anti-windup gains for the  $d$ -axis,  $q$ -axis, and field controllers, respectively.

## Assumptions and Limitations

- The plant model for the direct and quadrature axes can be approximated with a first-order system.

## Ports

### Input

#### **idqRef — Reference currents**

vector

Desired  $d$ - and  $q$ -axis currents for control of the asynchronous motor, in A.

Data Types: `single` | `double`

#### **idq — Measured currents**

vector

Actual  $d$ - and  $q$ -axis currents of the controlled asynchronous motor, in A.

Data Types: `single` | `double`

#### **vdqFF — Feedforward voltages**

vector

Feedforward pre-control voltages, in V.

Data Types: `single` | `double`

**VphMax — Maximum phase voltage**

scalar

Maximum allowable voltage in each phase, in V.

Data Types: `single` | `double`

**Reset — External reset**

scalar

External reset signal (rising edge) for integrators.

Data Types: `Boolean`

## Output

**vdqRef — Reference voltages**

vector

Desired  $d$ - and  $q$ -axis voltages for control of the asynchronous motor, in V.

Data Types: `single` | `double`

## Parameters

### Control Parameters

**D-axis current proportional gain —  $d$ -axis proportional gain**

1 (default)

Proportional gain for direct-axis current control.

**D-axis current integral gain —  $d$ -axis integral gain**

100 (default)

Integral gain for direct-axis current control.

**D-axis current anti-windup gain —  $d$ -axis anti-windup gain**

1 (default)

Anti-windup gain for direct-axis current control.



**Q-axis current proportional gain —  $q$ -axis proportional gain**

1 (default)

Proportional gain for quadrature-axis current control.

**Q-axis current integral gain —  $q$ -axis integral gain**

100 (default)

Integral gain for quadrature-axis current control.

**Q-axis current anti-windup gain —  $q$ -axis anti-windup gain**

1 (default)

Anti-windup gain for quadrature-axis current control.

**Sample time (-1 for inherited) — Block sample time**

-1 (default) | positive scalar

Time, in s, between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

If this block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly using a positive scalar.

**Axis prioritization — Axis prioritization for voltage limiter**

Q-axis (default) | D-axis | D-Q equivalence

Prioritize or maintain the ratio between the  $d$ - and  $q$ -axes when the block limits voltage.

**Enable pre-control voltage — Pre-control voltage**

on (default) | off

Enable or disable pre-control voltage.

## See Also

**Blocks**

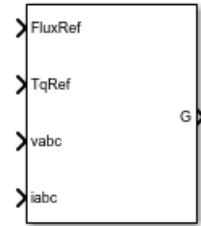
ASM Current Controller | ASM Direct Torque Control | ASM Field-Oriented Control | ASM Flux Observer | ASM Scalar Control

**Introduced in R2017b**

# ASM Direct Torque Control

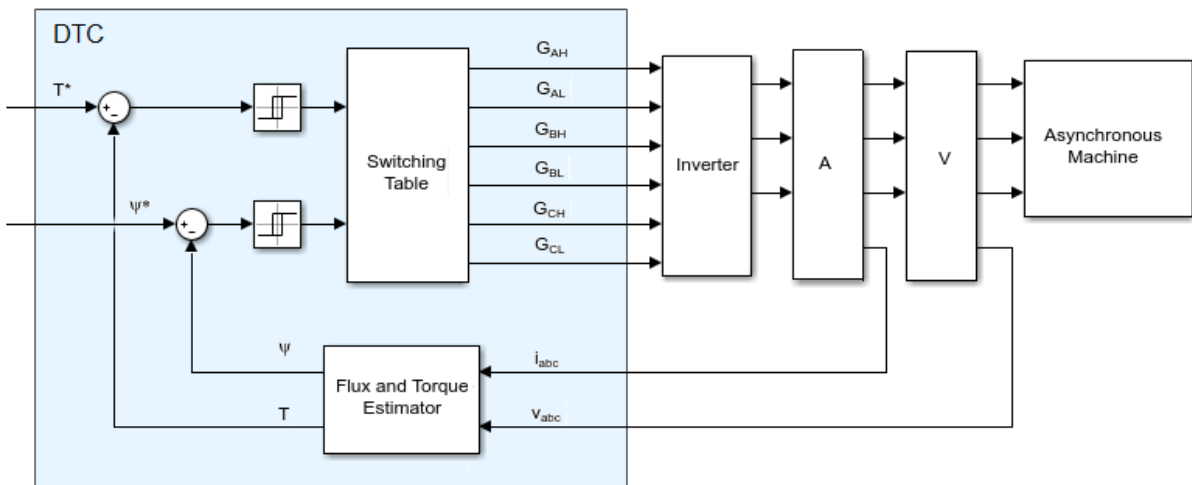
Asynchronous machine direct torque control

**Library:** Simscape / Electrical / Power Systems / Control / ASM Control



## Description

The ASM Direct Torque Control implements an asynchronous machine (ASM) direct torque control (DTC) structure. The figure shows the equivalent circuit for the ASM DTC.



## Equations

To estimate the torque and flux, the block discretizes the machine voltage equations in the stationary  $\alpha\beta$  reference frame using the backward Euler method. The discrete-time equations for stator fluxes in the  $\alpha\beta$  frame are:

$$\psi_{\alpha} = (v_{\alpha} - i_{\alpha} R_s) \frac{T_s z}{z - 1}$$

and

$$\psi_{\beta} = (v_{\beta} - i_{\beta} R_s) \frac{T_s z}{z - 1}$$

where:

- $v_{\alpha}$  is  $\alpha$ -axis voltage.
- $i_{\alpha}$  is  $\alpha$ -axis current.
- $R_s$  is the stator resistance.
- $\Psi_{\alpha}$  is the  $\alpha$ -axis stator flux.
- $v_{\beta}$  is  $\beta$ -axis voltage.
- $i_{\beta}$  is  $\beta$ -axis current.
- $\Psi_{\beta}$  is the  $\beta$ -axis stator flux.

The block calculates the torque and flux as:

$$T = \frac{3p}{2} (\psi_{\alpha} i_{\beta} - \psi_{\beta} i_{\alpha})$$

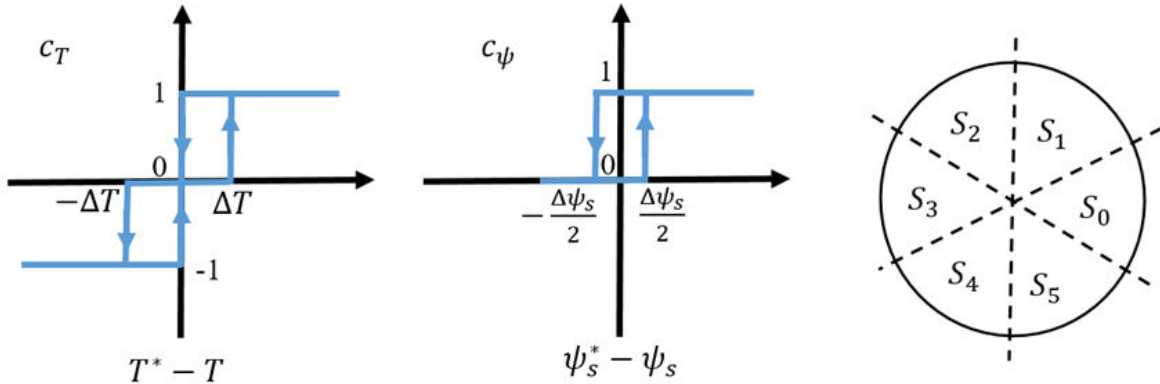
and

$$\psi_s = \sqrt{\psi_{\alpha}^2 + \psi_{\beta}^2}$$

where:

- $p$  is the number of pole pairs.
- $\Psi_s$  is the stator flux.

To detect flux and torque estimation errors, the block uses hysteresis comparators. The figure shows hysteresis comparators and the associated switching sectors.



The table shows the optimum switching for an inverter high-side system.

$c_\psi, c_T S(\theta)$		$S_0$	$S_1$	$S_2$	$S_3$	$S_4$	$S_5$
$c_\psi = 1$	$c_T = 1$	1, 1, 0	0, 1, 0	0, 1, 1	0, 0, 1	1, 0, 1	1, 0, 0
	$c_T = 0$	1, 1, 1	0, 0, 0	1, 1, 1	0, 0, 0	1, 1, 1	0, 0, 0
	$c_T = -1$	1, 0, 1	1, 0, 0	1, 1, 0	0, 1, 0	0, 1, 1	0, 0, 1
$c_\psi = 0$	$c_T = 1$	0, 1, 0	0, 1, 1	0, 0, 1	1, 0, 1	1, 0, 0	1, 1, 0
	$c_T = 0$	0, 0, 0	1, 1, 1	0, 0, 0	1, 1, 1	0, 0, 0	1, 1, 1
	$c_T = -1$	0, 0, 1	1, 0, 1	1, 0, 0	1, 1, 0	0, 1, 0	0, 1, 1

## Assumptions and Limitations

- The power inverter dead times are not considered. For hardware implementation, add the dead time externally.

## Ports

### Input

#### **FluxRef — Flux**

scalar

Reference stator flux.

Data Types: `single` | `double`

#### **TqRef — Torque**

scalar

Reference torque.

Data Types: `single` | `double`

#### **vabc — Voltage**

vector

Stator phase voltages.

Data Types: `single` | `double`

#### **iabc — Current**

vector

Stator phase currents.

Data Types: `single` | `double`

### Output

#### **G — Gate pulses**

vector | 0 or 1

Inverter gate pulses. The block does not consider any dead time.

Data Types: `single` | `double`

## Parameters

### **Stator resistance (Ohm) — Resistance**

0.25 (default) | positive scalar

Resistance of the machine stator.

### **Number of pole pairs — Pole number**

1 (default) | positive integer

Number of machine pole pairs.

### **Flux hysteresis bandwidth (Wb) — Flux**

0.02 (default) | positive scalar

Total bandwidth distributed symmetrically around the flux set point.

### **Torque hysteresis bandwidth (N\*m) — Torque**

10 (default) | positive scalar

Total bandwidth distributed symmetrically around the set point.

### **Sample time (-1 for inherited) — Block sample time**

-1 (default) | positive scalar

Time, in s, between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

If this block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly using a positive scalar.

## References

- [1] Takahashi, I., and T. Noguchi. "A New Quick-Response and High-Efficiency Control Strategy of an Induction Motor." *IEEE Transactions on Industry Applications*. Vol. IA-22, Number 5, 1986, pp. 820 - 827.

## **See Also**

### **Blocks**

ASM Current Controller | ASM Field-Oriented Control | ASM Flux Observer | ASM Scalar Control

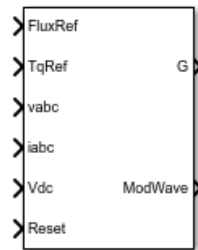
**Introduced in R2017b**



# ASM Direct Torque Control with Space Vector Modulator

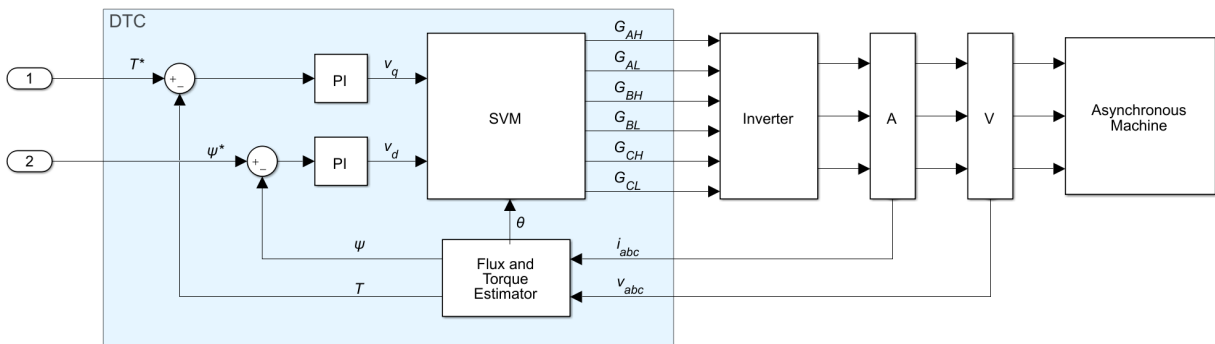
Asynchronous machine direct torque control structure with space vector modulator

**Library:** Simscape / Electrical / Power Systems / Control / ASM Control



## Description

The ASM Direct Torque Control with Space Modulator implements an asynchronous machine (ASM) direct torque control structure (DTC) with space vector modulator (SVM). Use this block to generate the gate pulses for an inverter controlling an ASM. This diagram shows the architecture of the block.



In the diagram:

- You provide the reference torque,  $T^*$ , and flux,  $\psi^*$ .

- The Flux and Torque Estimator estimates the actual torque,  $T$ , and flux,  $\psi$  from the measured phase currents,  $i_{abc}$ , and voltages,  $v_{abc}$ .
- Two PI controllers determine the reference  $d$  and  $q$  voltages,  $v_d$  and  $v_q$ , from the flux and torque errors, respectively.
- The SVM generates the gates pulses,  $G_{ij}$ , required to control an inverter driving the asynchronous machine. Subscript  $i$  corresponds to the phase ( $a$ ,  $b$ , or  $c$ ). Subscript  $j$  corresponds to the high,  $H$ , or low,  $L$ , signal.

## Flux and Torque Estimator

To estimate the torque and flux, the block discretizes the machine voltage equations in the stationary  $\alpha\beta$  reference frame using the backward Euler method. The discrete-time equations for stator fluxes in the  $\alpha\beta$  frame are:

$$\psi_{\alpha} = (v_{\alpha} - i_{\alpha}R_s) \frac{T_s z}{z-1}$$

$$\psi_{\beta} = (v_{\beta} - i_{\beta}R_s) \frac{T_s z}{z-1}$$

Where:

- $v_{\alpha}$  and  $v_{\beta}$  are the  $\alpha$ - and  $\beta$ -axis voltages, respectively.
- $i_{\alpha}$  and  $i_{\beta}$  are the  $\alpha$ - and  $\beta$ -axis currents, respectively.
- $\Psi_{\alpha}$  and  $\Psi_{\beta}$  are the  $\alpha$ - and  $\beta$ -axis stator fluxes, respectively.
- $R_s$  is the stator resistance.

The block calculates the torque and total stator flux as:

$$T = \frac{3p}{2} (\psi_{\alpha} i_{\beta} - \psi_{\beta} i_{\alpha})$$

$$\psi_s = \sqrt{\psi_{\alpha}^2 + \psi_{\beta}^2}$$

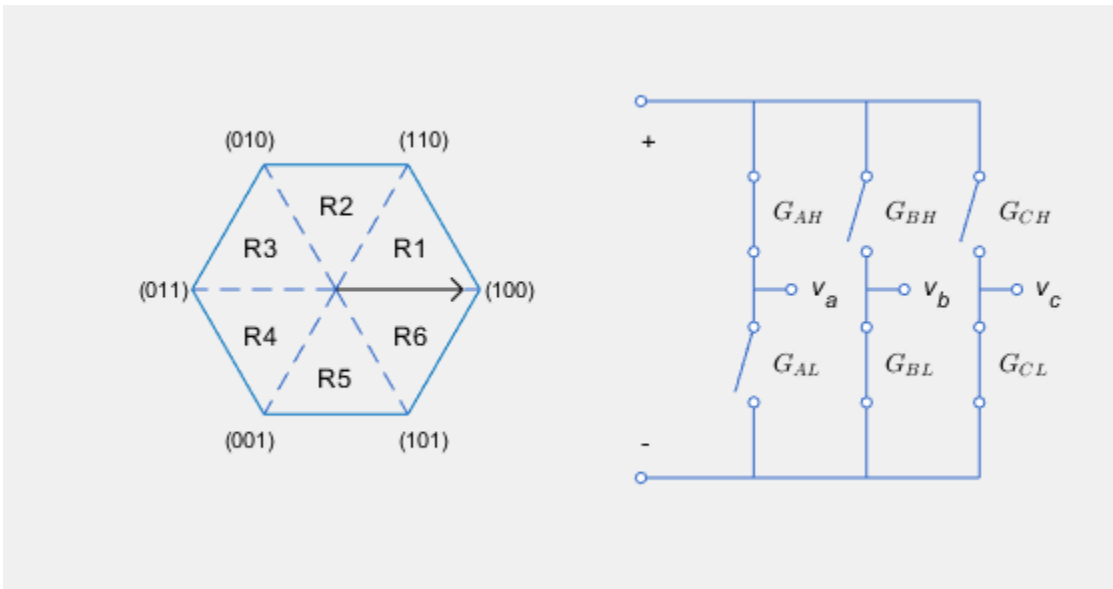
Where:

- $p$  is the number of pole pairs.

- $\Psi_s$  is the total stator flux.

## Space Vector Modulator

The SVM converts the desired voltages into gate pulses, which you use to control an inverter. This figure shows possible switching states of a three-phase inverter.



The hexagon represents the space vector diagram. Each of the six vertices represents a possible switching state ( $G_{AH}, G_{BH}, G_{CH}$ ) of the three-phase inverter. Each low gate takes the opposite state as its corresponding high gate. The inverter diagram illustrates the current state.

The rotating vector in the space vector diagram corresponds to the complex reference voltage vector, which rotates at the desired electrical frequency of the machine. In reality, the switching frequency is much faster than this electrical frequency. As a result, the inverter switches continually between the two states enclosing its current region  $R_i$ , and the zero state corresponding to  $(0,0,0)$ , to generate the desired voltages.

To learn about the implementation of this method, see the PWM Generator (Three-phase, Two-level) block.

## Ports

### Input

#### **FluxRef — Flux**

scalar

Reference stator flux.

Data Types: `single` | `double`

#### **TqRef — Torque**

scalar

Reference torque.

Data Types: `single` | `double`

#### **vabc — Voltage**

vector

Stator phase voltages.

Data Types: `single` | `double`

#### **iabc — Current**

vector

Stator phase currents.

Data Types: `single` | `double`

#### **Vdc — DC-link voltage signal**

scalar

DC-link voltage for the converter.

Data Types: `single` | `double`

#### **Reset — Controller reset**

scalar

Reset for the PI controller integrators.

Data Types: single | double

## Output

### **G — Gate pulses**

vector | 0 or 1

Inverter gate pulses. The block does not consider any dead time.

Data Types: single | double

### **ModWave — Modulation wave**

vector

Modulation wave you deploy to the hardware if you are generating code for a platform with PWM-capable hardware. Otherwise, this data is only for your reference.

## Parameters

### General

#### **Stator resistance (Ohm) — Resistance**

0.25 (default) | positive scalar

Resistance of the machine stator.

#### **Number of pole pairs — Pole number**

1 (default) | positive integer

Number of machine pole pairs.

#### **Inverter dc-link voltage threshold (V) — Voltage**

300 (default) | positive scalar

Voltage threshold to activate the power inverter.

#### **Fundamental sample time (s) — SVM sample time**

5e-6 (default) | positive scalar less than the control sample time

Sample time for the space vector modulator. Fundamental sample time must be less than the control sample time.

**Control sample time (s) — PI sample time**

5e-5 (default) | positive scalar greater than the fundamental sample time

Sample time for PI controllers. Control sample time must be greater than the fundamental sample time.

**Switching frequency (Hz) — Switching rate**

1000 (default) | positive scalar

Specify the rate at which you want the switches in the power converter to switch.

## **Control Parameters**

**Flux controller proportional gain — Flux PI proportional gain**

150 (default) | positive scalar

Proportional gain for the flux controller.

**Flux controller integral gain — Flux PI integral gain**

3000 (default) | positive scalar

Integral gain for the flux controller.

**Flux controller anti-windup gain — Flux PI anti-windup gain**

1 (default) | positive scalar

Anti-windup gain for the flux controller.

**Torque controller proportional gain — Torque PI proportional gain**

1 (default) | positive scalar

Proportional gain for the torque controller.

**Torque controller integral gain — Torque PI integral gain**

50 (default) | positive scalar

Integral gain for the torque controller.

**Torque controller anti-windup gain — Torque PI anti-windup gain**

1 (default) | positive scalar

Anti-windup gain for the torque controller.

### **Axis prioritization — Axis prioritization for voltage limiter**

Q-axis (default) | D-axis | D-Q equivalence

Prioritize or maintain ratio between  $d$ - and  $q$ -axes when the block limits voltage.

### **References**

- [1] Buja, G. S., and M. P. Kazmierkowski. "Direct Torque Control of PWM Inverter-Fed AC Motors—A Survey." *IEEE Transactions on Industrial Electronics* 51, no. 4, (2004): 744 - 757.

### **See Also**

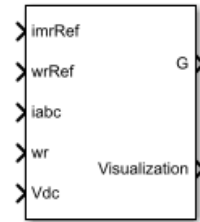
PWM Generator (Three-phase, Two-level) | ASM Direct Torque Control

**Introduced in R2018a**

# ASM Field-Oriented Control

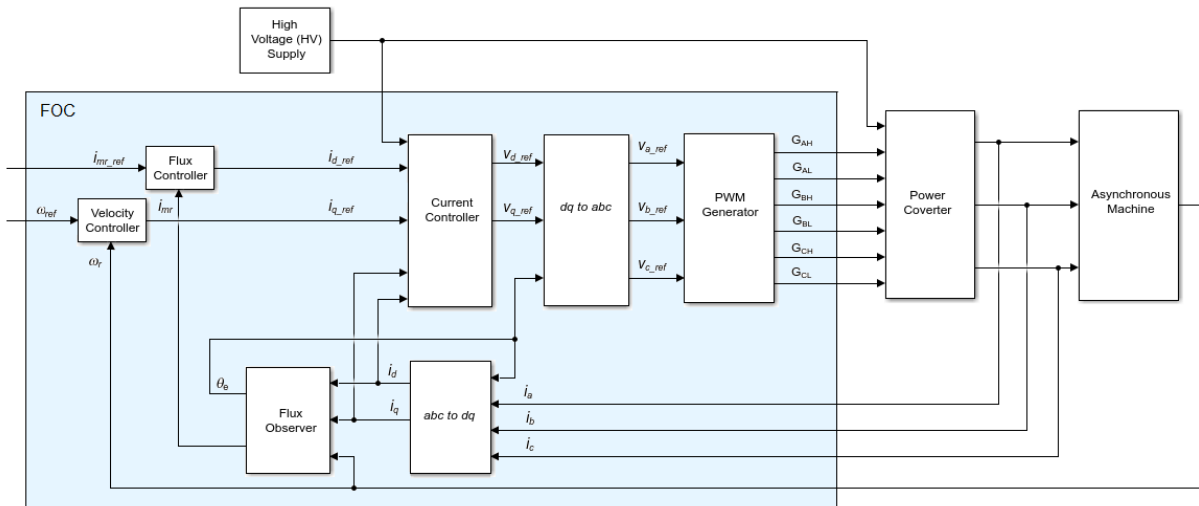
Asynchronous machine field-oriented control

**Library:** Simscape / Electrical / Power Systems / Control / ASM Control



## Description

The ASM Field-Oriented Controller block implements an asynchronous machine (ASM) field-oriented control (FOC) structure using the per-unit system. To decouple the torque and flux, FOC uses the rotor  $d$ - $q$  reference frame. The figure shows the control structure.



In the diagram:



- $\omega_r$  is the measured angular velocity.
- $\omega_{ref}$  is the reference angular velocity.
- $i_d$  and  $i_q$  are the  $d$ - and  $q$ -axis stator currents.
- $i_a$ ,  $i_b$ , and  $i_c$  are the  $a$ -,  $b$ - and  $c$ -phase stator winding currents.
- $i_{mr\_ref}$  is the reference magnetizing current.
- $i_{mr}$  is the magnetizing current.
- $v_d$  and  $v_q$  are the  $d$ - and  $q$ -axis stator voltages.
- $v_a$ ,  $v_b$ , and  $v_c$  are the  $a$ -,  $b$ - and  $c$ -phase stator winding voltages.
- $\theta_e$  is the rotor electrical angle.
- $G_{AH}$ ,  $G_{AL}$ ,  $G_{BH}$ ,  $G_{BL}$ ,  $G_{CH}$ , and  $G_{CL}$  are the  $a$ -,  $b$ - and  $c$ -phase high (H) and low(L) gate pulses.

## Assumptions and Limitations

- The machine parameters are known.
- The implementation uses the per-unit system.
- The control structure implementation uses a single sample rate.

## Ports

### Input

#### **imRef — Current**

scalar

Magnetizing reference current in the per-unit system.

Data Types: `single` | `double`

#### **wrRef — Velocity**

scalar

Rotor reference velocity in per-unit system.

Data Types: `single` | `double`

## **iabc — Current**

vector

Measured phase currents in the per-unit system.

Data Types: single | double

## **wr — Velocity**

scalar

Measured angular velocity in per-unit system.

Data Types: single | double

## **Vdc — Voltage**

scalar

Measured dc-link voltage, in V.

Data Types: single | double

## **Output**

### **G — Gate pulses**

vector | 0 or 1

Inverter gate pulses. The block does not consider any dead time.

Data Types: single | double

### **Visualization — Visualization signals**

vector

Bus containing signals for visualization.

Data Types: single | double | bus

# Parameters

## General

### **Rated voltage, rms line-to-line (V) – Voltage**

550 (default)

Nominal voltage.

### **Rated electrical frequency (Hz) – Frequency**

60 (default)

Nominal electrical frequency.

### **Rotor resistance, referred to the stator side (pu) – Resistance**

0.01 (default)

Rotor, stator-side resistance in the per-unit system.

### **Rotor leakage inductance, referred to the stator side (pu) – Inductance**

0.06 (default)

Rotor stator-side leakage inductance, in the pu-unit system.

### **Magnetizing inductance (pu) – Inductance**

2.7 (default)

Magnetizing inductance in the per-unit system.

### **Time constant for dq currents filters (s) – Time constant**

1e-4 (default)

Time constant for filtering the  $d$  and  $q$  currents.

### **Inverter dc-link voltage threshold (V) – Voltage**

500 (default)

Voltage threshold to activate the power inverter.

### **Fundamental sample time (s) – Time**

5e-6 (default) | positive scalar less than the control sample time

Fundamental sample time must be less than the control sample time.

**Control sample time (s) — Time**

5e-5 (default) | positive scalar greater than the fundamental sample time

Control sample time must be greater than the fundamental sample time.

**Outer Loop**

**Magnetizing current controller proportional gain — Gain**

10 (default)

Proportional gain for the magnetizing current controller.

**Magnetizing current controller integral gain — Gain**

1000 (default)

Integral gain for the magnetizing current controller.

**Magnetizing current controller integral anti-windup gain — Gain**

1000 (default)

Integral anti-windup gain for the magnetizing current controller.

**Speed controller proportional gain — Gain**

10 (default)

Proportional gain for the speed controller.

**Speed controller integral gain — Gain**

1000 (default)

Integral gain for the speed controller.

**Speed controller integral anti-windup gain — Gain**

1000 (default)

Integral anti-windup gain for the speed controller.

**Maximum d-axis current [pu] — Current**

2 (default)

Maximum current for the *d*-axis.

**Maximum q-axis current [pu] – Current**

2 (default)

Maximum current for the  $q$ -axis.**Inner Loop****Phase-a axis alignment –  $dq0$  reference frame alignment**

Q-axis (default) | D-axis

Align the  $a$ -phase vector of the  $abc$  reference frame to the  $d$ - or  $q$ -axis of the rotating reference frame.**D-axis current proportional gain – D-axis proportional gain**

1 (default) | positive number

Proportional gain of the PI controller used for direct-axis current control.

**D-axis current integral gain – D-axis integral gain**

100 (default) | positive number

Integrator gain of the PI controller used for direct-axis current control.

**D-axis current anti-windup gain – D-axis anti-windup gain**

1 (default) | positive number

Anti-windup gain of the PI controller used for direct-axis current control.

**Q-axis current proportional gain – Q-axis proportional gain**

1 (default) | positive number

Proportional gain of the PI controller used for quadrature-axis current control.

**Q-axis current integral gain – Q-axis integral gain**

100 (default) | positive number

Integrator gain of the PI controller used for quadrature-axis current control.

**Q-axis current anti-windup gain – Q-axis anti-windup gain**

1 (default) | positive number

Anti-windup gain of the PI controller used for quadrature-axis current control.

**Axis prioritization — Axis prioritization for voltage limiter**

Q-axis (default) | D-axis | D-Q equivalence

Prioritize or maintain ratio between  $d$ - and  $q$ -axes when block limits voltage.

**PWM**

**PWM method — Pulse width modulation method**

SVM: space vector modulation (default) | SPWM: sinusoidal PWM

Specify the waveform technique.

**Sampling mode — Wave-sampling method**

Natural (default) | Asymmetric | Symmetric

The sampling mode determines whether the block samples the modulation waveform when the waves intersect or when the carrier wave is at one or both of its boundary conditions.

**Switching frequency (Hz) — Switching rate**

1000 (default) | positive integer

Specify the rate at which you want the switches in the power converter to switch.

## See Also

**Blocks**

ASM Current Controller | ASM Direct Torque Control | ASM Flux Observer | ASM Scalar Control

**Introduced in R2017b**

# ASM Flux Observer

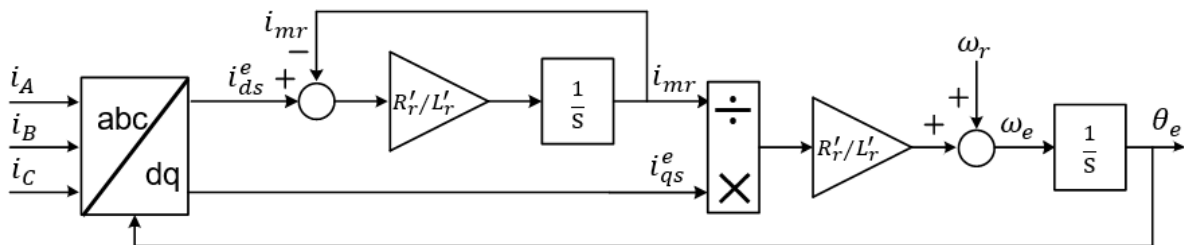
Asynchronous machine flux observer

**Library:** Simscape / Electrical / Power Systems / Control / Observers



## Description

The ASM Flux Observer block obtains the synchronous speed,  $\omega_e$ , and electrical angle,  $\theta_e$ , that are required for performing rotor field-oriented control (FOC). The figure shows the equivalent circuit for the observer.



## Equations

To determine the synchronous speed and electrical angle, the ASM Flux Observer block uses these relationships:

$$\lambda_{dr}^e = L_r' i_{dr}^e + L_M i_{ds}^e \triangleq L_M i_{mr},$$

$$0 = R_r' i_{dr}^e - (\omega_e - \omega_r) \lambda_{qr}^e + \frac{d\lambda_{dr}^e}{dt},$$

and

$$i'_{qr}{}^e = -\frac{L_M}{L_r'} i_{qs}{}^e,$$

in these combined forms:

$$i'_{dr}{}^e = \frac{L_M}{L_r'} (i_{mr} - i_{ds}{}^e)$$

$$\frac{di_{mr}}{dt} = \frac{R_r'}{L_r'} (i_{ds}{}^e - i_{mr})$$

and

$$\omega_e = \omega_r + \frac{R_r'}{L_r'} \frac{i_{qs}{}^e}{i_{mr}}$$

where:

- $\lambda'_{dr}{}^e$  is the  $d$ -axis rotor flux.
- $i_{mr}$  is the magnetizing current.
- $i_{ds}{}^e$  and  $i_{qs}{}^e$  are the  $d$ -axis and  $q$ -axis stator currents.
- $i'_{dr}{}^e$  and  $i'_{qr}{}^e$  are the  $d$ -axis and  $q$ -axis rotor currents.
- $\omega_e$  is the synchronous speed.
- $\omega_r$  is the mechanical rotational speed.
- $R_r'$  is the rotor resistance, referred to the stator side.
- $L_r'$  is the rotor leakage inductance, referred to the stator side.



- $L_M$  the magnetizing inductance.

## Ports

### Input

#### **iabc — Current**

vector

Measured stator currents in the per-unit system.

Data Types: `single` | `double`

#### **wr — Speed**

scalar

Measured rotational speed.

Data Types: `single` | `double`

### Output

#### **idqseF — Current**

vector

Filtered  $d$ -axis and  $q$ -axis stator currents in the synchronous reference frame.

Data Types: `single` | `double`

#### **imr — Current**

scalar

Magnetizing rotor current.

Data Types: `single` | `double`

#### **theta — Electrical angle**

scalar

Rotor electrical angle.

Data Types: single | double

**we — Synchronous speed**

scalar

Rotor synchronous speed.

Data Types: single | double

## Parameters

**Rated electrical frequency (Hz) — Frequency**

60 (default) | positive

Machine rated electrical frequency.

**Rotor resistance, referred to the stator side (pu) — Resistance**

0.01 (default) | positive

Rotor resistance, referred to the stator side, in the per-unit system.

**Rotor leakage inductance, referred to the stator side (pu) — Inductance**

0.06 (default) | positive

Rotor leakage inductance, referred to the stator side, in the per-unit system.

**Magnetizing inductance (pu) — Inductance**

2.7 (default) | positive

Magnetizing inductance in the per-unit system.

**Time constant for dq currents filters (s) — Time constant**

1e-4 (default) | 0 or positive

Time constant used to low-pass filter the  $d$ - $q$  currents.

**Sample time (-1 for inherited) — Sampling interval**

-1 (default) | default value or a positive number

Time interval between samples. If the block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step

model, specify the sample time explicitly. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

## References

- [1] Vas, P. *Electrical Machines and Drives: A Space-vector Theory Approach*. New York: Oxford University Press, 1992.

## See Also

### Blocks

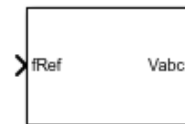
ASM Current Controller | ASM Direct Torque Control | ASM Field-Oriented Control | ASM Scalar Control

**Introduced in R2017b**

## ASM Scalar Control

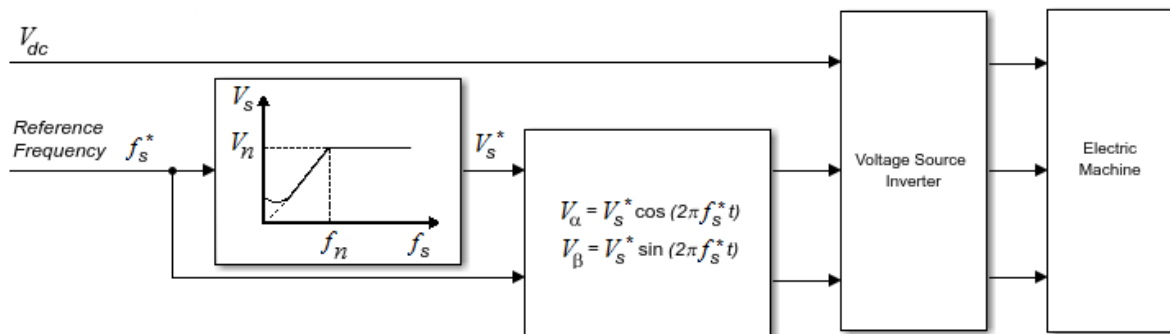
Asynchronous machine V/f control

**Library:** Simscape / Electrical / Power Systems / Control / ASM Control



### Description

The ASM Scalar Control block implements an asynchronous machine (ASM) scalar, that is V/f or V/Hz, control structure. The diagram shows the open-loop V/f control structure that the block implements.



### Equations

The ASM Scalar Control block computes the magnitude of the stator voltage based on the reference frequency,  $f_s^*$ , as:

$$V_s^* = \left( \frac{V_n - V_{min}}{f_n - f_{min}} \right) f_s^*$$

where:

- $V_n$  is the rated voltage.
- $V_{min}$  is the minimum voltage.
- $f_n$  is the rated electrical frequency.
- $f_{min}$  is the minimum frequency.

The voltage components in the stationary reference frame are:

$$V_\alpha = V_s^* \cos(2\pi f_s^* t)$$

and

$$V_\beta = V_s^* \sin(2\pi f_s^* t).$$

The block obtains  $V_{abc}$  from  $V_\alpha$  and  $V_\beta$  by using an inverse Clarke transformation.

## Ports

### Input

#### **fRef — Frequency**

scalar

Reference electrical frequency.

Example: Example

Data Types: `single` | `double`

### Output

#### **Vabc — Voltage**

vector

Reference phase voltages.

Example: Example

Data Types: single | double

## Parameters

### **Rated electrical frequency (Hz) — Frequency**

60 (default) | positive scalar

Nominal frequency.

### **Rated voltage (V) — Voltage**

550 (default) | positive and greater than the value of the **Minimum voltage (V)** parameter

Nominal voltage.

### **Minimum voltage (V) — Voltage**

10 (default) | zero or positive

Lower bound for the voltage.

### **Sample time (-1 for inherited) — Block sample time**

-1 (default) | positive scalar

Time, in s, between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

If this block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly using a positive scalar.

## See Also

### **Blocks**

ASM Current Controller | ASM Direct Torque Control | ASM Field-Oriented Control | ASM Flux Observer

### **Introduced in R2017b**

# Asynchronous Machine Measurement

Per-unit measurement from asynchronous machine



## Library

Simscape / Electrical / Power Systems / Machines

## Description

The Asynchronous Machine Measurement block outputs a per-unit measurement associated with a connected Asynchronous Machine Squirrel Cage or Asynchronous Machine Wound Rotor block. The input of the Asynchronous Machine Measurement block connects to the pu output port of the asynchronous machine block.

You set the **Output** parameter to a per-unit measurement associated with the asynchronous machine. Based on the value you select, the Asynchronous Machine Measurement block:

- Directly outputs the value of an element in the input signal vector
- Calculates the per-unit measurement by using values of elements in the input signal vector in mathematical expressions

The Asynchronous Machine Measurement block outputs a per-unit measurement from the asynchronous machine according to the output value expressions in the table. For example, when you set **Output** to **Stator d-axis voltage**, the block directly outputs the value of the pu\_vds element in the input signal vector. However, when you set **Output** to **Slip**, the block calculates the slip value by subtracting the value of the pu\_velocity element from 1.

<b>Output Parameter Setting</b>	<b>Output Value Expression</b>
Electrical torque	pu_torque
Rotor velocity	pu_velocity
Stator d-axis voltage	pu_vds
Stator q-axis voltage	pu_vqs
Stator zero-sequence voltage	pu_v0s
Stator d-axis current	pu_ids
Stator q-axis current	pu_iqs
Stator zero-sequence current	pu_i0s
Slip	1-pu_velocity
Apparent power	$\sqrt{pu\_Pt^2 + pu\_Qt^2}$
Real power	$pu\_Pt = (pu\_vds*pu\_ids) + (pu\_vqs*pu\_iqs) + 2(pu\_v0s*pu\_i0s)$
Reactive power	$pu\_Qt = (pu\_vqs*pu\_ids) - (pu\_vds*pu\_iqs)$
Terminal voltage	$\sqrt{pu\_vds^2 + pu\_vqs^2}$
Terminal current	$\sqrt{pu\_ids^2 + pu\_iqs^2}$
Power factor angle (rad)	power_factor_angle = atan2(pu_Qt, pu_Pt)
Power factor	cos(power_factor_angle)

## Parameters

### Output

Per-unit measurement from asynchronous machine. The default value is Electrical torque.

### Ports

The block has the following ports:



pu

Physical signal vector port associated with per-unit measurements from a connected asynchronous machine. The vector elements are:

- pu\_torque
- pu\_velocity
- pu\_vds
- pu\_vqs
- pu\_v0s
- pu\_ids
- pu\_iqs
- pu\_i0s

o

Per-unit measurement output port.

## See Also

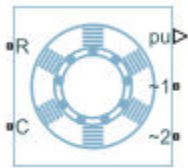
Asynchronous Machine Squirrel Cage (fundamental)

Asynchronous Machine Wound Rotor (fundamental)

**Introduced in R2013b**

# Asynchronous Machine Squirrel Cage (fundamental)

Squirrel-cage-rotor asynchronous machine with fundamental parameterization



## Library

Simscape / Electrical / Power Systems / Machines / Asynchronous Machine (Squirrel Cage)

## Description

The Asynchronous Machine Squirrel Cage (fundamental) block models a squirrel-cage-rotor asynchronous machine using fundamental parameters. A squirrel-cage-rotor asynchronous machine is a type of induction machine. All stator connections are accessible on the block. Therefore, you can model soft-start regimes using a switch between wye and delta configurations. If you need access to the rotor windings, use the Asynchronous Machine Wound Rotor (fundamental) or the Asynchronous Machine Wound Rotor (fundamental, SI) block instead.

Connect port ~1 to a three-phase circuit. To connect the stator in delta configuration, connect a Phase Permute block between ports ~1 and ~2. To connect the stator in wye configuration, connect port ~2 to a Grounded Neutral or a Floating Neutral block.

## Equations

The asynchronous machine equations are expressed with respect to a synchronous reference frame, defined by

$$\theta_e(t) = \int_0^t 2\pi f_{rated} dt,$$

where  $f_{rated}$  is the value of the **Rated electrical frequency** parameter.

The Park transformation maps stator equations to a reference frame that is stationary with respect to the rated electrical frequency. The Park transformation is defined by

$$P_s = \frac{2}{3} \begin{bmatrix} \cos \theta_e & \cos(\theta_e - \frac{2\pi}{3}) & \cos(\theta_e + \frac{2\pi}{3}) \\ -\sin \theta_e & -\sin(\theta_e - \frac{2\pi}{3}) & -\sin(\theta_e + \frac{2\pi}{3}) \\ \frac{1}{2} & \frac{1}{2} & \frac{1}{2} \end{bmatrix},$$

where  $\theta_e$  is the electrical angle.

The Park transformation is used to define the per-unit asynchronous machine equations. The stator voltage equations are defined by

$$v_{ds} = \frac{1}{\omega_{base}} \frac{d\psi_{ds}}{dt} - \omega \psi_{qs} + R_s i_{ds},$$

$$v_{qs} = \frac{1}{\omega_{base}} \frac{d\psi_{qs}}{dt} + \omega \psi_{ds} + R_s i_{qs},$$

and

$$v_{0s} = \frac{1}{\omega_{base}} \frac{d\psi_{0s}}{dt} + R_s i_{0s},$$

where:

- $v_{ds}$ ,  $v_{qs}$ , and  $v_{0s}$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator voltages, defined by

$$\begin{bmatrix} v_{ds} \\ v_{qs} \\ v_{0s} \end{bmatrix} = P_s \begin{bmatrix} v_a \\ v_b \\ v_c \end{bmatrix}.$$

$v_a$ ,  $v_b$ , and  $v_c$  are the stator voltages across ports ~1 and ~2.

- $\omega_{base}$  is the per-unit base electrical speed.
- $\psi_{ds}$ ,  $\psi_{qs}$ , and  $\psi_{0s}$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator flux linkages.
- $R_s$  is the stator resistance.
- $i_{ds}$ ,  $i_{qs}$ , and  $i_{0s}$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator currents defined by

$$\begin{bmatrix} i_{ds} \\ i_{qs} \\ i_{0s} \end{bmatrix} = P_s \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix}.$$

$i_a$ ,  $i_b$ , and  $i_c$  are the stator currents flowing from port ~1 to port ~2.

The rotor voltage equations are defined by

$$v_{dr} = \frac{1}{\omega_{base}} \frac{d\psi_{dr}}{dt} - (\omega - \omega_r)\psi_{qr} + R_{rd}i_{dr} = 0$$

and

$$v_{qr} = \frac{1}{\omega_{base}} \frac{d\psi_{qr}}{dt} + (\omega - \omega_r)\psi_{dr} + R_{rd}i_{qr} = 0,$$

where:

- $v_{dr}$  and  $v_{qr}$  are the  $d$ -axis and  $q$ -axis rotor voltages.
- $\psi_{dr}$  and  $\psi_{qr}$  are the  $d$ -axis and  $q$ -axis rotor flux linkages.
- $\omega$  is the per-unit synchronous speed. For a synchronous reference frame, the value is 1.
- $\omega_r$  is the per-unit mechanical rotational speed.
- $R_{rd}$  is the rotor resistance referred to the stator.

- $i_{dr}$  and  $i_{qr}$  are the  $d$ -axis and  $q$ -axis rotor currents.

The stator flux linkage equations are defined by

$$\psi_{ds} = L_{ss}i_{ds} + L_m i_{dr},$$

$$\psi_{qs} = L_{ss}i_{qs} + L_m i_{qr},$$

and

$$\psi_{0s} = L_{ss}i_{0s},$$

where  $L_{ss}$  is the stator self-inductance and  $L_m$  is the magnetizing inductance.

The rotor flux linkage equations are defined by

$$\psi_{dr} = L_{rrd}i_{dr} + L_m i_{ds}$$

and

$$\psi_{qr} = L_{rrd}i_{qr} + L_m i_{qs},$$

where  $L_{rrd}$  is the rotor self-inductance referred to the stator.

The rotor torque is defined by

$$T = \psi_{ds}i_{qs} - \psi_{qs}i_{ds}.$$

The stator self-inductance  $L_{ss}$ , stator leakage inductance  $L_{ls}$ , and magnetizing inductance  $L_m$  are related by

$$L_{ss} = L_{ls} + L_m.$$

The rotor self-inductance  $L_{rrd}$ , rotor leakage inductance  $L_{lrd}$ , and magnetizing inductance  $L_m$  are related by

$$L_{rrd} = L_{lrd} + L_m.$$

When a saturation curve is provided, the equations to determine the saturated magnetizing inductance as a function of magnetizing flux are:

$$L_{m\_sat} = f(\psi_m)$$

$$\psi_m = \sqrt{\psi_{dm}^2 + \psi_{qm}^2}$$

For no saturation, the equation reduces to

$$L_{m\_sat} = L_m$$

## Plotting and Display Options

You can perform plotting and display actions using the **Electrical** menu on the block context menu.

Right-click the block and, from the **Electrical** menu, select an option:

- **Display Base Values** — Displays the machine per-unit base values in the MATLAB® Command Window.
- **Plot Torque Speed (SI)** — Plots torque versus speed, both measured in SI units, in a MATLAB figure window using the current machine parameters.
- **Plot Torque Speed (pu)** — Plots torque versus speed, both measured in per-unit, in a MATLAB figure window using the current machine parameters.
- **Plot Open-Circuit Saturation** — Plots terminal voltage versus no-load stator current, both in per-unit, in a MATLAB figure window. The plot contains three traces:
  - Unsaturated — Stator magnetizing inductance (unsaturated).
  - Saturated — Open-circuit lookup table ( $v$  versus  $i$ ) you specify.
  - Derived — Open-circuit lookup table derived from the per-unit open-circuit lookup table ( $v$  versus  $i$ ) you specify. This data is used to calculate the saturated magnetizing inductance,  $L_{m\_sat}$ , and the saturation factor,  $K_s$ , versus magnetic flux linkage,  $\psi_m$ , characteristics.
- **Plot Saturation Factor** — Plots saturation factor,  $K_s$ , versus magnetic flux linkage,  $\psi_m$ , in a MATLAB figure window using the machine parameters. This parameter is derived from other parameters that you specify:

- No-load stator current saturation data,  $i$
- Terminal voltage saturation data,  $v$
- Leakage inductance,  $L_{ls}$
- **Plot Saturated Inductance** — Plots magnetizing inductance,  $L_{m\_sat}$ , versus magnetic flux linkage,  $\psi_m$ , in a MATLAB figure window using the machine parameters. This parameter is derived from other parameters that you specify:
  - No-load stator current saturation data,  $i$
  - Terminal voltage saturation data,  $v$
  - Leakage inductance,  $L_{ls}$

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Ports

The block has the following ports:

**R**

Mechanical rotational conserving port associated with the machine rotor.

**C**

Mechanical rotational conserving port associated with the machine case.

**~1**

Expandable three-phase port associated with the stator positive-end connections.

**~2**

Expandable three-phase port associated with the stator negative-end connections.

## **pu**

Physical signal vector port associated with the machine per-unit measurements. The vector elements are:

- pu\_torque
- pu\_velocity
- pu\_vds
- pu\_vqs
- pu\_v0s
- pu\_ids
- pu\_iqs
- pu\_i0s

## **Parameters**

All default parameter values are based on a machine delta-winding configuration.

- “Main” on page 1-66
- “Impedances” on page 1-67
- “Saturation” on page 1-68

## **Main**

### **Rated apparent power**

Rated apparent power of the asynchronous machine. The default value is 15e3 V\*A.

### **Rated voltage**

RMS line-line voltage. The default value is 220 V.

### **Rated electrical frequency**

Nominal electrical frequency corresponding to the rated apparent power. The default value is 60 Hz.

### **Number of pole pairs**

Number of machine pole pairs. The default value is 1.



**Squirrel cage**

Option to specify a single or double squirrel cage for the machine. The default value is `Single squirrel cage`.

Setting this parameter to `Double squirrel cage` exposes **Impedances** parameters for the second cage.

**Zero sequence**

Option to include or exclude zero-sequence terms.

- **Include** — Include zero-sequence terms. To prioritize model fidelity, use this default setting. Using this option results in an error for simulations that use the Partitioning solver. For more information, see “Increase Simulation Speed Using the Partitioning Solver” (Simscape).
- **Exclude** — Exclude zero-sequence terms. To prioritize simulation speed for desktop simulation or real-time deployment, select this option.

**Impedances****Stator resistance,  $R_s$  (pu)**

Stator resistance. The default value is `0.0258`.

**Stator leakage inductance,  $L_{ls}$  (pu)**

Stator leakage inductance. The default value is `0.0930`.

**Referred rotor resistance,  $R_r'$  (pu)**

Rotor resistance referred to the stator. The default value is `0.0145`.

This parameter is visible only when the **Main Squirrel cage** parameter is set to `Single squirrel cage`.

**Referred rotor leakage inductance,  $L_{lr}'$  (pu)**

Rotor leakage inductance referred to the stator. The default value is `0.0424`.

This parameter is visible only when the **Main Squirrel cage** parameter is set to `Single squirrel cage`.

**Referred rotor resistance in cage 1,  $R_{r1}'$  (pu)**

Rotor resistance in cage 1. The default value is `0.0145`.

This parameter is visible only when the **Main Squirrel cage** parameter is set to `Double squirrel cage`.

**Referred rotor leakage inductance, Llr1' (pu)**

Rotor leakage inductance in cage 1. The default value is 0.0424.

This parameter is visible only when the **Main Squirrel cage** parameter is set to Double squirrel cage.

**Referred rotor resistance in cage 2, Rr2' (pu)**

Rotor resistance in cage 2. The default value is 0.0145.

This parameter is visible only when the **Main Squirrel cage** parameter is set to Double squirrel cage.

**Referred rotor leakage inductance, Llr2' (pu)**

Rotor leakage inductance in cage 2. The default value is 0.0424.

This parameter is visible only when the **Main Squirrel cage** parameter is set to Double squirrel cage.

**Magnetizing inductance, Lm (pu)**

Magnetizing inductance, that is, the peak value of stator-rotor mutual inductance. The default value is 1.7562.

This parameter is hidden if you set the **Saturation Magnetic saturation representation** parameter to Per-unit open-circuit lookup table (v versus i).

**Stator zero-sequence inductance, L0 (pu)**

Stator zero-sequence inductance. The default value is 0.0930.

## Saturation

**Magnetic saturation representation**

Block magnetic saturation representation. Options are:

- None — The default value.
- Per-unit open-circuit lookup table (v versus i)

Other parameters are exposed or hidden when you set **Magnetic saturation representation** to Per-unit open-circuit lookup table (v versus i).

**Per-unit no-load stator current saturation data, i**

Current  $i$  data populates the voltage  $v$  versus field current  $i$  lookup table. This parameter must contain a vector with at least 10 elements. The default value is [0, .176, .396, .792, 1.1, 1.496, 2.2, 2.992, 4.18, 5.28].

This parameter is visible only when you set **Magnetic saturation representation** to Per-unit open-circuit lookup table ( $v$  versus  $i$ ).

**Per-unit terminal voltage saturation data, v**

Terminal voltage  $v$  data populates the voltage  $v$  versus current  $i$  lookup table. This parameter must contain a vector with at least 10 elements. The number of elements must match the number of elements in the vector for the **Per-unit no-load stator current saturation data, i** parameter. The default value is [0, .2309, .4041, .5196, .5774, .6351, .6928, .7506, .8083, .866] pu.

This parameter is visible only when you set **Magnetic saturation representation** to Per-unit open-circuit lookup table ( $v$  versus  $i$ ).

**References**

- [1] Kundur, P. *Power System Stability and Control*. New York, NY: McGraw Hill, 1993.
- [2] Lyshevski, S. E. *Electromechanical Systems, Electric Machines and Applied Mechatronics*. Boca Raton, FL: CRC Press, 1999.
- [3] Ojo, J. O., Consoli, A., and Lipo, T. A., "An improved model of saturated induction machines", *IEEE Transactions on Industry Applications*. Vol. 26, no. 2, pp. 212-221, 1990.

**See Also**

Asynchronous Machine Measurement | Asynchronous Machine Squirrel Cage (fundamental, SI) | Asynchronous Machine Wound Rotor (fundamental, SI) | Asynchronous Machine Wound Rotor (fundamental)

**Topics**

- "Expand and Collapse Three-Phase Ports on a Block"
- "Marine Full Electric Propulsion Power System"
- "Three-Phase Asynchronous Direct Online Motor Connected to Hydraulic Pump"

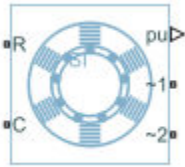
“Three-Phase Asynchronous Machine Starting”

“Three-Phase Asynchronous Wind Turbine Generator”

**Introduced in R2013b**

# Asynchronous Machine Squirrel Cage (fundamental, SI)

Squirrel-cage-rotor asynchronous machine with fundamental parameterization in SI units



## Library

Simscape / Electrical / Power Systems / Machines / Asynchronous Machine (Squirrel Cage)

## Description

The Asynchronous Machine Squirrel Cage (fundamental, SI) block models a squirrel-cage-rotor asynchronous machine using fundamental parameters expressed in the International System of Units (SI). A squirrel-cage asynchronous machine is a type of induction machine. All stator connections are accessible on the block. Therefore, you can model soft-start regimes using a switch between wye and delta configurations. If you need access to the rotor windings, use the Asynchronous Machine Wound Rotor (fundamental, SI) block or the Asynchronous Machine Wound Rotor (fundamental) block instead.

Connect port **~1** to a three-phase circuit. To connect the stator in delta configuration, connect a Phase Permute block between ports **~1** and **~2**. To connect the stator in wye configuration, connect port **~2** to a Grounded Neutral or a Floating Neutral block.

## Equations

The Asynchronous Machine Squirrel Cage (fundamental, SI) converts the SI values that you enter in the dialog box to per-unit values for simulation. For information on the

relationship between SI and per-unit machine parameters, see “Per-Unit Conversion for Machine Parameters”. For information on per-unit parameterization, see “Per-Unit System of Units”.

The asynchronous machine equations are expressed with respect to a synchronous reference frame, defined by

$$\theta_e(t) = \int_0^t 2\pi f_{rated} dt,$$

where  $f_{rated}$  is the value of the **Rated electrical frequency** parameter.

The Park transformation maps stator equations to a reference frame that is stationary with respect to the rated electrical frequency. The Park transformation is defined by

$$P_s = \frac{2}{3} \begin{bmatrix} \cos \theta_e & \cos(\theta_e - \frac{2\pi}{3}) & \cos(\theta_e + \frac{2\pi}{3}) \\ -\sin \theta_e & -\sin(\theta_e - \frac{2\pi}{3}) & -\sin(\theta_e + \frac{2\pi}{3}) \\ \frac{1}{2} & \frac{1}{2} & \frac{1}{2} \end{bmatrix},$$

where  $\theta_e$  is the electrical angle.

The Park transformation is used to define the per-unit asynchronous machine equations. The stator voltage equations are defined by

$$v_{ds} = \frac{1}{\omega_{base}} \frac{d\psi_{ds}}{dt} - \omega\psi_{qs} + R_s i_{ds},$$

$$v_{qs} = \frac{1}{\omega_{base}} \frac{d\psi_{qs}}{dt} + \omega\psi_{ds} + R_s i_{qs},$$

and

$$v_{0s} = \frac{1}{\omega_{base}} \frac{d\psi_{0s}}{dt} + R_s i_{0s},$$

where:

- $v_{ds}$ ,  $v_{qs}$ , and  $v_{0s}$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator voltages, defined by

$$\begin{bmatrix} v_{ds} \\ v_{qs} \\ v_{0s} \end{bmatrix} = P_s \begin{bmatrix} v_a \\ v_b \\ v_c \end{bmatrix}.$$

$v_a$ ,  $v_b$ , and  $v_c$  are the stator voltages across ports ~1 and ~2.

- $\omega_{base}$  is the per-unit base electrical speed.
- $\psi_{ds}$ ,  $\psi_{qs}$ , and  $\psi_{0s}$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator flux linkages.
- $R_s$  is the stator resistance.
- $i_{ds}$ ,  $i_{qs}$ , and  $i_{0s}$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator currents defined by

$$\begin{bmatrix} i_{ds} \\ i_{qs} \\ i_{0s} \end{bmatrix} = P_s \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix}.$$

$i_a$ ,  $i_b$ , and  $i_c$  are the stator currents flowing from port ~1 to port ~2.

The rotor voltage equations are defined by

$$v_{dr} = \frac{1}{\omega_{base}} \frac{d\psi_{dr}}{dt} - (\omega - \omega_r)\psi_{qr} + R_r i_{dr} = 0$$

and

$$v_{qr} = \frac{1}{\omega_{base}} \frac{d\psi_{qr}}{dt} + (\omega - \omega_r)\psi_{dr} + R_r i_{qr} = 0,$$

where:

- $v_{dr}$  and  $v_{qr}$  are the  $d$ -axis and  $q$ -axis rotor voltages.
- $\psi_{dr}$  and  $\psi_{qr}$  are the  $d$ -axis and  $q$ -axis rotor flux linkages.
- $\omega$  is the per-unit synchronous speed. For a synchronous reference frame, the value is 1.

- $\omega_r$  is the per-unit mechanical rotational speed.
- $R_{rd}$  is the rotor resistance referred to the stator.
- $i_{dr}$  and  $i_{qr}$  are the  $d$ -axis and  $q$ -axis rotor currents.

The stator flux linkage equations are defined by

$$\psi_{ds} = L_{ss}i_{ds} + L_m i_{dr},$$

$$\psi_{qs} = L_{ss}i_{qs} + L_m i_{qr},$$

and

$$\psi_{0s} = L_{ss}i_{0s},$$

where  $L_{ss}$  is the stator self-inductance and  $L_m$  is the magnetizing inductance.

The rotor flux linkage equations are defined by

$$\psi_{dr} = L_{rrd}i_{dr} + L_m i_{ds}$$

and

$$\psi_{qr} = L_{rrd}i_{qr} + L_m i_{qs},$$

where  $L_{rrd}$  is the rotor self-inductance referred to the stator.

The rotor torque is defined by

$$T = \psi_{ds}i_{qs} - \psi_{qs}i_{ds}.$$

The stator self-inductance  $L_{ss}$ , stator leakage inductance  $L_{ls}$ , and magnetizing inductance  $L_m$  are related by

$$L_{ss} = L_{ls} + L_m.$$

The rotor self-inductance  $L_{rrd}$ , rotor leakage inductance  $L_{lr}$ , and magnetizing inductance  $L_m$  are related by



$$L_{rrd} = L_{lrd} + L_m.$$

When a saturation curve is provided, the equations to determine the saturated magnetizing inductance as a function of magnetizing flux are:

$$L_{m\_sat} = f(\psi_m)$$

$$\psi_m = \sqrt{\psi_{dm}^2 + \psi_{qm}^2}$$

For no saturation, the equation reduces to

$$L_{m\_sat} = L_m$$

## Plotting and Display Options

You can perform plotting and display actions using the **Electrical** menu on the block context menu.

Right-click the block and, from the **Electrical** menu, select an option:

- **Display Base Values** — Displays the machine per-unit base values in the MATLAB Command Window.
- **Plot Torque Speed (SI)** — Plots torque versus speed, both measured in SI units, in a MATLAB figure window using the current machine parameters.
- **Plot Torque Speed (pu)** — Plots torque versus speed, both measured in per-unit, in a MATLAB figure window using the current machine parameters.
- **Plot Open-Circuit Saturation** — Plots terminal voltage versus no-load stator current, both in per-unit, in a MATLAB figure window. The plot contains three traces:
  - Unsaturated — Stator magnetizing inductance (unsaturated).
  - Saturated — Open-circuit lookup table ( $v$  versus  $i$ ) you specify.
  - Derived — Open-circuit lookup table derived from the per-unit open-circuit lookup table ( $v$  versus  $i$ ) you specify. This data is used to calculate the saturated magnetizing inductance,  $L_{m\_sat}$ , and the saturation factor,  $K_s$ , versus magnetic flux linkage,  $\psi_m$ , characteristics.

- **Plot Saturation Factor** — Plots saturation factor,  $K_s$ , versus magnetic flux linkage,  $\psi_m$ , in a MATLAB figure window using the machine parameters. This parameter is derived from other parameters that you specify:
  - No-load stator current saturation data,  $i$
  - Terminal voltage saturation data,  $v$
  - Leakage inductance,  $L_{ls}$
- **Plot Saturated Inductance** — Plots magnetizing inductance,  $L_{m\_sat}$ , versus magnetic flux linkage,  $\psi_m$ , in a MATLAB figure window using the machine parameters. This parameter is derived from other parameters that you specify:
  - No-load stator current saturation data,  $i$
  - Terminal voltage saturation data,  $v$
  - Leakage inductance,  $L_{ls}$

For this SI implementation,  $v$  is in V (phase-phase RMS) and  $i$  is in A (rms).

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Ports

**R**

Mechanical rotational conserving port associated with the machine rotor.

**C**

Mechanical rotational conserving port associated with the machine case.

**~1**

Expandable three-phase port associated with the stator positive-end connections.

~2

Expandable three-phase port associated with the stator negative-end connections.

### **pu**

Physical signal vector port associated with the machine per-unit measurements. The vector elements are:

- pu\_torque
- pu\_velocity
- pu\_vds
- pu\_vqs
- pu\_v0s
- pu\_ids
- pu\_iqs
- pu\_i0s

## **Parameters**

All default parameter values are based on a machine delta-winding configuration.

- “Main” on page 1-77
- “Impedances” on page 1-78
- “Saturation” on page 1-79

### **Main**

#### **Rated apparent power**

Rated apparent power of the asynchronous machine. The default value is 15e3 V\*A.

#### **Rated voltage**

RMS line-line voltage. The default value is 220 V.

#### **Rated electrical frequency**

Nominal electrical frequency corresponding to the rated apparent power. The default value is 60 Hz.

### **Number of pole pairs**

Number of machine pole pairs. The default value is 1.

### **Squirrel cage**

Option to specify a single or double squirrel cage for the machine. The default value is `Single squirrel cage`.

Setting this parameter to `Double squirrel cage` exposes **Impedances** parameters for the second cage.

### **Zero sequence**

Option to include or exclude zero-sequence terms.

- **Include** — Include zero-sequence terms. To prioritize model fidelity, use this default setting. Using this option results in an error for simulations that use the Partitioning solver. For more information, see “Increase Simulation Speed Using the Partitioning Solver” (Simscape).
- **Exclude** — Exclude zero-sequence terms. To prioritize simulation speed for desktop simulation or real-time deployment, select this option.

## **Impedances**

### **Stator resistance, $R_s$**

Stator resistance. The default value is 0.25 Ohm.

### **Stator leakage reactance, $X_{ls}$**

Stator leakage reactance. The default value is 0.9 Ohm.

### **Referred rotor resistance, $R_r'$**

Rotor resistance referred to the stator. The default value is 0.14 Ohm.

This parameter is visible only when the **Main Squirrel cage** parameter is set to `Single squirrel cage`.

### **Referred rotor leakage reactance, $X_{lr}'$**

Rotor leakage reactance referred to the stator. The default value is 0.41 Ohm.

This parameter is visible only when the **Main Squirrel cage** parameter is set to `Single squirrel cage`.

### **Referred rotor resistance, $R_{r1}'$**

Rotor resistance in cage 1. The default value is 0.14 Ohm.

This parameter is visible only when the **Main Squirrel cage** parameter is set to Double squirrel cage.

**Referred rotor leakage reactance,  $X_{lr1}$ '**

Rotor leakage reactance in cage 1. The default value is 0.41 Ohm.

This parameter is visible only when the **Main Squirrel cage** parameter is set to Double squirrel cage.

**Referred rotor resistance,  $R_{r2}$ '**

Rotor resistance in cage 2. The default value is 0.14 Ohm.

This parameter is visible only when the **Main Squirrel cage** parameter is set to Double squirrel cage.

**Referred rotor leakage reactance,  $X_{lr2}$ '**

Rotor leakage reactance in cage 2. The default value is 0.41 Ohm.

This parameter is visible only when the **Main Squirrel cage** parameter is set to Double squirrel cage.

**Magnetizing reactance,  $X_m$** 

Magnetizing reactance The default value is 17 Ohm.

This parameter is hidden if you set the **Saturation Magnetic saturation representation** parameter to Open-circuit lookup table (v versus i).

**Stator zero-sequence reactance,  $X_0$** 

Stator zero-sequence reactance. The default value is 0.9 Ohm.

## Saturation

**Magnetic saturation representation**

Block magnetic saturation representation. Options are:

- None — The default value.
- Open-circuit lookup table (v versus i)

Other parameters are exposed or hidden when you set **Magnetic saturation representation** to Open-circuit lookup table (v versus i).

## **No-load stator current saturation data, $i$ (rms)**

Current  $i$  data populates the voltage  $v$  versus field current  $i$  lookup table. This parameter must contain a vector with at least 10 elements. The default value is [0, 4, 9, 18, 25, 34, 50, 68, 95, 120] A.

This parameter is visible only when you set **Magnetic saturation representation** to Open-circuit lookup table ( $v$  versus  $i$ ).

## **Terminal voltage saturation data, $v$ (phase-phase, rms)**

Terminal voltage  $v$  data populates the voltage  $v$  versus current  $i$  lookup table. This parameter must contain a vector with at least 10 elements. The number of elements must match the number of elements in the vector for the **No-load stator current saturation data,  $I$  (rms)** parameter. The default value is [0, 88, 154, 198, 220, 242, 264, 286, 308, 330] V.

This parameter is visible only when you set **Magnetic saturation representation** to Open-circuit lookup table ( $v$  versus  $i$ ).

## **References**

- [1] Kundur, P. *Power System Stability and Control*. New York, NY: McGraw Hill, 1993.
- [2] Lyshevski, S. E. *Electromechanical Systems, Electric Machines and Applied Mechatronics*. Boca Raton, FL: CRC Press, 1999.
- [3] Ojo, J. O., Consoli, A., and Lipo, T. A., "An improved model of saturated induction machines", *IEEE Transactions on Industry Applications*. Vol. 26, no. 2, pp. 212-221, 1990.

## **See Also**

Asynchronous Machine Measurement | Asynchronous Machine Squirrel Cage (fundamental) | Asynchronous Machine Wound Rotor (fundamental) | Asynchronous Machine Wound Rotor (fundamental, SI)

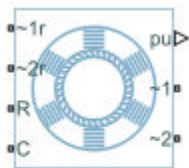
## **Topics**

"Expand and Collapse Three-Phase Ports on a Block"  
Three-Phase Asynchronous Machine Starting

**Introduced in R2015a**

## Asynchronous Machine Wound Rotor (fundamental)

Wound-rotor asynchronous machine with fundamental parameterization



### Library

Simscape / Electrical / Power Systems / Machines / Asynchronous Machine (Wound Rotor)

### Description

The Asynchronous Machine Wound Rotor (fundamental) block models a wound-rotor asynchronous machine using fundamental parameters. A wound-rotor asynchronous machine is a type of induction machine. All stator and rotor connections are accessible on the block. Therefore, you can model soft-start regimes using a switch between wye and delta configurations or by increasing rotor resistance. If you do not need access to the rotor windings, use the Asynchronous Machine Squirrel Cage (fundamental) block instead.

Connect port ~1 to a three-phase circuit. To connect the stator in delta configuration, connect a Phase Permute block between ports ~1 and ~2. To connect the stator in wye configuration, connect port ~2 to a Grounded Neutral or a Floating Neutral block. If you do not need to vary rotor resistance, connect rotor port ~1r to a Floating Neutral block and rotor port ~2r to a Grounded Neutral block.

The rotor circuit is referred to the stator. Therefore, when you use the block in a circuit, refer any additional circuit parameters to the stator.



## Equations

The asynchronous machine equations are expressed with respect to a synchronous reference frame, defined by

$$\theta_e(t) = \int_0^t 2\pi f_{rated} dt,$$

where  $f_{rated}$  is the value of the **Rated electrical frequency** parameter.

The Park transformation maps stator equations to a reference frame that is stationary with respect to the rated electrical frequency. The Park transformation is defined by

$$P_s = \frac{2}{3} \begin{bmatrix} \cos \theta_e & \cos(\theta_e - \frac{2\pi}{3}) & \cos(\theta_e + \frac{2\pi}{3}) \\ -\sin \theta_e & -\sin(\theta_e - \frac{2\pi}{3}) & -\sin(\theta_e + \frac{2\pi}{3}) \\ \frac{1}{2} & \frac{1}{2} & \frac{1}{2} \end{bmatrix},$$

where  $\theta_e$  is the electrical angle.

The rotor equations are mapped to another reference frame, defined by the difference between the electrical angle and the product of rotor angle  $\theta_r$  and number of pole pairs  $N$ :

$$P_r = \frac{2}{3} \begin{bmatrix} \cos(\theta_e - N\theta_r) & \cos(\theta_e - N\theta_r - \frac{2\pi}{3}) & \cos(\theta_e - N\theta_r + \frac{2\pi}{3}) \\ -\sin(\theta_e - N\theta_r) & -\sin(\theta_e - N\theta_r - \frac{2\pi}{3}) & -\sin(\theta_e - N\theta_r + \frac{2\pi}{3}) \\ \frac{1}{2} & \frac{1}{2} & \frac{1}{2} \end{bmatrix}.$$

The Park transformation is used to define the per-unit asynchronous machine equations. The stator voltage equations are defined by

$$v_{ds} = \frac{1}{\omega_{base}} \frac{d\psi_{ds}}{dt} - \omega\psi_{qs} + R_s i_{ds},$$

$$v_{qs} = \frac{1}{\omega_{base}} \frac{d\psi_{qs}}{dt} + \omega\psi_{ds} + R_s i_{qs},$$

and

$$v_{0s} = \frac{1}{\omega_{base}} \frac{d\psi_{0s}}{dt} + R_s i_{0s},$$

where:

- $v_{ds}$ ,  $v_{qs}$ , and  $v_{0s}$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator voltages, defined by

$$\begin{bmatrix} v_{ds} \\ v_{qs} \\ v_{0s} \end{bmatrix} = P_s \begin{bmatrix} v_a \\ v_b \\ v_c \end{bmatrix}.$$

$v_a$ ,  $v_b$ , and  $v_c$  are the stator voltages across ports **~1** and **~2**.

- $\omega_{base}$  is the per-unit base electrical speed.
- $\psi_{ds}$ ,  $\psi_{qs}$ , and  $\psi_{0s}$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator flux linkages.
- $R_s$  is the stator resistance.
- $i_{ds}$ ,  $i_{qs}$ , and  $i_{0s}$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator currents, defined by

$$\begin{bmatrix} i_{ds} \\ i_{qs} \\ i_{0s} \end{bmatrix} = P_s \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix}.$$

$i_a$ ,  $i_b$ , and  $i_c$  are the stator currents flowing from port **~1** to port **~2**.

The rotor voltage equations are defined by

$$v_{dr} = \frac{1}{\omega_{base}} \frac{d\psi_{dr}}{dt} - (\omega - \omega_r)\psi_{qr} + R_{rd}i_{dr},$$

$$v_{qr} = \frac{1}{\omega_{base}} \frac{d\psi_{qr}}{dt} + (\omega - \omega_r)\psi_{dr} + R_{rd}i_{qr}.$$

and

$$v_{0r} = \frac{1}{\omega_{base}} \frac{d\psi_{0r}}{dt} + R_{rd}i_{0s},$$

where:

- $v_{dr}$ ,  $v_{qr}$ , and  $v_{0r}$  are the  $d$ -axis,  $q$ -axis, and zero-sequence rotor voltages, defined by

$$\begin{bmatrix} v_{dr} \\ v_{qr} \\ v_{0r} \end{bmatrix} = P_r \begin{bmatrix} v_{ar} \\ v_{br} \\ v_{cr} \end{bmatrix}.$$

$v_{ar}$ ,  $v_{br}$ , and  $v_{cr}$  are the rotor voltages across ports  $\sim\mathbf{1r}$  and  $\sim\mathbf{2r}$ .

- $\psi_{dr}$ ,  $\psi_{qr}$ , and  $\psi_{0r}$  are the  $d$ -axis,  $q$ -axis, and zero-sequence rotor flux linkages.
- $\omega$  is the per-unit synchronous speed. For a synchronous reference frame, the value is 1.
- $\omega_r$  is the per-unit mechanical rotational speed.
- $R_{rd}$  is the rotor resistance referred to the stator.
- $i_{dr}$ ,  $i_{qr}$ , and  $i_{0r}$  are the  $d$ -axis,  $q$ -axis, and zero-sequence rotor currents, defined by

$$\begin{bmatrix} i_{dr} \\ i_{qr} \\ i_{0r} \end{bmatrix} = P_r \begin{bmatrix} i_{ar} \\ i_{br} \\ i_{cr} \end{bmatrix}.$$

$i_{ar}$ ,  $i_{br}$ , and  $i_{cr}$  are the rotor currents flowing from port  $\sim\mathbf{1r}$  to port  $\sim\mathbf{2r}$ .

The stator flux linkage equations are defined by

$$\psi_{ds} = L_{ss}i_{ds} + L_m i_{dr},$$

$$\psi_{qs} = L_{ss}i_{qs} + L_m i_{qr},$$

and

$$\psi_{0s} = L_{ss}i_{0s},$$

where  $L_{ss}$  is the stator self-inductance and  $L_m$  is the magnetizing inductance.

The rotor flux linkage equations are defined by

$$\psi_{dr} = L_{rrd}i_{dr} + L_m i_{ds}$$

$$\psi_{qr} = L_{rrd}i_{qr} + L_m i_{qs},$$

and

$$\psi_{0r} = L_{rrd}i_{0r},$$

where  $L_{rrd}$  is the rotor self-inductance referred to the stator.

The rotor torque is defined by

$$T = \psi_{ds}i_{qs} - \psi_{qs}i_{ds}.$$

The stator self-inductance  $L_{ss}$ , stator leakage inductance  $L_{ls}$ , and magnetizing inductance  $L_m$  are related by

$$L_{ss} = L_{ls} + L_m.$$

The rotor self-inductance  $L_{rrd}$ , rotor leakage inductance  $L_{lrd}$ , and magnetizing inductance  $L_m$  are related by

$$L_{rrd} = L_{lrd} + L_m.$$

When a saturation curve is provided, the equations to determine the saturated magnetizing inductance as a function of magnetizing flux are:

$$L_{m\_sat} = f(\psi_m)$$

$$\psi_m = \sqrt{\psi_{dm}^2 + \psi_{qm}^2}$$

For no saturation, the equation reduces to

$$L_{m\_sat} = L_m$$

## Plotting and Display Options

You can perform plotting and display actions using the **Electrical** menu on the block context menu.

Right-click the block and, from the **Electrical** menu, select an option:

- **Display Base Values** — Displays the machine per-unit base values in the MATLAB Command Window.
- **Plot Torque Speed (SI)** — Plots torque versus speed, both measured in SI units, in a MATLAB figure window using the current machine parameters.
- **Plot Torque Speed (pu)** — Plots torque versus speed, both measured in per-unit, in a MATLAB figure window using the current machine parameters.
- **Plot Open-Circuit Saturation** — Plots terminal voltage versus no-load stator current, both in per-unit, in a MATLAB figure window. The plot contains three traces:
  - Unsaturated — Stator magnetizing inductance (unsaturated).
  - Saturated — Open-circuit lookup table ( $v$  versus  $i$ ) you specify.
  - Derived — Open-circuit lookup table derived from the per-unit open-circuit lookup table ( $v$  versus  $i$ ) you specify. This data is used to calculate the saturated magnetizing inductance,  $L_{m,sat}$ , and the saturation factor,  $K_s$ , versus magnetic flux linkage,  $\psi_m$ , characteristics.
- **Plot Saturation Factor** — Plots saturation factor,  $K_s$ , versus magnetic flux linkage,  $\psi_m$ , in a MATLAB figure window using the machine parameters. This parameter is derived from other parameters that you specify:
  - No-load stator current saturation data,  $i$
  - Terminal voltage saturation data,  $v$
  - Leakage inductance,  $L_{ls}$
- **Plot Saturated Inductance** — Plots magnetizing inductance,  $L_{m,sat}$ , versus magnetic flux linkage,  $\psi_m$ , in a MATLAB figure window using the machine parameters. This parameter is derived from other parameters that you specify:
  - No-load stator current saturation data,  $i$
  - Terminal voltage saturation data,  $v$
  - Leakage inductance,  $L_{ls}$

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Ports

### R

Mechanical rotational conserving port associated with the machine rotor.

### C

Mechanical rotational conserving port associated with the machine case.

### ~1

Expandable three-phase port associated with the stator positive-end connections.

### ~2

Expandable three-phase port associated with the stator negative-end connections.

### ~1r

Expandable three-phase port associated with the rotor positive-end connections.

### ~2r

Expandable three-phase port associated with the rotor negative-end connections.

### pu

Physical signal vector port associated with the machine per-unit measurements. The vector elements are:

- pu\_torque
- pu\_velocity
- pu\_vds
- pu\_vqs

- pu\_v0s
- pu\_ids
- pu\_iqs
- pu\_i0s

## Parameters

All default parameter values are based on a machine delta-winding configuration.

- “Main” on page 1-89
- “Impedances” on page 1-90
- “Saturation” on page 1-90

### Main

#### Rated apparent power

Rated apparent power of the asynchronous machine. The default value is  $15e3 \text{ V}\cdot\text{A}$ .

#### Rated voltage

RMS line-line voltage. The default value is  $220 \text{ V}$ .

#### Rated electrical frequency

Nominal electrical frequency corresponding to the rated apparent power. The default value is  $60 \text{ Hz}$ .

#### Number of pole pairs

Number of machine pole pairs. The default value is  $1$ .

#### Zero sequence

Option to include or exclude zero-sequence terms.

- **Include** — Include zero-sequence terms. To prioritize model fidelity, use this default setting. Using this option results in an error for simulations that use the Partitioning solver. For more information, see “Increase Simulation Speed Using the Partitioning Solver” (Simscape).
- **Exclude** — Exclude zero-sequence terms. To prioritize simulation speed for desktop simulation or real-time deployment, select this option.

## Impedances

### Stator resistance, $R_s$ (pu)

Stator resistance. The default value is 0.0258.

### Stator leakage inductance, $L_{ls}$ (pu)

Stator leakage inductance. The default value is 0.0930.

### Referred rotor resistance, $R_r'$ (pu)

Rotor resistance referred to the stator. The default value is 0.0145.

### Referred rotor leakage inductance, $L_{lr}'$ (pu)

Rotor leakage inductance referred to the stator. The default value is 0.0424.

### Magnetizing inductance, $L_m$ (pu)

Magnetizing inductance, that is, the peak value of stator-rotor mutual inductance. The default value is 1.7562.

This parameter is hidden if you set the **Saturation Magnetic saturation representation** parameter to Per-unit open-circuit lookup table (v versus i).

### Stator zero-sequence inductance, $L_0$ (pu)

Stator zero-sequence inductance. The default value is 0.0930.

## Saturation

### Magnetic saturation representation

Block magnetic saturation representation. Options are:

- None — The default value.
- Per-unit open-circuit lookup table (v versus i)

Other parameters are exposed or hidden when you set **Magnetic saturation representation** to Per-unit open-circuit lookup table (v versus i).

### Per-unit no-load stator current saturation data, $i$

Current  $i$  data populates the voltage  $v$  versus field current  $i$  lookup table. This parameter must contain a vector with at least 10 elements. The default value is [0, .176, .396, .792, 1.1, 1.496, 2.2, 2.992, 4.18, 5.28].



This parameter is visible only when you set **Magnetic saturation representation** to Per-unit open-circuit lookup table (v versus i).

### **Per-unit terminal voltage saturation data, v**

Terminal voltage v data populates the voltage v versus current i lookup table. This parameter must contain a vector with at least 10 elements. The number of elements must match the number of elements in the vector for the **Per-unit no-load stator current saturation data, i** parameter. The default value is [0, .2309, .4041, .5196, .5774, .6351, .6928, .7506, .8083, .866] pu.

This parameter is visible only when you set **Magnetic saturation representation** to Per-unit open-circuit lookup table (v versus i).

## **References**

- [1] Kundur, P. *Power System Stability and Control*. New York, NY: McGraw Hill, 1993.
- [2] Lyshevski, S. E. *Electromechanical Systems, Electric Machines and Applied Mechatronics*. Boca Raton, FL: CRC Press, 1999.
- [3] Ojo, J. O., Consoli, A., and Lipo, T. A., "An improved model of saturated induction machines", *IEEE Transactions on Industry Applications*. Vol. 26, no. 2, pp. 212-221, 1990.

## **See Also**

Asynchronous Machine Measurement | Asynchronous Machine Squirrel Cage (fundamental) | Asynchronous Machine Squirrel Cage (fundamental, SI) | Asynchronous Machine Wound Rotor (fundamental, SI)

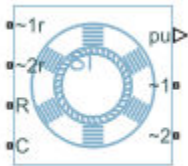
## **Topics**

"Expand and Collapse Three-Phase Ports on a Block"  
Three-Phase Asynchronous Machine Starting

## **Introduced in R2013b**

## Asynchronous Machine Wound Rotor (fundamental, SI)

Wound-rotor asynchronous machine with fundamental parameterization in SI units



### Library

Simscape / Electrical / Power Systems / Machines / Asynchronous Machine (Wound Rotor)

### Description

The Asynchronous Machine Wound Rotor (fundamental, SI) block models a wound-rotor asynchronous machine using fundamental parameters expressed in the International System of Units (SI). A wound-rotor asynchronous machine is a type of induction machine. All stator and rotor connections are accessible on the block. Therefore, you can model soft-start regimes using a switch between wye and delta configurations or by increasing rotor resistance. If you do not need access to the rotor windings, use the Asynchronous Machine Squirrel Cage (fundamental) or Asynchronous Machine Squirrel Cage (fundamental,SI) block instead.

Connect port **~1** to a three-phase circuit. To connect the stator in delta configuration, connect a Phase Permute block between ports **~1** and **~2**. To connect the stator in wye configuration, connect port **~2** to a Grounded Neutral or a Floating Neutral block. If you do not need to vary rotor resistance, connect rotor port **~1r** to a Floating Neutral block and rotor port **~2r** to a Grounded Neutral block.

The rotor circuit is referred to the stator. Therefore, when you use the block in a circuit, refer any additional circuit parameters to the stator.

## Equations

The Asynchronous Machine Wound Rotor (fundamental, SI) converts the SI values that you enter in the dialog box to per-unit values for simulation. For information on the relationship between SI and per-unit machine parameters, see “Per-Unit Conversion for Machine Parameters”. For information on per-unit parameterization, see “Per-Unit System of Units”.

The asynchronous machine equations are expressed with respect to a synchronous reference frame, defined by

$$\theta_e(t) = \int_0^t 2\pi f_{rated} dt,$$

where  $f_{rated}$  is the value of the **Rated electrical frequency** parameter.

The Park transformation maps stator equations to a reference frame that is stationary with respect to the rated electrical frequency. The Park transformation is defined by

$$P_s = \frac{2}{3} \begin{bmatrix} \cos \theta_e & \cos(\theta_e - \frac{2\pi}{3}) & \cos(\theta_e + \frac{2\pi}{3}) \\ -\sin \theta_e & -\sin(\theta_e - \frac{2\pi}{3}) & -\sin(\theta_e + \frac{2\pi}{3}) \\ \frac{1}{2} & \frac{1}{2} & \frac{1}{2} \end{bmatrix},$$

where  $\theta_e$  is the electrical angle.

The rotor equations are mapped to another reference frame, defined by the difference between the electrical angle and the product of rotor angle  $\theta_r$  and number of pole pairs N:

$$P_r = \frac{2}{3} \begin{bmatrix} \cos(\theta_e - N\theta_r) & \cos(\theta_e - N\theta_r - \frac{2\pi}{3}) & \cos(\theta_e - N\theta_r + \frac{2\pi}{3}) \\ -\sin(\theta_e - N\theta_r) & -\sin(\theta_e - N\theta_r - \frac{2\pi}{3}) & -\sin(\theta_e - N\theta_r + \frac{2\pi}{3}) \\ \frac{1}{2} & \frac{1}{2} & \frac{1}{2} \end{bmatrix}.$$

The Park transformation is used to define the per-unit asynchronous machine equations. The stator voltage equations are defined by

$$v_{ds} = \frac{1}{\omega_{base}} \frac{d\psi_{ds}}{dt} - \omega\psi_{qs} + R_s i_{ds},$$

$$v_{qs} = \frac{1}{\omega_{base}} \frac{d\psi_{qs}}{dt} + \omega\psi_{ds} + R_s i_{qs},$$

and

$$v_{0s} = \frac{1}{\omega_{base}} \frac{d\psi_{0s}}{dt} + R_s i_{0s},$$

where:

- $v_{ds}$ ,  $v_{qs}$ , and  $v_{0s}$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator voltages, defined by

$$\begin{bmatrix} v_{ds} \\ v_{qs} \\ v_{0s} \end{bmatrix} = P_s \begin{bmatrix} v_a \\ v_b \\ v_c \end{bmatrix}.$$

$v_a$ ,  $v_b$ , and  $v_c$  are the stator voltages across ports **~1** and **~2**.

- $\omega_{base}$  is the per-unit base electrical speed.
- $\psi_{ds}$ ,  $\psi_{qs}$ , and  $\psi_{0s}$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator flux linkages.
- $R_s$  is the stator resistance.
- $i_{ds}$ ,  $i_{qs}$ , and  $i_{0s}$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator currents, defined by

$$\begin{bmatrix} i_{ds} \\ i_{qs} \\ i_{0s} \end{bmatrix} = P_s \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix}.$$

$i_a$ ,  $i_b$ , and  $i_c$  are the stator currents flowing from port **~1** to port **~2**.

The rotor voltage equations are defined by

$$v_{dr} = \frac{1}{\omega_{base}} \frac{d\psi_{dr}}{dt} - (\omega - \omega_r)\psi_{qr} + R_{rd}i_{dr},$$

$$v_{qr} = \frac{1}{\omega_{base}} \frac{d\psi_{qr}}{dt} + (\omega - \omega_r)\psi_{dr} + R_{rd}i_{qr},$$

and

$$v_{0r} = \frac{1}{\omega_{base}} \frac{d\psi_{0r}}{dt} + R_{rd}i_{0s},$$

where:

- $v_{dr}$ ,  $v_{qr}$ , and  $v_{0r}$  are the  $d$ -axis,  $q$ -axis, and zero-sequence rotor voltages, defined by

$$\begin{bmatrix} v_{dr} \\ v_{qr} \\ v_{0r} \end{bmatrix} = P_r \begin{bmatrix} v_{ar} \\ v_{br} \\ v_{cr} \end{bmatrix}.$$

$v_{ar}$ ,  $v_{br}$ , and  $v_{cr}$  are the rotor voltages across ports  $\sim\mathbf{1r}$  and  $\sim\mathbf{2r}$ .

- $\psi_{dr}$ ,  $\psi_{qr}$ , and  $\psi_{0r}$  are the  $d$ -axis,  $q$ -axis, and zero-sequence rotor flux linkages.
- $\omega$  is the per-unit synchronous speed. For a synchronous reference frame, the value is 1.
- $\omega_r$  is the per-unit mechanical rotational speed.
- $R_{rd}$  is the rotor resistance referred to the stator.
- $i_{dr}$ ,  $i_{qr}$ , and  $i_{0r}$  are the  $d$ -axis,  $q$ -axis, and zero-sequence rotor currents, defined by

$$\begin{bmatrix} i_{dr} \\ i_{qr} \\ i_{0r} \end{bmatrix} = P_r \begin{bmatrix} i_{ar} \\ i_{br} \\ i_{cr} \end{bmatrix}.$$

$i_{ar}$ ,  $i_{br}$ , and  $i_{cr}$  are the rotor currents flowing from port  $\sim\mathbf{1r}$  to port  $\sim\mathbf{2r}$ .

The stator flux linkage equations are defined by

$$\psi_{ds} = L_{ss}i_{ds} + L_m i_{dr},$$

$$\psi_{qs} = L_{ss}i_{qs} + L_m i_{qr},$$

and

$$\psi_{0s} = L_{ss}i_{0s},$$

where  $L_{ss}$  is the stator self-inductance and  $L_m$  is the magnetizing inductance.

The rotor flux linkage equations are defined by

$$\psi_{dr} = L_{rrd}i_{dr} + L_m i_{ds}$$

$$\psi_{qr} = L_{rrd}i_{qr} + L_m i_{qs},$$

and

$$\psi_{0r} = L_{rrd}i_{0r},$$

where  $L_{rrd}$  is the rotor self-inductance referred to the stator.

The rotor torque is defined by

$$T = \psi_{ds}i_{qs} - \psi_{qs}i_{ds}.$$

The stator self-inductance  $L_{ss}$ , stator leakage inductance  $L_{ls}$ , and magnetizing inductance  $L_m$  are related by

$$L_{ss} = L_{ls} + L_m.$$

The rotor self-inductance  $L_{rrd}$ , rotor leakage inductance  $L_{lrd}$ , and magnetizing inductance  $L_m$  are related by

$$L_{rrd} = L_{lrd} + L_m.$$

When a saturation curve is provided, the equations to determine the saturated magnetizing inductance as a function of magnetizing flux are:

$$L_{m\_sat} = f(\psi_m)$$

$$\psi_m = \sqrt{\psi_{dm}^2 + \psi_{qm}^2}$$

For no saturation, the equation reduces to

$$L_{m\_sat} = L_m$$

## Plotting and Display Options

You can perform display and plotting actions using the **Electrical** menu on the block context menu.

Right-click the block and, from the **Electrical** menu, select an option:

- **Display Base Values** — Displays the machine per-unit base values in the MATLAB Command Window.
- **Plot Torque Speed (SI)** — Plots torque versus speed, both measured in SI units, in a MATLAB figure window using the current machine parameters.
- **Plot Torque Speed (pu)** — Plots torque versus speed, both measured in per-unit, in a MATLAB figure window using the current machine parameters.
- **Plot Open-Circuit Saturation** — Plots terminal voltage versus no-load stator current, both in per-unit, in a MATLAB figure window. The plot contains three traces:
  - Unsaturated — Stator magnetizing inductance (unsaturated).
  - Saturated — Open-circuit lookup table ( $v$  versus  $i$ ) you specify.
  - Derived — Open-circuit lookup table derived from the per-unit open-circuit lookup table ( $v$  versus  $i$ ) you specify. This data is used to calculate the saturated magnetizing inductance,  $L_{m\_sat}$ , and the saturation factor,  $K_s$ , versus magnetic flux linkage,  $\psi_m$ , characteristics.
- **Plot Saturation Factor** — Plots saturation factor,  $K_s$ , versus magnetic flux linkage,  $\psi_m$ , in a MATLAB figure window using the machine parameters. This parameter is derived from other parameters that you specify:
  - No-load stator current saturation data,  $i$
  - Terminal voltage saturation data,  $v$

- Leakage inductance,  $L_{ls}$
- **Plot Saturated Inductance** — Plots magnetizing inductance,  $L_{m\_sat}$ , versus magnetic flux linkage,  $\psi_m$ , in a MATLAB figure window using the machine parameters. This parameter is derived from other parameters that you specify:
  - No-load stator current saturation data,  $i$
  - Terminal voltage saturation data,  $v$
  - Leakage inductance,  $L_{ls}$

For this SI implementation,  $v$  is in V (phase-phase RMS) and  $i$  is in A (rms).

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Ports

The block has the following ports:

**R**

Mechanical rotational conserving port associated with the machine rotor.

**C**

Mechanical rotational conserving port associated with the machine case.

**~1**

Expandable three-phase port associated with the stator positive-end connections.

**~2**

Expandable three-phase port associated with the stator negative-end connections.

**~1r**

Expandable three-phase port associated with the rotor positive-end connections.



**~2r**

Expandable three-phase port associated with the rotor negative-end connections.

**pu**

Physical signal vector port associated with the machine per-unit measurements. The vector elements are:

- pu\_torque
- pu\_velocity
- pu\_vds
- pu\_vqs
- pu\_v0s
- pu\_ids
- pu\_iqs
- pu\_i0s

## Parameters

All default parameter values are based on a machine delta-winding configuration.

- “Main” on page 1-99
- “Impedances” on page 1-100
- “Saturation” on page 1-100

### Main

**Rated apparent power**

Rated apparent power of the asynchronous machine. The default value is 15e3 V\*A.

**Rated voltage**

RMS line-line voltage. The default value is 220 V.

**Rated electrical frequency**

Nominal electrical frequency corresponding to the rated apparent power. The default value is 60 Hz.

### **Number of pole pairs**

Number of machine pole pairs. The default value is 1.

### **Zero sequence**

Option to include or exclude zero-sequence terms.

- **Include** — Include zero-sequence terms. To prioritize model fidelity, use this default setting. Using this option results in an error for simulations that use the Partitioning solver. For more information, see “Increase Simulation Speed Using the Partitioning Solver” (Simscape).
- **Exclude** — Exclude zero-sequence terms. To prioritize simulation speed for desktop simulation or real-time deployment, select this option.

## **Impedances**

### **Stator resistance, $R_s$**

Stator resistance. The default value is 0.25 Ohm.

### **Stator leakage reactance, $X_{ls}$**

Stator leakage reactance. The default value is 0.9 Ohm.

### **Referred rotor resistance, $R_r'$**

Rotor resistance referred to the stator. The default value is 0.14 Ohm.

### **Referred rotor leakage reactance, $X_{lr}'$**

Rotor leakage reactance referred to the stator. The default value is 0.41 Ohm.

### **Magnetizing reactance, $X_m$**

Magnetizing reactance The default value is 17 Ohm.

This parameter is hidden if you set the **Saturation Magnetic saturation representation** parameter to Open-circuit lookup table (v versus i).

### **Stator zero-sequence reactance, $X_0$**

Stator zero-sequence reactance. The default value is 0.9 Ohm.

## **Saturation**

### **Magnetic saturation representation**

Block magnetic saturation representation. Options are:

- None — The default value.
- Open-circuit lookup table (v versus i)

Other parameters are exposed or hidden when you set **Magnetic saturation representation** to Open-circuit lookup table (v versus i).

#### **No-load stator current saturation data, i (rms)**

Current  $i$  data populates the voltage  $v$  versus field current  $i$  lookup table. This parameter must contain a vector with at least 10 elements. The default value is [0, 4, 9, 18, 25, 34, 50, 68, 95, 120] A.

This parameter is visible only when you set **Magnetic saturation representation** to Open-circuit lookup table (v versus i).

#### **Terminal voltage saturation data, v (phase-phase, rms)**

Terminal voltage  $v$  data populates the voltage  $v$  versus current  $i$  lookup table. This parameter must contain a vector with at least 10 elements. The number of elements must match the number of elements in the vector for the **No-load stator current saturation data, I (rms)** parameter. The default value is [0, 88, 154, 198, 220, 242, 264, 286, 308, 330] V.

This parameter is visible only when you set **Magnetic saturation representation** to Open-circuit lookup table (v versus i).

## **References**

- [1] Kundur, P. *Power System Stability and Control*. New York, NY: McGraw Hill, 1993.
- [2] Lyshevski, S. E. *Electromechanical Systems, Electric Machines and Applied Mechatronics*. Boca Raton, FL: CRC Press, 1999.
- [3] Ojo, J. O., Consoli, A., and Lipo, T. A., "An improved model of saturated induction machines", *IEEE Transactions on Industry Applications*. vol. 26, no. 2, pp. 212-221, 1990.

## **See Also**

Asynchronous Machine Measurement | Asynchronous Machine Squirrel Cage (fundamental) | Asynchronous Machine Squirrel Cage (fundamental, SI) | Asynchronous Machine Wound Rotor (fundamental)

## **Topics**

“Expand and Collapse Three-Phase Ports on a Block”  
Three-Phase Asynchronous Machine Starting

**Introduced in R2015a**

# Average-Value Chopper

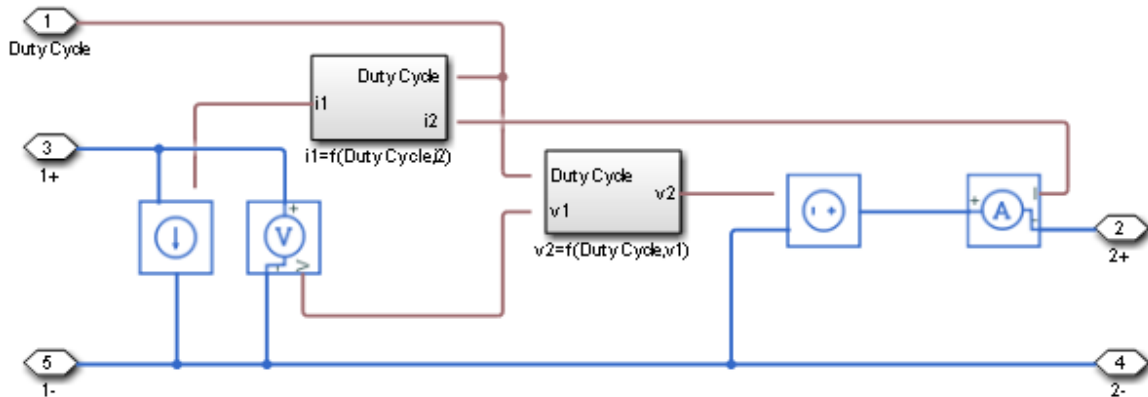
Average-value chopper

**Library:** Simscape / Electrical / Power Systems / Semiconductors



## Description

The Average-Value Chopper block represents a controlled average-value chopper. Use the duty cycle input to convert the electrical energy between the two sides. The figure shows the equivalent circuit for the block.



## Equations

The input current and output voltage depend on the chopper class that you specify.

### Voltage and Current Equations

Chopper Class	Quadrants	Output Voltage, $v_2$			Input Current, $i_1$		
		$i_2 < 0$	$i_2 = 0$	$i_2 > 0$	$i_2 < 0$	$i_2 = 0$	$i_2 > 0$
<b>A</b>	1 <sup>st</sup>	$v_2 = v_1$	$v_2 = DutyCycle \cdot v_1$		$i_1 = i_2$	$i_1 = DutyCycle \cdot i_2$	
<b>B</b>	2 <sup>nd</sup>	$v_2 = (1 - DutyCycle) \cdot v_1$		$v_2 = 0$	$i_1 = (1 - DutyCycle) \cdot i_2$		$i_1 = 0$
<b>C</b>	1 <sup>st</sup> and 2 <sup>nd</sup>	$v_2 = DutyCycle \cdot v_1$			$i_1 = DutyCycle \cdot i_2$		
<b>D</b>	1 <sup>st</sup> and 4 <sup>th</sup>	$v_2 = v_1$	$v_2 = (2 \cdot DutyCycle - 1) \cdot v_1$		$i_1 = i_2$	$i_1 = (2 \cdot DutyCycle - 1) \cdot i_2$	
<b>E</b>	Four	$v_2 = (2 \cdot DutyCycle - 1) \cdot v_1$			$i_1 = (2 \cdot DutyCycle - 1) \cdot i_2$		

### Limitations and Assumptions

- Input voltage,  $v_1$  is positive.
- Power losses are neglected.

## Ports

### Conserving

#### DutyCycle — Duty Cycle

electrical | scalar

Electrical conserving port associated with the duty cycle.

Data Types: double

#### 1+ — Positive DC voltage 1

electrical | scalar

Electrical conserving port associated with the positive terminal of the first DC voltage.

Data Types: double

### **1- — Negative DC voltage 1**

electrical | scalar

Electrical conserving port associated with the negative terminal of the first DC voltage.

Data Types: double

### **2+ — Positive DC voltage 2**

electrical | scalar

Electrical conserving port associated with the positive terminal of the second DC voltage.

Data Types: double

### **2- — Negative DC voltage 2**

electrical | scalar

Electrical conserving port associated with the negative terminal of the second DC voltage.

Data Types: double

## **Parameters**

### **Chopper type — Chopper class**

Class A - first quadrant (default) | Class B - second quadrant | Class C - first and second quadrant | Class D - first and fourth quadrant | Class E - four quadrant

Chopper class.

## **References**

[1] Trzynadlowski, A. M. *Introduction to Modern Power Electronics*. 2nd Ed. Hoboken, NJ: John Wiley & Sons Inc., 2010.

## **See Also**

Average-Value DC-DC Converter | Average-Value Inverter | Average-Value Rectifier | Four-Quadrant Chopper | One-Quadrant Chopper | Two-Quadrant Chopper

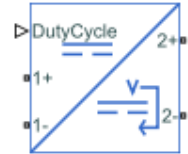
**Introduced in R2018b**



# Average-Value DC-DC Converter

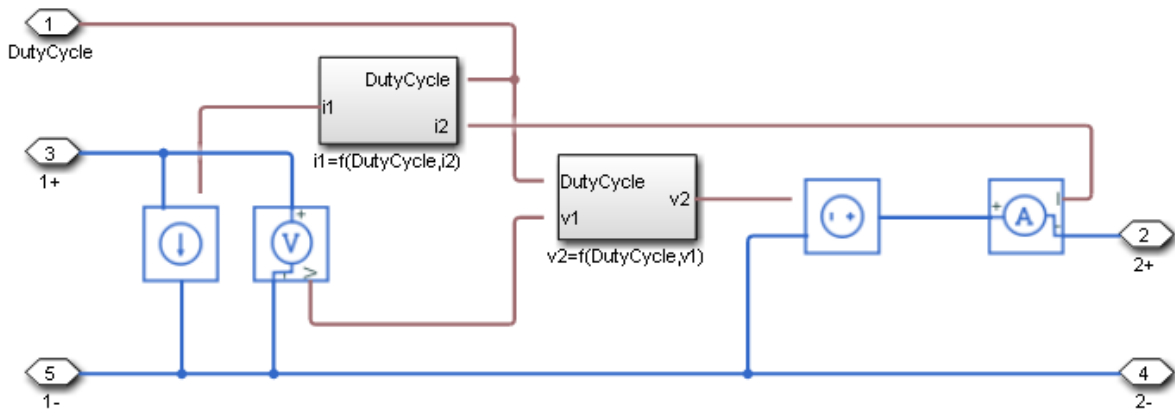
Average-value DC-DC converter

**Library:** Simscape / Electrical / Power Systems / Semiconductors



## Description

The Average-Value DC-DC Converter block represents a controlled average-value DC-DC converter. You can program the block as a buck converter, boost converter, or buck-boost converter. The diagram shows the equivalent circuit for the block. The converter contains a controlled current source and a controlled voltage source. Use the duty cycle port to convert the electrical energy between the connected components on either side of the converter.



## Equations

The input current and output voltage are a function of the duty cycle and depend on the converter type.

### Voltage and Current Equations

Converter Type	Output Voltage, $v_2$	Input Current, $i_1$
<b>Buck</b>	$v_2 = DutyCycle \cdot v_1$	$i_1 = DutyCycle \cdot i_2$
<b>Boost</b>	$v_2 = \frac{v_1}{1 - DutyCycle}$	$i_1 = \frac{i_2}{1 - DutyCycle}$
<b>Buck-Boost</b>	$v_2 = \frac{DutyCycle \cdot v_1}{1 - DutyCycle}$	$i_1 = \frac{DutyCycle \cdot i_2}{1 - DutyCycle}$

## Limitations and Assumptions

- The input voltage is positive.
- Power losses are neglected.
- All converter types use the same polarity for input and output.

## Ports

### Conserving

#### DutyCycle — Duty Cycle

electrical | scalar

Electrical conserving port associated with the duty cycle.

Data Types: double

#### 1+ — Positive DC voltage 1

electrical | scalar

Electrical conserving port associated with the positive terminal of the first DC voltage.

Data Types: double

### **1- — Negative DC voltage 1**

electrical | scalar

Electrical conserving port associated with the negative terminal of the first DC voltage.

Data Types: double

### **2+ — Positive DC voltage 2**

electrical | scalar

Electrical conserving port associated with the positive terminal of the second DC voltage.

Data Types: double

### **2- — Negative DC voltage 2**

electrical | scalar

Electrical conserving port associated with the negative terminal of the second DC voltage.

Data Types: double

## **Parameters**

### **Converter type — Converter type**

Buck converter (default) | Buck-boost converter | Boost converter

Type of converter.

## **See Also**

Average-Value Chopper | Average-Value Inverter | Average-Value Rectifier | Bidirectional DC-DC Converter | Boost Converter | Buck Converter | Buck-Boost Converter | Converter | Rectifier | Three-Level Converter

**Introduced in R2018b**

## Average-Value Inverter

Convert DC voltage to three-phase AC voltage with fixed power loss



## Library

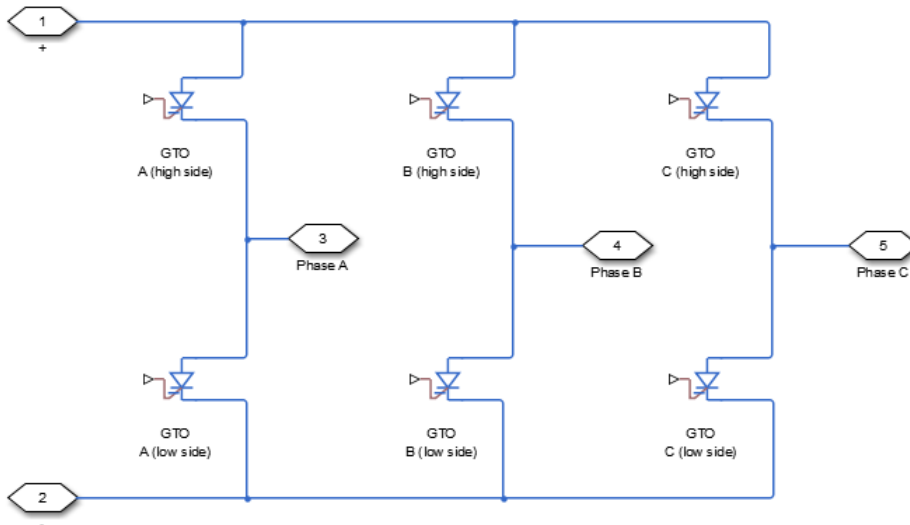
Simscape / Electrical / Power Systems / Semiconductors

## Description

The Average-Value Inverter block models an average-value, full-wave inverter. It converts DC voltage to three-phase AC voltages and converts three-phase AC power demand to DC power demand. The corresponding DC power demand is equal to the sum of the fixed power loss and the AC power demand.

You can use the Average-Value Inverter block only as a full-wave inverter. It behaves as a DC-voltage-controlled AC voltage source. The ratio you specify determines the ratio between the DC voltage and the AC voltage.

The figure shows the equivalent circuit for the inverter as a full-wave inverter. The Average-Value Inverter block does not yield the harmonics that are typically associated with the detailed representation, however, because it performs an average-value power conversion.



## Electrical Defining Equations

The voltages are defined by

$$V_{DC} = v_p - v_n,$$

$$V_{ref} = \frac{v_p + v_n}{2},$$

$$V_{RMS} = v_{ratio} V_{DC},$$

$$V_0 = \frac{\sqrt{2}}{\sqrt{3}} V_{RMS},$$

$$v_a = V_0 \sin(2\pi ft + \varphi) + v_{ref},$$

$$v_b = V_0 \sin(2\pi ft + \varphi - 120^\circ) + v_{ref},$$

and

$$v_c = V_0 \sin(2\pi ft + \varphi + 120^\circ) + v_{ref},$$

where:

- $v_p$  and  $v_n$  are the voltages at the positive and negative terminals of the inverter.
- $v_{DC}$  is the voltage difference between the positive and negative terminals of the inverter.
- $v_{ref}$  is the DC offset.
- $V_{ratio}$  is the ratio of rated AC voltage to rated DC voltage for the inverter. See the **Ratio of rated AC voltage to rated DC voltage** parameter in “Parameters” on page 1-113 for the  $V_{ratio}$  values for common inverter control modes.
- $V_{RMS}$  is the RMS AC line-line voltage.
- $V_0$  is the peak phase voltage.
- $f$  is the frequency.
- $t$  is the time.
- $\varphi$  is the phase shift.
- $v_a, v_b, v_c$  are the respective AC phase voltages.

The power, resistance, and currents are defined by

$$P_{AC} = -v_a i_a - v_b i_b - v_c i_c,$$

$$R_{DC} = \frac{v_{DC}^2}{P_{AC} + P_{fixed}},$$

and

$$i = \frac{v_{DC}}{R_{DC}},$$

where:

- $i_a, i_b,$  and  $i_c$  are the respective AC phase currents flowing into the inverter.

- $P_{AC}$  is the power output on the AC side.  $P_{AC}$  has a minimum limit of 0 W.
- $P_{fixed}$  is the fixed power loss that you specify on the block.
- $R_{DC}$  is the resistance on the DC side.
- $i$  is the current flowing from the positive to the negative terminals of the inverter.

The inverter starts to create an AC voltage, that is turns on, when the DC supply voltage is above the value that you specify for **DC voltage for turn on** parameter. It stops inverting, that is turns off, when the DC supply voltage falls below the value that you specify for **DC voltage for turn off** parameter. When the inverter turns off, the block sets the output AC current to zero.

## Parameters

### Rated AC frequency

AC frequency, specified in Hz (where Hz is defined as  $1/s$ ). For example, kHz and MHz are valid units, but rad/s is not. The default value is 60 Hz.

### Phase shift

Phase shift in angular units. The default value is 0 deg.

### Ratio of rated AC voltage to rated DC voltage

The table shows ratios for common three-phase two-level inverter control modes. The default value is  $\sqrt{6}/\pi$ .

For 180° and 120° conduction modes, the listed voltages are the fundamental RMS values of line-line voltages. For other methods, the listed voltages are the maximum fundamental RMS values of line-line voltages.

You can control the output voltage of the inverter according to specific requirements. DPWM includes 30° DPWM, 60° DPWM, and 120° DPWM. For details, see references [3] and [4].

Control Method	$V_{RMS}$ (line-line)	Ratio of $V_{RMS}$ (line-line) to $v_{DC}$
180° conduction mode [1]	$\frac{\sqrt{6}}{\pi}V_{DC}$	0.7797

<b>Control Method</b>	<b><math>V_{RMS}</math> (line-line)</b>	<b>Ratio of <math>V_{RMS}</math> (line-line) to <math>v_{DC}</math></b>
120° conduction mode [1]	$\frac{3}{\sqrt{2}\pi}V_{DC}$	0.6752
Hysteresis current control [2]	$\left(\frac{\sqrt{3}}{\sqrt{2}}\right)\left(\frac{2V_{DC}}{\pi}\right)$	0.7797
Sinusoidal PWM (SPMW) [2]	$\left(\frac{\sqrt{3}}{\sqrt{2}}\right)\left(\frac{V_{DC}}{2}\right)$	0.6124
Space vector modulation (SVM) [2]	$\left(\frac{\sqrt{3}}{\sqrt{2}}\right)\left(\frac{V_{DC}}{\sqrt{3}}\right)$	0.7071
Discontinuous PWM (DPWM) [3], [4]	$\left(\frac{\sqrt{3}}{\sqrt{2}}\right)\left(\frac{V_{DC}}{\sqrt{3}}\right)$	0.7071
Convert to the original AC voltage of the average-value rectifier	$\left(\frac{\pi}{\sqrt{2}}\right)\left(\frac{V_{DC}}{3}\right)$	0.7405

**Fixed power loss**

Minimum power drawn on the DC side. The default value is 1e3 W.

**DC voltage for turn on**

When the DC supply voltage rises above this value, the inverter produces an AC output voltage.

**DC voltage for turn off**

When the DC supply voltage falls below this value, the inverter turns off and the block sets the output AC currents to zero.

**Ports**

The block has the following ports:



- + Electrical conserving port associated with the positive terminal
- Electrical conserving port associated with the negative terminal
- ~ Expandable three-phase port

## References

- [1] Rashid, M. H. *Pulse-Width-Modulation Inverters*. Upper Saddle River, NJ: Prentice-Hall, 2004, pp. 237-248.
- [2] Krause, P. C., O. Wasynczuk, and S. D. Sudhoff. *Analysis of Electric Machinery and Drive Systems*. Piscataway, NJ: IEEE Press, 2002.
- [3] Chung, D. W., J. S. Kim, and S. K. Kul. "Unified voltage modulation technique for real-time three-phase power conversion." *IEEE Transactions on Industry Applications*. Vol. 34, no. 2, 1998, pp. 374-380.
- [4] Hava, A. M., R. J. Kerkman, and T. A. Lipo. "Simple analytical and graphical methods for carrier-based PWM-VSI drives." *IEEE Transactions on Power Electronics*. Vol. 14, 1999, no. 1, pp. 49-61.

## See Also

Average-Value Chopper | Average-Value DC-DC Converter | Average-Value Rectifier | Bidirectional DC-DC Converter | Boost Converter | Buck Converter | Buck-Boost Converter | Converter | Rectifier | Three-Level Converter

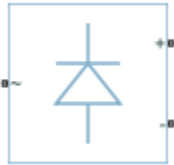
## Topics

"Expand and Collapse Three-Phase Ports on a Block"

**Introduced in R2015a**

## Average-Value Rectifier

Convert three-phase AC voltage to DC voltage with fixed power loss



### Library

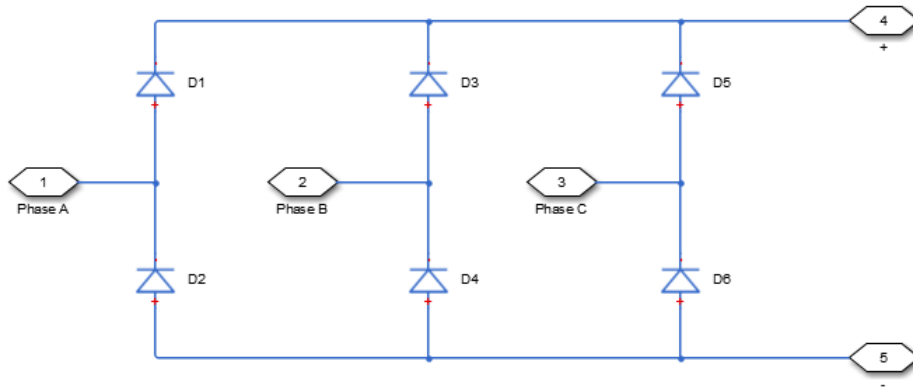
Simscape / Electrical / Power Systems / Semiconductors

### Description

The Average-Value Rectifier block models an average-value, full-wave, six-pulse rectifier. It converts instantaneous three-phase AC voltages to DC voltage and DC power demand to three-phase AC power demand. The corresponding AC power demand is equal to the sum of the fixed power loss and the DC power demand.

You can use the Average-Value Rectifier block only as a six-pulse rectifier. You cannot combine two Average-Value Rectifier blocks to represent a twelve-pulse rectifier.

The figure shows the equivalent circuit for the rectifier as a full-wave, six-pulse rectifier. The Average-Value Rectifier block does not yield the harmonics that are typically associated with the detailed representation, however, because it performs an average-value power conversion.



## Electrical Defining Equations

The voltages are defined by:

$$v_{ref} = \frac{v_a + v_b + v_c}{3},$$

$$V_{RMS} = \sqrt{\frac{(v_a - v_b)^2 + (v_b - v_c)^2 + (v_c - v_a)^2}{3}},$$

$$v_{DC} = 3 \frac{\sqrt{2}}{\pi} V_{RMS},$$

$$v_p = v_{ref} + \frac{v_{DC}}{2},$$

and

$$v_n = v_{ref} - \frac{v_{DC}}{2},$$

where:

- $v_a, v_b, v_c$  are the respective AC phase voltages.
- $v_{ref}$  is the DC offset on the AC side. In a balanced AC power system with no DC bias,  $v_{DC}$  is 0 V.
- $V_{RMS}$  is the RMS AC line-line voltage.
- $v_{DC}$  is the voltage difference between the positive and negative terminals of the rectifier.
- $3\sqrt{2}/\pi$  is the  $v_{DC}/V_{RMS}$  ratio for a full-wave, six-pulse rectifier.
- $v_p, v_n$  are the voltages at the positive and negative terminals of the rectifier.

The resistance, power, and currents are defined by

$$R_{fixed} = \frac{V_{Rated}^2}{P_{fixed}},$$

$$P_{DC} = -v_p i_p - v_n i_n,$$

$$R_{AC} = \frac{V_{RMS}^2}{P_{DC} + \frac{V_{RMS}^2}{R_{fixed}}},$$

and

$$[i_a \quad i_b \quad i_c] = \frac{[v_a \quad v_b \quad v_c] - v_{ref}}{R_{AC}},$$

where:

- $V_{Rated}$  is the rated AC voltage that you specify on the block mask.
- $P_{fixed}$  is the fixed power loss that you specify on the block mask.
- $R_{fixed}$  is the fixed per-phase series resistance in an equivalent wye-connected load.
- $i_p, i_n$  are the currents flowing into the positive and negative terminals of the rectifier.

- $P_{DC}$  is the power output on the DC side.  $P_{DC}$  has a minimum limit of 0 W.
- $R_{AC}$  is the per-phase series resistance in an equivalent wye-connected load.
- $i_a, i_b, i_c$  are the respective AC phase currents flowing into the rectifier.

## Parameters

### Rated AC voltage

Rated voltage of the AC system. The default value is 4160 V.

### Fixed power loss

Minimum power drawn on the AC side at rated AC voltage. When the instantaneous AC voltage is equal to the value you specify for the **Rated AC voltage**, the AC power demand equals the value you specify for the **Fixed power loss** plus DC power demand. The default value is 1e3 W.

## Ports

The block has the following ports:

~

Expandable three-phase port

+

Electrical conserving port associated with the positive terminal

-

Electrical conserving port associated with the negative terminal

## See Also

[Average-Value Chopper](#) | [Average-Value DC-DC Converter](#) | [Average-Value Inverter](#) | [Bidirectional DC-DC Converter](#) | [Boost Converter](#) | [Buck Converter](#) | [Buck-Boost Converter](#) | [Converter](#) | [Rectifier](#) | [Three-Level Converter](#)

## Topics

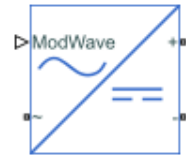
“Expand and Collapse Three-Phase Ports on a Block”

**Introduced in R2014b**

# Average-Value Voltage Source Converter

Bidirectional AC/DC average-value voltage source converter

**Library:** Simscape / Electrical / Power Systems / Semiconductors



## Description

The Average-Value Voltage Source Converter block converts electrical energy from AC to DC voltage or from DC to AC voltage according to an input three-phase modulation wave. The corresponding input power is equal to the sum of the fixed power loss and the output power.

## Ports

### Input

**ModWave — Modulation wave**

vector

Physical signal input port associated with the normalized modulation wave.

### Conserving

**~ — Voltage**

electrical

Expandable electrical conserving port associated with voltage. For more information, see three-phase port.

**+ — Positive terminal**

electrical

Electrical conserving port associated with the positive terminal.

**- — Negative terminal**

electrical

Electrical conserving port associated with the negative terminal.

## Parameters

**Fixed power loss — Power loss**

1 W (default)

Fixed power loss on semiconductor components, in W. The input power is equal to the fixed power loss plus output power.

## See Also

[Average-Value Chopper](#) | [Average-Value Inverter](#) | [Average-Value Rectifier](#) | [Voltage Source](#)

**Introduced in R2018a**



# Band-Limited Op-Amp

Model band-limited operational amplifier



## Library

Integrated Circuits

## Description

The Band-Limited Op-Amp block models a band-limited operational amplifier. If the voltages at the positive and negative ports are  $V_p$  and  $V_m$ , respectively, the output voltage is:

$$V_{out} = \frac{A(V_p - V_m)}{\frac{s}{2\pi f} + 1} - I_{out} * R_{out}$$

where:

- $A$  is the gain.
- $R_{out}$  is the output resistance.
- $I_{out}$  is the output current.
- $s$  is the Laplace operator.
- $f$  is the 3-dB bandwidth.

The input current is:

$$\frac{V_p - V_m}{R_{in}}$$

where  $R_{in}$  is the input resistance.

The block does not use the initial condition you specify using the **Initial output voltage, V0** parameter if you select the **Start simulation from steady state** check box in the Simscape Solver Configuration block.

## Parameters

### Gain, A

The open-loop gain of the operational amplifier. The default value is 1000.

### Input resistance, Rin

The resistance at the input of the operational amplifier that the block uses to calculate the input current. The default value is  $1e+06 \Omega$ .

### Output resistance, Rout

The resistance at the output of the operational amplifier that the block uses to calculate the drop in output voltage due to the output current. The default value is  $100 \Omega$ .

### Minimum output, Vmin

The lower limit on the operational amplifier no-load output voltage. The default value is -15 V.

### Maximum output, Vmax

The upper limit on the operational amplifier no-load output voltage. The default value is 15 V.

### Maximum slew rate, Vdot

The maximum positive or negative rate of change of output voltage magnitude. The default value is 1000 V/s.

### Bandwidth, f

The open-loop bandwidth, that is, the frequency at which the gain drops by 3 dB compared to the low-frequency gain, A. The default value is  $1e+05$  Hz.

### Initial output voltage, V0

The output voltage at the start of the simulation when the output current is zero. The default value is 0 V.

**Note** This parameter value does not account for the voltage drop across the output resistor.

---

## Ports

The block has the following ports:

+

Positive electrical voltage

-

Negative electrical voltage

OUT

Output voltage

## See Also

[Finite-Gain Op-Amp](#) | [Fully Differential Op-Amp](#) | [Op-Amp](#)

# Battery

Simple battery model

**Library:** Simscape / Electrical / Electronics and Mechatronics / Sources  
Simscape / Electrical / Power Systems / Sources



## Description

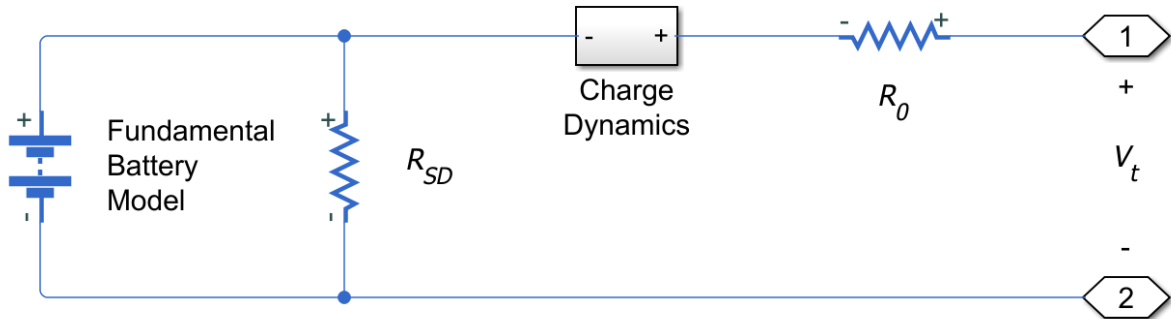
The Battery block represents a simple battery model. The block has four modeling variants, accessible by right-clicking the block in your block diagram and then selecting the appropriate option from the context menu, under **Simscape > Block choices**:

- **Uninstrumented | No thermal port** — Basic model that does not output battery charge level or simulate thermal effects. This modeling variant is the default.
- **Uninstrumented | Show thermal port** — Model with exposed thermal port. This model does not measure internal charge level of the battery.
- **Instrumented | No thermal port** — Model with exposed charge output port. This model does not simulate thermal effects.
- **Instrumented | Show thermal port** — Model that lets you measure internal charge level of the battery and simulate thermal effects. Both the thermal port and the charge output port are exposed.

The instrumented variants have an extra physical signal port that outputs the internal state of charge. Use this functionality to change load behavior as a function of state of charge, without the complexity of building a charge state estimator.

The thermal port variants expose a thermal port, which represents the battery thermal mass. When you select this option, provide additional parameters to define battery behavior at a second temperature. For more information, see “Modeling Thermal Effects” on page 1-129.

The battery equivalent circuit is made up of the fundamental battery model, the self-discharge resistance  $R_{SD}$ , the charge dynamics model, and the series resistance  $R_0$ .



## Battery Model

If you select **Infinite** for the **Battery charge capacity** parameter, the block models the battery as a series resistor and a constant voltage source. If you select **Finite** for the **Battery charge capacity** parameter, the block models the battery as a series resistor and a charge-dependent voltage source. In the finite case, the voltage is a function of charge and has the following relationship:

$$V = V_0 \left( \frac{\text{SOC}}{1 - \beta(1 - \text{SOC})} \right)$$

where:

- SOC (state-of-charge) is the ratio of current charge to rated battery capacity.
- $V_0$  is the voltage when the battery is fully charged at no load, as defined by the **Nominal voltage** parameter.
- $\beta$  is a constant that is calculated so that the battery voltage is  $V1$  when the charge is  $AH1$ . Specify the voltage  $V1$  and ampere-hour rating  $AH1$  using block parameters.  $AH1$  is the charge when the no-load (open-circuit) voltage is  $V1$ , and  $V1$  is less than the nominal voltage.

The equation defines an approximate relationship between voltage and remaining charge. This approximation replicates the increasing rate of voltage drop at low charge values, and ensures that the battery voltage becomes zero when the charge level is zero. The advantage of this model is that it requires few parameters, which are readily available on most datasheets.

## Modeling Battery Fade

For battery models with finite battery charge capacity, you can model battery performance deterioration depending on the number of discharge cycles. This deterioration is referred to as battery fade. To enable battery fade, set the **Battery fade** parameter to **Enabled**. This setting exposes additional parameters in the **Fade** section.

The block implements battery fade by scaling certain battery parameter values that you specify in the **Main** section, depending on the number of completed discharge cycles. The block uses multipliers  $\lambda_{AH}$ ,  $\lambda_{R0}$ , and  $\lambda_{V1}$  on the **Ampere-hour rating**, **Internal resistance**, and **Voltage V1 < Vnom when charge is AH1** parameter values, respectively. These multipliers, in turn, depend on the number of discharge cycles:

$$\lambda_{AH} = 1 - k_1 N^{0.5}$$

$$\lambda_{R0} = 1 + k_2 N^{0.5}$$

$$\lambda_{V1} = 1 - k_3 N$$

$$N = N_0 + \frac{1}{AH} \int_0^t \frac{i(t) \cdot H(i(t))}{\lambda_{AH}(t)} dt$$

where:

- $\lambda_{AH}$  is the multiplier for battery nominal capacity.
- $\lambda_{R0}$  is the multiplier for battery series resistance.
- $\lambda_{V1}$  is the multiplier for voltage  $V1$ .
- $N$  is the number of discharge cycles completed.
- $N_0$  is the number of full discharge cycles completed before the start of the simulation.
- $AH$  is the rated battery capacity in ampere-hours.
- $i(t)$  is the instantaneous battery output current.
- $H(i(t))$  is the Heaviside function of the instantaneous battery output current. This function returns 0 if the argument is negative, and 1 if the argument is positive.

The block calculates the coefficients  $k_1$ ,  $k_2$ , and  $k_3$  by substituting the parameter values you provide in the **Fade** section into these battery equations. For example, the default set of block parameters corresponds to the following coefficient values:

- $k_1 = 1e-2$
- $k_2 = 1e-3$
- $k_3 = 1e-3$

You can also define a starting point for a simulation based on the previous charge-discharge history by using the high-priority variable **Discharge cycles**. For more information, see “Variables” on page 1-132.

## Modeling Thermal Effects

For thermal variants of the block, you provide additional parameters to define battery behavior at a second temperature. The extended equations for the voltage when the thermal port is exposed are:

$$V = V_{or} \left( \frac{\text{SOC}}{1 - \beta(1 - \text{SOC})} \right)$$

$$V_{or} = V_0 (1 + \lambda_v (T - T_1))$$

where:

- $T$  is the battery temperature.
- $T_1$  is the nominal measurement temperature.
- $\lambda_v$  is the parameter temperature dependence coefficient for  $V_0$ .
- $\beta$  is calculated in the same way as “Battery Model” on page 1-127, using the temperature-modified nominal voltage  $V_{or}$ .

The internal series resistance, self-discharge resistance, and any charge-dynamic resistances are also functions of temperature:

$$R_T = R (1 + \lambda_r (T - T_1))$$

where  $\lambda_r$  is the parameter temperature dependence coefficient.

All the temperature dependence coefficients are determined from the corresponding values you provide at the nominal and second measurement temperatures. If you include charge dynamics in the model, the time constants vary with temperature in the same way.

The battery temperature is determined from a summation of all the ohmic losses included in the model:

$$M_{th}\dot{T} = \sum_i V_{T,i}^2 / R_{T,i}$$

where:

- $M_{th}$  is the battery thermal mass.
- $i$  corresponds to the  $i$ th ohmic loss contributor. Depending on how you have configured the block, the losses include:
  - Series resistance
  - Self-discharge resistance
  - First charge dynamics segment
  - Second charge dynamics segment
  - Third charge dynamics segment
  - Fourth charge dynamics segment
  - Fifth charge dynamics segment
- $V_{T,i}$  is the voltage drop across resistor  $i$ .
- $R_{T,i}$  is resistor  $i$ .

## Modeling Charge Dynamics

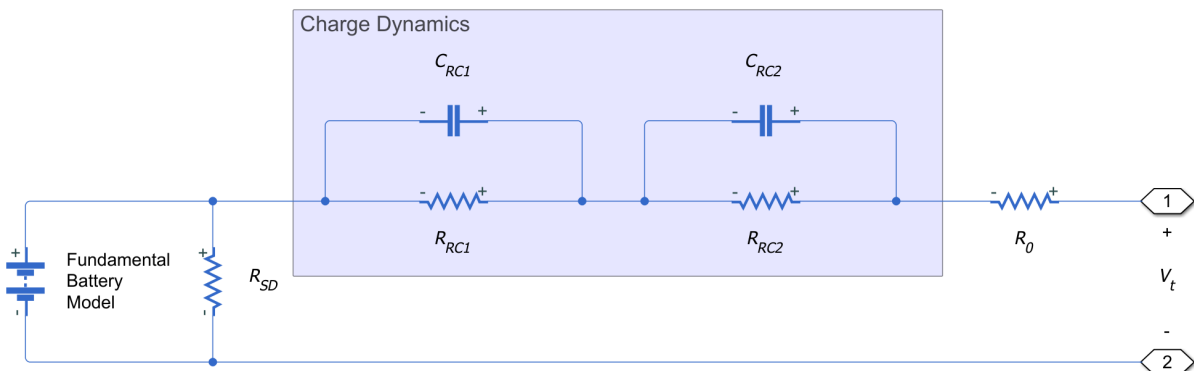
You can model battery charge dynamics using the **Charge dynamics** parameter:

- **No dynamics** — The equivalent circuit contains no parallel RC sections. There is no delay between terminal voltage and internal charging voltage of the battery.
- **One time-constant dynamics** — The equivalent circuit contains one parallel RC section. Specify the time constant using the **First time constant** parameter.
- **Two time-constant dynamics** — The equivalent circuit contains two parallel RC sections. Specify the time constants using the **First time constant** and **Second time constant** parameters.



- Three time-constant dynamics — The equivalent circuit contains three parallel RC sections. Specify the time constants using the **First time constant**, **Second time constant**, and **Third time constant** parameters.
- Four time-constant dynamics — The equivalent circuit contains four parallel RC sections. Specify the time constants using the **First time constant**, **Second time constant**, **Third time constant**, and **Fourth time constant** parameters.
- Five time-constant dynamics — The equivalent circuit contains five parallel RC sections. Specify the time constants using the **First time constant**, **Second time constant**, **Third time constant**, **Fourth time constant**, and **Fifth time constant** parameters.

This figure shows the equivalent circuit for the block configured with two time-constant dynamics.



In the diagram:

- $R_{RC1}$  and  $R_{RC2}$  are the parallel RC resistances. Specify these values with the **First polarization resistance** and **Second polarization resistance** parameters, respectively.
- $C_{RC1}$  and  $C_{RC2}$  are the parallel RC capacitances. The time constant  $\tau$  for each parallel section relates the  $R$  and  $C$  values using the relationship  $C = \tau / R$ . Specify  $\tau$  for each section using the **First time constant** and **Second time constant** parameters, respectively.
- $R_0$  is the series resistance. Specify this value with the **Internal resistance** parameter.

## Plotting Voltage-Charge Characteristics

A quick plot feature lets you visualize the voltage-charge characteristic for the battery model parameter values. To plot the characteristics, right-click a Battery block in your model and, from the context menu, select **Electrical > Basic characteristics**. The software automatically computes a set of bias conditions, based on the block parameter values, and opens a figure window containing a plot of no-load voltage versus the state-of-charge (SOC) for the block. For more information, see “Plot Basic Characteristics for Battery Blocks”.

## Variables

Use the **Variables** section of the block interface to set the priority and initial target values for the block variables prior to simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** section lists all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

When you model battery fade, the **Discharge cycles** variable lets you specify the number of charge-discharge cycles completed prior to the start of simulation. If you disable battery fade modeling, this variable is not used by the block.

## Basic Assumptions and Limitations

- The self-discharge resistance is assumed not to depend strongly on the number of discharge cycles.
- For the thermal variant of the battery, you provide fade data only for the reference temperature operation. The block applies the same derived  $\lambda_{AH}$ ,  $\lambda_{RO}$ , and  $\lambda_{V1}$  multipliers to parameter values corresponding to the second temperature.
- When using the thermal block variants, use caution when operating at temperatures outside of the temperature range bounded by the **Measurement temperature** and **Second measurement temperature** values. The block uses linear interpolation for the derived equation coefficients, and simulation results can become nonphysical outside of the specified range. The block checks that the internal series resistance, self-discharge resistance, and nominal voltage always remain positive. If there is a violation, the block issues error messages.

## Ports

### Output

#### **q — Battery charge level, C**

physical signal

Physical signal port that outputs the internal charge, in the units of coulomb (C). Use this output port to change load behavior as a function of charge, without the complexity of building a charge state estimator.

#### **Dependencies**

Enabled for the instrumented variants of the block: **Instrumented | No thermal port** and **Instrumented | Show thermal port**.

### Conserving

#### **+ — Positive terminal**

electrical

Electrical conserving port associated with the battery positive terminal.

#### **- — Negative terminal**

electrical

Electrical conserving port associated with the battery negative terminal.

#### **H — Battery thermal mass**

thermal

Thermal conserving port that represents the battery thermal mass. When you expose this port, provide additional parameters to define battery behavior at a second temperature. For more information, see “Modeling Thermal Effects” on page 1-129.

#### **Dependencies**

Enabled for the thermal variants of the block: **Uninstrumented | Show thermal port** and **Instrumented | Show thermal port**.

## Parameters

### Main

#### **Nominal voltage — Output voltage when battery is fully charged**

12 V (default) | positive number

The no-load voltage across the battery when it is fully charged.

#### **Internal resistance — Battery internal resistance**

2  $\Omega$  (default) | positive number

Internal connection resistance of the battery.

#### **Battery charge capacity — Select battery model**

Infinite (default) | Finite

Select one of the options for modeling the charge capacity of the battery:

- **Infinite** — The battery voltage is independent of charge drawn from the battery.
- **Finite** — The battery voltage decreases as charge decreases.

#### **Ampere-hour rating — Nominal battery capacity when fully charged**

50 hr\*A (default) | positive number

The maximum (nominal) battery charge in ampere-hours. To specify a target value for the initial battery charge at the start of simulation, use the high-priority **Charge** variable. For more information, see “Variables” on page 1-132.

#### **Dependencies**

Enabled when the **Battery charge capacity** parameter is set to Finite.

#### **Voltage V1 < Vnom when charge is AH1 — Output voltage at charge level AH1**

11.5 V (default) | positive number

The fundamental battery output voltage when the charge level is AH1, as specified by the **Charge AH1 when no-load volts are V1** parameter.

#### **Dependencies**

Enabled when the **Battery charge capacity** parameter is set to Finite.

**Charge AH1 when no-load volts are V1 — Charge level when the no-load output voltage is V1**

25 hr\*A (default) | positive number

The battery charge level corresponding to the no-load output voltage specified by the **Voltage V1 < Vnom when charge is AH1** parameter.

**Dependencies**

Enabled when the **Battery charge capacity** parameter is set to Finite.

**Self-discharge — Select whether to model the self-discharge resistance of the battery**

Disabled (default) | Enabled

Select whether to model the self-discharge resistance of the battery. The block models this effect as a resistor across the terminals of the fundamental battery model.

As temperature increases, self-discharge resistance decreases, causing self-discharge to increase. If the decrease in resistance is too fast, thermal runaway of the battery and numerical instability can occur. You can resolve this by doing any of the following:

- Decrease the thermal resistance
- Decrease the gradient of the self-discharge resistance with respect to temperature
- Increase the self-discharge resistance

**Dependencies**

Enabled when the **Battery charge capacity** parameter is set to Finite.

**Self-discharge resistance — Resistance that represents battery self-discharge**2000  $\Omega$  (default) | positive number

The resistance across the fundamental battery model that represents battery self-discharge.

**Dependencies**

Enabled when the **Self-discharge** parameter is set to Enabled.

**Measurement temperature — Temperature at which the block parameters are measured**

298.15 K (default) | positive number

Temperature  $T_1$ , at which the block parameters in the **Main** section are measured. For more information, see “Modeling Thermal Effects” on page 1-129.

**Dependencies**

Enabled for blocks with exposed thermal port.

**Dynamics**

**Charge dynamics — Battery charge dynamics model**

No dynamics (default) | One time-constant dynamics | Two time-constant dynamics | Three time-constant dynamics | Four time-constant dynamics | Five time-constant dynamics

Select how to model battery charge dynamics. This parameter determines the number of parallel RC sections in the equivalent circuit:

- **No dynamics** — The equivalent circuit contains no parallel RC sections. There is no delay between terminal voltage and internal charging voltage of the battery.
- **One time-constant dynamics** — The equivalent circuit contains one parallel RC section. Specify the time constant using the **First time constant** parameter.
- **Two time-constant dynamics** — The equivalent circuit contains two parallel RC sections. Specify the time constants using the **First time constant** and **Second time constant** parameters.
- **Three time-constant dynamics** — The equivalent circuit contains three parallel RC sections. Specify the time constants using the **First time constant**, **Second time constant**, and **Third time constant** parameters.
- **Four time-constant dynamics** — The equivalent circuit contains four parallel RC sections. Specify the time constants using the **First time constant**, **Second time constant**, **Third time constant**, and **Fourth time constant** parameters.
- **Five time-constant dynamics** — The equivalent circuit contains five parallel RC sections. Specify the time constants using the **First time constant**, **Second time constant**, **Third time constant**, **Fourth time constant**, and **Fifth time constant** parameters.

**First polarization resistance — First RC resistance**

0.005  $\Omega$  (default) | positive number

The resistance of the first parallel RC section. This parameter primarily affects the ohmic losses of the RC section.

**Dependencies**

To enable this parameter, set **Charge dynamics** to One time-constant dynamics, Two time-constant dynamics, Three time-constant dynamics, Four time-constant dynamics, or Five time-constant dynamics.

**First time constant — First RC time constant**

30 s (default) | positive number

The time constant of the first parallel RC section. This value is equal to  $RC$  and affects the dynamics of the RC section.

**Dependencies**

To enable this parameter, set **Charge dynamics** to One time-constant dynamics, Two time-constant dynamics, Three time-constant dynamics, Four time-constant dynamics, or Five time-constant dynamics.

**Second polarization resistance — Second RC resistance**

0.005  $\Omega$  (default) | positive number

The resistance of the second parallel RC section. This parameter primarily affects the ohmic losses of the RC section.

**Dependencies**

To enable this parameter, set **Charge dynamics** to Two time-constant dynamics, Three time-constant dynamics, Four time-constant dynamics, or Five time-constant dynamics.

**Second time constant — Second RC time constant**

30 s (default) | positive number

The time constant of the second parallel RC section. This value is equal to  $RC$  and affects the dynamics of the RC section.

**Dependencies**

To enable this parameter, set **Charge dynamics** to Two time-constant dynamics, Three time-constant dynamics, Four time-constant dynamics, or Five time-constant dynamics.

**Third polarization resistance — Third RC resistance**

0.005  $\Omega$  (default) | positive number

The resistance of the third parallel RC section. This parameter primarily affects the ohmic losses of the RC section.

**Dependencies**

To enable this parameter, set **Charge dynamics** to Three time-constant dynamics, Four time-constant dynamics, or Five time-constant dynamics.

**Third time constant — Third RC time constant**

30 s (default) | positive number

The time constant of the third parallel RC section. This value is equal to  $RC$  and affects the dynamics of the RC section.

**Dependencies**

To enable this parameter, set **Charge dynamics** to Three time-constant dynamics, Four time-constant dynamics, or Five time-constant dynamics.

**Fourth polarization resistance — Fourth RC resistance**

0.005  $\Omega$  (default) | positive number

The resistance of the fourth parallel RC section. This parameter primarily affects the ohmic losses of the RC section.

**Dependencies**

To enable this parameter, set **Charge dynamics** to Four time-constant dynamics or Five time-constant dynamics.

**Fourth time constant — Fourth RC time constant**

30 s (default) | positive number

The time constant of the fourth parallel RC section. This value is equal to  $RC$  and affects the dynamics of the RC section.



**Dependencies**

To enable this parameter, set **Charge dynamics** to Four time-constant dynamics or Five time-constant dynamics.

**Fifth polarization resistance — Fifth RC resistance**

0.005  $\Omega$  (default) | positive number

The resistance of the fifth parallel RC section. This parameter primarily affects the ohmic losses of the RC section.

**Dependencies**

To enable this parameter, set **Charge dynamics** to Five time-constant dynamics.

**Fifth time constant — Fifth RC time constant**

30 s (default) | positive number

The time constant of the fifth parallel RC section. This value is equal to  $RC$  and affects the dynamics of the RC section.

**Dependencies**

To enable this parameter, set **Charge dynamics** to Five time-constant dynamics.

**Fade****Battery fade — Select whether to model battery performance deterioration with aging**

Disabled (default) | Enabled

Select whether to include battery fade modeling:

- **Disabled** — The battery performance is not age dependent.
- **Enabled** — The battery performance changes depending on the number of completed charge-discharge cycles. Selecting this option exposes additional parameters in this section, which define the battery performance after a certain number of discharge cycles. The block uses these parameter values to calculate the scaling coefficients  $k_1$ ,  $k_2$ , and  $k_3$ . For more information, see “Modeling Battery Fade” on page 1-128.

**Dependencies**

Enabled when the **Battery charge capacity** parameter in the **Main** section is set to Finite. If **Battery charge capacity** is Infinite, the **Fade** section is empty.

**Number of discharge cycles, N — Number of cycles that defines a second set of data points**

100 (default) | positive number

The number of charge-discharge cycles after which the other parameters in this section are measured. This second set of data points defines the scaling coefficients  $k_1$ ,  $k_2$ , and  $k_3$ , used in modeling battery fade.

**Dependencies**

Enabled when the **Battery fade** parameter is set to Enabled.

**Ampere-hour rating after N discharge cycles — Maximum battery capacity after N discharge cycles**

45 hr\*A (default) | positive number

The maximum battery charge, in ampere-hours, after the number of discharge cycles specified by the **Number of discharge cycles, N** parameter.

**Dependencies**

Enabled when the **Battery fade** parameter is set to Enabled.

**Internal resistance after N discharge cycles — Battery internal resistance after N discharge cycles**

2.02  $\Omega$  (default) | positive number

The battery internal resistance after the number of discharge cycles specified by the **Number of discharge cycles, N** parameter.

**Dependencies**

Enabled when the **Battery fade** parameter is set to Enabled.

**Voltage V1 at charge AH1 after N discharge cycles — Output voltage at charge level AH1 after N discharge cycles**

10.35 V (default) | positive number

The fundamental battery model output voltage, at charge level AH1, after the number of discharge cycles specified by the **Number of discharge cycles, N** parameter.

**Dependencies**

Enabled when the **Battery fade** parameter is set to Enabled.

**Temperature Dependence**

This section appears only for blocks with exposed thermal port. For more information, see “Modeling Thermal Effects” on page 1-129.

**Nominal voltage at second measurement temperature — Output voltage when battery is fully charged**

12 V (default) | positive number

The no-load voltage across the battery at the second measurement temperature when it is fully charged.

**Internal resistance at second measurement temperature — Battery internal resistance**

2.2  $\Omega$  (default) | positive number

Internal connection resistance of the battery at the second measurement temperature.

**Voltage V1 at second measurement temperature — Output voltage at charge level AH1**

11.4 V (default) | positive number

The fundamental battery model output voltage at the second measurement temperature and at charge level AH1, as specified by the **Charge AH1 when no-load volts are V1** parameter.

**Dependencies**

Enabled when the **Battery charge capacity** parameter in the **Main** section is set to Finite.

**Self-discharge resistance at second measurement temperature — Resistance that represents battery self-discharge**

2200  $\Omega$  (default) | positive number

The resistance across the fundamental battery model at the second measurement temperature. This resistance represents the self-discharge.

**Dependencies**

Enabled when the **Self-discharge resistance** parameter in the **Main** section is set to Enabled.

**First polarization resistance at second measurement temperature — First RC resistance at second measurement temperature**

0.005  $\Omega$  (default) | positive number

The resistance of the first parallel RC section at the second measurement temperature.

**Dependencies**

To enable this parameter, set **Charge dynamics** to One time-constant dynamics, Two time-constant dynamics, Three time-constant dynamics, Four time-constant dynamics, or Five time-constant dynamics.

**First time constant at second measurement temperature — First RC time constant at second measurement temperature**

30 s (default) | positive number

The time constant of the first parallel RC section at the second measurement temperature.

**Dependencies**

To enable this parameter, set **Charge dynamics** to One time-constant dynamics, Two time-constant dynamics, Three time-constant dynamics, Four time-constant dynamics, or Five time-constant dynamics.

**Second polarization resistance at second measurement temperature — Second RC resistance at second measurement temperature**

0.005  $\Omega$  (default) | positive number

The resistance of the second parallel RC section at the second measurement temperature.

**Dependencies**

To enable this parameter, set **Charge dynamics** to Two time-constant dynamics, Three time-constant dynamics, Four time-constant dynamics, or Five time-constant dynamics.

**Second time constant at second measurement temperature — Second RC time constant at second measurement temperature**

30 s (default) | positive number

The time constant of the second parallel RC section at the second measurement temperature.

**Dependencies**

To enable this parameter, set **Charge dynamics** to Two time-constant dynamics, Three time-constant dynamics, Four time-constant dynamics, or Five time-constant dynamics.

**Third polarization resistance at second measurement temperature — Third RC resistance at second measurement temperature**

0.005  $\Omega$  (default) | positive number

The resistance of the third parallel RC section at the second measurement temperature.

**Dependencies**

To enable this parameter, set **Charge dynamics** to Three time-constant dynamics, Four time-constant dynamics, or Five time-constant dynamics.

**Third time constant at second measurement temperature — Third RC time constant at second measurement temperature**

30 s (default) | positive number

The time constant of the third parallel RC section at the second measurement temperature.

**Dependencies**

To enable this parameter, set **Charge dynamics** to Three time-constant dynamics, Four time-constant dynamics, or Five time-constant dynamics.

**Fourth polarization resistance at second measurement temperature — Fourth RC resistance at second measurement temperature**

0.005  $\Omega$  (default) | positive number

The resistance of the fourth parallel RC section at the second measurement temperature.

### **Dependencies**

To enable this parameter, set **Charge dynamics** to Four time-constant dynamics or Five time-constant dynamics.

### **Fourth time constant at second measurement temperature — Fourth RC time constant at second measurement temperature**

30 s (default) | positive number

The time constant of the fourth parallel RC section at the second measurement temperature.

### **Dependencies**

To enable this parameter, set **Charge dynamics** to Four time-constant dynamics or Five time-constant dynamics.

### **Fifth polarization resistance at second measurement temperature — Fifth RC resistance at second measurement temperature**

0.005  $\Omega$  (default) | positive number

The resistance of the fifth parallel RC section at the second measurement temperature.

### **Dependencies**

To enable this parameter, set **Charge dynamics** to Five time-constant dynamics.

### **Fifth time constant at second measurement temperature — Fifth RC time constant at second measurement temperature**

30 s (default) | positive number

The time constant of the fifth parallel RC section at the second measurement temperature.

### **Dependencies**

To enable this parameter, set **Charge dynamics** to Five time-constant dynamics.

### **Second measurement temperature — Temperature at which the block parameters in this section are measured**

273.15 K (default) | positive number

Temperature  $T_2$ , at which the block parameters in the **Temperature Dependence** section are measured. For more information, see “Modeling Thermal Effects” on page 1-129.

To specify the initial temperature at the start of simulation, use the high-priority **Temperature** variable. For more information, see “Variables” on page 1-132.

## Thermal Port

This section appears only for blocks with exposed thermal port. For more information, see “Modeling Thermal Effects” on page 1-129.

### **Thermal mass – Thermal mass associated with the thermal port**

30000 J/K (default) | positive number

Thermal mass associated with the thermal port H. It represents the energy required to raise the temperature of the thermal port by one degree.

## References

- [1] Ramadass, P., B. Haran, R. E. White, and B. N. Popov. “Mathematical modeling of the capacity fade of Li-ion cells.” *Journal of Power Sources*. 123 (2003), pp. 230â€”240.
- [2] Ning, G., B. Haran, and B. N. Popov. “Capacity fade study of lithium-ion batteries cycled at high discharge rates.” *Journal of Power Sources*. 117 (2003), pp. 160â€”169.

## See Also

Battery (Table-Based) | Controlled Voltage Source | DC Voltage Source

**Introduced in R2008b**

## Battery (Table-Based)

Tabulated battery model

**Library:** Simscape / Electrical / Electronics and Mechatronics / Sources  
Simscape / Electrical / Power Systems / Sources

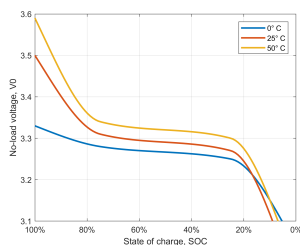


### Description

The Battery (Table-Based) block represents a high-fidelity battery model. The block calculates no-load voltage as a function of charge level and temperature using lookup tables and includes several modeling options:

- Self-discharge
- Battery fade
- Charge dynamics

The plot shows a battery whose performance varies with temperature and state of charge changes, as typically found on a datasheet.



Use this block to parameterize batteries with complex no-load voltage behavior from datasheets or experimental results. For a simpler representation of a battery, see the Battery block.

The Battery (Table-Based) block has four modeling variants, accessible by right-clicking the block in your block diagram and then selecting the appropriate option from the context menu, under **Simscape > Block choices**:

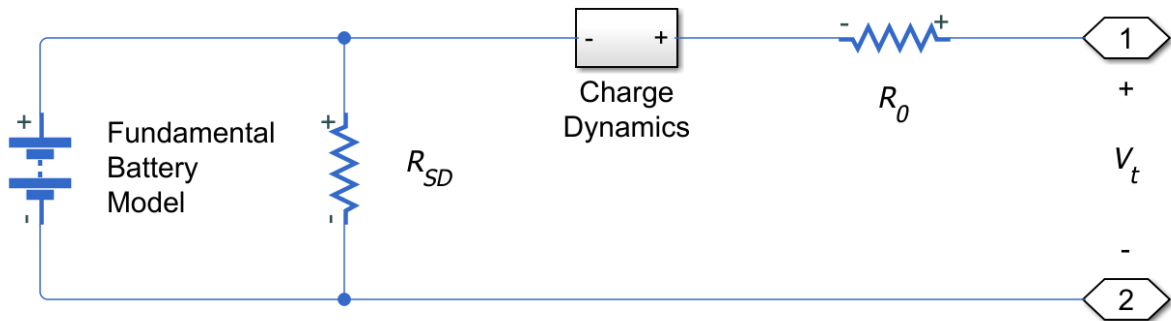


- **Uninstrumented | No thermal port** — Basic model that does not output battery charge level and simulates at a fixed temperature. This modeling variant is the default.
- **Uninstrumented | Show thermal port** — Model with exposed thermal port. This model does not output internal charge level of the battery.
- **Instrumented | No thermal port** — Model with exposed charge output port. This model uses a fixed temperature throughout the simulation.
- **Instrumented | Show thermal port** — Model that lets you output internal charge level of the battery. Both the thermal port and the charge output port are exposed.

The instrumented variants have an extra physical signal port that outputs the internal state of charge. Use this functionality to change load behavior as a function of state of charge, without the complexity of building a charge state estimator.

The thermal port variants expose a thermal port, which represents the battery thermal mass.

The battery equivalent circuit is made up of the fundamental battery model, the self-discharge resistance  $R_{SD}$ , the charge dynamics model, and the series resistance  $R_0$ .



## Battery Model

The block calculates the no-load voltage, or the voltage across the fundamental battery model by interpolation:

$$v_0 = v_0(\text{SOC}, T)$$

Where:

- $v_0$  is the no-load voltage of the battery. Specify the grid of lookup values using the **No-load voltage, V0(SOC,T)** parameter.
- SOC is the ratio of current charge to rated battery capacity. Specify the SOC breakpoints using the **Vector of state-of-charge values, SOC** parameter.
- $T$  is the battery temperature. Specify the  $T$  breakpoints using the **Vector of temperatures, T** parameter.

The block also models the series resistance  $R_0$  as a function of state of charge and temperature. Specify the grid of lookup values for the series resistance using the **Terminal resistance, R0(SOC,T)** parameter.

## Modeling Self-Discharge

When the battery terminals are open-circuit, it is still possible for internal currents to discharge the battery. This behavior is called self-discharge. To enable this effect, set the **Self-discharge** parameter to Enabled.

The block models these internal currents with a temperature-dependent resistance  $R_{SD}(T)$  across the terminals of the fundamental battery model. You can specify the lookup values for this resistance using the **Self-discharge resistance, Rleak(T)** parameter.

## Modeling Charge Dynamics

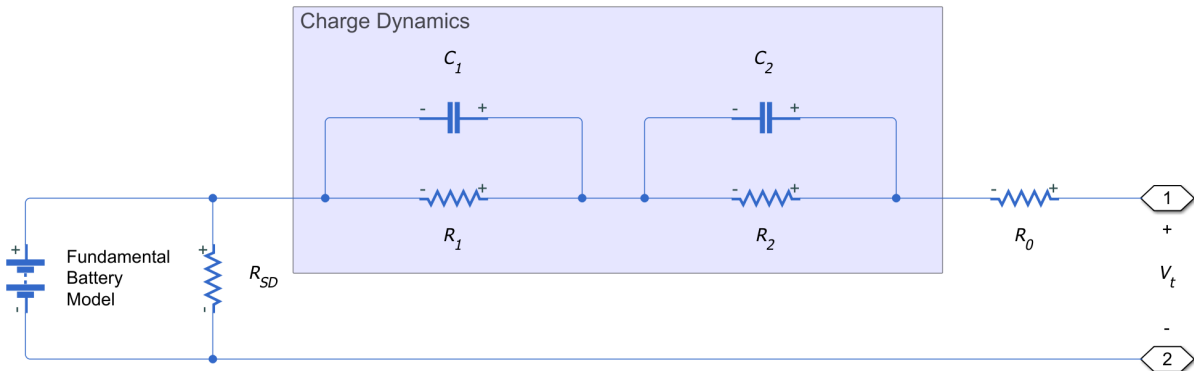
Batteries exhibit greater charging/discharging currents immediately after a change in load than they do after an extended period. This time-varying property is a result of battery charge dynamics and is modeled using parallel RC sections in the equivalent circuit.

You can model battery charge dynamics using the **Charge dynamics** parameter:

- **No dynamics** — The equivalent circuit contains no parallel RC sections. There is no delay between terminal voltage and internal charging voltage of the battery.
- **One time-constant dynamics** — The equivalent circuit contains one parallel RC section. Specify the time constant using the **First time constant, tau1(SOC,T)** parameter.
- **Two time-constant dynamics** — The equivalent circuit contains two parallel RC sections. Specify the time constants using the **First time constant, tau1(SOC,T)** and **Second time constant, tau2(SOC,T)** parameters.

- Three time-constant dynamics — The equivalent circuit contains three parallel RC sections. Specify the time constants using the **First time constant,  $\tau_1(\text{SOC}, T)$** , **Second time constant,  $\tau_2(\text{SOC}, T)$** , and **Third time constant,  $\tau_3(\text{SOC}, T)$**  parameters.
- Four time-constant dynamics — The equivalent circuit contains four parallel RC sections. Specify the time constants using the **First time constant,  $\tau_1(\text{SOC}, T)$** , **Second time constant,  $\tau_2(\text{SOC}, T)$** , **Third time constant,  $\tau_3(\text{SOC}, T)$** , and **Fourth time constant,  $\tau_4(\text{SOC}, T)$**  parameters.
- Five time-constant dynamics — The equivalent circuit contains five parallel RC sections. Specify the time constants using the **First time constant,  $\tau_1(\text{SOC}, T)$** , **Second time constant,  $\tau_2(\text{SOC}, T)$** , **Third time constant,  $\tau_3(\text{SOC}, T)$** , **Fourth time constant,  $\tau_4(\text{SOC}, T)$** , and **Fifth time constant,  $\tau_5(\text{SOC}, T)$**  parameters.

This diagram shows the equivalent circuit for the block configured with two time-constant dynamics.



In the diagram:

- $R_1$  and  $R_2$  are the parallel RC resistances. Specify these values with the **First polarization resistance,  $R_1(\text{SOC}, T)$**  and **Second polarization resistance,  $R_2(\text{SOC}, T)$**  parameters, respectively.
- $C_1$  and  $C_2$  are the parallel RC capacitances. The time constant  $\tau$  for each parallel section relates the  $R$  and  $C$  values using the relationship  $C = \tau / R$ . Specify  $\tau$  for each section using the **First time constant,  $\tau_1(\text{SOC}, T)$**  and **Second time constant,  $\tau_2(\text{SOC}, T)$**  parameters, respectively.

- $R_0$  is the series resistance. Specify this value with the **Internal resistance, R0(SOC,T)** parameter.

## Modeling Battery Fade

Battery fade is the deterioration of battery performance over repeated charge and discharge cycles. The no-load voltage across the fundamental battery model fades proportionally with the number of discharge cycles  $n$ :

$$v_{0,\text{fade}} = v_0 \left( 1 + \frac{\delta_{v_0}}{100} \frac{n}{N} \right)$$

Where:

- $N$  is the reference number of discharge cycles over which you specify percent change of various battery parameters. Set this value using the **Number of discharge cycles, N** parameter.
- $\delta_{v_0}$  is the percent change in no-load voltage after  $N$  discharge cycles. Specify  $\delta_{v_0}$  using the **Change in no-load voltage after N discharge cycles (%)** parameter.

The nominal charge, from which state of charge is calculated, fades with the square root of number of discharge cycles:

$$q_{\text{nom},\text{fade}} = q_{\text{nom}} \left( 1 + \frac{\delta_{AH}}{100} \sqrt{\frac{n}{N}} \right)$$

Where:

- $q_{\text{nom}}$  is the ampere-hour rating of the battery. Specify this value using the **Ampere-hour rating, AH(T)** parameter.
- $\delta_{AH}$  is the percent change in ampere-hour rating of the battery after  $N$  discharge cycles.

All resistances in the battery model also fade with the square root of the number of discharge cycles:

$$R_{i,\text{fade}} = R_i \left( 1 + \frac{\delta_{R_i}}{100} \sqrt{\frac{n}{N}} \right)$$

Where:

- $R_i$  is the  $i^{\text{th}}$  resistance
- $\delta_{R_i}$  is the percent change in this resistance over  $N$  cycles

Depending on how you have configured the block, the resistances might include:

- The series resistance — Specify the percent change over  $N$  cycles using the **Change in terminal resistance after N discharge cycles (%)** parameter.
- The self-discharge resistance — Specify the percent change over  $N$  cycles using the **Change in self-discharge resistance after N discharge cycles (%)** parameter.
- The first charge dynamics resistance — Specify the percent change over  $N$  cycles using the **Change in first polarization resistance after N discharge cycles (%)** parameter.
- The second charge dynamics resistance — Specify the percent change over  $N$  cycles using the **Change in second polarization resistance after N discharge cycles (%)** parameter.
- The third charge dynamics resistance — Specify the percent change over  $N$  cycles using the **Change in third polarization resistance after N discharge cycles (%)** parameter.
- The fourth charge dynamics resistance — Specify the percent change over  $N$  cycles using the **Change in fourth polarization resistance after N discharge cycles (%)** parameter.
- The fifth charge dynamics resistance — Specify the percent change over  $N$  cycles using the **Change in fifth polarization resistance after N discharge cycles (%)** parameter.

## Modeling Thermal Effects

For thermal variants of the block, the battery temperature is determined from a summation of all the ohmic losses included in the model:

$$M_{th} \dot{T} = \sum_i V_{T,i}^2 / R_{T,i}$$

Where:

- $M_{th}$  is the battery thermal mass.
- $i$  corresponds to the  $i^{th}$  ohmic loss contributor. Depending on how you have configured the block, the losses might include:
  - The series resistance
  - The self-discharge resistance
  - The first charge dynamics segment
  - The second charge dynamics segment
  - The third charge dynamics segment
  - The fourth charge dynamics segment
  - The fifth charge dynamics segment
- $V_{T_i}$  is the voltage drop across resistor  $i$ .
- $R_{T_i}$  is resistor  $i$ .

## Plotting Voltage-Charge Characteristics

A quick plot feature lets you visualize the voltage-charge characteristic for the battery model parameter values. To plot the characteristics, right-click a Battery (Table-Based) block in your model and, from the context menu, select **Electrical > Basic characteristics**. The software automatically computes a set of bias conditions, based on the block parameter values, and opens a figure window containing a plot of no-load voltage versus the state-of-charge (SOC) for the block. For more information, see “Plot Basic Characteristics for Battery Blocks”.

## Ports

### Output

#### SOC — Battery charge level, C

physical signal

Physical signal port that outputs the internal state of charge. Use this output port to change load behavior as a function of state of charge, without the complexity of building a charge state estimator.

### Dependencies

Enabled for the instrumented variants of the block: **Instrumented | No thermal port** and **Instrumented | Show thermal port**.

## Conserving

### + – Positive terminal

electrical

Electrical conserving port associated with the battery positive terminal.

### - – Negative terminal

electrical

Electrical conserving port associated with the battery negative terminal.

### H – Battery thermal mass

thermal

Thermal conserving port that represents the battery thermal mass.

### Dependencies

Enabled for the thermal variants of the block: **Uninstrumented | Show thermal port** and **Instrumented | Show thermal port**.

## Parameters

### Main

#### Vector of state-of-charge values, SOC – SOC breakpoints

[0, .25, .75, 1] (default) | vector

Vector of state-of-charge breakpoints defining the points at which you specify lookup data. This vector must be strictly ascending.

#### Vector of temperatures, T – T breakpoints

[273.15, 298.15, 323.15] K (default) | vector of positive numbers

Vector of temperature breakpoints defining the points at which you specify lookup data. This vector must be positive and strictly ascending.

**No-load voltage,  $V_0(\text{SOC}, T)$  —  $V_0$  lookup table**

[3.2, 3.1, 3.14; 3.25, 3.27, 3.3; 3.28, 3.31, 3.34; 3.33, 3.5, 3.59]  
V (default) | matrix of nonnegative numbers

Lookup data for no-load voltages across the fundamental battery model at the specified SOC and temperature breakpoints.

**Terminal resistance,  $R_0(\text{SOC}, T)$  —  $R_0$  lookup table**

[.03, .015, .002; .04, .017, .008; .039, .012, .006; .027, .013, .021] Ohm (default) | matrix of nonnegative numbers

Lookup data for series resistance of the battery at the specified SOC and temperature breakpoints.

**Ampere-hour rating,  $AH(T)$  — AH breakpoints**

[2.9, 4.1, 4.2] hr\*A (default) | vector of nonnegative numbers

Lookup data for the ampere-hour rating of the battery at the specified temperature breakpoints. The block calculates the state of charge by dividing the accumulated charge by this value. The block calculates accumulated charge by integrating the battery current.

**Self-discharge — Select whether to model the self-discharge resistance of the battery**

Disabled (default) | Enabled

Select whether to model the self-discharge resistance of the battery. The block models this effect as a resistor across the terminals of the fundamental battery model.

As temperature increases, self-discharge resistance decreases, causing self-discharge to increase. If the decrease in resistance is too fast, thermal runaway of the battery and numerical instability can occur. You can resolve this instability by making any of these changes:

- Decrease the thermal resistance
- Decrease the gradient of the self-discharge resistance with respect to temperature
- Increase the self-discharge resistance



### Self-discharge resistance, $R_{leak}(T)$ — Resistance that represents battery self-discharge

[8000, 7000, 6000] Ohm (default) | vector of positive numbers

Lookup data for self-discharge resistance of the battery at the specified temperature breakpoints. This resistance acts across the terminals of the fundamental battery model.

#### Dependencies

Enabled when the **Self-discharge** parameter is set to Enabled.

## Dynamics

### Charge dynamics — Battery charge dynamics model

No dynamics (default) | One time-constant dynamics | Two time-constant dynamics | Three time-constant dynamics | Four time-constant dynamics | Five time-constant dynamics

Select how to model battery charge dynamics. This parameter determines the number of parallel RC sections in the equivalent circuit:

- **No dynamics** — The equivalent circuit contains no parallel RC sections. There is no delay between terminal voltage and internal charging voltage of the battery.
- **One time-constant dynamics** — The equivalent circuit contains one parallel RC section. Specify the time constant using the **First time constant,  $\tau_1(\text{SOC}, T)$**  parameter.
- **Two time-constant dynamics** — The equivalent circuit contains two parallel RC sections. Specify the time constants using the **First time constant,  $\tau_1(\text{SOC}, T)$**  and **Second time constant,  $\tau_2(\text{SOC}, T)$**  parameters.
- **Three time-constant dynamics** — The equivalent circuit contains three parallel RC sections. Specify the time constants using the **First time constant,  $\tau_1(\text{SOC}, T)$** , **Second time constant,  $\tau_2(\text{SOC}, T)$** , and **Third time constant,  $\tau_3(\text{SOC}, T)$**  parameters.
- **Four time-constant dynamics** — The equivalent circuit contains four parallel RC sections. Specify the time constants using the **First time constant,  $\tau_1(\text{SOC}, T)$** , **Second time constant,  $\tau_2(\text{SOC}, T)$** , **Third time constant,  $\tau_3(\text{SOC}, T)$** , and **Fourth time constant,  $\tau_4(\text{SOC}, T)$**  parameters.
- **Five time-constant dynamics** — The equivalent circuit contains five parallel RC sections. Specify the time constants using the **First time constant,  $\tau_1(\text{SOC}, T)$** ,

**Second time constant, tau2(SOC,T), Third time constant, tau3(SOC,T), Fourth time constant, tau4(SOC,T), and Fifth time constant, tau5(SOC,T) parameters.**

**First polarization resistance, R1(SOC, T) — First RC resistance**

[.089, .076, .01; .042, .022, .099; .019, .007, .002; .051, .043, .029] Ohm (default) | matrix of positive numbers

Lookup data for the first parallel RC resistance at the specified SOC and temperature breakpoints. This parameter primarily affects the ohmic losses of the RC section.

**Dependencies**

To enable this parameter, set **Charge dynamics** to One time-constant dynamics, Two time-constant dynamics, Three time-constant dynamics, Four time-constant dynamics, or Five time-constant dynamics.

**First time constant, tau1(SOC,T) — First RC time constant**

[44, 148, 235; 93, 110, 1000; 19, 27, 133; .5, 22, 3] s (default) | matrix of positive numbers

Lookup data for the first parallel RC time constant at the specified SOC and temperature breakpoints.

**Dependencies**

To enable this parameter, set **Charge dynamics** to One time-constant dynamics, Two time-constant dynamics, Three time-constant dynamics, Four time-constant dynamics, or Five time-constant dynamics.

**Second polarization resistance, R2(SOC, T) — Second RC resistance**

[.014, .382, .407; .028, .006, .007; .014, .007, .006; .333, .956, .912] Ohm (default) | matrix of positive numbers

Lookup data for the second parallel RC resistance at the specified SOC and temperature breakpoints. This parameter primarily affects the ohmic losses of the RC section.

**Dependencies**

To enable this parameter, set **Charge dynamics** to Two time-constant dynamics, Three time-constant dynamics, Four time-constant dynamics, or Five time-constant dynamics.

**Second time constant, tau2(SOC,T) – Second RC time constant**

[1, 44, 5644; 11, 24, 506; 2, 14, 330; 3310, 13419, 30216] s (default) | matrix of positive numbers

Lookup data for the second parallel RC time constant at the specified SOC and temperature breakpoints.

**Dependencies**

To enable this parameter, set **Charge dynamics** to Two time-constant dynamics, Three time-constant dynamics, Four time-constant dynamics, or Five time-constant dynamics.

**Third polarization resistance, R3(SOC,T) – Third RC resistance**

[.014, .382, .407; .028, .006, .007; .014, .007, .006; .333, .956, .912] Ohm (default) | matrix of positive numbers

Lookup data for the third parallel RC resistance at the specified SOC and temperature breakpoints. This parameter primarily affects the ohmic losses of the RC section.

**Dependencies**

To enable this parameter, set **Charge dynamics** to Three time-constant dynamics, Four time-constant dynamics, or Five time-constant dynamics.

**Third time constant, tau3(SOC,T) – Third RC time constant**

[1, 44, 5644; 11, 24, 506; 2, 14, 330; 3310, 13419, 30216] s (default) | matrix of positive numbers

Lookup data for the third parallel RC time constant at the specified SOC and temperature breakpoints.

**Dependencies**

To enable this parameter, set **Charge dynamics** to Three time-constant dynamics, Four time-constant dynamics, or Five time-constant dynamics.

**Fourth polarization resistance, R4(SOC,T) – Fourth RC resistance**

[.014, .382, .407; .028, .006, .007; .014, .007, .006; .333, .956, .912] Ohm (default) | matrix of positive numbers

Lookup data for the fourth parallel RC resistance at the specified SOC and temperature breakpoints. This parameter primarily affects the ohmic losses of the RC section.

## Dependencies

To enable this parameter, set **Charge dynamics** to Four time-constant dynamics or Five time-constant dynamics.

### Fourth time constant, $\tau_4(\text{SOC}, T)$ — Fourth RC time constant

[1, 44, 5644; 11, 24, 506; 2, 14, 330; 3310, 13419, 30216] s (default) | matrix of positive numbers

Lookup data for the fourth parallel RC time constant at the specified SOC and temperature breakpoints.

## Dependencies

To enable this parameter, set **Charge dynamics** to Four time-constant dynamics or Five time-constant dynamics.

### Fifth polarization resistance, $R_5(\text{SOC}, T)$ — Fifth RC resistance

[.014, .382, .407; .028, .006, .007; .014, .007, .006; .333, .956, .912] Ohm (default) | matrix of positive numbers

Lookup data for the fifth parallel RC resistance at the specified SOC and temperature breakpoints. This parameter primarily affects the ohmic losses of the RC section.

## Dependencies

To enable this parameter, set **Charge dynamics** to Five time-constant dynamics.

### Fifth time constant, $\tau_5(\text{SOC}, T)$ — Fifth RC time constant

[1, 44, 5644; 11, 24, 506; 2, 14, 330; 3310, 13419, 30216] s (default) | matrix of positive numbers

Lookup data for the fifth parallel RC time constant at the specified SOC and temperature breakpoints.

## Dependencies

To enable this parameter, set **Charge dynamics** to Five time-constant dynamics.

## Fade

### Number of discharge cycles, $N$ — Reference number of cycles for percent change calculations

100 (default) | number from 1 to Inf

The number of charge-discharge cycles over which the specified percent changes occur.

**Change in no-load voltage after  $N$  discharge cycles (%) – Percent change in no-load voltage after  $N$  cycles**

0 (default) | scalar

Percent change in the no-load voltage after the battery undergoes  $N$  discharge cycles.

**Change in terminal resistance after  $N$  discharge cycles (%) – Percent change in series resistance after  $N$  cycles**

0 (default) | scalar

Percent change in the series resistance after the battery undergoes  $N$  discharge cycles.

**Change in ampere-hour rating after  $N$  discharge cycles (%) – Percent change in ampere-hour rating after  $N$  cycles**

0 (default) | scalar

Percent change in the ampere-hour rating after the battery undergoes  $N$  discharge cycles.

**Change in self-discharge resistance after  $N$  discharge cycles (%) – Percent change in self-discharge resistance after  $N$  cycles**

0 (default) | scalar

Percent change in the self-discharge resistance after the battery undergoes  $N$  discharge cycles.

**Dependencies**

To enable this parameter, set **Self-discharge** to Enabled.

**Change in first polarization resistance after  $N$  discharge cycles (%) – Percent change in first RC resistance after  $N$  cycles**

0 (default) | scalar

Percent change in the first RC resistance after the battery undergoes  $N$  discharge cycles.

**Dependencies**

To enable this parameter, set **Charge dynamics** to One time-constant dynamics, Two time-constant dynamics, Three time-constant dynamics, Four time-constant dynamics, or Five time-constant dynamics.

**Change in second polarization resistance after  $N$  discharge cycles (%) — Percent change in second RC resistance after  $N$  cycles**

0 (default) | scalar

Percent change in the second RC resistance after the battery undergoes  $N$  discharge cycles.

**Dependencies**

To enable this parameter, set **Charge dynamics** to Two time-constant dynamics, Three time-constant dynamics, Four time-constant dynamics, or Five time-constant dynamics.

**Change in third polarization resistance after  $N$  discharge cycles (%) — Percent change in third RC resistance after  $N$  cycles**

0 (default) | scalar

Percent change in the third RC resistance after the battery undergoes  $N$  discharge cycles.

**Dependencies**

To enable this parameter, set **Charge dynamics** to Three time-constant dynamics, Four time-constant dynamics, or Five time-constant dynamics.

**Change in fourth polarization resistance after  $N$  discharge cycles (%) — Percent change in fourth RC resistance after  $N$  cycles**

0 (default) | scalar

Percent change in the fourth RC resistance after the battery undergoes  $N$  discharge cycles.

**Dependencies**

To enable this parameter, set **Charge dynamics** to Four time-constant dynamics, or Five time-constant dynamics.

**Change in fifth polarization resistance after  $N$  discharge cycles (%) — Percent change in fifth RC resistance after  $N$  cycles**

0 (default) | scalar

Percent change in the fifth RC resistance after the battery undergoes  $N$  discharge cycles.

**Dependencies**

To enable this parameter, set **Charge dynamics** to Five time-constant dynamics.

## Thermal

### **Simulation temperature — Battery temperature**

298.15 K (default) | positive number

Battery temperature used in lookup tables throughout simulation.

#### **Dependencies**

This section appears only for blocks without an exposed thermal port. For more information, see “Modeling Thermal Effects” on page 1-129.

### **Thermal mass — Thermal mass associated with the thermal port**

30000 J/K (default)

Thermal mass associated with the thermal port H. It represents the energy required to raise the temperature of the thermal port by one degree.

#### **Dependencies**

Enabled for blocks with an exposed thermal port. For more information, see “Modeling Thermal Effects” on page 1-129.

## See Also

Battery

**Introduced in R2018a**

## Bidirectional DC-DC Converter

Controller-driven bidirectional DC-DC step-up and step-down voltage regulator

**Library:** Simscape / Electrical / Power Systems / Semiconductors



### Description

The Bidirectional DC-DC Converter block represents a converter that steps up or steps down DC voltage from either side of the converter to the other as driven by an attached controller and gate-signal generator. Bidirectional DC-DC converters are useful for switching between energy storage and use, for example, in electric vehicles.

The Bidirectional DC-DC Converter block allows you to model a nonisolated converter with two switching devices or an isolated converter with six switching devices. Options for the type of switching devices are:

- GTO — Gate turn-off thyristor. For information on the I-V characteristic of the device, see GTO.
- Ideal semiconductor switch — For information on the I-V characteristic of the device, see Ideal Semiconductor Switch.
- IGBT — Insulated-gate bipolar transistor. For information on the I-V characteristic of the device, see IGBT.
- MOSFET — N-channel metal-oxide-semiconductor field-effect transistor. For information on the I-V characteristic of the device, see MOSFET.
- Thyristor — For information on the I-V characteristic of the device, see Thyristor.

### Model

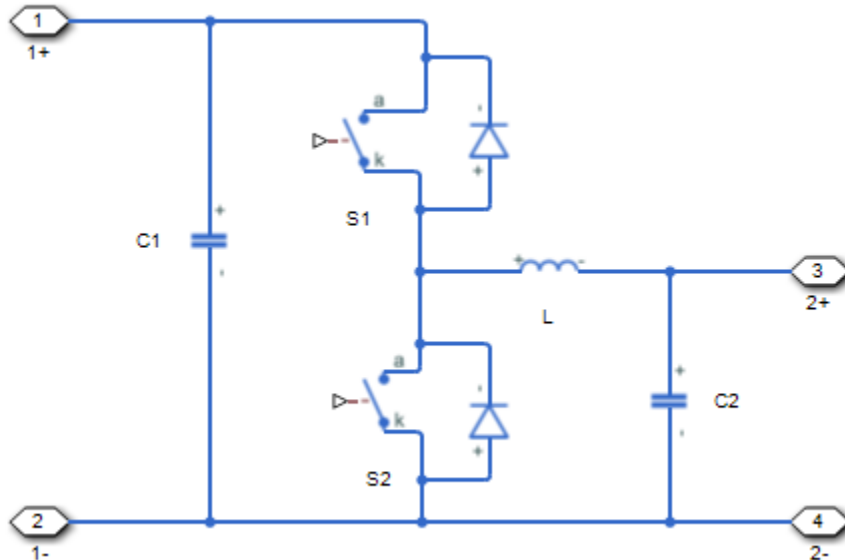
Each of two model variants for the Bidirectional DC-DC Converter block corresponds to a **Block choice** option. To access the block choices, in the model window, right-click the block, and then use either of these methods:



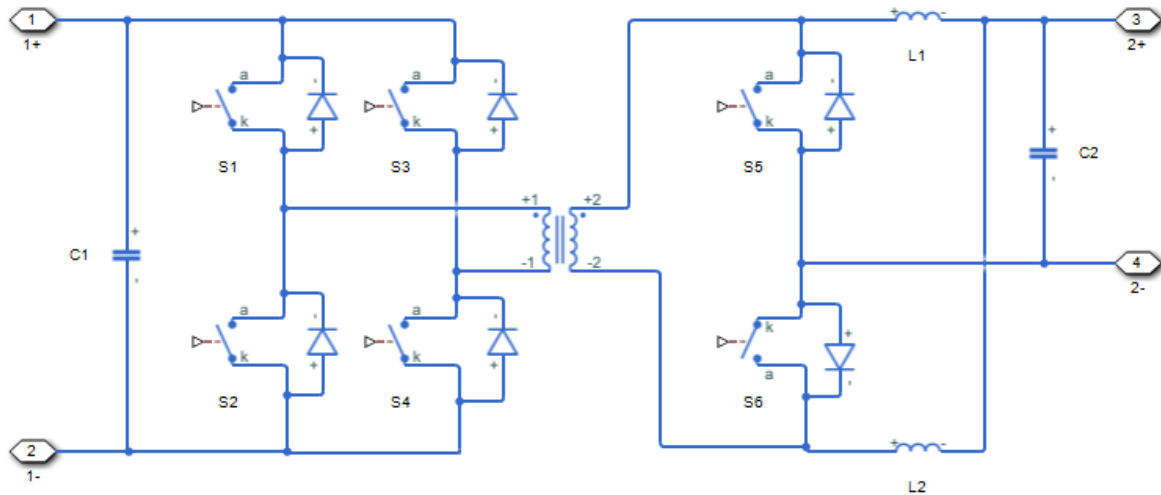
- From the context menu, select **Simscape > Block choices**.
- On the Simulink® Editor menu bar, select **View > Property Inspector**. In the **Property Inspector** window, click the value of the **Block choice**.

The model variants are:

- Nonisolated converter — Bidirectional DC-DC converter without an electrical barrier. This model variant contains an inductor, two capacitors, and two switches that are of the same device type. This block choice is the default.



- Isolated converter — Bidirectional DC-DC converter with an electrical barrier. This model variant contains four additional switches that form a full bridge. The full bridge is on the input or high-voltage (HV) side of the converter. The other two switches are on the output or low-voltage (LV) side of the converter. You can select different semiconductor types for the HV and LV switching devices. For example, you can use a GTO for the HV switching devices and an IGBT for the LV switching devices. To provide separation between the input and output voltages, the model uses a high-frequency transformer.



## Protection

The block contains an integral protection diode for each switching device. The integral diode protects the semiconductor device by providing a conduction path for reverse current. An inductive load can produce a high reverse-voltage spike when the semiconductor device suddenly switches off the voltage supply to the load.

To configure the internal protection diode block, use the **Protection Diode** parameters. This table shows how to set the **Model dynamics** parameter based on your goals.

Goals	Value to Select	Integral Protection Diode
Prioritize simulation speed.	Protection diode with no dynamics	The Diode block
Prioritize model fidelity by precisely specifying reverse-mode charge dynamics.	Protection diode with charge dynamics	The Commutation Diode block

You can also include a snubber circuit for each switching device. Snubber circuits contain a series-connected resistor and capacitor. They protect switching devices against high voltages that inductive loads produce when the device turns off the voltage supply to the

load. Snubber circuits also prevent excessive rates of current change when a switching device turns on.

To include and configure a snubber circuit for each switching device, use the **Snubbers** parameters.

## Gate Control

To connect Simulink gate-control voltage signals to the gate ports of the switching devices:

- 1 Convert each voltage signal using a Simulink-PS Converter block.
- 2 Multiplex the converted gate signals into a single vector. For a nonisolated converter model, use a Two-Pulse Gate Multiplexer block. For an isolated converter model, use a Six-Pulse Gate Multiplexer block.
- 3 Connect the vector signal to the **G** port.

## Assumptions

A source impedance or a nonzero equivalent-series resistance (ESR) is connected to the left side of the Bidirectional DC-DC Converter block.

## Ports

### Conserving

#### **G — Switching device gate control**

electrical | vector

Electrical conserving port associated with the gate terminals of the switching devices.

Data Types: double

#### **1+ — Positive DC voltage 1**

electrical | scalar

Electrical conserving port associated with the positive terminal of the first DC voltage.

Data Types: double

**1- — Negative DC voltage 1**

electrical | scalar

Electrical conserving port associated with the negative terminal of the first DC voltage.

Data Types: double

**2+ — Positive DC voltage 2**

electrical | scalar

Electrical conserving port associated with the positive terminal of the second DC voltage.

Data Types: double

**2- — Negative DC voltage 2**

electrical | scalar

Electrical conserving port associated with the negative terminal of the second DC voltage.

Data Types: double

## Parameters

### Switching Devices

These tables show how the visibility of **Switching Devices** parameters depends on the converter model and switching devices that you select. To learn how to read the table, see “Parameter Dependencies” on page A-2.

### Nonisolated Converter Switching Devices Parameter Dependencies

Parameters and Options				
Switching device				
<i>Ideal Semiconductor Switch</i>	<i>GTO</i>	<i>IGBT</i>	<i>MOSFET</i>	<i>Thyristor</i>
On-state resistance	Forward voltage	Forward voltage	Drain-source on resistance	Forward voltage
Off-state conductance	On-state resistance	On-state resistance	Off-state conductance	On-state resistance
Threshold voltage	Off-state conductance	Off-state conductance	Threshold voltage	Off-state conductance
	Gate trigger voltage, $V_{gt}$	Threshold voltage		Gate trigger voltage, $V_{gt}$
	Gate turn-off voltage, $V_{gt\_off}$			Gate turn-off voltage, $V_{gt\_off}$
	Holding current			Holding current

### Isolated Converter Switching Devices Parameter Dependencies

<b>Parameters and Options</b>				
<b>Switching device HV</b>				
<i>Ideal Semiconductor Switch</i>	<i>GTO</i>	<i>IGBT</i>	<i>MOSFET</i>	<i>Thyristor</i>
On-state resistance HV	Forward voltage HV	Forward voltage HV	Drain-source on resistance HV	Forward voltage HV
Off-state conductance HV	On-state resistance HV	On-state resistance HV	Off-state conductance HV	On-state resistance HV
Threshold voltage HV	Off-state conductance HV	Off-state conductance HV	Threshold voltage HV	Off-state conductance HV
	Gate trigger voltage HV, Vgt_hv	Threshold voltage HV		Gate trigger voltage HV, Vgt_hv
	Gate turn-off voltage HV, Vgt_off_hv			Gate turn-off voltage HV, Vgt_off_hv
	Holding current HV			Holding current HV
<b>Switching device LV</b>				
<i>Ideal Semiconductor Switch</i>	<i>GTO</i>	<i>IGBT</i>	<i>MOSFET</i>	<i>Thyristor</i>
On-state resistance LV	Forward voltage LV	Forward voltage LV	Drain-source on resistance LV	Forward voltage LV
Off-state conductance LV	On-state resistance LV	On-state resistance LV	Off-state conductance LV	On-state resistance LV
Threshold voltage LV	Off-state conductance LV	Off-state conductance LV	Threshold voltage LV	Off-state conductance LV
	Gate trigger voltage LV, Vgt	Threshold voltage LV		Gate trigger voltage LV, Vgt

Parameters and Options				
	Gate turn-off voltage LV, Vgt_off_lv			Gate turn-off voltage LV, Vgt_off_lv
	Holding current LV			Holding current LV

### Switching device – Switch type

Ideal Semiconductor Switch (default) | GTO | IGBT | MOSFET | Thyristor

Switching device type for the nonisolated converter model.

### Dependencies

See the Nonisolated Converter Switching Devices Parameter Dependencies table.

### Forward voltage – Voltage

0.8 V (default) | scalar

For the different switching device types, the **Forward voltage** is taken as:

- GTO — Minimum voltage required across the anode and cathode block ports for the gradient of the device I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**
- IGBT — Minimum voltage required across the collector and emitter block ports for the gradient of the diode i-v characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**
- Thyristor — Minimum voltage required for the device to turn on

### Dependencies

See the Nonisolated Converter Switching Devices Parameter Dependencies table.

### On-state resistance – Resistance

0.001 Ohm (default) | scalar

For the different switching device types, the **On-state resistance** is taken as:

- GTO — Rate of change of voltage versus current above the forward voltage
- Ideal semiconductor switch — Anode-cathode resistance when the device is on

- IGBT — Collector-emitter resistance when the device is on
- Thyristor — Anode-cathode resistance when the device is on

### **Dependencies**

See the Nonisolated Converter Switching Devices Parameter Dependencies table.

### **Drain-source on resistance — Resistance**

0.001 Ohm (default) | scalar

Resistance between the drain and the source, which also depends on the gate-to-source voltage.

### **Dependencies**

See the Nonisolated Converter Switching Devices Parameter Dependencies table.

### **Off-state conductance — Conductance**

1e-5 1/Ohm (default) | scalar

Conductance when the device is off. The value must be less than  $1/R$ , where  $R$  is the value of **On-state resistance**.

For the different switching device types, the **On-state resistance** is taken as:

- GTO — Anode-cathode conductance
- Ideal semiconductor switch — Anode-cathode conductance
- IGBT — Collector-emitter conductance
- MOSFET — Drain-source conductance
- Thyristor — Anode-cathode conductance

### **Dependencies**

See the Nonisolated Converter Switching Devices Parameter Dependencies table.

### **Threshold voltage — Voltage threshold**

6 V (default) | scalar

Gate voltage threshold. The device turns on when the gate voltage is above this value. For the different switching device types, the device voltage of interest is:



- Ideal semiconductor switch — Gate-emitter voltage
- IGBT — Gate-cathode voltage
- MOSFET — Gate-source voltage

### Dependencies

See the Nonisolated Converter Switching Devices Parameter Dependencies table.

### Gate trigger voltage, $V_{gt}$ — Voltage threshold

1 V (default) | scalar

Gate-cathode voltage threshold. The device turns on when the gate-cathode voltage is above this value.

### Dependencies

See the Nonisolated Converter Switching Devices Parameter Dependencies table.

### Gate turn-off voltage, $V_{gt\_off}$ — Voltage threshold

-1 V (default) | scalar

Gate-cathode voltage threshold. The device turns off when the gate-cathode voltage is below this value.

### Dependencies

See the Nonisolated Converter Switching Devices Parameter Dependencies table.

### Holding current — Current threshold

1 A (default) | scalar

Gate current threshold. The device stays on when the current is above this value, even when the gate-cathode voltage falls below the gate trigger voltage.

### Dependencies

See the Nonisolated Converter Switching Devices Parameter Dependencies table.

### Switching device HV — Switch

Ideal Semiconductor Switch (default) | GTO | IGBT | MOSFET | Thyristor

Switching device type for the high-voltage side of the isolated converter model.

## Dependencies

See the Isolated Converter Switching Devices Parameter Dependencies table.

## Forward voltage HV — Voltage

0.8 Ohm (default) | scalar

For the different switching device types, the **Forward voltage HV** is taken as:

- GTO — Minimum voltage required across the anode and cathode block ports for the gradient of the device I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**
- IGBT — Minimum voltage required across the collector and emitter block ports for the gradient of the diode i-v characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**
- Thyristor — Minimum voltage required for the device to turn on

## Dependencies

See the Isolated Converter Switching Devices Parameter Dependencies table.

## Drain-source on resistance HV — Resistance

0.001 Ohm (default) | scalar

Resistance between the drain and the source, which also depends on the gate-to-source voltage.

## Dependencies

See the Isolated Converter Switching Devices Parameter Dependencies table.

## On-state resistance HV — Resistance

0.001 Ohm (default) | scalar

For the different switching device types, the **On-state resistance HV** is taken as:

- GTO — Rate of change of voltage versus current above the forward voltage
- Ideal semiconductor switch — Anode-cathode resistance when the device is on
- IGBT — Collector-emitter resistance when the device is on
- Thyristor — Anode-cathode resistance when the device is on

**Dependencies**

See the Isolated Converter Switching Devices Parameter Dependencies table.

**Off-state conductance HV – Conductance**

1e-5 1/0hm (default) | scalar

Conductance when the device is off. The value must be less than  $1/R$ , where  $R$  is the value of **On-state resistance HV**.

For the different switching device types, the **On-state resistance HV** is taken as:

- GTO — Anode-cathode conductance
- Ideal semiconductor switch — Anode-cathode conductance
- IGBT — Collector-emitter conductance
- MOSFET — Drain-source conductance
- Thyristor — Anode-cathode conductance

**Dependencies**

See the Isolated Converter Switching Devices Parameter Dependencies table.

**Threshold voltage HV – Voltage threshold**

6 V (default) | scalar

Gate voltage threshold. The device turns on when the gate voltage is above this value. For the different switching device types, the device voltage of interest is:

- Ideal semiconductor switch — Gate-emitter voltage
- IGBT — Gate-cathode voltage
- MOSFET — Gate-source voltage

**Dependencies**

See the Isolated Converter Switching Devices Parameter Dependencies table.

**Gate trigger voltage HV, Vgt\_hv – Voltage threshold**

1 V (default) | scalar

Gate-cathode voltage threshold. The device turns on when the gate-cathode voltage is above this value.

## Dependencies

See the Isolated Converter Switching Devices Parameter Dependencies table.

## Gate turn-off voltage HV, `Vgt_off_hv` — Voltage threshold

-1 V (default) | scalar

Gate-cathode voltage threshold. The device turns off when the gate-cathode voltage is below this value.

## Dependencies

See the Isolated Converter Switching Devices Parameter Dependencies table.

## Holding current HV — Current threshold

1 A (default) | scalar

Gate current threshold. The device stays on when the current is above this value, even when the gate-cathode voltage falls below the gate trigger voltage.

## Dependencies

See the Isolated Converter Switching Devices Parameter Dependencies table.

## Switching device LV — Switch

Ideal Semiconductor Switch (default) | GTO | IGBT | MOSFET | Thyristor

Switching device type for the low-voltage side of the isolated converter model.

## Dependencies

See the Isolated Converter Switching Devices Parameter Dependencies table.

## Forward voltage LV — Voltage

0.8 Ohm (default) | scalar

For the different switching device types, the **Forward voltage LV** is taken as:

- GTO — Minimum voltage required across the anode and cathode block ports for the gradient of the device I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**
- IGBT — Minimum voltage required across the collector and emitter block ports for the gradient of the diode i-v characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**

- Thyristor — Minimum voltage required for the device to turn on

### Dependencies

See the Isolated Converter Switching Devices Parameter Dependencies table.

### Drain-source on resistance LV — Resistance

0.001 Ohm (default) | scalar

Resistance between the drain and the source, which also depends on the gate-to-source voltage.

### Dependencies

See the Isolated Converter Switching Devices Parameter Dependencies table.

### On-state resistance LV — Resistance

0.001 Ohm (default) | scalar

For the different switching device types, the **On-state resistance LV** is taken as:

- GTO — Rate of change of voltage versus current above the forward voltage
- Ideal semiconductor switch — Anode-cathode resistance when the device is on
- IGBT — Collector-emitter resistance when the device is on
- Thyristor — Anode-cathode resistance when the device is on

### Dependencies

See the Isolated Converter Switching Devices Parameter Dependencies table.

### Off-state conductance LV — Conductance

1e-5 1/Ohm (default) | scalar

Conductance when the device is off. The value must be less than  $1/R$ , where  $R$  is the value of **On-state resistance LV**.

For the different switching device types, the **On-state resistance LV** is taken as:

- GTO — Anode-cathode conductance
- Ideal semiconductor switch — Anode-cathode conductance
- IGBT — Collector-emitter conductance

- MOSFET — Drain-source conductance
- Thyristor — Anode-cathode conductance

### **Dependencies**

See the Isolated Converter Switching Devices Parameter Dependencies table.

### **Threshold voltage LV — Voltage threshold**

6 V (default) | scalar

Gate voltage threshold. The device turns on when the gate voltage is above this value. For the different switching device types, the device voltage of interest is:

- Ideal semiconductor switch — Gate-emitter voltage
- IGBT — Gate-cathode voltage
- MOSFET — Gate-source voltage

### **Dependencies**

See the Isolated Converter Switching Devices Parameter Dependencies table.

### **Gate trigger voltage LV, Vgt\_lv — Voltage threshold**

1 V (default) | scalar

Gate-cathode voltage threshold. The device turns on when the gate-cathode voltage is above this value.

### **Dependencies**

See the Isolated Converter Switching Devices Parameter Dependencies table.

### **Gate turn-off voltage LV, Vgt\_off\_lv — Voltage threshold**

-1 V (default) | scalar

Gate-cathode voltage threshold. The device turns off when the gate-cathode voltage is below this value.

### **Dependencies**

See the Isolated Converter Switching Devices Parameter Dependencies table.

### **Holding current LV — Current threshold**

1 A (default) | scalar

Gate current threshold. The device stays on when the current is above this value, even when the gate-cathode voltage falls below the gate trigger voltage.

### Dependencies

See the Isolated Converter Switching Devices Parameter Dependencies table.

## Protection Diode

The visibility of **Protection Diode** parameters depends on how you configure the protection diode **Model dynamics** and **Reverse recovery time parameterization** parameters. To learn how to read this table, see “Parameter Dependencies” on page A-2.

### Protection Diode Parameter Dependencies

Parameters and Options			
<b>Model dynamics</b>			
<i>Protection diode with no dynamics</i>	<i>Protection diode with charge dynamics</i>		
Forward voltage	Forward voltage		
On resistance	On resistance		
Off conductance	Off conductance		
	Junction capacitance		
	Peak reverse current, iRM		
	Initial forward current when measuring iRM		
	Rate of change of current when measuring iRM		
	<b>Reverse recovery time parameterization</b>		
	<i>Specify stretch factor</i>	<i>Specify reverse recovery time directly</i>	<i>Specify reverse recovery charge</i>
	Reverse recovery time stretch factor	Reverse recovery time, trr	Reverse recovery charge, Qrr

### Model dynamics – Diode model

Protection diode with no dynamics (default) | Protection diode with charge dynamics

Diode type. The options are:

- **Diode with no dynamics** — Select this option to prioritize simulation speed using the Diode block.
- **Diode with charge dynamics** — Select this option to prioritize model fidelity in terms of reverse mode charge dynamics using the Commutation Diode block.

### **Dependencies**

See the Protection Diode Parameter Dependencies table.

### **Forward voltage — Voltage**

0.8 V (default) | scalar

Minimum voltage required across the positive and negative block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**.

### **On resistance — Resistance**

0.001 Ohm (default) | scalar

Rate of change of voltage versus current above the **Forward voltage**.

### **Off conductance — Conductance**

1e-5 1/Ohm (default) | scalar

Conductance of the reverse-biased diode.

### **Junction capacitance — Capacitance**

50 nF (default) | scalar

Diode junction capacitance.

### **Dependencies**

See the Protection Diode Parameter Dependencies table.

### **Peak reverse current, $i_{RM}$ — Current**

-235 A (default) | scalar less than 0

Peak reverse current measured by an external test circuit.

### **Dependencies**

See the Protection Diode Parameter Dependencies table.



**Initial forward current when measuring  $i_{RM}$  – Current**

300 A (default) | scalar greater than 0

Initial forward current when measuring peak reverse current. This value must be greater than zero.

**Dependencies**

See the Protection Diode Parameter Dependencies table.

**Rate of change of current when measuring  $i_{RM}$  – Current change rate**

-50 A/us (default) | scalar

Rate of change of current when measuring peak reverse current.

**Dependencies**

See the Protection Diode Parameter Dependencies table.

**Reverse recovery time parameterization – Recovery-time model**

Specify stretch factor (default) | Specify reverse recovery time directly | Specify reverse recovery charge

Model for parameterizing the recovery time. When you select **Specify stretch factor** or **Specify reverse recovery charge**, you can specify a value that the block uses to derive the reverse recovery time. For more information on these options, see “Alternatives to Specifying  $t_{rr}$  Directly” on page 1-339.

**Dependencies**

See the Protection Diode Parameter Dependencies table.

**Reverse recovery time stretch factor – Stretch factor**

3 (default) | scalar greater than 1

Value that the block uses to calculate **Reverse recovery time,  $t_{rr}$** . Specifying the stretch factor is an easier way to parameterize the reverse recovery time than specifying the reverse recovery charge. The larger the value of the stretch factor, the longer it takes for the reverse recovery current to dissipate.

**Dependencies**

See the Protection Diode Parameter Dependencies table.

## Reverse recovery time, $t_{rr}$ — Time

15 us (default) | scalar

Interval between the time when the current initially goes to zero (when the diode turns off) and the time when the current falls to less than 10 percent of the peak reverse current.

The value of the **Reverse recovery time,  $t_{rr}$**  parameter must be greater than the value of the **Peak reverse current,  $i_{RM}$**  parameter divided by the value of the **Rate of change of current when measuring  $i_{RM}$**  parameter.

### Dependencies

See the Protection Diode Parameter Dependencies table.

## Reverse recovery charge, $Q_{rr}$ — Charge

1500 s\*uA (default) | scalar

Value that the block uses to calculate **Reverse recovery time,  $t_{rr}$** . Use this parameter if the data sheet for your diode device specifies a value for the reverse recovery charge instead of a value for the reverse recovery time.

The reverse recovery charge is the total charge that continues to dissipate when the

diode turns off. The value must be less than  $-\frac{i_{RM}^2}{2a}$ ,

where:

- $i_{RM}$  is the value specified for **Peak reverse current,  $i_{RM}$** .
- $a$  is the value specified for **Rate of change of current when measuring  $i_{RM}$** .

### Dependencies

See the Protection Diode Parameter Dependencies table.

## Transformer

The **Transformer** parameters are only visible when **Block choice** is set to Isolated converter.

## Transformer inductance $L_1$ — Inductance

10 H (default) | positive scalar

Self-inductance of the first winding of the transformer.

**Dependencies**

This parameter is only visible when **Block choice** is set to `Isolated converter`.

**Transformer inductance L2 – Inductance**

0.1 H (default) | positive scalar

Self-inductance of the second winding of the transformer.

**Dependencies**

This parameter is only visible when **Block choice** is set to `Isolated converter`.

**Transformer coefficient of coupling – Coupling coefficient**

0.9 (default) | positive scalar greater than zero and less than 1

Defines the mutual inductance of the transformer.

**Dependencies**

This parameter is only visible when **Block choice** is set to `Isolated converter`.

## LC Parameters

**Inductance, L – Inductance**

1e-6 H (default) | positive scalar

Converter inductance. For the isolated converter model variant, the two inductors are identical.

**Capacitance, C1 – Capacitance**

1e-7 F (default) | positive scalar

Capacitance of the first DC terminal.

**Capacitance, C2 – Capacitance**

1e-7 F (default) | positive scalar

Capacitance of the second DC terminal.

**C1 effective series resistance – Resistance**

1e-6 Ohm (default) | zero or positive scalar

Series resistance of capacitor  $C1$ .

**C2 effective series resistance — Resistance**

1e-6 Ohm (default) | zero or positive scalar

Series resistance of capacitor  $C2$ .

**Snubbers**

The table summarizes the **Snubbers** parameter dependencies. To learn how to read the table, see “Parameter Dependencies” on page A-2.

**Snubbers Parameter Dependencies**

<b>Snubbers Parameter Dependencies</b>			
<b>Block choice</b>			
<i>Nonisolated Converter</i>		<i>Isolated Converter</i>	
<b>Snubber</b>		<b>Snubber HV</b>	
<i>None</i>	<i>RC Snubber</i>	<b>None</b>	<i>RC Snubber</i>
	Snubber resistance		Snubber resistance HV
	Snubber capacitance		Snubber capacitance HV
		<b>Snubber LV</b>	
		<i>None</i>	<i>RC Snubber</i>
			Snubber resistance LV
			Snubber capacitance LV

**Snubber — Snubber model**

None (default) | RC snubber

Snubber for each switching device.

**Dependencies**

See the Snubbers Parameter Dependencies table.

**Snubber resistance — Resistance**

0.1 Ohm (default) | scalar

Resistance of the snubbers.

**Dependencies**

See the Snubbers Parameter Dependencies table.

**Snubber capacitance — Capacitance**

1e-7 F (default) | scalar

Capacitance of the snubbers.

**Dependencies**

See the Snubbers Parameter Dependencies table.

**Snubber HV — Snubber model**

None (default) | RC snubber

HV snubber for each switching device.

**Dependencies**

See the Snubbers Parameter Dependencies table.

**Snubber resistance HV — Resistance**

0.1 Ohm (default) | scalar

Resistance of the high-voltage snubbers.

**Dependencies**

See the Snubbers Parameter Dependencies table.

**Snubber capacitance HV — Capacitance**

1e-7 F (default) | scalar

Capacitance of the high-voltage snubbers.

**Dependencies**

See the Snubbers Parameter Dependencies table.

## **Snubber LV — Snubber model**

None (default) | RC snubber

LV snubber for each switching device.

### **Dependencies**

See the Snubbers Parameter Dependencies table.

## **Snubber resistance LV — Resistance**

0.1 Ohm (default) | scalar

Resistance of the low-voltage snubbers.

### **Dependencies**

See the Snubbers Parameter Dependencies table.

## **Snubber capacitance LV — Capacitance**

1e-7 F (default) | scalar

Capacitance of the low-voltage snubbers.

### **Dependencies**

See the Snubbers Parameter Dependencies table.

## **References**

- [1] Saleh, M., Y. Esa, Y. Mhandi, W. Brandauer, and A. Mohamed. *Design and implementation of CCNY DC microgrid testbed*. Industry Applications Society Annual Meeting. Portland, OR: 2016, pp 1-7.
- [2] Kutkut, N. H., and G. Luckjiff. *Current mode control of a full bridge DC-to-DC converter with a two inductor rectifier*. Power Electronics Specialists Conference. Saint Louis, MO: 1997, pp 203-209.
- [3] Nene, H. *Digital control of a bi-directional DC-DC converter for automotive applications*. Twenty-Eighth Annual IEEE Applied Power Electronics Conference and Exposition (APEC). Long Beach, CA: 2013, pp 1360-1365.

## See Also

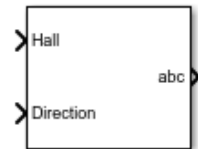
Average-Value DC-DC Converter | Boost Converter | Buck Converter | Buck-Boost Converter | Converter | GTO | IGBT | Ideal Semiconductor Switch | MOSFET | PWM Generator | PWM Generator (Three-phase, Two-level) | Six-Pulse Gate Multiplexer | Three-Level Converter | Thyristor

**Introduced in R2018a**

## BLDC Commutation Logic

Commutation logic for brushless DC motors

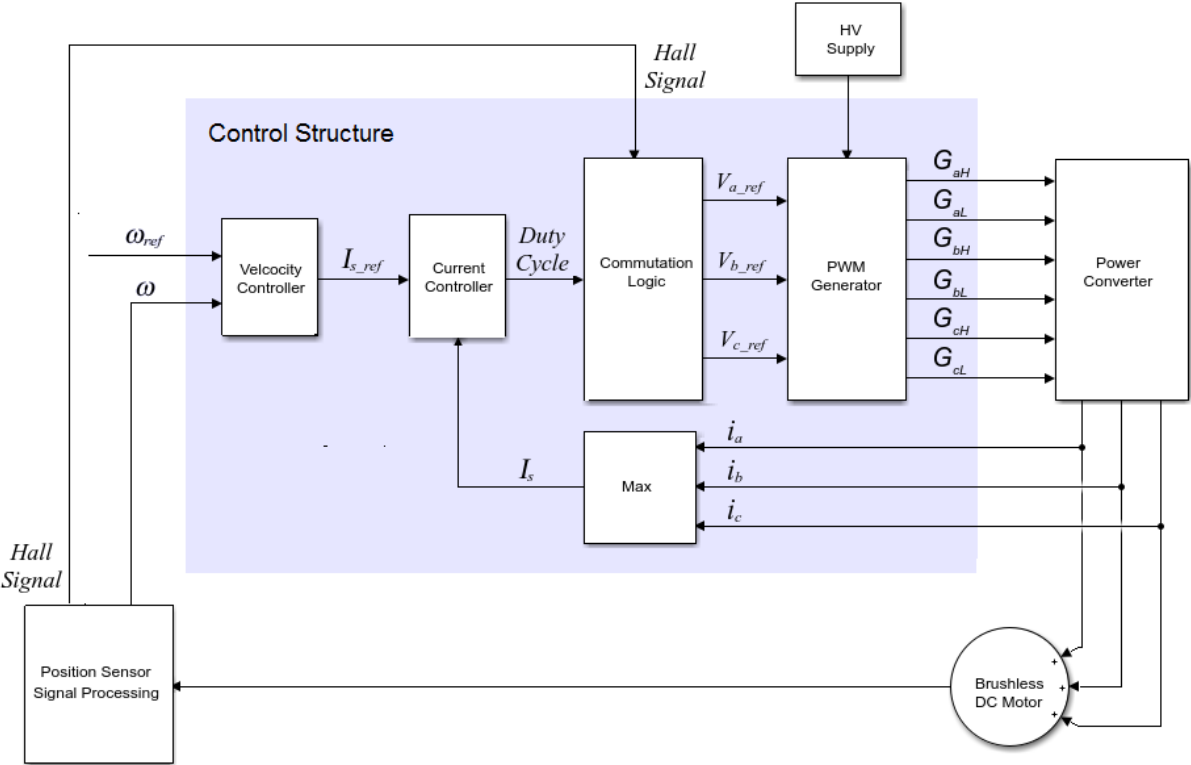
**Library:** Simscape / Electrical / Power Systems / Control /  
BLDC Control



### Description

The BLDC Commutation Logic block implements a commutation logic for brushless DC motors as part of this control algorithm.





The commutation logic is based on the Hall signals as summarized in this table.

Hall Sensors			Motor Phases		
Hall a	Hall b	Hall c	Phase a	Phase b	Phase c
0	0	0	0	0	0
1	1	0	0	1	-1
0	1	0	-1	1	0
0	1	1	-1	0	1
0	0	1	0	-1	1
1	0	1	1	-1	0

Hall Sensors			Motor Phases		
1	0	0	1	0	-1
1	1	1	0	0	0

## Ports

### Input

#### **Hall — Hall sensor**

vector

Hall sensor data.

Data Types: single | double

#### **Direction — Motor direction**

scalar

Direction of motor rotation.

Data Types: single | double

### Output

#### **abc — Motor phase**

vector

Motor phase indicated by the commutation logic.

Data Types: single | double

## Parameters

#### **Sample time (-1 for inherited) — Block sample time**

-1 (default) | positive scalar

Time, in s, between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

If this block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly using a positive scalar.

## References

- [1] Stirban, A., I. Boldea, and G. D. Andreescu. "Motion-Sensorless Control of BLDC-PM Motor With Offline FEM-Information-Assisted Position and Speed Observer." *IEEE Transactions on Industry Applications*. 48, no. 6 (2012): 1950-1958.

## See Also

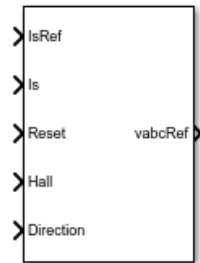
BLDC Current Controller | BLDC Current Controller with PWM Generation | Brushless DC Motor

**Introduced in R2018a**

## BLDC Current Controller

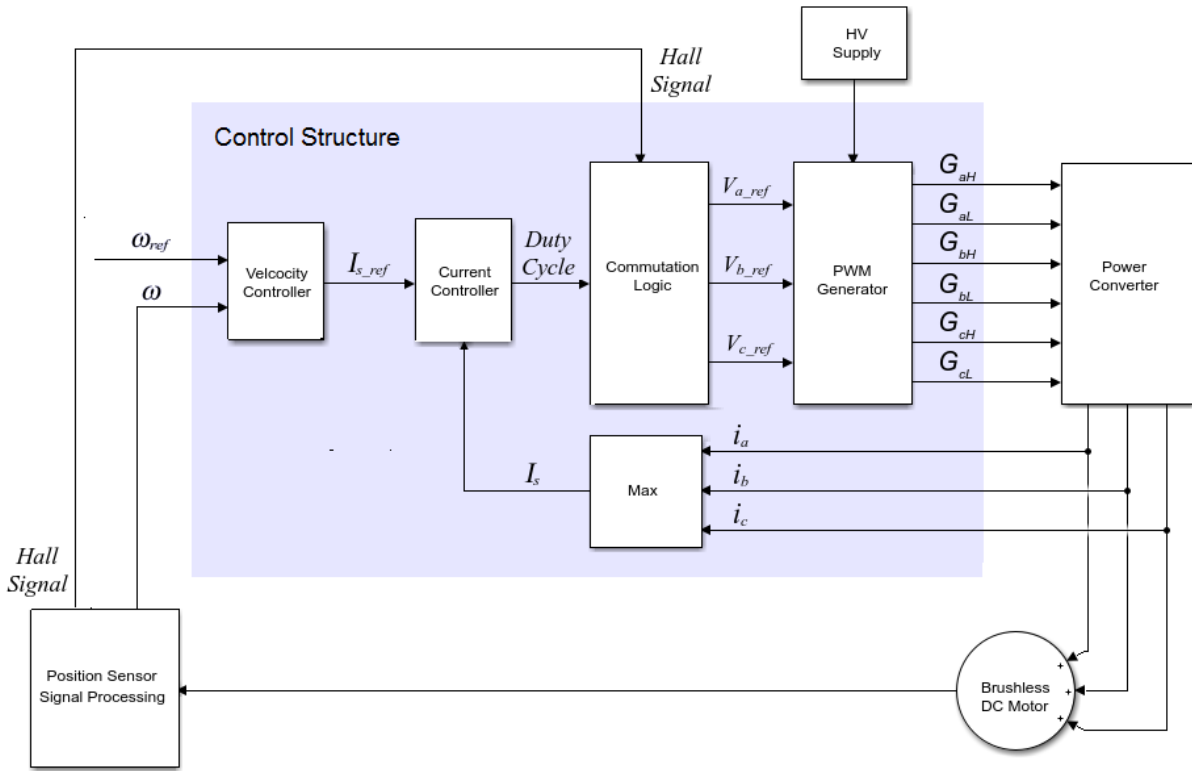
Discrete-time PI-based Brushless DC Motor current control

**Library:** Simscape / Electrical / Power Systems / Control / BLDC Control



### Description

The BLDC Current Controller block uses this algorithm to control current in a DC brushless motor.



## Equations

The BLDC Current Controller produces the duty cycle for a Brushless DC Motor block by implementing proportional-integral (PI) current control using this equation.

$$D = \left( K_p + K_i \frac{T_s z}{z - 1} \right) (I_{s\_ref} - I_s)$$

Where:

- $D$  is the duty cycle.
- $K_p$  is the proportional gain.

- $K_i$  is the integral gain.
- $T_s$  is the time period.
- $I_{s\_ref}$  is the reference current.
- $I_s$  is the measured current.
- $G_{zc}$  is the zero cancellation polynomial.

The closed-loop transfer function for the PI control algorithm yields a zero that can be cancelled by using zero-cancellation in the feedforward path. The zero-cancellation transfer function in discrete-time is:

$$G_{zc}(z) = \frac{\frac{T_s K_i}{K_p}}{z + \left( \frac{T_s - \frac{K_p}{K_i}}{\frac{K_p}{K_i}} \right)}$$

The block obtains control signals for the three phases by multiplying the duty cycle by the commutation signals. The resulting three control signals are normalized over the interval [-1, 1].

## Ports

### Input

#### **IsRef — Reference current**

scalar

Reference current for control.

Data Types: `single` | `double`

#### **Is — Measured current**

scalar

Actual current.

Data Types: single | double

### **Reset — External reset**

scalar

External reset signal (rising edge) for the integrator.

Data Types: single | double

### **Hall — Hall sensor**

vector

Hall sensor data.

Data Types: single | double

### **Direction — Motor direction**

scalar

Direction of motor rotation.

Data Types: single | double

## **Output**

### **vabcRef — Reference voltage**

vector

Reference voltage for the *a*-, *b*-, and *c*-phases.

Data Types: single | double

## **Parameters**

### **Proportional gain — Controller proportional gain, $K_p$**

1 (default) | positive scalar

Proportional gain,  $K_p$ , of the controller.

### **Integral gain — Integral gain, $K_i$**

5 (default) | positive scalar

Integral gain,  $K_i$ , of the controller.

## **Anti-windup gain — Anti-windup gain, $K_{aw}$**

1 (default) | positive scalar

Anti-windup gain,  $K_{aw}$ , of the controller.

## **Sample time (-1 for inherited) — Block sample time**

-1 (default) | positive scalar

Time, in s, between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

If this block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly using a positive scalar.

### **Dependencies**

If you set **Sample time (-1 for inherited)** to -1 and select the **Enable zero cancellation** option, the **Discretization sample time** parameter becomes visible.

## **Discretization sample time — Sample time for discretization**

0.001 (default) | positive scalar

Time, in s, between consecutive discretizations. Discretization is required for zero cancellation.

### **Dependencies**

This parameter is only visible when both these conditions are met:

- **Sample time** is set to -1.
- **Enable zero cancellation** is selected.

## **Enable zero cancellation — Feedforward zero cancellation**

off (default) | on

Option to use zero cancellation on the feedforward path.

### **Dependencies**

If you select the **Enable zero cancellation** option and set **Sample time (-1 for inherited)** to -1, the **Discretization sample time** parameter becomes visible.



## References

- [1] Stirban, A., I. Boldea, and G. D. Andreescu. "Motion-Sensorless Control of BLDC-PM Motor With Offline FEM-Information-Assisted Position and Speed Observer." *IEEE Transactions on Industry Applications*. 48, no. 6 (2012): 1950-1958.

## See Also

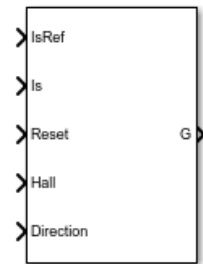
BLDC Commutation Logic | BLDC Current Controller with PWM Generation | Brushless DC Motor

**Introduced in R2018a**

## BLDC Current Controller with PWM Generation

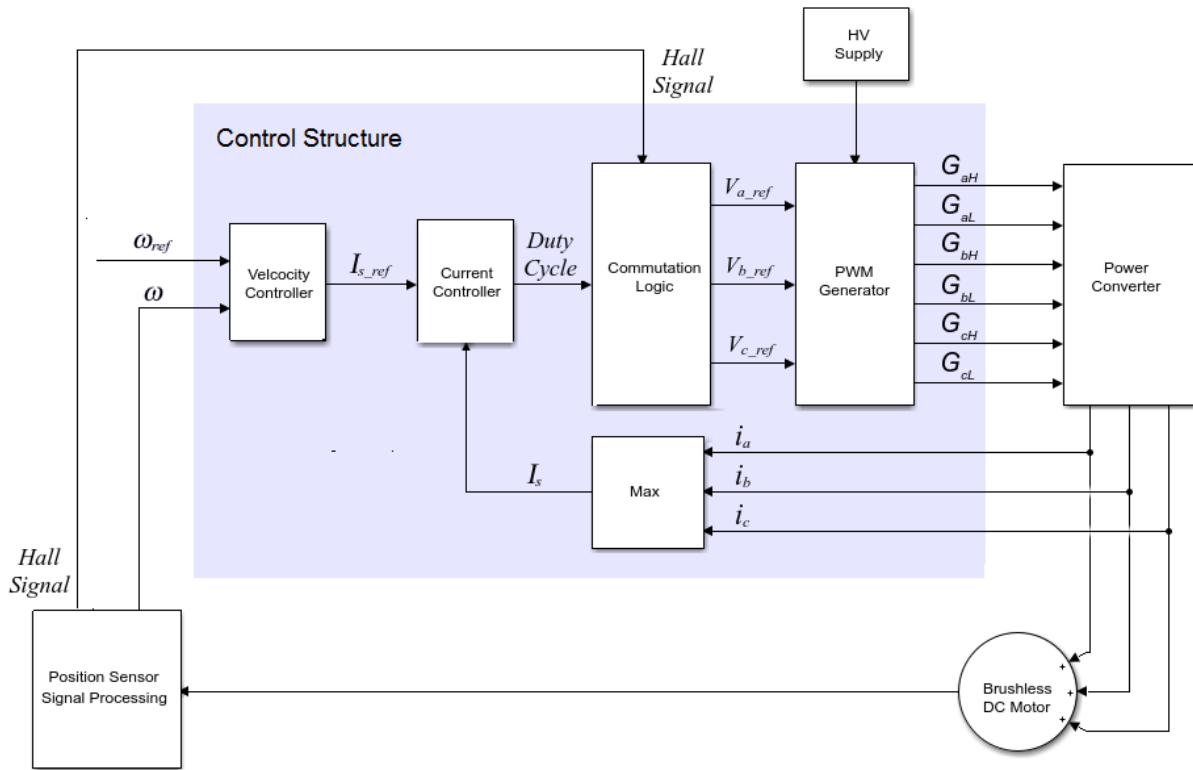
Brushless DC motor current control with pulse width modulation generation

**Library:** Simscape / Electrical / Power Systems / Control /  
BLDC Control



### Description

The BLDC Current Controller with PWM Generation block generates a pulse width modulation (PWM) signal and controls current in a brushless DC motor. The controller uses this algorithm.



## Equations

The BLDC Current Controller produces the duty cycle for a Brushless DC Motor block by implementing proportional-integral (PI) current control using this equation

$$D = \left( K_p + K_i \frac{T_s z}{z - 1} \right) (I_{s\_ref} - I_s)$$

where:

- $D$  is the duty cycle.
- $K_p$  is the proportional gain.

- $K_i$  is the integral gain.
- $T_s$  is the time period.
- $I_{s\_ref}$  is the reference current.
- $I_s$  is the measured current.
- $G_{zc}$  is the zero cancellation polynomial.

The closed-loop transfer function for the PI control algorithm yields a zero that can be cancelled by using zero-cancellation block in the feedforward path. The zero-cancellation transfer function in discrete-time is:

$$G_{zc}(z) = \frac{\frac{T_s K_i}{K_p}}{z + \left( \frac{T_s - \frac{K_p}{K_i}}{\frac{K_p}{K_i}} \right)}$$

The block obtains control signals for the three phases by multiplying the duty cycle by the commutation signals. The resulting three control signals are normalized over the interval  $[-1, 1]$ .

The PWM generator outputs a 1 when the value of the control signal is greater than the carrier counter value. Otherwise, the PWM generator outputs a 0.

## Ports

### Input

#### **IsRef — Reference current**

scalar

Reference current for control.

Data Types: `single` | `double`

**Is — Measured current**

scalar

Actual current.

Data Types: single | double

**Reset — External reset**

scalar

External reset signal (rising edge) for the integrator.

Data Types: single | double

**Hall — Hall sensor**

vector

Hall sensor data.

Data Types: single | double

**Direction — Motor direction**

scalar

Direction of motor rotation.

Data Types: single | double

**Output****G — Gate pulses**

vector

Pulse waveforms that determine switching behavior in the attached block.

Data Types: single | double

## Parameters

### Control Parameters

#### **Proportional gain — Controller proportional gain, $K_p$**

1 (default) | positive scalar

Proportional gain,  $K_p$ , of the controller.

#### **Integral gain — Integral gain, $K_i$**

5 (default) | positive scalar

Integral gain,  $K_i$ , of the controller.

#### **Anti-windup gain — Anti-windup gain, $K_{aw}$**

1 (default) | positive scalar

Anti-windup gain,  $K_{aw}$ , of the controller.

#### **Sample time (-1 for inherited) — Block sample time**

-1 (default) | positive scalar

Time, in s, between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

If this block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly using a positive scalar.

#### **Dependencies**

If you set **Sample time (-1 for inherited)** to -1 and select the **Enable zero cancellation** option, the **Discretization sample time** parameter becomes visible.

#### **Discretization sample time — Sample time for discretization**

0.001 (default) | positive scalar

Time, in s, between consecutive discretizations. Discretization is required for zero cancellation.

**Dependencies**

This parameter is only visible when both these conditions are met:

- **Sample time** is set to -1.
- **Enable zero cancellation** is selected.

**Enable zero cancellation – Feedforward zero cancellation**

off (default) | on

Option to use zero cancellation on the feedforward path.

**Dependencies**

If you select the **Enable zero cancellation** option and set **Sample time (-1 for inherited)** to -1, the **Discretization sample time** parameter becomes visible.

**PWM Generator****Carrier counter – Carrier counter model**

Up (default) | Down | Up-Down

Use the carrier counter strategy to change the initial behavior of the PWM output:

- Up counter — PWM output begins at the start of the on state.
- Down counter — PWM output begins at the start of the off state.
- Up-down counter — PWM output begins in the middle of the on state.

**Timer period (s) – PWM timer period**

0.001 (default) | positive scalar

Pulse width modulation timer period,  $T_{per}$ , in seconds.

**Fundamental sample time (s) – Sample time for PWM generation**

0.0001 (default) | positive scalar

Time, in s, between consecutive PWM generator executions. During execution, the block produces PWM output and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

To ensure adequate resolution in the generated PWM signal, set the fundamental sample time so that  $0 < T_{s\_pwm} \leq 10T_{per}$ , where:

- $T_{s\_pwm}$  is the **Fundamental sample time (s)**.
- $T_{per}$  is the **Timer period (s)**.

## References

- [1] Stirban, A., I. Boldea, and G. D. Andreescu. "Motion-Sensorless Control of BLDC-PM Motor With Offline FEM-Information-Assisted Position and Speed Observer." *IEEE Transactions on Industry Applications*. 48, no. 6 (2012): 1950-1958.

## See Also

BLDC Commutation Logic | BLDC Current Controller | Brushless DC Motor

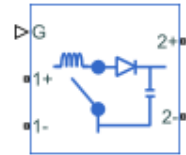
**Introduced in R2018a**



# Boost Converter

Controller-driven DC-DC step-up voltage regulator

**Library:** Simscape / Electrical / Power Systems / Semiconductors



## Description

The Boost Converter block represents a converter that steps up DC voltage as driven by an attached controller and gate-signal generator. Boost converters are also known as step-up voltage regulators because they increase voltage magnitude.

The Boost Converter block allows you to model an asynchronous converter with one switching device or a synchronous converter with two switching devices. Options for the type of switching devices are:

- GTO — Gate turn-off thyristor. For information on the I-V characteristic of the device, see GTO.
- Ideal semiconductor switch — For information on the I-V characteristic of the device, see Ideal Semiconductor Switch.
- IGBT — Insulated-gate bipolar transistor. For information on the I-V characteristic of the device, see IGBT.
- MOSFET — N-channel metal-oxide-semiconductor field-effect transistor. For information on the I-V characteristic of the device, see MOSFET.
- Thyristor — For information on the I-V characteristic of the device, see Thyristor.

## Model

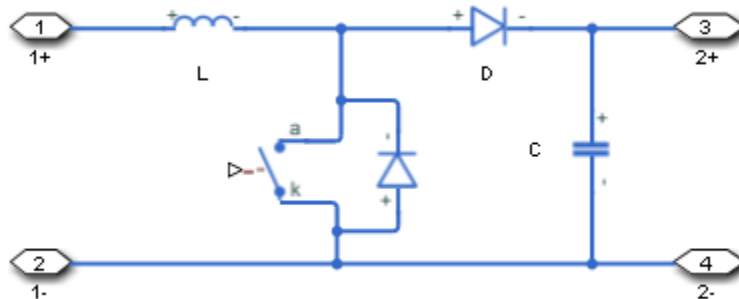
Each of three model variants for the Boost Converter block corresponds to a **Block choice** option. To access the block choices, in the model window, right-click the block, and then use either of these methods:

- From the context menu, select **Simscape > Block choices**.
- On the Simulink Editor menu bar, select **View > Property Inspector**. In the **Property Inspector** window, click the value of the **Block choice**.

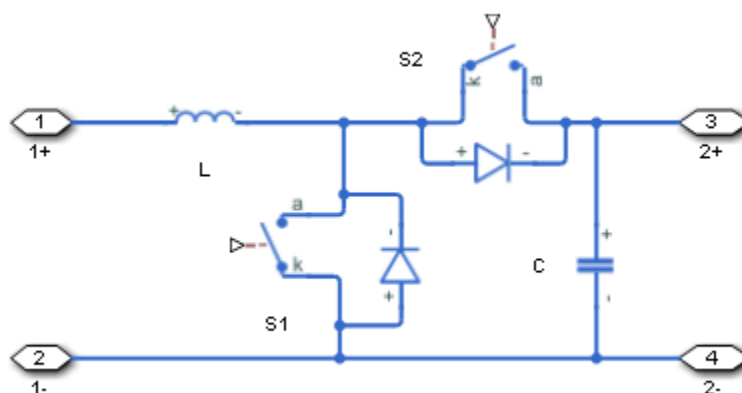
The model variants are:

- PS control port — Asynchronous converter with a physical signal port. This block choice is the default.
- Electrical control ports — Asynchronous converter with one positive and one negative electrical conserving port. To control switching device gates using Simscape Electrical Electronics and Mechatronics blocks, select this option.
- Synchronous converter — Synchronous converter with an electrical conserving port.

The asynchronous boost converter models contain an inductor, a switching device, a diode, and an output capacitor.



The synchronous boost converter model contains an inductor, two switching devices, and an output capacitor.



In each case, the capacitor smooths the output voltage.

## Protection

For the synchronous converter model, you can include an integral protection diodes. Integral diodes protect the semiconductor device by providing a conduction path for reverse current. An inductive load can produce a high reverse-voltage spike when the semiconductor device suddenly switches off the voltage supply to the load.

To include and configure the internal protection diodes, use the **Diode** parameters. This table shows how to set the **Model dynamics** parameter based on your goals.

Goals		Value to Select	Integral Protection Diode
Do not include protection.		None	None
Include protection.	Prioritize simulation speed.	Protection diode with no dynamics	The Diode block

Goals		Value to Select	Integral Protection Diode
	Prioritize model fidelity by precisely specifying reverse-mode charge dynamics.	Protection diode with charge dynamics	The Commutation Diode block

You can also include a snubber circuit for each switching device. Snubber circuits contain a series-connected resistor and capacitor. They protect switching devices against high voltages that inductive loads produce when the device turns off the voltage supply to the load. Snubber circuits also prevent excessive rates of current change when a switching device turns on.

To include and configure a snubber circuit for each switching device, use the **Snubbers** parameters.

## Gate Control

To connect gate-control voltage signals to the gate ports of the switching devices, for the:

- PS control port model:
  - 1 Convert a Simulink gate-control voltage signal to a physical signal using a Simulink-PS Converter block.
  - 2 Connect the Simulink-PS Converter block to the **G** port.
- Electrical control ports model:
  - 1 Connect a Simscape electrical-domain positive DC voltage signal to the **G+** port.
  - 2 Connect the Simscape electrical-domain negative DC voltage signal to the **G-** port.
- Synchronous converter model:
  - 1 Convert each Simulink gate-control voltage signal to a physical signal using Simulink-PS Converter blocks.
  - 2 Multiplex the converted gate-control signals into a single vector using a Two-Pulse Gate Multiplexer.
  - 3 Connect the vector signal to the **G** port.

## Ports

### Input

#### **G — Switching device gate control**

physical signal | vector

Physical signal port associated with the gate terminals of the switching device.

#### **Dependencies**

This port is enabled only for the PS control port block choice.

Data Types: double

### Conserving

#### **G — Switching device gate control**

electrical | vector

Electrical conserving port associated with the gate terminals of the switching devices.

#### **Dependencies**

This port is enabled only for the Synchronous converter block choice.

Data Types: double

#### **G+ — Switching device gate control positive terminal**

electrical | scalar

Positive electrical conserving port associated with the positive gate terminal of the switching device.

#### **Dependencies**

This port is enabled only for the Electrical control ports block choice.

Data Types: double

#### **G- — Switching device gate control negative terminal**

electrical | scalar

Negative electrical conserving port associated with the negative gate terminal of the switching device.

**Dependencies**

This port is enabled only for the `Electrical control ports` block choice.

Data Types: `double`

**1+ — Positive DC voltage 1**

`electrical` | `scalar`

Electrical conserving port associated with the positive terminal of the first DC voltage.

Data Types: `double`

**1- — Negative DC voltage 1**

`electrical` | `scalar`

Electrical conserving port associated with the negative terminal of the first DC voltage.

Data Types: `double`

**2+ — Positive DC voltage 2**

`electrical` | `scalar`

Electrical conserving port associated with the positive terminal of the second DC voltage.

Data Types: `double`

**2- — Negative DC voltage 2**

`electrical` | `scalar`

Electrical conserving port associated with the negative terminal of the second DC voltage.

Data Types: `double`

## Parameters

### Switching Devices

This table shows how the visibility of **Switching Devices** parameters depends on the **Switching device** that you select. To learn how to read the table, see “Parameter Dependencies” on page A-2.

#### Switching Devices Parameter Dependencies

Parameters and Options				
Switching device				
<i>Ideal Semiconductor Switch</i>	<i>GTO</i>	<i>IGBT</i>	<i>MOSFET</i>	<i>Thyristor</i>
On-state resistance	Forward voltage	Forward voltage	Drain-source on resistance	Forward voltage
Off-state conductance	On-state resistance	On-state resistance	Off-state conductance	On-state resistance
Threshold voltage	Off-state conductance	Off-state conductance	Threshold voltage	Off-state conductance
	Gate trigger voltage, Vgt	Threshold voltage		Gate trigger voltage, Vgt
	Gate turn-off voltage, Vgt_off			Gate turn-off voltage, Vgt_off
	Holding current			Holding current

#### Switching device – Switch type

`Ideal Semiconductor Switch` (default) | `GTO` | `IGBT` | `MOSFET` | `Thyristor`

Switching device type for the converter. For the synchronous model, the switches are identical.

#### Dependencies

See the Switching Devices Parameter Dependencies table.

## Forward voltage — Voltage

0.8 V (default) | scalar

For the different switching device types, the **Forward voltage** is taken as:

- GTO — Minimum voltage required across the anode and cathode block ports for the gradient of the device I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**
- IGBT — Minimum voltage required across the collector and emitter block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**
- Thyristor — Minimum voltage required for the device to turn on

### Dependencies

See the Switching Devices Parameter Dependencies table.

## On-state resistance — Resistance

0.001 Ohm (default) | scalar

For the different switching device types, the **On-state resistance** is taken as:

- GTO — Rate of change of voltage versus current above the forward voltage
- Ideal semiconductor switch — Anode-cathode resistance when the device is on
- IGBT — Collector-emitter resistance when the device is on
- Thyristor — Anode-cathode resistance when the device is on

### Dependencies

See the Switching Devices Parameter Dependencies table.

## Drain-source on resistance — Resistance

0.001 Ohm (default) | scalar

Resistance between the drain and the source, which also depends on the gate-to-source voltage.

### Dependencies

See the Switching Devices Parameter Dependencies table.



**Off-state conductance — Conductance**

1e-5 1/0hm (default) | scalar

Conductance when the device is off. The value must be less than  $1/R$ , where  $R$  is the value of **On-state resistance**.

For the different switching device types, the **On-state resistance** is taken as:

- GTO — Anode-cathode conductance
- Ideal semiconductor switch — Anode-cathode conductance
- IGBT — Collector-emitter conductance
- MOSFET — Drain-source conductance
- Thyristor — Anode-cathode conductance

**Dependencies**

See the Switching Devices Parameter Dependencies table.

**Threshold voltage — Voltage threshold**

6 V (default) | scalar

Gate voltage threshold. The device turns on when the gate voltage is above this value. For the different switching device types, the device voltage of interest is:

- Ideal semiconductor switch — Gate-emitter voltage
- IGBT — Gate-cathode voltage
- MOSFET — Gate-source voltage

**Dependencies**

See the Switching Devices Parameter Dependencies table.

**Gate trigger voltage, Vgt — Voltage threshold**

1 V (default) | scalar

Gate-cathode voltage threshold. The device turns on when the gate-cathode voltage is above this value.

**Dependencies**

See the Switching Devices Parameter Dependencies table.

**Gate turn-off voltage, Vgt\_off — Voltage threshold**

-1 V (default) | scalar

Gate-cathode voltage threshold. The device turns off when the gate-cathode voltage is below this value.

**Dependencies**

See the Switching Devices Parameter Dependencies table.

**Holding current — Current threshold**

1 A (default) | scalar

Gate current threshold. The device stays on when the current is above this value, even when the gate-cathode voltage falls below the gate trigger voltage.

**Dependencies**

See the Switching Devices Parameter Dependencies table.

**Diode**

This table shows how the visibility of **Diode** parameters depends on how you configure the **Block choice**, **Model dynamics**, and **Reverse recovery time parameterization** parameters. To learn how to read this table, see “Parameter Dependencies” on page A-2.

## Diode Parameter Dependencies

Parameters and Options					
<b>Block choice</b>					
<i>PS control port or Electrical control ports</i>			<i>Synchronous converter</i>		
<b>Model dynamics</b>			<b>Model dynamics</b>		
<i>Diode with no dynamics</i>	<i>Diode with charge dynamics</i>		<i>None</i>	<i>Protection diode with no dynamics</i>	<i>Protection diode with charge dynamics</i>
Forward voltage	Forward voltage		Forward voltage	Forward voltage	
On resistance	On resistance		On resistance	On resistance	
Off conductance	Off conductance		Off conductance	Off conductance	
	Junction capacitance			Junction capacitance	
	Peak reverse current, iRM			Peak reverse current, iRM	
	Initial forward current when measuring iRM			Initial forward current when measuring iRM	
	Rate of change of current when measuring iRM			Rate of change of current when measuring iRM	
	<b>Reverse recovery time parameterization</b>			<b>Reverse recovery time parameterization</b>	

Parameters and Options								
	<i>Specify stretch factor</i>	<i>Specify reverse recovery time directly</i>	<i>Specify reverse recovery charge</i>			<i>Specify stretch factor</i>	<i>Specify reverse recovery time directly</i>	<i>Specify reverse recovery charge</i>
	Reverse recovery time stretch factor	Reverse recovery time, trr	Reverse recovery charge, Qrr			Reverse recovery time stretch factor	Reverse recovery time, trr	Reverse recovery charge, Qrr

**Model dynamics — Diode model**

Protection diode with no dynamics (default) | Protection diode with charge dynamics | None

Diode type. The options are:

- **None** — This option is the default for the synchronous converter, but is not available for the asynchronous converter.
- **Diode with no dynamics** — Select this option to prioritize simulation speed using the Diode block. This option is the default for the asynchronous converter.
- **Diode with charge dynamics** — Select this option to prioritize model fidelity in terms of reverse mode charge dynamics using the Commutation Diode block.

**Dependencies**

See the Diode Parameter Dependencies table.

**Forward voltage — Voltage**

0.8 V (default) | scalar

Minimum voltage required across the positive and negative block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**.

**On resistance — Resistance**

0.001 Ohm (default) | scalar

Rate of change of voltage versus current above the **Forward voltage**.

**Off conductance — Conductance**

1e-5 1/Ohm (default) | scalar

Conductance of the reverse-biased diode.

**Junction capacitance — Capacitance**

50 nF (default) | scalar

Diode junction capacitance.

**Dependencies**

See the Diode Parameter Dependencies table.

**Peak reverse current,  $i_{RM}$  — Current**

-235 A (default) | scalar less than 0

Peak reverse current measured by an external test circuit.

**Dependencies**

See the Diode Parameter Dependencies table.

**Initial forward current when measuring  $i_{RM}$  — Current**

300 A (default) | scalar greater than 0

Initial forward current when measuring peak reverse current. This value must be greater than zero.

**Dependencies**

See the Diode Parameter Dependencies table.

**Rate of change of current when measuring  $i_{RM}$  — Current change rate**

-50 A/us (default) | scalar

Rate of change of current when measuring peak reverse current.

### **Dependencies**

See the Diode Parameter Dependencies table.

### **Reverse recovery time parameterization — Recovery-time model**

Specify stretch factor (default) | Specify reverse recovery time directly  
| Specify reverse recovery charge

Model for parameterizing the recovery time. When you select `Specify stretch factor` or `Specify reverse recovery charge`, you can specify a value that the block uses to derive the reverse recovery time.

### **Dependencies**

See the Diode Parameter Dependencies table.

### **Reverse recovery time stretch factor — Stretch factor**

3 (default) | scalar greater than 1

Value that the block uses to calculate **Reverse recovery time, trr**. Specifying the stretch factor is an easier way to parameterize the reverse recovery time than specifying the reverse recovery charge. The larger the value of the stretch factor, the longer it takes for the reverse recovery current to dissipate.

### **Dependencies**

See the Diode Parameter Dependencies table.

### **Reverse recovery time, trr — Time**

15 us (default) | scalar

Interval between the time when the current initially goes to zero (when the diode turns off) and the time when the current falls to less than 10 percent of the peak reverse current.

The value of the **Reverse recovery time, trr** parameter must be greater than the value of the **Peak reverse current, iRM** parameter divided by the value of the **Rate of change of current when measuring iRM** parameter.

### **Dependencies**

See the Diode Parameter Dependencies table.

**Reverse recovery charge, Q<sub>rr</sub> – Charge**

1500 s\*uA (default) | scalar

Value that the block uses to calculate **Reverse recovery time, t<sub>rr</sub>**. Use this parameter if the data sheet for your diode device specifies a value for the reverse recovery charge instead of a value for the reverse recovery time.

The reverse recovery charge is the total charge that continues to dissipate when the

diode turns off. The value must be less than  $-\frac{i_{RM}^2}{2a}$ ,

where:

- $i_{RM}$  is the value specified for **Peak reverse current, i<sub>RM</sub>**.
- $a$  is the value specified for **Rate of change of current when measuring i<sub>RM</sub>**.

**Dependencies**

See the Diode Parameter Dependencies table.

**LC Parameters****Inductance – Inductance**

1e-6 H (default) | positive scalar

Inductance.

**Capacitance – Capacitance**

1e-7 F (default) | positive scalar

Capacitance.

**Capacitor effective series resistance – Capacitor resistance**

1e-6 Ohm (default) | zero or positive scalar

Series resistance of the capacitor.

**Snubbers**

The table summarizes the **Snubbers** parameter dependencies. To learn how to read the table, see “Parameter Dependencies” on page A-2.

### Snubbers Parameter Dependencies

Snubbers Parameter Dependencies	
<b>Snubber</b>	
None	RC Snubber
	Snubber resistance
	Snubber capacitance

#### Snubber — Snubber model

None (default) | RC snubber

Switching device snubber.

#### Dependencies

See the Snubbers Parameter Dependencies table.

#### Snubber resistance — Resistance

0.1 (default) | Ohm | scalar

Resistance of the switching device snubber.

#### Dependencies

See the Snubbers Parameter Dependencies table.

#### Snubber capacitance — Capacitance

1e-7 (default) | F | scalar

Capacitance of the switching device snubber.

#### Dependencies

See the Snubbers Parameter Dependencies table.

### References

- [1] Trzynadlowski, A. M. *Introduction to Modern Power Electronics, 2nd Edition*. Hoboken, NJ: John Wiley & Sons Inc., 2010.



- [2] Han, D. and B. Sarlioglu, "Deadtime Effect on GaN-Based Synchronous Boost Converter and Analytical Model for Optimal Deadtime Selection." *IEEE Transactions on Power Electronics*. Vol. 31, Number 1, 2016, pp 601-612.

## See Also

[Average-Value DC-DC Converter](#) | [Bidirectional DC-DC Converter](#) | [Buck Converter](#) | [Buck-Boost Converter](#) | [Converter](#) | [GTO](#) | [IGBT](#) | [Ideal Semiconductor Switch](#) | [MOSFET](#) | [PWM Generator](#) | [PWM Generator \(Three-phase, Two-level\)](#) | [Six-Pulse Gate Multiplexer](#) | [Three-Level Converter](#) | [Thyristor](#)

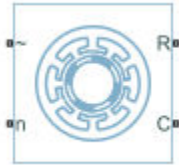
## Topics

["Alternatives to Specifying trr Directly"](#) on page 1-339

**Introduced in R2018a**

## Brushless DC Motor

Three-winding brushless DC motor with trapezoidal flux distribution



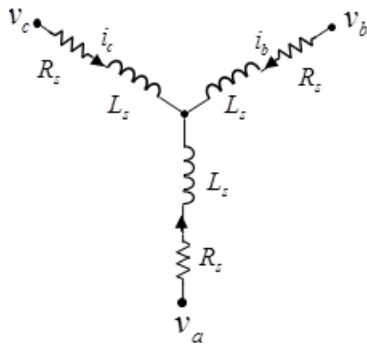
## Library

Simscape / Electrical / Power Systems / Machines / Permanent Magnet Rotor

## Description

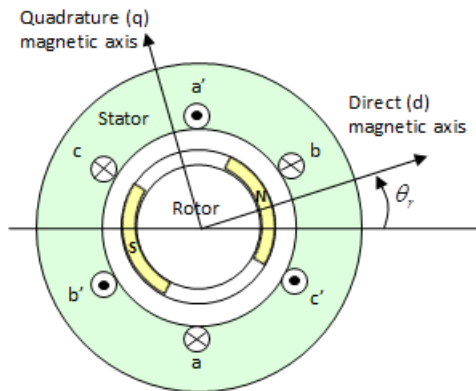
The Brushless DC Motor block models a permanent magnet synchronous machine with a three-phase wye-wound stator. The block has four options for defining the permanent magnet flux distribution as a function of rotor angle. Two options allow for simple parameterization by assuming a perfect trapezoid for the back emf. For simple parameterization, you specify either the flux linkage or the rotor-induced back emf. The other two options give more accurate results using tabulated data that you specify. For more accurate results, you specify either the flux linkage partial derivative or the measured back emf constant for a given rotor speed.

The figure shows the equivalent electrical circuit for the stator windings.



## Motor Construction

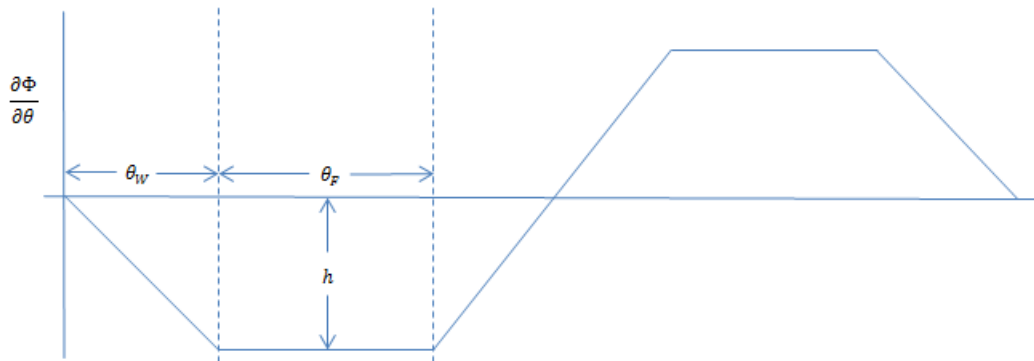
This figure shows the motor construction with a single pole-pair on the rotor.



For the axes convention in the preceding figure, the  $a$ -phase and permanent magnet fluxes are aligned when rotor angle  $\theta_r$  is zero. The block supports a second rotor-axis definition. For the second definition, the rotor angle is the angle between the  $a$ -phase magnetic axis and the rotor  $q$ -axis.

## Trapezoidal Rate of Change of Flux

The rotor magnetic field due to the permanent magnets create a trapezoidal rate of change of flux with rotor angle. The figure shows this rate of change of flux.



Back emf is the rate of change of flux, defined by

$$\frac{d\Phi}{dt} = \frac{\partial\Phi}{\partial\theta} \frac{d\theta}{dt} = \frac{\partial\Phi}{\partial\theta} \omega,$$

where:

- $\Phi$  is the permanent magnet flux linkage.
- $\theta$  is the rotor angle.
- $\omega$  is the mechanical rotational speed.

The height  $h$  of the trapezoidal rate of change of flux profile is derived from the permanent magnet peak flux.

Integrating  $\frac{\partial\Phi}{\partial\theta}$  over the range 0 to  $\pi/2$ ,

$$\Phi_{max} = \frac{h}{2}(\theta_F + \theta_W),$$

where:

- $\Phi_{max}$  is the permanent magnet flux linkage.
- $h$  is the rate of change of flux profile height.
- $\theta_F$  is the rotor angle range over which the back emf that the permanent magnet flux induces in the stator is constant.

- $\theta_W$  is the rotor angle range over which back emf increases or decreases linearly when the rotor moves at constant speed.

Rearranging the preceding equation,

$$h = 2\Phi_{max} / (\theta_F + \theta_W).$$

## Electrical Defining Equations

Voltages across the stator windings are defined by

$$\begin{bmatrix} v_a \\ v_b \\ v_c \end{bmatrix} = \begin{bmatrix} R_s & 0 & 0 \\ 0 & R_s & 0 \\ 0 & 0 & R_s \end{bmatrix} \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix} + \begin{bmatrix} \frac{d\psi_a}{dt} \\ \frac{d\psi_b}{dt} \\ \frac{d\psi_c}{dt} \end{bmatrix},$$

where:

- $v_a$ ,  $v_b$ , and  $v_c$  are the external voltages applied to the three motor electrical connections.
- $R_s$  is the equivalent resistance of each stator winding.
- $i_a$ ,  $i_b$ , and  $i_c$  are the currents flowing in the stator windings.
- 

$$\frac{d\psi_a}{dt}, \frac{d\psi_b}{dt}, \text{ and } \frac{d\psi_c}{dt}$$

are the rates of change of magnetic flux in each stator winding.

The permanent magnet and the three windings contribute to the total flux linking each winding. The total flux is defined by

$$\begin{bmatrix} \psi_a \\ \psi_b \\ \psi_c \end{bmatrix} = \begin{bmatrix} L_{aa} & L_{ab} & L_{ac} \\ L_{ba} & L_{bb} & L_{bc} \\ L_{ca} & L_{cb} & L_{cc} \end{bmatrix} \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix} + \begin{bmatrix} \psi_{am} \\ \psi_{bm} \\ \psi_{cm} \end{bmatrix},$$

where:

- $\psi_a$ ,  $\psi_b$ , and  $\psi_c$  are the total fluxes linking each stator winding.
- $L_{aa}$ ,  $L_{bb}$ , and  $L_{cc}$  are the self-inductances of the stator windings.
- $L_{ab}$ ,  $L_{ac}$ ,  $L_{ba}$ , etc. are the mutual inductances of the stator windings.
- $\psi_{am}$ ,  $\psi_{bm}$ , and  $\psi_{cm}$  are the permanent magnet fluxes linking the stator windings.

The inductances in the stator windings are functions of rotor angle, defined by

$$L_{aa} = L_s + L_m \cos(2\theta_r),$$

$$L_{bb} = L_s + L_m \cos(2(\theta_r - 2\pi/3)),$$

$$L_{cc} = L_s + L_m \cos(2(\theta_r + 2\pi/3)),$$

$$L_{ab} = L_{ba} = -M_s - L_m \cos(2(\theta_r + \pi/6)),$$

$$L_{bc} = L_{cb} = -M_s - L_m \cos(2(\theta_r + \pi/6 - 2\pi/3)),$$

and

$$L_{ca} = L_{ac} = -M_s - L_m \cos(2(\theta_r + \pi/6 + 2\pi/3)),$$

where:

- $L_s$  is the stator self-inductance per phase — The average self-inductance of each of the stator windings.
- $L_m$  is the stator inductance fluctuation — The amplitude of the fluctuation in self-inductance and mutual inductance with changing rotor angle.
- $M_s$  is the stator mutual inductance — The average mutual inductance between the stator windings.

The permanent magnet flux linking each stator winding follows the trapezoidal profile shown in the figure. The block implements the trapezoidal profile using lookup tables to calculate permanent magnet flux values.

## Simplified Equations

The defining voltage and torque equations for the block are

$$\begin{bmatrix} v_d \\ v_q \\ v_0 \end{bmatrix} = P \begin{bmatrix} v_a \\ v_b \\ v_c \end{bmatrix} - N\omega \begin{bmatrix} \frac{\partial \psi_{am}}{\partial \theta_r} \\ \frac{\partial \psi_{bm}}{\partial \theta_r} \\ \frac{\partial \psi_{cm}}{\partial \theta_r} \end{bmatrix},$$

$$v_d = R_s i_d + L_d \frac{di_d}{dt} - N\omega i_q L_q,$$

$$v_q = R_s i_q + L_q \frac{di_q}{dt} + N\omega i_d L_d,$$

$$v_0 = R_s i_0 + L_0 \frac{di_0}{dt},$$

and

$$T = \frac{3}{2} N (i_q i_d L_d - i_d i_q L_q) + [i_a \quad i_b \quad i_c] \begin{bmatrix} \frac{\partial \psi_{am}}{\partial \theta_r} \\ \frac{\partial \psi_{bm}}{\partial \theta_r} \\ \frac{\partial \psi_{cm}}{\partial \theta_r} \end{bmatrix},$$

where:

- $v_d$ ,  $v_q$ , and  $v_0$  are the  $d$ -axis,  $q$ -axis, and zero-sequence voltages.
- $P$  is Park's Transformation, defined by

$$P = 2/3 \begin{bmatrix} \cos \theta_e & \cos(\theta_e - 2\pi/3) & \cos(\theta_e + 2\pi/3) \\ -\sin \theta_e & -\sin(\theta_e - 2\pi/3) & -\sin(\theta_e + 2\pi/3) \\ 0.5 & 0.5 & 0.5 \end{bmatrix}.$$

- $N$  is the number of rotor permanent magnet pole pairs.

- $\omega$  is the rotor mechanical rotational speed.

- 

$$\frac{\partial \psi_{am}}{\partial \theta_r}, \frac{\partial \psi_{bm}}{\partial \theta_r}, \text{ and } \frac{\partial \psi_{cm}}{\partial \theta_r}$$

are the partial derivatives of instantaneous permanent magnet flux linking each phase winding.

- $i_d$ ,  $i_q$ , and  $i_0$  are the  $d$ -axis,  $q$ -axis, and zero-sequence currents, defined by

$$\begin{bmatrix} i_d \\ i_q \end{bmatrix} = P \begin{bmatrix} i_a \\ i_b \end{bmatrix}.$$

- $L_d = L_s + M_s + 3/2 L_m$ .  $L_d$  is the stator  $d$ -axis inductance.

- $L_q = L_s + M_s - 3/2 L_m$ .  $L_q$  is the stator  $q$ -axis inductance.

- $L_0 = L_s - 2M_s$ .  $L_0$  is the stator zero-sequence inductance.

- $T$  is the rotor torque. Torque flows from the motor case (block physical port C) to the motor rotor (block physical port R).

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.



## Ports

~

Expandable three-phase port

n

Electrical conserving port associated with the neutral phase

R

Mechanical rotational conserving port associated with the motor rotor

C

Mechanical rotational conserving port associated with the motor case

## Parameters

### Rotor

#### Back EMF profile

Parameterization for defining the permanent magnet flux distribution as a function of rotor angle. Choose:

- **Perfect trapezoid - specify maximum flux linkage** to specify the maximum flux linkage for the permanent magnet and the rotor angle where the back emf is constant. The block assumes a perfect trapezoid for the back emf. This is the default value.
- **Perfect trapezoid - specify maximum rotor-induced back emf** to specify the maximum rotor-induced back emf and the corresponding rotor speed. The block assumes a perfect trapezoid for the back emf.
- **Tabulated - specify flux partial derivative with respect to rotor angle** to specify values for the partial derivative of flux linkage and the corresponding rotor angles.
- **Tabulated - specify rotor-induced back emf as a function of rotor angle** to specify the measured back emf constant and the corresponding rotor speed and angles.

**Maximum permanent magnet flux linkage**

Peak permanent magnet flux linkage with any of the stator windings. This parameter is visible only when **Back EMF profile** is set to Perfect trapezoid - specify maximum flux linkage. The default value is 0.03 Wb.

**Rotor angle over which back emf is constant**

Rotor angle range over which the permanent magnet flux linking the stator winding is constant. This angle is  $\theta_F$  in the figure that shows the “Trapezoidal Rate of Change of Flux” on page 1-221. This parameter is visible only when **Back EMF profile** is set to Perfect trapezoid - specify maximum flux linkage or Perfect trapezoid - specify maximum rotor-induced back emf. The default value is  $\pi / 12$  rad.

**Maximum rotor-induced back emf**

Peak rotor-induced back emf into the stator windings. This parameter is visible only when **Back EMF profile** is set to Perfect trapezoid - specify maximum rotor-induced back emf. The default value is 9.6 V.

**Rotor-induced back emf**

Vector of values for the rotor-induced back emf as a function of rotor angle. The first and last values must be the same, and are normally both zero. For more information, see the **Corresponding rotor angles** parameter. First and last values are the same because flux is cyclic with period  $2\pi / N$ , where  $N$  is the number of permanent magnet pole pairs. This parameter is visible only when **Back EMF profile** is set to Tabulated - specify rotor-induced back emf as a function of rotor angle. The default value is [0.0, -9.6, -9.6, 9.6, 9.6, 0.0] V.

**Flux linkage partial derivative with respect to rotor angle**

Vector of values for the partial derivative of flux linkage (where flux linkage is flux times number of winding turns) with respect to rotor angle. The first and last values must be the same, and are normally both zero. For more information, see the **Corresponding rotor angles** parameter. First and last values are the same because flux is cyclic with period  $2\pi / N$ , where  $N$  is the number of permanent magnet pole pairs. This parameter is visible only when **Back EMF profile** is set to Tabulated - specify flux partial derivative with respect to rotor angle. The default value is [0.0, -0.1528, -0.1528, 0.1528, 0.1528, 0.0] Wb/rad.

**Corresponding rotor angles**

Vector of rotor angles where the flux linkage partial derivative or rotor-induced back emf is defined. Rotor angle is defined as the angle between the  $a$ -phase magnetic axis and the  $d$ -axis. That is, when the angle is zero, the magnetic fields due to the rotor

and the  $a$ -phase winding align. This definition is used regardless of your block setting for rotor angle definition. The first value is zero, and the last value is  $2\pi / N$ , where  $N$  is the number of permanent magnet pole pairs. This parameter is visible only when **Back EMF profile** is set to **Tabulated - specify flux partial derivative with respect to rotor angle** or **Tabulated - specify rotor-induced back emf as a function of rotor angle**. The default value is [0, 7.5, 22.5, 37.5, 52.5, 60] deg.

### **Rotor speed used for back emf measurement**

Specify the rotor speed corresponding to the maximum rotor-induced back emf. This parameter is visible only when **Back EMF profile** is set to **Perfect trapezoid - specify maximum rotor-induced back emf** or **Tabulated - specify rotor-induced back emf as a function of rotor angle**. The default value is 600 rpm.

### **Number of pole pairs**

Number of permanent magnet pole pairs on the rotor. The default value is 6.

### **Zero sequence**

Option to include or exclude zero-sequence terms.

- **Include** — Include zero-sequence terms. To prioritize model fidelity, use this default setting. Using this option results in an error for simulations that use the Partitioning solver. For more information, see “Increase Simulation Speed Using the Partitioning Solver” (Simscape).
- **Exclude** — Exclude zero-sequence terms. To prioritize simulation speed for desktop simulation or real-time deployment, select this option.

### **Rotor angle definition**

Reference point for the rotor angle measurement. The default value is **Angle between the a-phase magnetic axis and the d-axis**. This definition is shown in the “Motor Construction” on page 1-221 figure. When you select this value, the rotor and  $a$ -phase fluxes are aligned when the rotor angle is zero.

The other value you can choose for this parameter is **Angle between the a-phase magnetic axis and the q-axis**. When you select this value, the  $a$ -phase current generates maximum torque when the rotor angle is zero.

## **Stator**

### **Stator parameterization**

Choose Specify Ld, Lq, and L0, the default value, or Specify Ls, Lm, and Ms.

### **Stator d-axis inductance, Ld**

D-axis inductance. This parameter is visible only if you set **Stator parameterization** to Specify Ld, Lq, and L0. The default value is 0.00022 H.

### **Stator q-axis inductance, Lq**

Q-axis inductance. This parameter is visible only if you set **Stator parameterization** to Specify Ld, Lq, and L0. The default value is 0.00022 H.

### **Stator zero-sequence inductance, L0**

Zero-sequence inductance. This parameter is visible only if you set **Stator parameterization** to Specify Ld, Lq, and L0. The default value is 0.00016 H.

### **Stator self-inductance per phase, Ls**

Average self-inductance of each of the three stator windings. This parameter is visible only if you set **Stator parameterization** to Specify Ls, Lm, and Ms. The default value is 0.00002 H.

### **Stator inductance fluctuation, Lm**

Amplitude of the fluctuation in self-inductance and mutual inductance of the stator windings with rotor angle. This parameter is visible only if you set **Stator parameterization** to Specify Ls, Lm, and Ms. The default value is 0 H.

### **Stator mutual inductance, Ms**

Average mutual inductance between the stator windings. This parameter is visible only if you set **Stator parameterization** to Specify Ls, Lm, and Ms. The default value is 0.00002 H.

### **Stator resistance per phase, Rs**

Resistance of each of the stator windings. The default value is 0.013 Ohm.

## **Mechanical**

### **Rotor Inertia**

Inertia of the rotor attached to mechanical translational port **R**. The default value is 0.01 kg\*m<sup>2</sup>. The value can be zero.

**Rotor damping**

Rotary damping. The default value is 0 N\*m/(rad/s).

**References**

- [1] Kundur, P. *Power System Stability and Control*. New York, NY: McGraw Hill, 1993.
- [2] Anderson, P. M. *Analysis of Faulted Power Systems*. Hoboken, NJ: Wiley-IEEE Press, 1995.

**See Also****Simscape Blocks**

Hybrid Excitation Synchronous Machine | Permanent Magnet Synchronous Motor

**Blocks**

BLDC Commutation Logic | BLDC Current Controller | BLDC Current Controller with PWM Generation

**Topics**

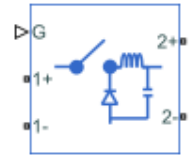
“Expand and Collapse Three-Phase Ports on a Block”

**Introduced in R2013b**

## Buck Converter

Controller-driven DC-DC step-down voltage regulator

**Library:** Simscape / Electrical / Power Systems / Semiconductors



## Description

The Buck Converter block represents a converter that steps down DC voltage as driven by an attached controller and gate-signal generator. Buck converters are also known as step-down voltage regulators because they decrease voltage magnitude.

The Buck Converter block allows you to model an asynchronous converter with one switching device or a synchronous converter with two switching devices. Options for the type of switching devices are:

- GTO — Gate turn-off thyristor. For information on the I-V characteristic of the device, see GTO.
- Ideal semiconductor switch — For information on the I-V characteristic of the device, see Ideal Semiconductor Switch.
- IGBT — Insulated-gate bipolar transistor. For information on the I-V characteristic of the device, see IGBT.
- MOSFET — N-channel metal-oxide-semiconductor field-effect transistor. For information on the I-V characteristic of the device, see MOSFET.
- Thyristor — For information on the I-V characteristic of the device, see Thyristor.

## Model

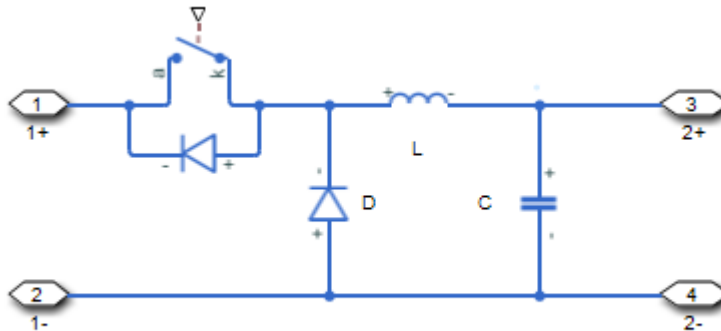
Each of three model variants for the Buck Converter block corresponds to a **Block choice** option. To access the block choices, in the model window, right-click the block, and then use either of these methods:

- From the context menu, select **Simscape > Block choices**.
- On the Simulink Editor menu bar, select **View > Property Inspector**. In the **Property Inspector** window, click the value of the **Block choice**.

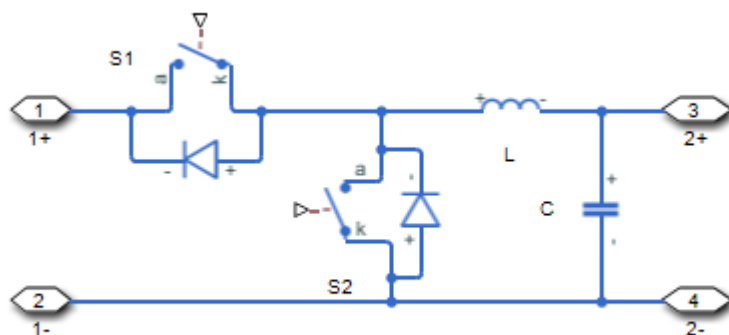
The model variants are:

- PS control port — Asynchronous converter with a physical signal port. This block choice is the default.
- Electrical control ports — Asynchronous converter with one positive and one negative electrical conserving port. To control switching device gates using Simscape Electrical Electronics and Mechatronics blocks, select this option.
- Synchronous converter — Synchronous converter with an electrical conserving port.

The asynchronous buck converter models contain a switching device, a diode, an inductor, and an output capacitor.



The synchronous buck converter model contains two switching devices, an inductor, and an output capacitor.



In each case, the capacitor smoothes the output voltage.

## Protection

For the synchronous converter model, you can include an integral protection diode for the S2 switching device. An integral diode protects the semiconductor device by providing a conduction path for reverse current. An inductive load can produce a high reverse-voltage spike when the semiconductor device suddenly switches off the voltage supply to the load.

To include and configure the internal protection diode block, use the **Diode** parameters. This table shows how to set the **Model dynamics** parameter based on your goals.

Goals		Value to Select	Integral Protection Diode
Do not include protection.		None	None
Include protection.	Prioritize simulation speed.	Protection diode with no dynamics	The Diode block
	Prioritize model fidelity by precisely specifying reverse-mode charge dynamics.	Protection diode with charge dynamics	The Commutation Diode block



You can also include a snubber circuit for each switching device. Snubber circuits contain a series-connected resistor and capacitor. They protect switching devices against high voltages that inductive loads produce when the device turns off the voltage supply to the load. Snubber circuits also prevent excessive rates of current change when a switching device turns on.

To include and configure a snubber circuit for each switching device, use the **Snubbers** parameters.

## Gate Control

To connect gate-control voltage signals to the gate ports of the switching devices, for the:

- PS control port model:
  - 1 Convert a Simulink gate-control voltage signal to a physical signal using a Simulink-PS Converter block.
  - 2 Connect the Simulink-PS Converter block to the **G** port.
- Electrical control ports model:
  - 1 Connect a Simscape electrical-domain positive DC voltage signal to the **G+** port.
  - 2 Connect the Simscape electrical-domain negative DC voltage signal to the **G-** port.
- Synchronous converter model:
  - 1 Convert each Simulink gate-control voltage signal to a physical signal using Simulink-PS Converter blocks.
  - 2 Multiplex the converted gate-control signals into a single vector using a Two-Pulse Gate Multiplexer.
  - 3 Connect the vector signal to the **G** port.

## Ports

### Input

#### **G** — Switching device gate control

physical signal | vector

Physical signal port associated with the gate terminals of the switching device.

### **Dependencies**

This port is enabled only for the PS control port block choice.

Data Types: double

## **Conserving**

### **G — Switching device gate control**

electrical | vector

Electrical conserving port associated with the gate terminals of the switching devices.

### **Dependencies**

This port is enabled only for the Synchronous converter block choice.

Data Types: double

### **G+ — Switching device gate control positive terminal**

electrical | scalar

Positive electrical conserving port associated with the positive gate terminal of the switching device.

### **Dependencies**

This port is enabled only for the Electrical control ports block choice.

Data Types: double

### **G- — Switching device gate control negative terminal**

electrical | scalar

Negative electrical conserving port associated with the negative gate terminal of the switching device.

### **Dependencies**

This port is enabled only for the Electrical control ports block choice.

Data Types: double

### **1+ — Positive DC voltage 1**

electrical | scalar

Electrical conserving port associated with the positive terminal of the first DC voltage.

Data Types: double

### **1- — Negative DC voltage 1**

electrical | scalar

Electrical conserving port associated with the negative terminal of the first DC voltage.

Data Types: double

### **2+ — Positive DC voltage 2**

electrical | scalar

Electrical conserving port associated with the positive terminal of the second DC voltage.

Data Types: double

### **2- — Negative DC voltage 2**

electrical | scalar

Electrical conserving port associated with the negative terminal of the second DC voltage.

Data Types: double

## **Parameters**

### **Switching Devices**

This table shows how the visibility of **Switching Devices** parameters depends on the **Switching device** that you select. To learn how to read the table, see “Parameter Dependencies” on page A-2.

### Switching Devices Parameter Dependencies

Parameters and Options				
Switching device				
<i>Ideal Semiconductor Switch</i>	<i>GTO</i>	<i>IGBT</i>	<i>MOSFET</i>	<i>Thyristor</i>
On-state resistance	Forward voltage	Forward voltage	Drain-source on resistance	Forward voltage
Off-state conductance	On-state resistance	On-state resistance	Off-state conductance	On-state resistance
Threshold voltage	Off-state conductance	Off-state conductance	Threshold voltage	Off-state conductance
	Gate trigger voltage, Vgt	Threshold voltage		Gate trigger voltage, Vgt
	Gate turn-off voltage, Vgt_off			Gate turn-off voltage, Vgt_off
	Holding current			Holding current

#### Switching device — Switch type

*Ideal Semiconductor Switch* (default) | *GTO* | *IGBT* | *MOSFET* | *Thyristor*

Switching device type for the converter. For the synchronous model, the switches are identical.

#### Dependencies

See the Switching Devices Parameter Dependencies table.

#### Forward voltage — Voltage

0.8 V (default) | scalar

For the different switching device types, the **Forward voltage** is taken as:

- *GTO* — Minimum voltage required across the anode and cathode block ports for the gradient of the device I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**

- IGBT — Minimum voltage required across the collector and emitter block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**
- Thyristor — Minimum voltage required for the device to turn on

### Dependencies

See the Switching Devices Parameter Dependencies table.

### On-state resistance — Resistance

0.001 Ohm (default) | scalar

For the different switching device types, the **On-state resistance** is taken as:

- GTO — Rate of change of voltage versus current above the forward voltage
- Ideal semiconductor switch — Anode-cathode resistance when the device is on
- IGBT — Collector-emitter resistance when the device is on
- Thyristor — Anode-cathode resistance when the device is on

### Dependencies

See the Switching Devices Parameter Dependencies table.

### Drain-source on resistance — Resistance

0.001 Ohm (default) | scalar

Resistance between the drain and the source, which also depends on the gate-to-source voltage.

### Dependencies

See the Switching Devices Parameter Dependencies table.

### Off-state conductance — Conductance

1e-5 1/Ohm (default) | scalar

Conductance when the device is off. The value must be less than  $1/R$ , where  $R$  is the value of **On-state resistance**.

For the different switching device types, the **On-state resistance** is taken as:

- GTO — Anode-cathode conductance

- Ideal semiconductor switch — Anode-cathode conductance
- IGBT — Collector-emitter conductance
- MOSFET — Drain-source conductance
- Thyristor — Anode-cathode conductance

### **Dependencies**

See the Switching Devices Parameter Dependencies table.

### **Threshold voltage — Voltage threshold**

6 V (default) | scalar

Gate voltage threshold. The device turns on when the gate voltage is above this value. For the different switching device types, the device voltage of interest is:

- Ideal semiconductor switch — Gate-emitter voltage
- IGBT — Gate-cathode voltage
- MOSFET — Gate-source voltage

### **Dependencies**

See the Switching Devices Parameter Dependencies table.

### **Gate trigger voltage, $V_{gt}$ — Voltage threshold**

1 V (default) | scalar

Gate-cathode voltage threshold. The device turns on when the gate-cathode voltage is above this value.

### **Dependencies**

See the Switching Devices Parameter Dependencies table.

### **Gate turn-off voltage, $V_{gt\_off}$ — Voltage threshold**

-1 V (default) | scalar

Gate-cathode voltage threshold. The device turns off when the gate-cathode voltage is below this value.

### **Dependencies**

See the Switching Devices Parameter Dependencies table.

**Holding current — Current threshold**

1 A (default) | scalar

Gate current threshold. The device stays on when the current is above this value, even when the gate-cathode voltage falls below the gate trigger voltage.

**Dependencies**

See the Switching Devices Parameter Dependencies table.

**Diode**

This table shows how the visibility of **Diode** parameters depends on how you configure the **Block choice**, **Model dynamics**, and **Reverse recovery time parameterization** parameters. To learn how to read this table, see “Parameter Dependencies” on page A-2.

**Diode Parameter Dependencies**

<b>Parameters and Options</b>					
<b>Block choice</b>					
<i>PS control port or Electrical control ports</i>			<i>Synchronous converter</i>		
<b>Model dynamics</b>			<b>Model dynamics</b>		
<i>Diode with no dynamics</i>	<i>Diode with charge dynamics</i>		<i>None</i>	<i>Protection diode with no dynamics</i>	<i>Protection diode with charge dynamics</i>
Forward voltage	Forward voltage		Forward voltage	Forward voltage	
On resistance	On resistance		On resistance	On resistance	
Off conductance	Off conductance		Off conductance	Off conductance	
	Junction capacitance			Junction capacitance	
	Peak reverse current, iRM			Peak reverse current, iRM	
	Initial forward current when measuring iRM			Initial forward current when measuring iRM	
	Rate of change of current when measuring iRM			Rate of change of current when measuring iRM	
	<b>Reverse recovery time parameterization</b>			<b>Reverse recovery time parameterization</b>	



Parameters and Options								
	<i>Specify stretch factor</i>	<i>Specify reverse recovery time directly</i>	<i>Specify reverse recovery charge</i>			<i>Specify stretch factor</i>	<i>Specify reverse recovery time directly</i>	<i>Specify reverse recovery charge</i>
	Reverse recovery time stretch factor	Reverse recovery time, trr	Reverse recovery charge, Qrr			Reverse recovery time stretch factor	Reverse recovery time, trr	Reverse recovery charge, Qrr

### Model dynamics — Diode model

Protection diode with no dynamics (default) | Protection diode with charge dynamics | None

Diode type. The options are:

- **None** — This option is the default for the synchronous converter, but is not available for the asynchronous converter.
- **Diode with no dynamics** — Select this option to prioritize simulation speed using the Diode block. This option is the default for the asynchronous converter.
- **Diode with charge dynamics** — Select this option to prioritize model fidelity in terms of reverse mode charge dynamics using the Commutation Diode block.

### Dependencies

See the Diode Parameter Dependencies table.

### Forward voltage — Voltage

0.8 V (default) | scalar

Minimum voltage required across the positive and negative block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**.

**On resistance — Resistance**

0.001 Ohm (default) | scalar

Rate of change of voltage versus current above the **Forward voltage**.

**Off conductance — Conductance**

1e-5 1/Ohm (default) | scalar

Conductance of the reverse-biased diode.

**Junction capacitance — Capacitance**

50 nF (default) | scalar

Diode junction capacitance.

**Dependencies**

See the Diode Parameter Dependencies table.

**Peak reverse current,  $i_{RM}$  — Current**

-235 A (default) | scalar less than 0

Peak reverse current measured by an external test circuit.

**Dependencies**

See the Diode Parameter Dependencies table.

**Initial forward current when measuring  $i_{RM}$  — Current**

300 A (default) | scalar greater than 0

Initial forward current when measuring peak reverse current. This value must be greater than zero.

**Dependencies**

See the Diode Parameter Dependencies table.

**Rate of change of current when measuring  $i_{RM}$  — Current change rate**

-50 A/us (default) | scalar

Rate of change of current when measuring peak reverse current.

### Dependencies

See the Diode Parameter Dependencies table.

### Reverse recovery time parameterization – Recovery-time model

Specify stretch factor (default) | Specify reverse recovery time directly  
| Specify reverse recovery charge

Model for parameterizing the recovery time. When you select `Specify stretch factor` or `Specify reverse recovery charge`, you can specify a value that the block uses to derive the reverse recovery time.

### Dependencies

See the Diode Parameter Dependencies table.

### Reverse recovery time stretch factor – Stretch factor

3 (default) | scalar greater than 1

Value that the block uses to calculate **Reverse recovery time, trr**. Specifying the stretch factor is an easier way to parameterize the reverse recovery time than specifying the reverse recovery charge. The larger the value of the stretch factor, the longer it takes for the reverse recovery current to dissipate.

### Dependencies

See the Diode Parameter Dependencies table.

### Reverse recovery time, trr – Time

15 us (default) | scalar

Interval between the time when the current initially goes to zero (when the diode turns off) and the time when the current falls to less than 10 percent of the peak reverse current.

The value of the **Reverse recovery time, trr** parameter must be greater than the value of the **Peak reverse current, iRM** parameter divided by the value of the **Rate of change of current when measuring iRM** parameter.

### Dependencies

See the Diode Parameter Dependencies table.

## Reverse recovery charge, $Q_{rr}$ — Charge

1500 s\*uA (default) | scalar

Value that the block uses to calculate **Reverse recovery time,  $t_{rr}$** . Use this parameter if the data sheet for your diode device specifies a value for the reverse recovery charge instead of a value for the reverse recovery time.

The reverse recovery charge is the total charge that continues to dissipate when the

diode turns off. The value must be less than  $-\frac{i_{RM}^2}{2a}$ ,

where:

- $i_{RM}$  is the value specified for **Peak reverse current,  $i_{RM}$** .
- $a$  is the value specified for **Rate of change of current when measuring  $i_{RM}$** .

### Dependencies

See the Diode Parameter Dependencies table.

## LC Filter

### Inductance — Filter inductance

1e-6 H (default) | positive scalar

Inductance of the LC filter.

### Capacitance — Filter capacitance

1e-7 F (default) | positive scalar

Capacitance of the LC filter.

### Capacitor effective series resistance — Capacitor resistance

1e-6 Ohm (default) | zero or positive scalar

Series resistance of the capacitor.

## Snubbers

The table summarizes the **Snubbers** parameter dependencies. To learn how to read the table, see “Parameter Dependencies” on page A-2.

## Snubbers Parameter Dependencies

Snubbers Parameter Dependencies	
Snubber	
None	RC Snubber
	Snubber resistance
	Snubber capacitance

### Snubber — Snubber model

None (default) | RC snubber

Switching device snubber.

#### Dependencies

See the Snubbers Parameter Dependencies table.

### Snubber resistance — Resistance

0.1 (default) | Ohm | scalar

Resistance of the switching device snubber.

#### Dependencies

See the Snubbers Parameter Dependencies table.

### Snubber capacitance — Capacitance

1e-7 (default) | F | scalar

Capacitance of the switching device snubber.

#### Dependencies

See the Snubbers Parameter Dependencies table.

## References

- [1] Trzynadlowski, A. M. *Introduction to Modern Power Electronics, 2nd Edition*. Hoboken, NJ: John Wiley & Sons Inc., 2010.
- [2] Hedayati, M. H., P. Bharadwaj, and V. John. "Hybrid synchronous DC-DC buck power converter using Si and GaN transistors." *IEEE International Conference on Power*

*Electronics, Drives and Energy Systems (PEDES)*. Trivandrum, India: 2016, pp 1-6.

## **See Also**

Average-Value DC-DC Converter | Bidirectional DC-DC Converter | Boost Converter | Buck-Boost Converter | Converter | GTO | IGBT | Ideal Semiconductor Switch | MOSFET | PWM Generator | PWM Generator (Three-phase, Two-level) | Six-Pulse Gate Multiplexer | Three-Level Converter | Thyristor

## **Topics**

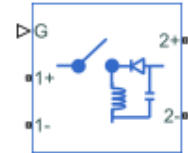
“Alternatives to Specifying  $t_{rr}$  Directly” on page 1-339

**Introduced in R2018a**

# Buck-Boost Converter

Controller-driven DC-DC inverting or four-switch step-up or step-down voltage regulator

**Library:** Simscape / Electrical / Power Systems / Semiconductors



## Description

The Buck-Boost Converter block represents a DC-DC converter that can either step up or step down DC voltage from one side of the converter to the other as driven by an attached controller and gate-signal generator. Buck-boost converters are also known as step-up/step-down voltage regulators because they can increase or decrease voltage magnitude.

The block can also invert voltage so that the polarity of the output voltage is the opposite of the polarity of the input voltage. The magnitude of the output voltage depends on the duty cycle.

The Buck-Boost Converter block allows you to model an inverting buck-boost converter with one switching device or a buck-boost converter with four switching devices. Options for the type of switching devices are:

- GTO — Gate turn-off thyristor. For information on the I-V characteristic of the device, see GTO.
- Ideal semiconductor switch — For information on the I-V characteristic of the device, see Ideal Semiconductor Switch.
- IGBT — Insulated-gate bipolar transistor. For information on the I-V characteristic of the device, see IGBT.
- MOSFET — N-channel metal-oxide-semiconductor field-effect transistor. For information on the I-V characteristic of the device, see MOSFET.
- Thyristor — For information on the I-V characteristic of the device, see Thyristor.

## Model

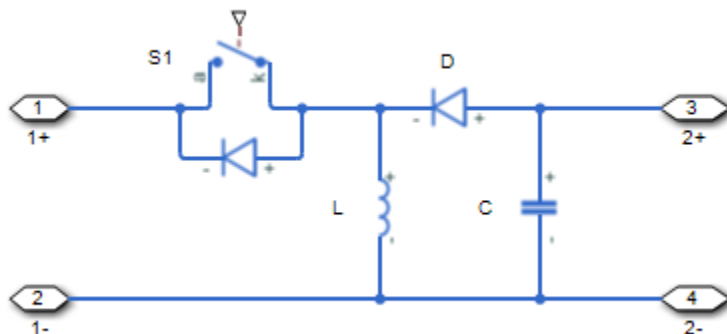
Each of three model variants for the Buck-Boost Converter block corresponds to a **Block choice** option. To access the block choices, in the model window, right-click the block, and then use either of these methods:

- From the context menu, select **Simscape > Block choices**.
- On the Simulink Editor menu bar, select **View > Property Inspector**. In the **Property Inspector** window, click the value of the **Block choice**.

The model variants are:

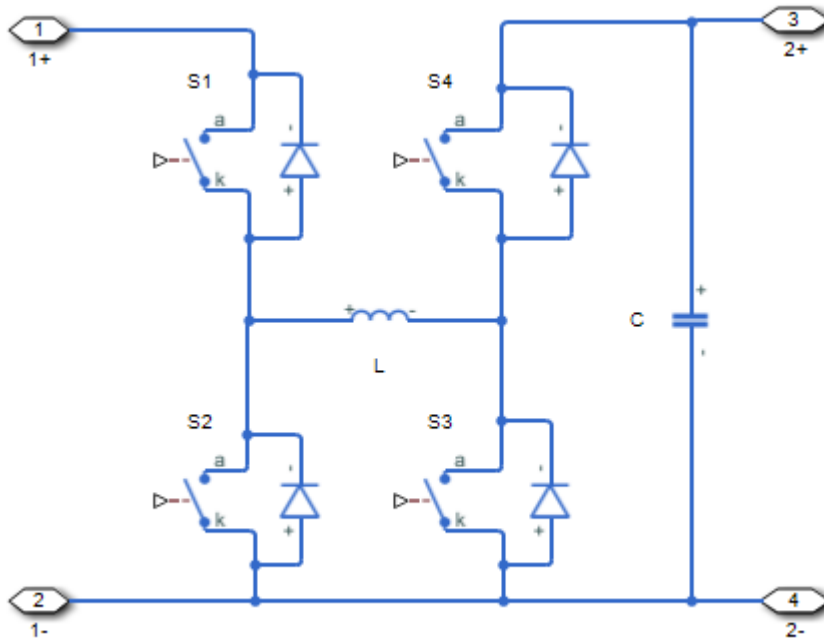
- PS control port — Inverting buck-boost converter with a physical signal port. This block choice is the default.
- Electrical control ports — Inverting buck-boost converter with one positive and one negative electrical conserving ports. To control switching device gates using Simscape Electrical Electronics and Mechatronics blocks, select this option.
- Four-switch converter — Four-switch buck-boost converter with an electrical conserving port.

The inverting converter models contain a switching device, a diode, an inductor, and an output capacitor.



The four-switch converter model contains four switching devices, an inductor, and an output capacitor.





In each case, the capacitor smoothes the output voltage.

## Protection

You can include a snubber circuit for each switching device. Snubber circuits contain a series-connected resistor and capacitor. They protect switching devices against high voltages that inductive loads produce when the device turns off the voltage supply to the load. Snubber circuits also prevent excessive rates of current change when a switching device turns on.

To include and configure a snubber circuit for each switching device, use the **Snubbers** parameters.

## Gate Control

To connect gate-control voltage signals to the gate ports of the switching devices, for the:

- PS control port model:
  - 1 Convert a Simulink gate-control voltage signal to a physical signal using a Simulink-PS Converter block.
  - 2 Connect the Simulink-PS Converter block to the **G** port.
- Electrical control ports model:
  - 1 Connect a Simscape electrical-domain positive DC voltage signal to the **G+** port.
  - 2 Connect the Simscape electrical-domain negative DC voltage signal to the **G-** port.
- Synchronous converter model:
  - 1 Convert each Simulink gate-control voltage signal to a physical signal using Simulink-PS Converter blocks.
  - 2 Multiplex the converted gate-control signals into a single vector using a Four-Pulse Gate Multiplexer
  - 3 Connect the vector signal to the **G** port.

## Ports

### Input

#### **G — Switching device gate control**

physical signal | vector

Physical signal port associated with the gate terminals of the switching device.

#### **Dependencies**

This port is enabled only for the PS control port block choice.

Data Types: double

### Conserving

#### **G — Switching device gate control**

electrical | vector

Electrical conserving port associated with the gate terminals of the switching devices.

**Dependencies**

This port is enabled only for the `Four-switch converter` block choice.

Data Types: `double`

**G+ — Switching device gate control positive terminal**

`electrical` | `scalar`

Positive electrical conserving port associated with the positive gate terminal of the switching device.

**Dependencies**

This port is enabled only for the `Electrical control ports` block choice.

Data Types: `double`

**G- — Switching device gate control negative terminal**

`electrical` | `scalar`

Negative electrical conserving port associated with the negative gate terminal of the switching device.

**Dependencies**

This port is enabled only for the `Electrical control ports` block choice.

Data Types: `double`

**1+ — Positive DC voltage 1**

`electrical` | `scalar`

Electrical conserving port associated with the positive terminal of the first DC voltage.

Data Types: `double`

**1- — Negative DC voltage 1**

`electrical` | `scalar`

Electrical conserving port associated with the negative terminal of the first DC voltage.

Data Types: `double`

**2+ — Positive DC voltage 2**

`electrical` | `scalar`

Electrical conserving port associated with the positive terminal of the second DC voltage.

Data Types: double

## **2- — Negative DC voltage 2**

electrical | scalar

Electrical conserving port associated with the negative terminal of the second DC voltage.

Data Types: double

## **Parameters**

### **Switching Devices**

This table shows how the visibility of **Switching Devices** parameters depends on the **Switching device** that you select. To learn how to read the table, see “Parameter Dependencies” on page A-2.

## Switching Devices Parameter Dependencies

Parameters and Options				
Switching device				
<i>Ideal Semiconductor Switch</i>	<i>GTO</i>	<i>IGBT</i>	<i>MOSFET</i>	<i>Thyristor</i>
On-state resistance	Forward voltage	Forward voltage	Drain-source on resistance	Forward voltage
Off-state conductance	On-state resistance	On-state resistance	Off-state conductance	On-state resistance
Threshold voltage	Off-state conductance	Off-state conductance	Threshold voltage	Off-state conductance
	Gate trigger voltage, Vgt	Threshold voltage		Gate trigger voltage, Vgt
	Gate turn-off voltage, Vgt_off			Gate turn-off voltage, Vgt_off
	Holding current			Holding current

### Switching device – Switch type

*Ideal Semiconductor Switch* (default) | *GTO* | *IGBT* | *MOSFET* | *Thyristor*

Switching device type for the converter. For the four-switch model, the switches are identical.

model.

### Dependencies

See the Switching Devices Parameter Dependencies table.

### Forward voltage – Voltage

0.8 V (default) | scalar

For the different switching device types, the **Forward voltage** is taken as:

- GTO — Minimum voltage required across the anode and cathode block ports for the gradient of the device I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**

- IGBT — Minimum voltage required across the collector and emitter block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**
- Thyristor — Minimum voltage required for the device to turn on

### Dependencies

See the Switching Devices Parameter Dependencies table.

### On-state resistance — Resistance

0.001 Ohm (default) | scalar

For the different switching device types, the **On-state resistance** is taken as:

- GTO — Rate of change of voltage versus current above the forward voltage
- Ideal semiconductor switch — Anode-cathode resistance when the device is on
- IGBT — Collector-emitter resistance when the device is on
- Thyristor — Anode-cathode resistance when the device is on

### Dependencies

See the Switching Devices Parameter Dependencies table.

### Drain-source on resistance — Resistance

0.001 Ohm (default) | scalar

Resistance between the drain and the source, which also depends on the gate-to-source voltage.

### Dependencies

See the Switching Devices Parameter Dependencies table.

### Off-state conductance — Conductance

1e-5 1/Ohm (default) | scalar

Conductance when the device is off. The value must be less than  $1/R$ , where  $R$  is the value of **On-state resistance**.

For the different switching device types, the **On-state resistance** is taken as:

- GTO — Anode-cathode conductance

- Ideal semiconductor switch — Anode-cathode conductance
- IGBT — Collector-emitter conductance
- MOSFET — Drain-source conductance
- Thyristor — Anode-cathode conductance

### Dependencies

See the Switching Devices Parameter Dependencies table.

### Threshold voltage — Voltage threshold

6 V (default) | scalar

Gate voltage threshold. The device turns on when the gate voltage is above this value. For the different switching device types, the device voltage of interest is:

- Ideal semiconductor switch — Gate-emitter voltage
- IGBT — Gate-cathode voltage
- MOSFET — Gate-source voltage

### Dependencies

See the Switching Devices Parameter Dependencies table.

### Gate trigger voltage, $V_{gt}$ — Voltage threshold

1 V (default) | scalar

Gate-cathode voltage threshold. The device turns on when the gate-cathode voltage is above this value.

### Dependencies

See the Switching Devices Parameter Dependencies table.

### Gate turn-off voltage, $V_{gt\_off}$ — Voltage threshold

-1 V (default) | scalar

Gate-cathode voltage threshold. The device turns off when the gate-cathode voltage is below this value.

### Dependencies

See the Switching Devices Parameter Dependencies table.

**Holding current — Current threshold**

1 A (default) | scalar

Gate current threshold. The device stays on when the current is above this value, even when the gate-cathode voltage falls below the gate trigger voltage.

**Dependencies**

See the Switching Devices Parameter Dependencies table.

**Diode**

This table shows how the visibility of **Diode** parameters depends on how you configure the **Model dynamics** and **Reverse recovery time parameterization** parameters. To learn how to read this table, see “Parameter Dependencies” on page A-2.

**Diode Parameter Dependencies**

<b>Parameters and Options</b>				
<b>Model dynamics</b>				
<i>Diode with no dynamics</i>	<i>Diode with charge dynamics</i>			
Forward voltage	Forward voltage			
On resistance	On resistance			
Off conductance	Off conductance			
	Junction capacitance			
	Peak reverse current, iRM			
	Initial forward current when measuring iRM			
	Rate of change of current when measuring iRM			
	<b>Reverse recovery time parameterization</b>			
	<i>Specify stretch factor</i>	<i>Specify reverse recovery time directly</i>	<i>Specify reverse recovery charge</i>	
	Reverse recovery time stretch factor	Reverse recovery time, trr	Reverse recovery charge, Qrr	



**Model dynamics — Diode model**

Diode with no dynamics (default) | Diode with charge dynamics

Diode type. The options are:

- **Diode with no dynamics** — Select this option to prioritize simulation speed using the Diode block.
- **Diode with charge dynamics** — Select this option to prioritize model fidelity in terms of reverse mode charge dynamics using the Commutation Diode block.

**Dependencies**

See the Diode Parameter Dependencies table.

**Forward voltage — Voltage**

0.8 V (default) | scalar

Minimum voltage required across the positive and negative block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**.

**On resistance — Resistance**

0.001 Ohm (default) | scalar

Rate of change of voltage versus current above the **Forward voltage**.**Off conductance — Conductance**

1e-5 1/Ohm (default) | scalar

Conductance of the reverse-biased diode.

**Junction capacitance — Capacitance**

50 nF (default) | scalar

Diode junction capacitance.

**Dependencies**

See the Diode Parameter Dependencies table.

**Peak reverse current,  $i_{RM}$  — Current**

-235 A (default) | scalar less than 0

Peak reverse current measured by an external test circuit.

## Dependencies

See the Diode Parameter Dependencies table.

## Initial forward current when measuring $i_{RM}$ — Current

300 A (default) | scalar greater than 0

Initial forward current when measuring peak reverse current. This value must be greater than zero.

## Dependencies

See the Diode Parameter Dependencies table.

## Rate of change of current when measuring $i_{RM}$ — Current change rate

-50 A/us (default) | scalar

Rate of change of current when measuring peak reverse current.

## Dependencies

See the Diode Parameter Dependencies table.

## Reverse recovery time parameterization — Recovery-time model

Specify stretch factor (default) | Specify reverse recovery time directly  
| Specify reverse recovery charge

Model for parameterizing the recovery time. When you select Specify stretch factor or Specify reverse recovery charge, you can specify a value that the block uses to derive the reverse recovery time.

## Dependencies

See the Diode Parameter Dependencies table.

## Reverse recovery time stretch factor — Stretch factor

3 (default) | scalar greater than 1

Value that the block uses to calculate **Reverse recovery time,  $t_{rr}$** . Specifying the stretch factor is an easier way to parameterize the reverse recovery time than specifying the reverse recovery charge. The larger the value of the stretch factor, the longer it takes for the reverse recovery current to dissipate.

**Dependencies**

See the Diode Parameter Dependencies table.

**Reverse recovery time,  $t_{rr}$  – Time**

15  $\mu$ s (default) | scalar

Interval between the time when the current initially goes to zero (when the diode turns off) and the time when the current falls to less than 10 percent of the peak reverse current.

The value of the **Reverse recovery time,  $t_{rr}$**  parameter must be greater than the value of the **Peak reverse current,  $i_{RM}$**  parameter divided by the value of the **Rate of change of current when measuring  $i_{RM}$**  parameter.

**Dependencies**

See the Diode Parameter Dependencies table.

**Reverse recovery charge,  $Q_{rr}$  – Charge**

1500  $s \cdot \mu$ A (default) | scalar

Value that the block uses to calculate **Reverse recovery time,  $t_{rr}$** . Use this parameter if the data sheet for your diode device specifies a value for the reverse recovery charge instead of a value for the reverse recovery time.

The reverse recovery charge is the total charge that continues to dissipate when the

diode turns off. The value must be less than  $-\frac{i_{RM}^2}{2a}$ ,

where:

- $i_{RM}$  is the value specified for **Peak reverse current,  $i_{RM}$** .
- $a$  is the value specified for **Rate of change of current when measuring  $i_{RM}$** .

**Dependencies**

See the Diode Parameter Dependencies table.

## LC Parameters

### Inductance — Inductance

1e-6 H (default) | positive scalar

Inductance.

### Capacitance — Capacitance

1e-7 F (default) | positive scalar

Capacitance.

### Capacitor effective series resistance — Capacitor resistance

1e-6 Ohm (default) | zero or positive scalar

Series resistance of the capacitor.

## Snubbers

The table summarizes the **Snubbers** parameter dependencies. To learn how to read the table, see “Parameter Dependencies” on page A-2.

### Snubbers Parameter Dependencies

Snubbers Parameter Dependencies	
Snubber	
None	<i>RC Snubber</i>
	Snubber resistance
	Snubber capacitance

### Snubber — Snubber model

None (default) | RC snubber

Switching device snubber.

### Dependencies

See the Snubbers Parameter Dependencies table.

### Snubber resistance — Resistance

0.1 (default) | Ohm | scalar

Resistance of the switching device snubber.

### **Dependencies**

See the Snubbers Parameter Dependencies table.

### **Snubber capacitance — Capacitance**

1e-7 (default) | F | scalar

Capacitance of the switching device snubber.

### **Dependencies**

See the Snubbers Parameter Dependencies table.

## **References**

- [1] Trzynadlowski, A. M. *Introduction to Modern Power Electronics, 2nd Edition*. Hoboken, NJ: John Wiley & Sons Inc., 2010.
- [2] Xiaoyong, R., Z. Tang, X. Ruan, J. Wei and G. Hua. *Four Switch Buck-Boost Converter for Telecom DC-DC power supply applications*. Twenty-Third Annual IEEE Applied Power Electronics Conference and Exposition. Austin, TX: 2008, pp 1527-1530.

## **See Also**

Average-Value DC-DC Converter | Bidirectional DC-DC Converter | Boost Converter | Buck Converter | Converter | GTO | IGBT | Ideal Semiconductor Switch | MOSFET | PWM Generator | PWM Generator (Three-phase, Two-level) | Six-Pulse Gate Multiplexer | Three-Level Converter | Thyristor

## **Topics**

“Alternatives to Specifying trr Directly” on page 1-339

## **Introduced in R2018a**

# Capacitor

Capacitor model including nonlinearity, tolerance, operational limits, and fault behavior



## Library

Passive Devices

## Description

The Capacitor block lets you model linear and nonlinear (table-based) capacitors, including polar capacitors. Optionally, you can also model the following effects:

- “Tolerances” on page 1-265
- “Operating Limits” on page 1-266
- “Faults” on page 1-266

You can turn these modeling options on and off independently of each other. When all the additional options are turned off, the component behavior is identical to the Simscape Foundation library Capacitor block.

In its simplest form, the Capacitor block models a linear capacitor, described with the following equation:

$$I = C \frac{dV}{dt}$$

where:

- $I$  is the current.
- $C$  is the capacitance.

- $V$  is the voltage.
- $t$  is the time.

To model a nonlinear or polar capacitor, set the **Enable table-based capacitance** parameter to **Yes - use table-based capacitance** and provide a lookup table of capacitance-voltage values:

- For polar capacitors, where this lookup table is asymmetric with respect to the applied terminal voltage, set the **Symmetric C-V table** parameter to **No - use C-V data as-is**.
- For other types of nonlinear capacitor, ensure symmetry of the capacitance with regards to the applied terminal voltage by setting the **Symmetric C-V table** parameter to **Yes - use voltage magnitude when computing C**.

## Tolerances

You can apply tolerances to the nominal value you provide for the **Capacitance** parameter. Datasheets typically provide a tolerance percentage for a given capacitor type. The table shows how the block applies tolerances and calculates capacitance based on the selected **Tolerance application** option.

Option	Capacitance Value
None – use nominal value	$C$
Random tolerance	Uniform distribution: $C \cdot (1 - tol + 2 \cdot tol \cdot rand)$ Gaussian distribution: $C \cdot (1 + tol \cdot randn / nSigma)$
Apply maximum tolerance value	$C \cdot (1 + tol)$
Apply minimum tolerance value	$C \cdot (1 - tol)$

In the table,

- $C$  is the **Capacitance** parameter value, nominal capacitance.
- $tol$  is the fractional tolerance, **Capacitance tolerance (%) / 100**.
- $nSigma$  is the value you provide for the **Number of standard deviations for quoted tolerance** parameter.

- `rand` and `randn` are standard MATLAB functions for generating uniform and normal distribution random numbers.

## Operating Limits

You can specify operating limits in terms of maximum working voltage and the maximum (instantaneous) power dissipation in the series resistance and in the parallel conductance of the capacitor.

For polar capacitors, you can define the working voltage range in such a way that the block provides a warning, or an error, if the polarity of the applied voltage becomes incorrect.

When an operating limit is exceeded, the block can either generate a warning or stop the simulation with an error. For more information, see “Operating Limits” on page 1-269.

## Faults

Instantaneous changes in capacitor parameters are unphysical. Therefore, when the Capacitor block enters the faulted state, the capacitance, resistance, and conductance transition to their faulted values over a period of time, according to the following formula:

$$CurrentValue = FaultedValue - (FaultedValue - UnfaultedValue) \cdot \operatorname{sech}(\Delta t / \tau)$$

where:

- $\Delta t$  is the time since the onset of the fault condition.
- $\tau$  is the user-defined time constant associated with the fault transition.

The block can trigger the start of fault transition:

- At a specific time
- When terminal voltage is outside the permissible voltage range for longer than a specific time interval

You can enable or disable these trigger mechanisms separately, or use them together if more than one trigger mechanism is required in a simulation. When more than one mechanism is enabled, the first mechanism to trigger the fault transition takes precedence. In other words, component fails no more than once per simulation.



You can also choose whether to issue an assertion when a fault occurs, by using the **Reporting when a fault occurs** parameter. The assertion can take the form of a warning or an error. By default, the block does not issue an assertion.

## Variables

Use the **Variables** section of the block interface to set the priority and initial target values for the block variables prior to simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

The **Capacitor voltage** variable lets you specify a high-priority target for the initial capacitor voltage at the start of simulation.

## Parameters

- “Main” on page 1-267
- “Operating Limits” on page 1-269
- “Faults” on page 1-270

## Main

### Enable table-based capacitance

Select the type of capacitor:

- **No - use constant capacitance** — Model a linear capacitor, with nominal capacitance defined by the **Capacitance** parameter value. This is the default.
- **Yes - use table-based capacitance** — Model a nonlinear capacitor, where the nominal capacitance value changes based on the value of applied terminal voltage.

### Capacitance

The nominal capacitance value for linear capacitor. This parameter is visible only if you select **No - use constant capacitance** for the **Enable table-based capacitance** parameter. Capacitance value must be greater than zero. The default value is 1  $\mu\text{F}$ .

### Capacitance values

The vector of capacitance values, for table lookup based on the corresponding voltage value. This parameter is visible only if you select **Yes - use table-based**

capacitance for the **Enable table-based capacitance** parameter. Capacitance values must be greater than 0. The vector length must be the same as the voltage vector length. The default value is [1e-5 1e-6] F.

### **Corresponding voltage values**

The input vector of voltage values for table-based capacitance calculation. This parameter is visible only if you select **Yes - use table-based capacitance** for the **Enable table-based capacitance** parameter. The vector length must be greater than or equal to 2, and the values must be strictly monotonic, either increasing or decreasing. The default value is [0 10] V.

### **Symmetric C-V table**

This parameter is visible only if you select **Yes - use table-based capacitance** for the **Enable table-based capacitance** parameter. Specify how to use the table data:

- **Yes - use voltage magnitude when computing C** — Use this option to ensure symmetry of the capacitance with regards to the applied terminal voltage. This is the default.
- **No - use C-V data as-is** — Use this option to model polar capacitors. For example, with default parameter values for table-based capacitance, applied voltage of -10 V would produce nominal capacitance of 1e-6 F. However, if you select **No - use C-V data as-is** for the **Symmetric C-V table** parameter, the resulting capacitance value is 1e-5 F, because the block uses the nearest input value for extrapolation.

### **Capacitance tolerance (%)**

The capacitor tolerance as defined on the manufacturer datasheet. For table-based capacitors, this tolerance is applied to the entire table at once. The default value is 5%.

### **Tolerance application**

Select how to apply tolerance during simulation:

- **None – use nominal value** — The block does not apply tolerance, uses the nominal capacitance value. This is the default.
- **Random tolerance** — The block applies random offset to the capacitance value, within the tolerance value limit. You can choose Uniform or Gaussian distribution for calculating the random number by using the **Tolerance distribution** parameter.

- **Apply maximum tolerance value** — The capacitance is increased by the specified tolerance percent value.
- **Apply minimum tolerance value** — The capacitance is decreased by the specified tolerance percent value.

### **Tolerance distribution**

This parameter is visible only if you select **Random** tolerance for the **Tolerance application** parameter. Select the distribution type:

- **Uniform** — Uniform distribution. This is the default.
- **Gaussian** — Gaussian distribution.

### **Number of standard deviations for quoted tolerance**

Number of standard deviations for calculating the Gaussian random number. This parameter is visible only if you select **Gaussian** for the **Tolerance distribution** parameter. The default value is 4.

### **Series resistance**

Simulation of some circuits may require the presence of the small series resistance. Equivalent series resistance (ESR) is sometimes specified on manufacturer datasheets. If not, you can define this resistance via the dissipation factor (DF), which is also shown on many datasheets. The relationship is  $DF = 2\pi \cdot f \cdot C \cdot ESR$ , where  $f$  is signal frequency. The default value is 1  $\mu\Omega$ .

### **Parallel conductance**

Parallel leakage path associated with the capacitor. For capacitors connected in series, the presence of a small parallel conductance can help with convergence. The default value is 0 1/ $\Omega$ .

## **Operating Limits**

### **Enable operating limits**

Select **Yes** to enable reporting when the operational limits are exceeded. The associated parameters become visible on the **Operating Limits** tab to let you select the reporting method and specify the operating limits in terms of power and working voltage. The default value is **No**.

### **Reporting when operating limits exceeded**

Select what happens when an operating limit is exceeded:

- Warn — The block issues a warning. This is the default.
- Error — Simulation stops with an error.

### **Working voltage range**

Range of voltage values allowed for normal block operation, specified as a vector of size 2. The default value is [-25 25] V.

### **Power rating**

Maximum instantaneous power dissipation in the resistance and conductance elements associated with the capacitor. The default value is 1 W.

## **Faults**

### **Enable faults**

Select Yes to enable faults modeling. The associated parameters become visible on the **Faults** tab to let you select the reporting method and specify the trigger mechanism (temporal or behavioral). You can enable these trigger mechanisms separately or use them together. The default value is No.

### **Reporting when a fault occurs**

Choose whether to issue an assertion when a fault occurs:

- None — The block does not issue an assertion. This is the default.
- Warn — The block issues a warning.
- Error — Simulation stops with an error.

### **Faulted capacitance as % of unfaulted**

Relative change in the capacitance when the block is in the faulted state, as compared to the unfaulted state. For table-based capacitances, the relative change is applied to all elements of the vector. The default value is 100, which means that the faulted capacitance is equal to the unfaulted capacitance.

### **Faulted series resistance**

Equivalent series resistance of the capacitor when the block is in the faulted state. The default value is 1 m $\Omega$ .

### **Faulted parallel conductance**

Parallel leakage conductance of the capacitor when the block is in the faulted state. The default value is 0 1/ $\Omega$ .

**Fault transition time constant**

Time constant associated with the transition to the faulted state, as described in “Faults” on page 1-266. The default value is 1 ms.

**Enable temporal fault trigger**

Select Yes to enable time-based fault triggering. The default value is No.

**Simulation time for a fault event**

Set the simulation time at which you want the block to start entering the fault state. This parameter is visible only if the **Enable temporal fault trigger** parameter is set to Yes. The default value is 1 s.

**Enable behavioral fault trigger**

Select Yes to enable behavioral fault triggering. The default value is No.

**Permissible voltage range**

Specify the minimum and maximum permissible voltage. If the voltage value is outside this range for longer than the **Time to fail when exceeding voltage range** parameter value, then the block starts entering the fault state. This parameter is visible only if the **Enable behavioral fault trigger** parameter is set to Yes. The default value is [ -100 100 ] V.

**Time to fail when exceeding voltage range**

Set the maximum length of time that the voltage can be outside the permissible voltage range without triggering the fault. This parameter is visible only if the **Enable behavioral fault trigger** parameter is set to Yes. The default value is 1 s.

## Ports

The block has the following ports:

+

Positive electrical port

-

Negative electrical port

## See Also

Fault | Inductor | Resistor

## Cauer Thermal Model Element

Heat transfer through an individual layer of a semiconductor module

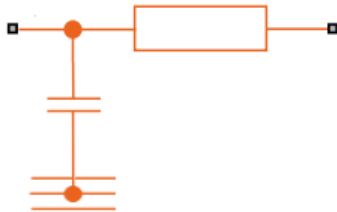


### Library

Simscape / Electrical / Power Systems / Semiconductors / Fundamental Components / Thermal

### Description

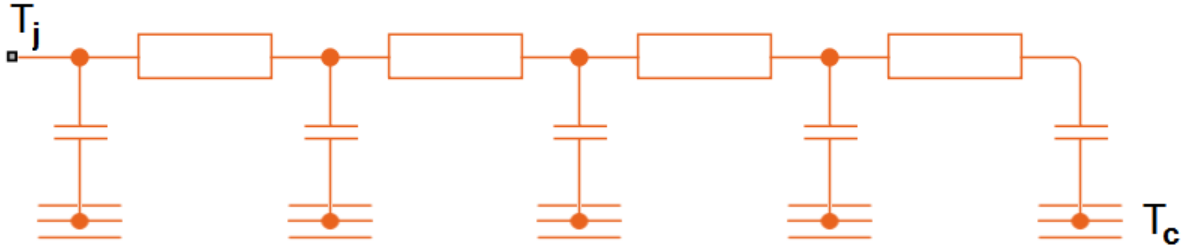
The Cauer Thermal Model Element block represents heat transfer through an individual layer of a semiconductor module. The figure shows an equivalent circuit for a Cauer Thermal Model Element block.



A Cauer thermal model represents the multiple layers that constitute the packaging of a semiconductor. Layers include chip, solder, substrate, solder, and base. Other terms that describe a Cauer thermal model are:

- Continued fraction circuit
- T model
- Ladder network

To create a Cauer thermal model, connect multiple instances of the Cauer Thermal Model Element block in series. In the figure of the Cauer thermal model,  $T_j$  is the junction temperature and  $T_c$  is the base plate temperature.



The defining equations for the Cauer Thermal Model Element block are:

$$C_{thermal} = \frac{\tau}{R_{thermal}},$$

$$Q_{AB} = \frac{T_{AB}}{R_{thermal}},$$

and

$$Q_{AR} = C_{thermal} \frac{dT_{AR}}{dt},$$

where:

- $C_{thermal}$  is the thermal capacity.
- $\tau$  is the thermal time constant.
- $R_{thermal}$  is the thermal resistance.
- $Q_{AB}$  is the heat flow through the material.
- $T_{AB}$  is the temperature difference between the material layers.
- $Q_{AR}$  is the heat flow through the thermal capacity.
- $T_{AR}$  is the temperature drop across the thermal capacity.

## Parameters

- “Parameters Tab” on page 1-274
- “Variables Tab” on page 1-274

### Parameters Tab

#### Thermal resistance

The default value for the thermal resistance,  $R_{thermal}$ , is  $5e-3$  K/W.

#### Thermal time constant

The default value for the thermal time constant,  $\tau$ , is  $0.1$  s.

### Variables Tab

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Ports

The block has the following ports:

A

Thermal conserving port associated with the first surface of the individual layer of the semiconductor.

R

Thermal conserving port associated with the chosen thermal reference.

B

Thermal conserving port associated with the second surface of the individual layer of the semiconductor.



## References

- [1] Schütze, T. *AN2008-03: Thermal equivalent circuit models*. Application Note. V1.0. Germany: Infineon Technologies AG, 2008.

## See Also

Foster Thermal Model | Thermal Resistor

## Topics

“Quantifying IGBT Thermal Losses”

**Introduced in R2016a**

## Change Detector

Boolean signal change detector

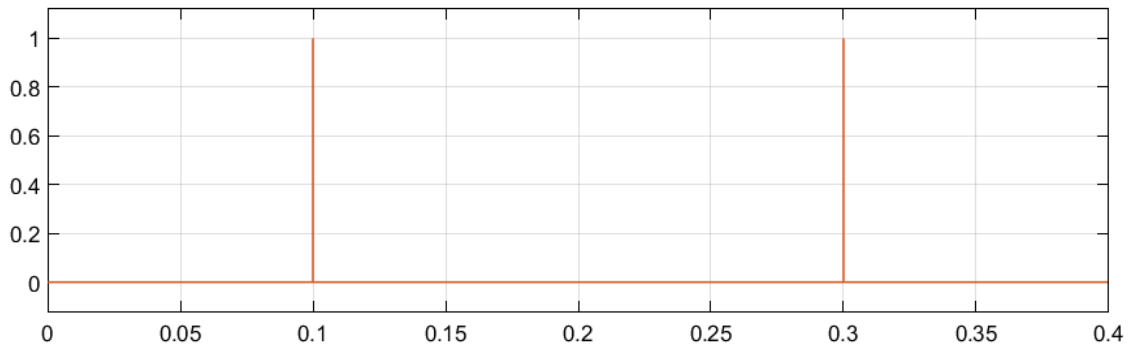
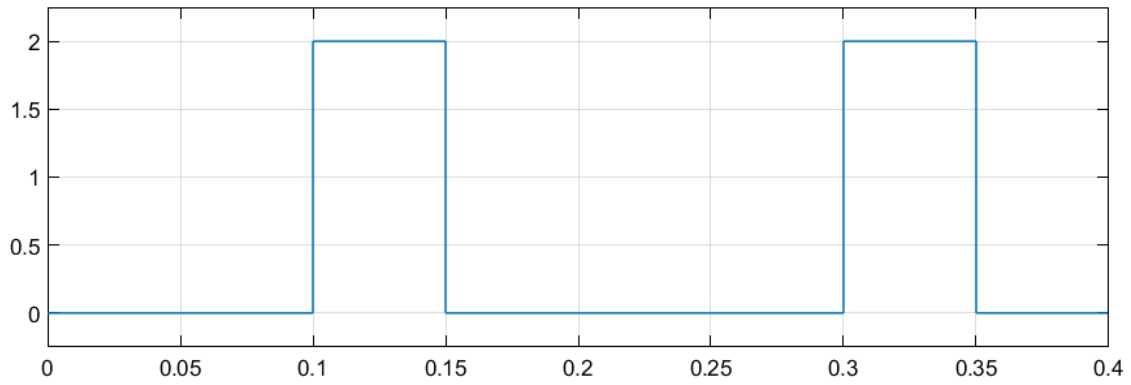
**Library:**           Simscape / Electrical / Power Systems / Control /  
                          General Control



## Description

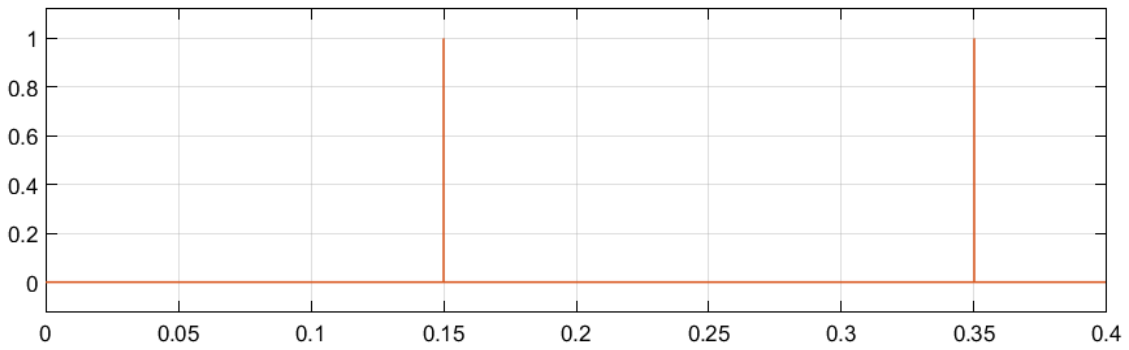
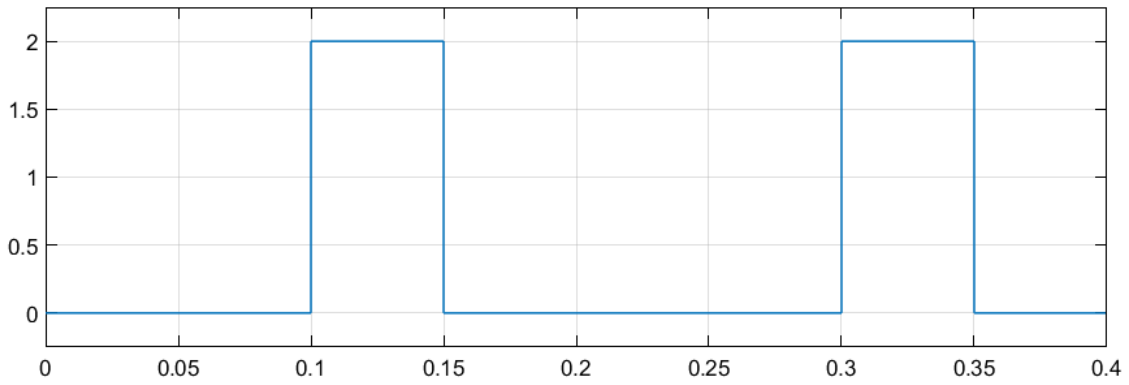
The Change Detector block outputs a Boolean response of `true` when it detects a change in the Boolean input signal that meets one of these change criteria:

- Rising edge — The input goes from false to true.

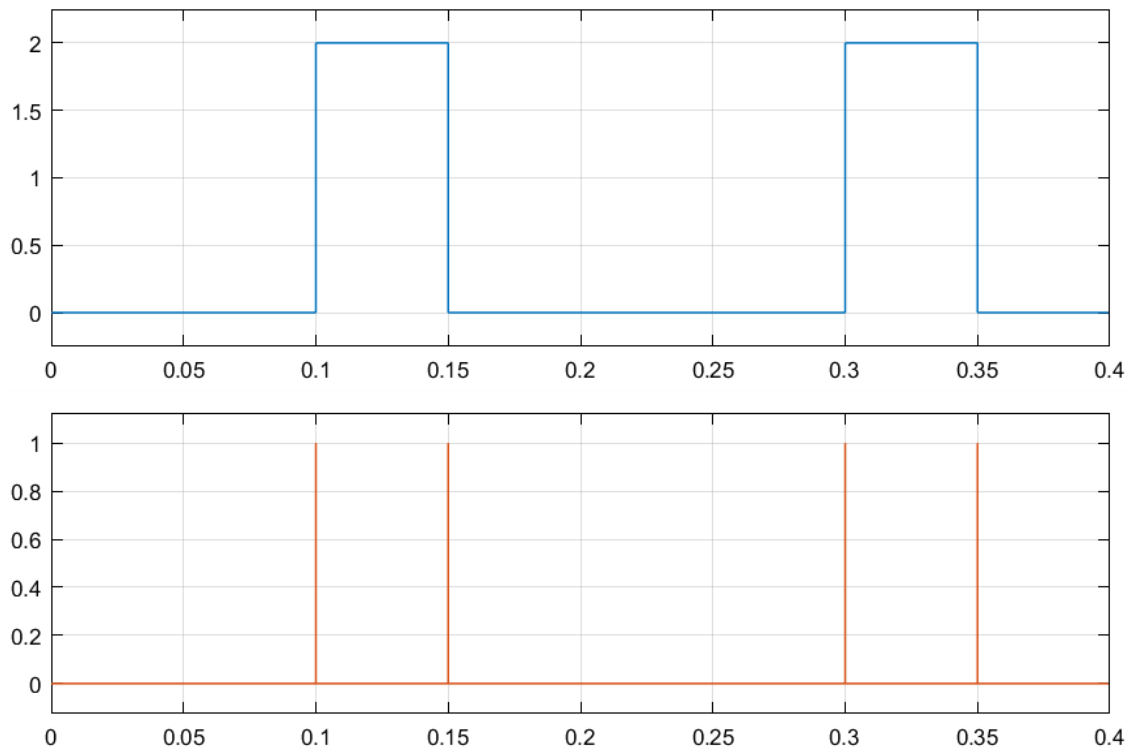


Offset=0

- Falling edge — The input goes from true to false.



- Either edge — The input goes from true to false or from false to true.



## Ports

### Input

**u** – Boolean input

0 or 1

Input Boolean signal. If false, 0. If true, 1.

Data Types: Boolean

## Output

### **y — Change report**

0 or 1

Output is true, 1, when the block detects a change that corresponds to the specified criteria (rising, falling, or either edge). Otherwise, output is false, 0.

Data Types: Boolean

## Parameters

### **Change detection — Change criteria**

Rising edge (default) | Falling edge | Either edge

Criteria for change detection.

### **Initial condition — Initial Boolean value**

0 (default) | 1

Initial value of the previous input. If the input at the start of simulation is different from the initial condition value, the block detects an edge.

### **Sample time (-1 for inherited) — Block sample time**

-1 (default) | 0 | positive scalar

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For inherited discrete-time operation, specify -1. For discrete-time operation, specify a positive integer. For continuous-time operation, specify 0.

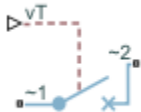
If this block is in a masked subsystem, or other variant subsystem that allows you to switch between continuous operation and discrete operation, promote the sample time parameter. Promoting the sample time parameter ensures correct switching between the continuous and discrete implementations of the block. For more information, see “Promote Parameter to Mask” (Simulink).

## **See Also**

**Introduced in R2018b**

# Circuit Breaker

Three-phase circuit breaker controlled by external signal



## Library

Simscape / Electrical / Power Systems / Switches & Breakers

## Description

The Circuit Breaker block models a three-phase circuit breaker that uses an external signal and phase current information to break an electrical circuit.

The table shows how the external signal  $vT$  controls the block behavior.

Condition	Block Behavior	Resistance Parameter Used
$vT < \text{Threshold}$	The breaker is closed. Each phase in the composite three-phase port ~1 connects to the corresponding phase in the port ~2.	<b>Closed Resistance</b>
$vT \geq \text{Threshold}$	When the current in any phase of the composite port ~1 crosses zero, the phase disconnects from the corresponding phase at port ~2. The breaker is open.	<b>Open Conductance</b>



## Parameters

### Closed resistance

Resistance between ports ~1 and ~2 when the breaker is closed. The default value is 0.001 Ohm.

### Open conductance

Conductance between ports ~1 and ~2 when the breaker is open. The default value is 1e-6 1/Ohm.

### Threshold

Threshold voltage for the control port vT. The block uses the threshold voltage and the value of vT at the start of the simulation to determine whether the breaker is initially open or closed. When the voltage rises above the threshold, the breaker opens each phase as its current crosses zero. When the control port voltage falls below the threshold, the breaker closes. The default value is 0.5 V.

## Ports

The block has the following ports:

~1

Expandable three-phase port

~2

Expandable three-phase port

vT

Scalar control port, which is either a physical signal or an electrical port.

## See Also

Single-Phase Circuit Breaker | Single-Phase Circuit Breaker (with arc)

## Topics

“Three-Phase Synchronous Machine Control”

“Three-Phase Custom Simplified Synchronous Machine”

“Marine Full Electric Propulsion Power System”

“Switch Between Physical Signal and Electrical Ports”  
“Expand and Collapse Three-Phase Ports on a Block”

**Introduced in R2013b**

# Clarke to Park Angle Transform

Implement  $\alpha\beta 0$  to  $dq0$  transform

**Library:** Simscape / Electrical / Power Systems / Control /  
Mathematical Transforms

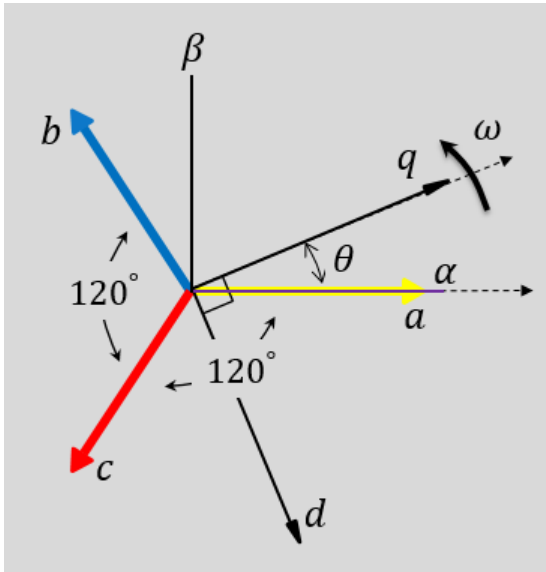


## Description

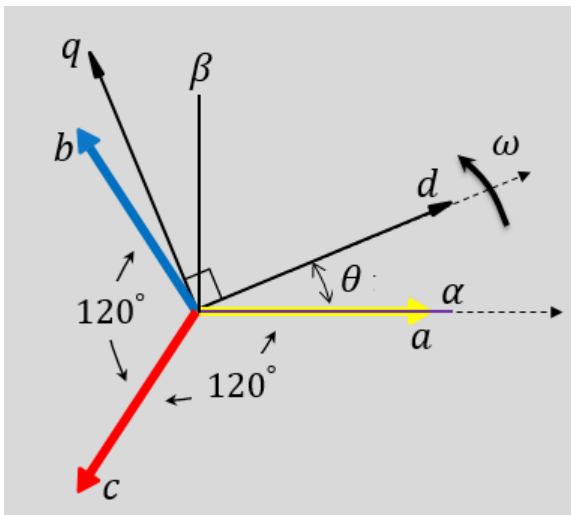
The Clarke to Park Angle Transform block converts the alpha, beta, and zero components in a stationary reference frame to direct, quadrature, and zero components in a rotating reference frame. For balanced three-phase systems, the zero components are equal to zero.

You can configure the block to align the phase  $a$ -axis of the three-phase system to either the  $q$ - or  $d$ -axis of the rotating reference frame at time,  $t = 0$ . The figures show the direction of the magnetic axes of the stator windings in the three-phase system, a stationary  $\alpha\beta 0$  reference frame, and a rotating  $dq0$  reference frame where:

- The  $a$ -axis and the  $q$ -axis are initially aligned.



- The  $a$ -axis and the  $d$ -axis are initially aligned.

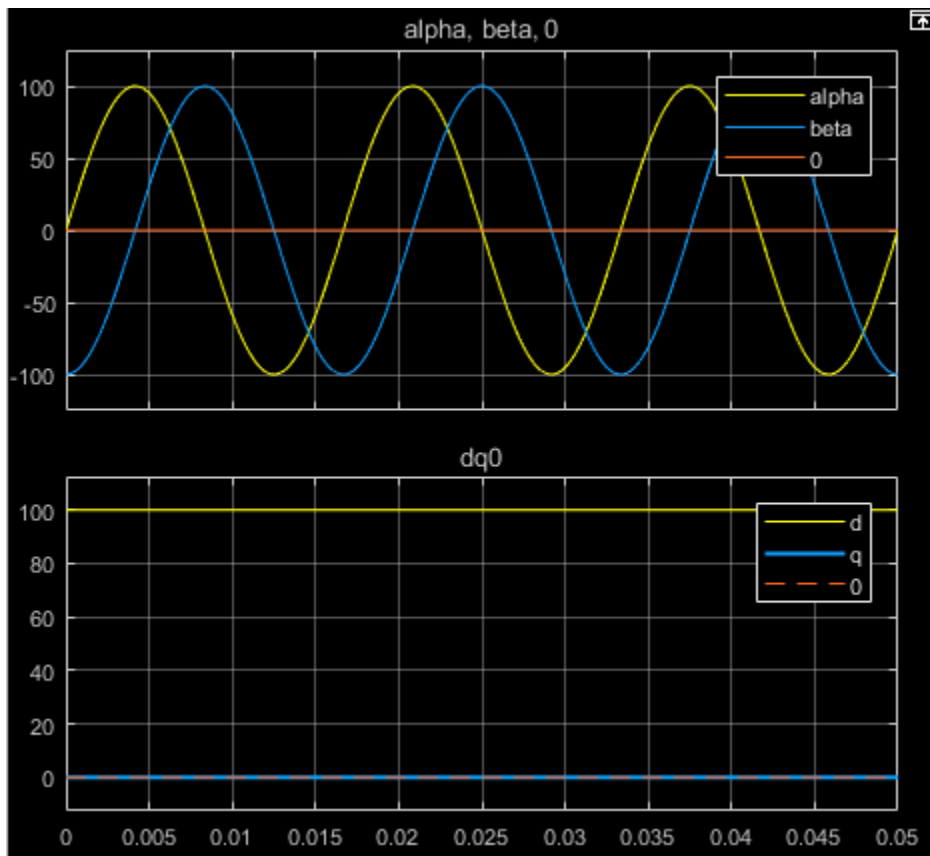


In both cases, the angle  $\theta = \omega t$ , where

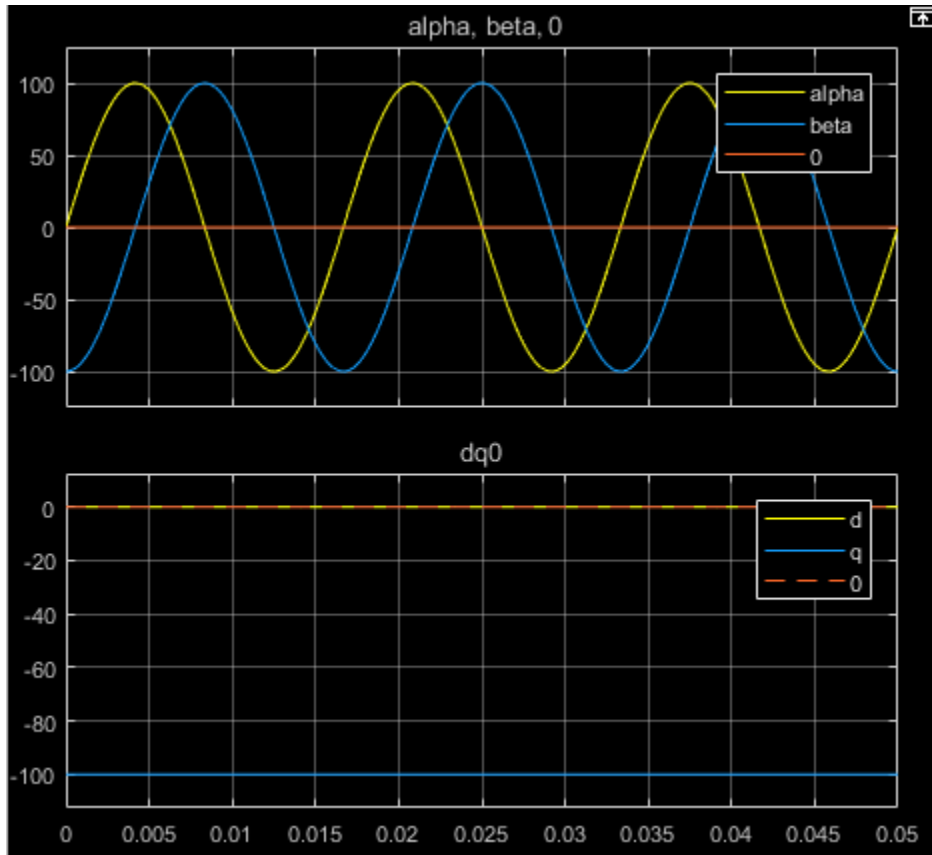
- $\theta$  is the angle between the  $a$  and  $q$  axes for the  $q$ -axis alignment or the angle between the  $a$  and  $d$  axes for the  $d$ -axis alignment.
- $\omega$  is the rotational speed of the  $d$ - $q$  reference frame.
- $t$  is the time, in s, from the initial alignment.

The figures show the time-response of the individual components of equivalent balanced  $\alpha\beta 0$  and  $dq0$  for an:

- Alignment of the  $a$ -phase vector to the  $q$ -axis



- Alignment of the  $a$ -phase vector to the  $d$ -axis



## Equations

The Clarke to Park Angle Transform block implements the transform for an  $a$ -phase to  $q$ -axis alignment as

$$\begin{bmatrix} d \\ q \\ 0 \end{bmatrix} = \begin{bmatrix} \sin(\theta) & -\cos(\theta) & 0 \\ \cos(\theta) & \sin(\theta) & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} \alpha \\ \beta \\ 0 \end{bmatrix}$$

where:

- $\alpha$  and  $\beta$  are the alpha-axis and beta-axis components of the two-phase system in the stationary reference frame.
- $0$  is the zero component.
- $d$  and  $q$  are the direct-axis and quadrature-axis components of the two-axis system in the rotating reference frame.

For an  $a$ -phase to  $d$ -axis alignment, the block implements the transform using this equation:

$$\begin{bmatrix} d \\ q \\ 0 \end{bmatrix} = \begin{bmatrix} \cos(\theta) & \sin(\theta) & 0 \\ -\sin(\theta) & \cos(\theta) & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} \alpha \\ \beta \\ 0 \end{bmatrix}$$

## Ports

### Input

#### $\alpha\beta0$ — $\alpha$ - $\beta$ axis and zero components

vector

Alpha-axis,  $\alpha$ , beta-axis,  $\beta$ , and zero components of the two-phase system in the stationary reference frame.

Data Types: single | double

#### $\theta_{abc}$ — Rotational angle

scalar | in radians

Angular position of the rotating reference frame. The value of this parameter is equal to the polar distance from the vector of the  $a$ -phase in the  $abc$  reference frame to the initially aligned axis of the  $dq0$  reference frame.

Data Types: single | double

### Output

#### $dq0$ — $d$ - $q$ axis and zero components

vector

Direct-axis and quadrature-axis components and the zero component of the system in the rotating reference frame.

Data Types: `single` | `double`

## Parameters

### Phase-a axis alignment — $dq0$ reference frame alignment

Q-axis (default) | D-axis

Align the  $a$ -phase vector of the  $abc$  reference frame to the  $d$ - or  $q$ -axis of the rotating reference frame.

## References

[1] Krause, P., O. Wasynczuk, S. D. Sudhoff, and S. Pekarek. *Analysis of Electric Machinery and Drive Systems*. Piscataway, NJ: Wiley-IEEE Press, 2013.

## See Also

### Blocks

Clarke Transform | Inverse Clarke Transform | Inverse Park Transform | Park Transform | Park to Clarke Angle Transform

**Introduced in R2017b**



# Clarke Transform

Implement  $abc$  to  $\alpha\beta 0$  transform

**Library:** Simscape / Electrical / Power Systems / Control /  
Mathematical Transforms

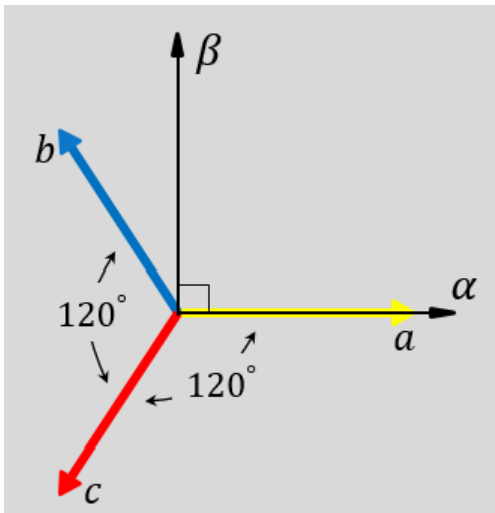


## Description

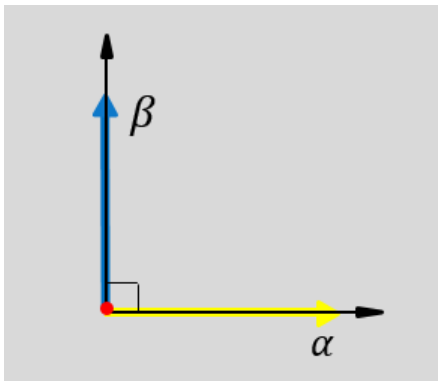
The Clarke Transform block converts the time-domain components of a three-phase system in an  $abc$  reference frame to components in a stationary  $\alpha\beta 0$  reference frame. The block can preserve the active and reactive powers with the powers of the system in the  $abc$  reference frame by implementing a power invariant version of the Clarke transform. For a balanced system, the zero component is equal to zero.

The figures show:

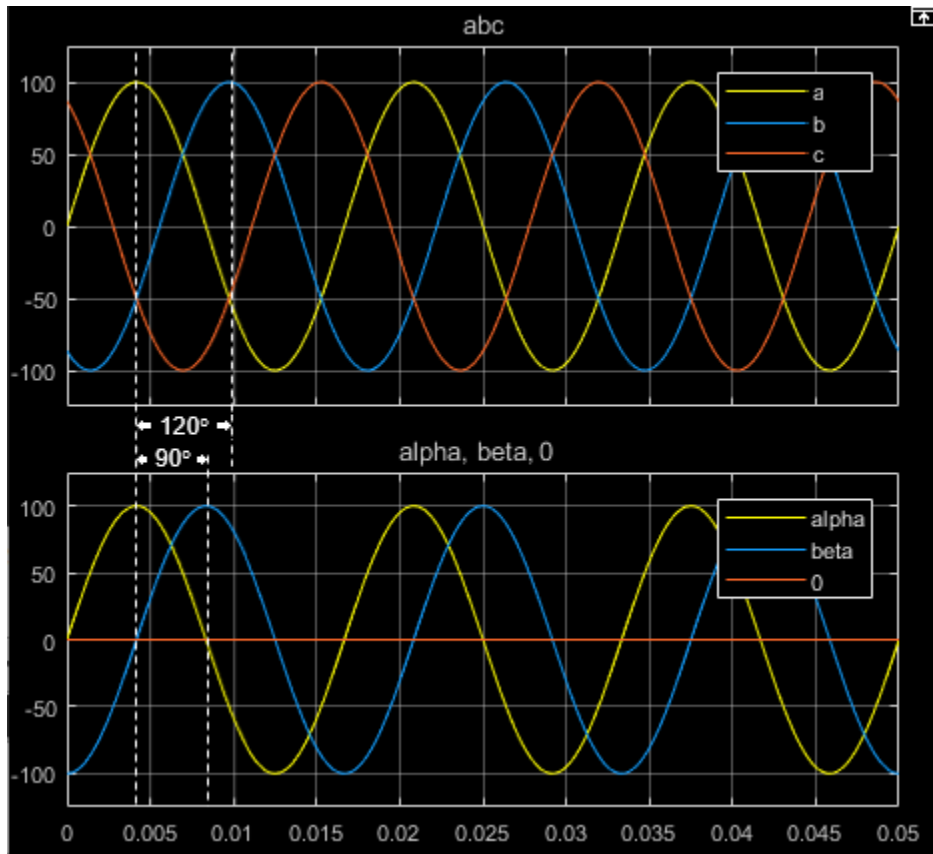
- The direction of the magnetic axes of the stator windings in the  $abc$  reference frame and the stationary  $\alpha\beta 0$  reference frame



- Equivalent  $\alpha$ ,  $\beta$ , and zero components in the stationary reference frame



- The time-response of the individual components of equivalent balanced  $abc$  and  $\alpha\beta 0$  systems



## Equations

The block implements the Clarke transform as

$$\begin{bmatrix} \alpha \\ \beta \\ 0 \end{bmatrix} = \frac{2}{3} \begin{bmatrix} 1 & -\frac{1}{2} & -\frac{1}{2} \\ 0 & \frac{\sqrt{3}}{2} & -\frac{\sqrt{3}}{2} \\ \frac{1}{2} & \frac{1}{2} & \frac{1}{2} \end{bmatrix} \begin{bmatrix} a \\ b \\ c \end{bmatrix},$$

where:

- $a$ ,  $b$ , and  $c$  are the components of the three-phase system in the  $abc$  reference frame.
- $\alpha$  and  $\beta$  are the components of the two-axis system in the stationary reference frame.
- $0$  is the zero component of the two-axis system in the stationary reference frame.

The block implements the power invariant version of the Clarke transform as

$$\begin{bmatrix} \alpha \\ \beta \\ 0 \end{bmatrix} = \sqrt{\frac{2}{3}} \begin{bmatrix} 1 & -\frac{1}{2} & -\frac{1}{2} \\ 0 & \frac{\sqrt{3}}{2} & -\frac{\sqrt{3}}{2} \\ \frac{1}{\sqrt{2}} & \frac{1}{\sqrt{2}} & \frac{1}{\sqrt{2}} \end{bmatrix} \begin{bmatrix} a \\ b \\ c \end{bmatrix}.$$

## Ports

### Input

#### **abc — a-, b-, and c-phase components**

vector

Components of the three-phase system in the  $abc$  reference frame.

Data Types: single | double

### Output

#### **$\alpha\beta 0$ — $\alpha$ - $\beta$ axis and zero components**

vector

Alpha-axis component,  $\alpha$ , beta-axis component  $\beta$ , and zero component in the stationary reference frame.

Data Types: single | double

## Parameters

### **Power Invariant** — Power invariant transform

off (default) | on

Preserve the active and reactive power of the system in the *abc* reference frame.

## References

- [1] Krause, P., O. Wasynczuk, S. D. Sudhoff, and S. Pekarek. *Analysis of Electric Machinery and Drive Systems*. Piscataway, NJ: Wiley-IEEE Press, 2013.

## See Also

### **Blocks**

Clarke to Park Angle Transform | Inverse Clarke Transform | Inverse Park Transform | Park Transform | Park to Clarke Angle Transform

**Introduced in R2017b**

## CMOS AND

Model CMOS AND gate behaviorally



### Library

Logic

### Description

The CMOS AND block represents a CMOS AND logic gate behaviorally:

- The block output logic level is **HIGH** if the logic levels of both of the gate inputs are 1.
- The block output logic level is **LOW** otherwise.

The block determines the logic levels of the gate inputs as follows:

- If the gate voltage is greater than the threshold voltage, the block interprets the input as logic 1.
- Otherwise, the block interprets the input as logic 0.

The threshold voltage is the voltage value at midpoint between the **High level input voltage** parameter value and the **Low level input voltage** parameter value.

---

**Note** To improve simulation speed, the block does not model all the internal individual MOSFET devices that make up the gate. See “Basic Assumptions and Limitations” on page 1-297 for details.

---

The block models the gate as follows:

- The gate inputs have infinite resistance and finite or zero capacitance.

- The gate output offers a selection of two models: **Linear** and **Quadratic**. For more information, see “Selecting the Output Model for Logic Blocks”. Use the **Output current-voltage relationship** parameter to specify the output model.
- You can specify propagation delay for both output models. For **Linear** output, the block sets the value of the gate output capacitor such that the resistor-capacitor time constant equals the **Propagation delay** parameter value. For **Quadratic** output, the gate input demand is lagged to approximate the **Propagation delay** parameter value.

The block output voltage depends on the output model selected:

- For **Linear** model, output high is the **High level output voltage** parameter value, and output low is the **Low level output voltage** parameter value.
- For **Quadratic** model, the output voltage for High and Low states is a function of the output current, as explained in “Quadratic Model Output and Parameters”. For zero load current, output high is  $V_{cc}$  (the **Supply voltage** parameter value), and output low is zero volts.

## Basic Assumptions and Limitations

The block does not model the internal individual MOSFET devices that make up the gate (except for the final MOSFET pair if you select the **Quadratic** option for the **Output current-voltage relationship** parameter). This limitation has the following implications:

- The block does not accurately model the gate's response to input noise and inputs that are around the logic threshold voltage.
- The block does not accurately model dynamic response.

Circuits that involve a feedback path around a set of logic gates may require a nonzero propagation delay to be set on one or more gates.

## Parameters

- “Inputs Tab” on page 1-298
- “Outputs Tab” on page 1-298
- “Initial Conditions Tab” on page 1-300

## Inputs Tab

### Low level input voltage

Voltage value below which the block interprets the input voltage as logic LOW. The default value is 2 V.

### High level input voltage

Voltage value above which the block interprets the input voltage as logic HIGH. The default value is 3 V.

### Average input capacitance

Fixed capacitance that approximates the input capacitance for a MOSFET gate. The MOSFET capacitance depends on the applied voltage. When you drive this block with another gate, the **Average input capacitance** produces a rise time similar to that of the MOSFET. You can usually find this capacitance value on a manufacturer datasheet. The default value is 5 pF. Setting this value to zero may result in faster simulation times.

## Outputs Tab

### Output current-voltage relationship

Select the output model, **Linear** or **Quadratic**. The default value is **Linear**.

### Low level output voltage

Voltage value at the output when the output logic level is LOW. The default value is 0 V. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter.

### High level output voltage

Voltage value at the output when the output logic level is HIGH. The default value is 5 V. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter.

### Output resistance

Value of the series output resistor that is used to model the drop in output voltage resulting from the output current. The default value is 25  $\Omega$ . You can derive this value from a datasheet by dividing the high-level output voltage by the maximum low-level output current. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter.



### Supply voltage

Supply voltage value applied to the gate in your circuit. The default value is 5 V. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Measurement voltage

The gate supply voltage for which mask data output resistances and currents are defined. The default value is 5 V. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Logic HIGH output resistance at zero current and at I<sub>OH</sub>

A row vector [  $R_{OH1}$   $R_{OH2}$  ] of two resistance values. The first value  $R_{OH1}$  is the gradient of the output voltage-current relationship when the gate is logic HIGH and there is no output current. The second value  $R_{OH2}$  is the gradient of the output voltage-current relationship when the gate is logic HIGH and the output current is  $I_{OH}$ . The default value is [ 25 250 ]  $\Omega$ . This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Logic HIGH output current I<sub>OH</sub> when shorted to ground

The resulting current when the gate is in the logic HIGH state, but the load forces the output voltage to zero. The default value is 63 mA. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Logic LOW output resistance at zero current and at I<sub>OL</sub>

A row vector [  $R_{OL1}$   $R_{OL2}$  ] of two resistance values. The first value  $R_{OL1}$  is the gradient of the output voltage-current relationship when the gate is logic LOW and there is no output current. The second value  $R_{OL2}$  is the gradient of the output voltage-current relationship when the gate is logic LOW and the output current is  $I_{OL}$ . The default value is [ 30 800 ]  $\Omega$ . This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Logic LOW output current I<sub>OL</sub> when shorted to V<sub>cc</sub>

The resulting current when the gate is in the logic LOW state, but the load forces the output voltage to the supply voltage  $V_{cc}$ . The default value is -45 mA. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### **Propagation delay**

Time it takes for the output to swing from LOW to HIGH or HIGH to LOW after the input logic levels change. The default value is 25 ns.

### **Protection diode on resistance**

The gradient of the voltage-current relationship for the protection diodes when forward biased. The default value is 5  $\Omega$ . This parameter is available when you select the Quadratic option for the **Output current-voltage relationship** parameter.

### **Protection diode forward voltage**

The voltage above which the protection diode is turned on. The default value is 0.6 V. This parameter is available when you select the Quadratic option for the **Output current-voltage relationship** parameter.

## **Initial Conditions Tab**

### **Output initial state**

Specify whether the initial output state of the block is High or Low. This parameter is used for both linear and quadratic output states, provided that the **Propagation delay** parameter is greater than zero and the Solver Configuration block does not have the **Start simulation from steady state** option selected. The default value is Low.

## **Ports**

The block has the following ports:

A

Electrical input port

B

Electrical input port

J

Electrical output port

# CMOS Buffer

Model CMOS Buffer gate behaviorally



## Library

Logic

## Description

The CMOS Buffer block represents a CMOS Buffer logic gate behaviorally:

- The block output logic level is **HIGH** if the logic level of the gate input is 1.
- The block output logic level is **LOW** otherwise.

The block determines the logic levels of the gate inputs as follows:

- If the gate voltage is greater than the threshold voltage, the block interprets the input as logic 1.
- Otherwise, the block interprets the input as logic 0.

The threshold voltage is the voltage value at midpoint between the **High level input voltage** parameter value and the **Low level input voltage** parameter value.

---

**Note** To improve simulation speed, the block does not model all the internal individual MOSFET devices that make up the gate. See “Basic Assumptions and Limitations” on page 1-302 for details.

---

The block models the gate as follows:

- The gate inputs have infinite resistance and finite or zero capacitance.
- The gate output offers a selection of two models: **Linear** and **Quadratic**. For more information, see “Selecting the Output Model for Logic Blocks”. Use the **Output current-voltage relationship** parameter to specify the output model.
- You can specify propagation delay for both output models. For **Linear** output, the block sets the value of the gate output capacitor such that the resistor-capacitor time constant equals the **Propagation delay** parameter value. For **Quadratic** output, the gate input demand is lagged to approximate the **Propagation delay** parameter value.

The block output voltage depends on the output model selected:

- For **Linear** model, output high is the **High level output voltage** parameter value, and output low is the **Low level output voltage** parameter value.
- For **Quadratic** model, the output voltage for High and Low states is a function of the output current, as explained in “Quadratic Model Output and Parameters”. For zero load current, output high is  $V_{cc}$  (the **Supply voltage** parameter value), and output low is zero volts.

## Basic Assumptions and Limitations

The block does not model the internal individual MOSFET devices that make up the gate (except for the final MOSFET pair if you select the **Quadratic** option for the **Output current-voltage relationship** parameter). This limitation has the following implications:

- The block does not accurately model the gate's response to input noise and inputs that are around the logic threshold voltage.
- The block does not accurately model dynamic response.

Circuits that involve a feedback path around a set of logic gates may require a nonzero propagation delay to be set on one or more gates.

## Parameters

- “Inputs Tab” on page 1-303
- “Outputs Tab” on page 1-303
- “Initial Conditions Tab” on page 1-305

## Inputs Tab

### Low level input voltage

Voltage value below which the block interprets the input voltage as logic LOW. The default value is 2 V.

### High level input voltage

Voltage value above which the block interprets the input voltage as logic HIGH. The default value is 3 V.

### Average input capacitance

Fixed capacitance that approximates the input capacitance for a MOSFET gate. The MOSFET capacitance depends on the applied voltage. When you drive this block with another gate, the **Average input capacitance** produces a rise time similar to that of the MOSFET. You can usually find this capacitance value on a manufacturer datasheet. The default value is 5 pF. Setting this value to zero may result in faster simulation times.

## Outputs Tab

### Output current-voltage relationship

Select the output model, **Linear** or **Quadratic**. The default value is **Linear**.

### Low level output voltage

Voltage value at the output when the output logic level is LOW. The default value is 0 V. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter.

### High level output voltage

Voltage value at the output when the output logic level is HIGH. The default value is 5 V. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter.

### Output resistance

Value of the series output resistor that is used to model the drop in output voltage resulting from the output current. The default value is 25  $\Omega$ . You can derive this value from a datasheet by dividing the high-level output voltage by the maximum low-level output current. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter.

### **Supply voltage**

Supply voltage value applied to the gate in your circuit. The default value is 5 V. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### **Measurement voltage**

The gate supply voltage for which mask data output resistances and currents are defined. The default value is 5 V. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### **Logic HIGH output resistance at zero current and at I<sub>OH</sub>**

A row vector [  $R_{OH1}$   $R_{OH2}$  ] of two resistance values. The first value  $R_{OH1}$  is the gradient of the output voltage-current relationship when the gate is logic HIGH and there is no output current. The second value  $R_{OH2}$  is the gradient of the output voltage-current relationship when the gate is logic HIGH and the output current is  $I_{OH}$ . The default value is [ 25 250 ]  $\Omega$ . This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### **Logic HIGH output current I<sub>OH</sub> when shorted to ground**

The resulting current when the gate is in the logic HIGH state, but the load forces the output voltage to zero. The default value is 63 mA. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### **Logic LOW output resistance at zero current and at I<sub>OL</sub>**

A row vector [  $R_{OL1}$   $R_{OL2}$  ] of two resistance values. The first value  $R_{OL1}$  is the gradient of the output voltage-current relationship when the gate is logic LOW and there is no output current. The second value  $R_{OL2}$  is the gradient of the output voltage-current relationship when the gate is logic LOW and the output current is  $I_{OL}$ . The default value is [ 30 800 ]  $\Omega$ . This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### **Logic LOW output current I<sub>OL</sub> when shorted to Vcc**

The resulting current when the gate is in the logic LOW state, but the load forces the output voltage to the supply voltage  $V_{cc}$ . The default value is -45 mA. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

**Propagation delay**

Time it takes for the output to swing from LOW to HIGH or HIGH to LOW after the input logic levels change. The default value is 25 ns.

**Protection diode on resistance**

The gradient of the voltage-current relationship for the protection diodes when forward biased. The default value is 5  $\Omega$ . This parameter is available when you select the Quadratic option for the **Output current-voltage relationship** parameter.

**Protection diode forward voltage**

The voltage above which the protection diode is turned on. The default value is 0.6 V. This parameter is available when you select the Quadratic option for the **Output current-voltage relationship** parameter.

**Initial Conditions Tab****Output initial state**

Specify whether the initial output state of the block is High or Low. This parameter is used for both linear and quadratic output states, provided that the **Propagation delay** parameter is greater than zero and the Solver Configuration block does not have the **Start simulation from steady state** option selected. The default value is Low.

**Ports**

The block has the following ports:

A

Electrical input port

J

Electrical output port

## CMOS NAND

Model CMOS NAND gate behaviorally



### Library

Logic

### Description

The CMOS NAND block represents a CMOS NAND logic gate behaviorally:

- The block output logic level is **HIGH** if the logic levels of both of the gate inputs are 0.
- The block output logic level is **LOW** otherwise.

The block determines the logic levels of the gate inputs as follows:

- If the gate voltage is greater than the threshold voltage, the block interprets the input as logic 1.
- Otherwise, the block interprets the input as logic 0.

The threshold voltage is the voltage value at midpoint between the **High level input voltage** parameter value and the **Low level input voltage** parameter value.

---

**Note** To improve simulation speed, the block does not model all the internal individual MOSFET devices that make up the gate. See “Basic Assumptions and Limitations” on page 1-307 for details.

---

The block models the gate as follows:

- The gate inputs have infinite resistance and finite or zero capacitance.



- The gate output offers a selection of two models: **Linear** and **Quadratic**. For more information, see “Selecting the Output Model for Logic Blocks”. Use the **Output current-voltage relationship** parameter to specify the output model.
- You can specify propagation delay for both output models. For **Linear** output, the block sets the value of the gate output capacitor such that the resistor-capacitor time constant equals the **Propagation delay** parameter value. For **Quadratic** output, the gate input demand is lagged to approximate the **Propagation delay** parameter value.

The block output voltage depends on the output model selected:

- For **Linear** model, output high is the **High level output voltage** parameter value, and output low is the **Low level output voltage** parameter value.
- For **Quadratic** model, the output voltage for High and Low states is a function of the output current, as explained in “Quadratic Model Output and Parameters”. For zero load current, output high is  $V_{cc}$  (the **Supply voltage** parameter value), and output low is zero volts.

## Basic Assumptions and Limitations

The block does not model the internal individual MOSFET devices that make up the gate (except for the final MOSFET pair if you select the **Quadratic** option for the **Output current-voltage relationship** parameter). This limitation has the following implications:

- The block does not accurately model the gate's response to input noise and inputs that are around the logic threshold voltage.
- The block does not accurately model dynamic response.

Circuits that involve a feedback path around a set of logic gates may require a nonzero propagation delay to be set on one or more gates.

## Parameters

- “Inputs Tab” on page 1-308
- “Outputs Tab” on page 1-308
- “Initial Conditions Tab” on page 1-310

## Inputs Tab

### Low level input voltage

Voltage value below which the block interprets the input voltage as logic LOW. The default value is 2 V.

### High level input voltage

Voltage value above which the block interprets the input voltage as logic HIGH. The default value is 3 V.

### Average input capacitance

Fixed capacitance that approximates the input capacitance for a MOSFET gate. The MOSFET capacitance depends on the applied voltage. When you drive this block with another gate, the **Average input capacitance** produces a rise time similar to that of the MOSFET. You can usually find this capacitance value on a manufacturer datasheet. The default value is 5 pF. Setting this value to zero may result in faster simulation times.

## Outputs Tab

### Output current-voltage relationship

Select the output model, **Linear** or **Quadratic**. The default value is **Linear**.

### Low level output voltage

Voltage value at the output when the output logic level is LOW. The default value is 0 V. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter.

### High level output voltage

Voltage value at the output when the output logic level is HIGH. The default value is 5 V. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter.

### Output resistance

Value of the series output resistor that is used to model the drop in output voltage resulting from the output current. The default value is 25  $\Omega$ . You can derive this value from a datasheet by dividing the high-level output voltage by the maximum low-level output current. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter.

### Supply voltage

Supply voltage value applied to the gate in your circuit. The default value is 5 V. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Measurement voltage

The gate supply voltage for which mask data output resistances and currents are defined. The default value is 5 V. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Logic HIGH output resistance at zero current and at $I_{OH}$

A row vector [  $R_{OH1}$   $R_{OH2}$  ] of two resistance values. The first value  $R_{OH1}$  is the gradient of the output voltage-current relationship when the gate is logic HIGH and there is no output current. The second value  $R_{OH2}$  is the gradient of the output voltage-current relationship when the gate is logic HIGH and the output current is  $I_{OH}$ . The default value is [ 25 250 ]  $\Omega$ . This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Logic HIGH output current $I_{OH}$ when shorted to ground

The resulting current when the gate is in the logic HIGH state, but the load forces the output voltage to zero. The default value is 63 mA. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Logic LOW output resistance at zero current and at $I_{OL}$

A row vector [  $R_{OL1}$   $R_{OL2}$  ] of two resistance values. The first value  $R_{OL1}$  is the gradient of the output voltage-current relationship when the gate is logic LOW and there is no output current. The second value  $R_{OL2}$  is the gradient of the output voltage-current relationship when the gate is logic LOW and the output current is  $I_{OL}$ . The default value is [ 30 800 ]  $\Omega$ . This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Logic LOW output current $I_{OL}$ when shorted to Vcc

The resulting current when the gate is in the logic LOW state, but the load forces the output voltage to the supply voltage Vcc. The default value is -45 mA. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### **Propagation delay**

Time it takes for the output to swing from LOW to HIGH or HIGH to LOW after the input logic levels change. The default value is 25 ns.

### **Protection diode on resistance**

The gradient of the voltage-current relationship for the protection diodes when forward biased. The default value is 5  $\Omega$ . This parameter is available when you select the Quadratic option for the **Output current-voltage relationship** parameter.

### **Protection diode forward voltage**

The voltage above which the protection diode is turned on. The default value is 0.6 V. This parameter is available when you select the Quadratic option for the **Output current-voltage relationship** parameter.

## **Initial Conditions Tab**

### **Output initial state**

Specify whether the initial output state of the block is High or Low. This parameter is used for both linear and quadratic output states, provided that the **Propagation delay** parameter is greater than zero and the Solver Configuration block does not have the **Start simulation from steady state** option selected. The default value is Low.

## **Ports**

The block has the following ports:

A

Electrical input port

B

Electrical input port

J

Electrical output port

# CMOS NOR

Model CMOS NOR gate behaviorally



## Library

Logic

## Description

The CMOS NOR block represents a CMOS NOR logic gate behaviorally:

- The block output logic level is **LOW** if the logic levels of any of the gate inputs are 1.
- The block output logic level is **HIGH** otherwise.

The block determines the logic levels of the gate inputs as follows:

- If the gate voltage is greater than the threshold voltage, the block interprets the input as logic 1.
- Otherwise, the block interprets the input as logic 0.

The threshold voltage is the voltage value at midpoint between the **High level input voltage** parameter value and the **Low level input voltage** parameter value.

---

**Note** To improve simulation speed, the block does not model all the internal individual MOSFET devices that make up the gate. See “Basic Assumptions and Limitations” on page 1-312 for details.

---

The block models the gate as follows:

- The gate inputs have infinite resistance and finite or zero capacitance.

- The gate output offers a selection of two models: **Linear** and **Quadratic**. For more information, see “Selecting the Output Model for Logic Blocks”. Use the **Output current-voltage relationship** parameter to specify the output model.
- You can specify propagation delay for both output models. For **Linear** output, the block sets the value of the gate output capacitor such that the resistor-capacitor time constant equals the **Propagation delay** parameter value. For **Quadratic** output, the gate input demand is lagged to approximate the **Propagation delay** parameter value.

The block output voltage depends on the output model selected:

- For **Linear** model, output high is the **High level output voltage** parameter value, and output low is the **Low level output voltage** parameter value.
- For **Quadratic** model, the output voltage for High and Low states is a function of the output current, as explained in “Quadratic Model Output and Parameters”. For zero load current, output high is  $V_{cc}$  (the **Supply voltage** parameter value), and output low is zero volts.

## Basic Assumptions and Limitations

The block does not model the internal individual MOSFET devices that make up the gate (except for the final MOSFET pair if you select the **Quadratic** option for the **Output current-voltage relationship** parameter). This limitation has the following implications:

- The block does not accurately model the gate's response to input noise and inputs that are around the logic threshold voltage.
- The block does not accurately model dynamic response.

Circuits that involve a feedback path around a set of logic gates may require a nonzero propagation delay to be set on one or more gates.

## Parameters

- “Inputs Tab” on page 1-313
- “Outputs Tab” on page 1-313
- “Initial Conditions Tab” on page 1-315

## Inputs Tab

### Low level input voltage

Voltage value below which the block interprets the input voltage as logic LOW. The default value is 2 V.

### High level input voltage

Voltage value above which the block interprets the input voltage as logic HIGH. The default value is 3 V.

### Average input capacitance

Fixed capacitance that approximates the input capacitance for a MOSFET gate. The MOSFET capacitance depends on the applied voltage. When you drive this block with another gate, the **Average input capacitance** produces a rise time similar to that of the MOSFET. You can usually find this capacitance value on a manufacturer datasheet. The default value is 5 pF. Setting this value to zero may result in faster simulation times.

## Outputs Tab

### Output current-voltage relationship

Select the output model, **Linear** or **Quadratic**. The default value is **Linear**.

### Low level output voltage

Voltage value at the output when the output logic level is LOW. The default value is 0 V. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter.

### High level output voltage

Voltage value at the output when the output logic level is HIGH. The default value is 5 V. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter.

### Output resistance

Value of the series output resistor that is used to model the drop in output voltage resulting from the output current. The default value is 25  $\Omega$ . You can derive this value from a datasheet by dividing the high-level output voltage by the maximum low-level output current. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter.

**Supply voltage**

Supply voltage value applied to the gate in your circuit. The default value is 5 V. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

**Measurement voltage**

The gate supply voltage for which mask data output resistances and currents are defined. The default value is 5 V. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

**Logic HIGH output resistance at zero current and at I<sub>OH</sub>**

A row vector [  $R_{OH1}$   $R_{OH2}$  ] of two resistance values. The first value  $R_{OH1}$  is the gradient of the output voltage-current relationship when the gate is logic HIGH and there is no output current. The second value  $R_{OH2}$  is the gradient of the output voltage-current relationship when the gate is logic HIGH and the output current is  $I_{OH}$ . The default value is [ 25 250 ]  $\Omega$ . This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

**Logic HIGH output current I<sub>OH</sub> when shorted to ground**

The resulting current when the gate is in the logic HIGH state, but the load forces the output voltage to zero. The default value is 63 mA. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

**Logic LOW output resistance at zero current and at I<sub>OL</sub>**

A row vector [  $R_{OL1}$   $R_{OL2}$  ] of two resistance values. The first value  $R_{OL1}$  is the gradient of the output voltage-current relationship when the gate is logic LOW and there is no output current. The second value  $R_{OL2}$  is the gradient of the output voltage-current relationship when the gate is logic LOW and the output current is  $I_{OL}$ . The default value is [ 30 800 ]  $\Omega$ . This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

**Logic LOW output current I<sub>OL</sub> when shorted to Vcc**

The resulting current when the gate is in the logic LOW state, but the load forces the output voltage to the supply voltage  $V_{cc}$ . The default value is -45 mA. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.



**Propagation delay**

Time it takes for the output to swing from LOW to HIGH or HIGH to LOW after the input logic levels change. The default value is 25 ns.

**Protection diode on resistance**

The gradient of the voltage-current relationship for the protection diodes when forward biased. The default value is 5  $\Omega$ . This parameter is available when you select the Quadratic option for the **Output current-voltage relationship** parameter.

**Protection diode forward voltage**

The voltage above which the protection diode is turned on. The default value is 0.6 V. This parameter is available when you select the Quadratic option for the **Output current-voltage relationship** parameter.

**Initial Conditions Tab****Output initial state**

Specify whether the initial output state of the block is High or Low. This parameter is used for both linear and quadratic output states, provided that the **Propagation delay** parameter is greater than zero and the Solver Configuration block does not have the **Start simulation from steady state** option selected. The default value is Low.

**Ports**

The block has the following ports:

A

Electrical input port

B

Electrical input port

J

Electrical output port

## CMOS NOT

Model CMOS NOT gate behaviorally



### Library

Logic

### Description

The CMOS NOT block represents a CMOS NOT logic gate behaviorally:

- The block output logic level is **HIGH** if the logic level of the gate input is 0.
- The block output logic level is **LOW** otherwise.

The block determines the logic levels of the gate inputs as follows:

- If the gate voltage is greater than the threshold voltage, the block interprets the input as logic 1.
- Otherwise, the block interprets the input as logic 0.

The threshold voltage is the voltage value at midpoint between the **High level input voltage** parameter value and the **Low level input voltage** parameter value.

---

**Note** To improve simulation speed, the block does not model all the internal individual MOSFET devices that make up the gate. See “Basic Assumptions and Limitations” on page 1-317 for details.

---

The block models the gate as follows:

- The gate inputs have infinite resistance and finite or zero capacitance.
- The gate output offers a selection of two models: **Linear** and **Quadratic**. For more information, see “Selecting the Output Model for Logic Blocks”. Use the **Output current-voltage relationship** parameter to specify the output model.
- You can specify propagation delay for both output models. For **Linear** output, the block sets the value of the gate output capacitor such that the resistor-capacitor time constant equals the **Propagation delay** parameter value. For **Quadratic** output, the gate input demand is lagged to approximate the **Propagation delay** parameter value.

The block output voltage depends on the output model selected:

- For **Linear** model, output high is the **High level output voltage** parameter value, and output low is the **Low level output voltage** parameter value.
- For **Quadratic** model, the output voltage for High and Low states is a function of the output current, as explained in “Quadratic Model Output and Parameters”. For zero load current, output high is  $V_{cc}$  (the **Supply voltage** parameter value), and output low is zero volts.

## Basic Assumptions and Limitations

The block does not model the internal individual MOSFET devices that make up the gate (except for the final MOSFET pair if you select the **Quadratic** option for the **Output current-voltage relationship** parameter). This limitation has the following implications:

- The block does not accurately model the gate's response to input noise and inputs that are around the logic threshold voltage.
- The block does not accurately model dynamic response.

Circuits that involve a feedback path around a set of logic gates may require a nonzero propagation delay to be set on one or more gates.

## Parameters

- “Inputs Tab” on page 1-318
- “Outputs Tab” on page 1-318
- “Initial Conditions Tab” on page 1-320

## Inputs Tab

### Low level input voltage

Voltage value below which the block interprets the input voltage as logic LOW. The default value is 2 V.

### High level input voltage

Voltage value above which the block interprets the input voltage as logic HIGH. The default value is 3 V.

### Average input capacitance

Fixed capacitance that approximates the input capacitance for a MOSFET gate. The MOSFET capacitance depends on the applied voltage. When you drive this block with another gate, the **Average input capacitance** produces a rise time similar to that of the MOSFET. You can usually find this capacitance value on a manufacturer datasheet. The default value is 5 pF. Setting this value to zero may result in faster simulation times.

## Outputs Tab

### Output current-voltage relationship

Select the output model, **Linear** or **Quadratic**. The default value is **Linear**.

### Low level output voltage

Voltage value at the output when the output logic level is LOW. The default value is 0 V. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter.

### High level output voltage

Voltage value at the output when the output logic level is HIGH. The default value is 5 V. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter.

### Output resistance

Value of the series output resistor that is used to model the drop in output voltage resulting from the output current. The default value is 25  $\Omega$ . You can derive this value from a datasheet by dividing the high-level output voltage by the maximum low-level output current. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter.

### Supply voltage

Supply voltage value applied to the gate in your circuit. The default value is 5 V. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Measurement voltage

The gate supply voltage for which mask data output resistances and currents are defined. The default value is 5 V. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Logic HIGH output resistance at zero current and at I<sub>OH</sub>

A row vector [  $R_{OH1}$   $R_{OH2}$  ] of two resistance values. The first value  $R_{OH1}$  is the gradient of the output voltage-current relationship when the gate is logic HIGH and there is no output current. The second value  $R_{OH2}$  is the gradient of the output voltage-current relationship when the gate is logic HIGH and the output current is  $I_{OH}$ . The default value is [ 25 250 ]  $\Omega$ . This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Logic HIGH output current I<sub>OH</sub> when shorted to ground

The resulting current when the gate is in the logic HIGH state, but the load forces the output voltage to zero. The default value is 63 mA. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Logic LOW output resistance at zero current and at I<sub>OL</sub>

A row vector [  $R_{OL1}$   $R_{OL2}$  ] of two resistance values. The first value  $R_{OL1}$  is the gradient of the output voltage-current relationship when the gate is logic LOW and there is no output current. The second value  $R_{OL2}$  is the gradient of the output voltage-current relationship when the gate is logic LOW and the output current is  $I_{OL}$ . The default value is [ 30 800 ]  $\Omega$ . This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Logic LOW output current I<sub>OL</sub> when shorted to Vcc

The resulting current when the gate is in the logic LOW state, but the load forces the output voltage to the supply voltage  $V_{cc}$ . The default value is -45 mA. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### **Propagation delay**

Time it takes for the output to swing from LOW to HIGH or HIGH to LOW after the input logic levels change. The default value is 25 ns.

### **Protection diode on resistance**

The gradient of the voltage-current relationship for the protection diodes when forward biased. The default value is 5  $\Omega$ . This parameter is available when you select the Quadratic option for the **Output current-voltage relationship** parameter.

### **Protection diode forward voltage**

The voltage above which the protection diode is turned on. The default value is 0.6 V. This parameter is available when you select the Quadratic option for the **Output current-voltage relationship** parameter.

## **Initial Conditions Tab**

### **Output initial state**

Specify whether the initial output state of the block is High or Low. This parameter is used for both linear and quadratic output states, provided that the **Propagation delay** parameter is greater than zero and the Solver Configuration block does not have the **Start simulation from steady state** option selected. The default value is Low.

## **Ports**

The block has the following ports:

A

Electrical input port

J

Electrical output port

# CMOS OR

Model CMOS OR gate behaviorally



## Library

Logic

## Description

The CMOS OR block represents a CMOS OR logic gate behaviorally:

- The block output logic level is **HIGH** if the logic levels of any of the gate inputs are 1.
- The block output logic level is **LOW** otherwise.

The block determines the logic levels of the gate inputs as follows:

- If the gate voltage is greater than the threshold voltage, the block interprets the input as logic 1.
- Otherwise, the block interprets the input as logic 0.

The threshold voltage is the voltage value at midpoint between the **High level input voltage** parameter value and the **Low level input voltage** parameter value.

---

**Note** To improve simulation speed, the block does not model all the internal individual MOSFET devices that make up the gate. See “Basic Assumptions and Limitations” on page 1-322 for details.

---

The block models the gate as follows:

- The gate inputs have infinite resistance and finite or zero capacitance.

- The gate output offers a selection of two models: **Linear** and **Quadratic**. For more information, see “Selecting the Output Model for Logic Blocks”. Use the **Output current-voltage relationship** parameter to specify the output model.
- You can specify propagation delay for both output models. For **Linear** output, the block sets the value of the gate output capacitor such that the resistor-capacitor time constant equals the **Propagation delay** parameter value. For **Quadratic** output, the gate input demand is lagged to approximate the **Propagation delay** parameter value.

The block output voltage depends on the output model selected:

- For **Linear** model, output high is the **High level output voltage** parameter value, and output low is the **Low level output voltage** parameter value.
- For **Quadratic** model, the output voltage for High and Low states is a function of the output current, as explained in “Quadratic Model Output and Parameters”. For zero load current, output high is  $V_{cc}$  (the **Supply voltage** parameter value), and output low is zero volts.

## Basic Assumptions and Limitations

The block does not model the internal individual MOSFET devices that make up the gate (except for the final MOSFET pair if you select the **Quadratic** option for the **Output current-voltage relationship** parameter). This limitation has the following implications:

- The block does not accurately model the gate's response to input noise and inputs that are around the logic threshold voltage.
- The block does not accurately model dynamic response.

Circuits that involve a feedback path around a set of logic gates may require a nonzero propagation delay to be set on one or more gates.

## Parameters

- “Inputs Tab” on page 1-323
- “Outputs Tab” on page 1-323
- “Initial Conditions Tab” on page 1-325



## Inputs Tab

### Low level input voltage

Voltage value below which the block interprets the input voltage as logic LOW. The default value is 2 V.

### High level input voltage

Voltage value above which the block interprets the input voltage as logic HIGH. The default value is 3 V.

### Average input capacitance

Fixed capacitance that approximates the input capacitance for a MOSFET gate. The MOSFET capacitance depends on the applied voltage. When you drive this block with another gate, the **Average input capacitance** produces a rise time similar to that of the MOSFET. You can usually find this capacitance value on a manufacturer datasheet. The default value is 5 pF. Setting this value to zero may result in faster simulation times.

## Outputs Tab

### Output current-voltage relationship

Select the output model, **Linear** or **Quadratic**. The default value is **Linear**.

### Low level output voltage

Voltage value at the output when the output logic level is LOW. The default value is 0 V. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter.

### High level output voltage

Voltage value at the output when the output logic level is HIGH. The default value is 5 V. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter.

### Output resistance

Value of the series output resistor that is used to model the drop in output voltage resulting from the output current. The default value is 25  $\Omega$ . You can derive this value from a datasheet by dividing the high-level output voltage by the maximum low-level output current. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter.

### **Supply voltage**

Supply voltage value applied to the gate in your circuit. The default value is 5 V. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### **Measurement voltage**

The gate supply voltage for which mask data output resistances and currents are defined. The default value is 5 V. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### **Logic HIGH output resistance at zero current and at I<sub>OH</sub>**

A row vector [  $R_{OH1}$   $R_{OH2}$  ] of two resistance values. The first value  $R_{OH1}$  is the gradient of the output voltage-current relationship when the gate is logic HIGH and there is no output current. The second value  $R_{OH2}$  is the gradient of the output voltage-current relationship when the gate is logic HIGH and the output current is  $I_{OH}$ . The default value is [ 25 250 ]  $\Omega$ . This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### **Logic HIGH output current I<sub>OH</sub> when shorted to ground**

The resulting current when the gate is in the logic HIGH state, but the load forces the output voltage to zero. The default value is 63 mA. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### **Logic LOW output resistance at zero current and at I<sub>OL</sub>**

A row vector [  $R_{OL1}$   $R_{OL2}$  ] of two resistance values. The first value  $R_{OL1}$  is the gradient of the output voltage-current relationship when the gate is logic LOW and there is no output current. The second value  $R_{OL2}$  is the gradient of the output voltage-current relationship when the gate is logic LOW and the output current is  $I_{OL}$ . The default value is [ 30 800 ]  $\Omega$ . This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### **Logic LOW output current I<sub>OL</sub> when shorted to Vcc**

The resulting current when the gate is in the logic LOW state, but the load forces the output voltage to the supply voltage  $V_{cc}$ . The default value is -45 mA. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

**Propagation delay**

Time it takes for the output to swing from LOW to HIGH or HIGH to LOW after the input logic levels change. The default value is 25 ns.

**Protection diode on resistance**

The gradient of the voltage-current relationship for the protection diodes when forward biased. The default value is 5  $\Omega$ . This parameter is available when you select the Quadratic option for the **Output current-voltage relationship** parameter.

**Protection diode forward voltage**

The voltage above which the protection diode is turned on. The default value is 0.6 V. This parameter is available when you select the Quadratic option for the **Output current-voltage relationship** parameter.

**Initial Conditions Tab****Output initial state**

Specify whether the initial output state of the block is High or Low. This parameter is used for both linear and quadratic output states, provided that the **Propagation delay** parameter is greater than zero and the Solver Configuration block does not have the **Start simulation from steady state** option selected. The default value is Low.

**Ports**

The block has the following ports:

A

Electrical input port

B

Electrical input port

J

Electrical output port

## CMOS XOR

Model CMOS XOR gate behaviorally



## Library

Logic

## Description

The CMOS XOR block represents a CMOS XOR logic gate behaviorally:

- The block output logic level is **HIGH** if the logic level of exactly one of the gate inputs is 1.
- The block output logic level is **LOW** otherwise.

The block determines the logic levels of the gate inputs as follows:

- If the gate voltage is greater than the threshold voltage, the block interprets the input as logic 1.
- Otherwise, the block interprets the input as logic 0.

The threshold voltage is the voltage value at midpoint between the **High level input voltage** parameter value and the **Low level input voltage** parameter value.

---

**Note** To improve simulation speed, the block does not model all the internal individual MOSFET devices that make up the gate. See “Basic Assumptions and Limitations” on page 1-327 for details.

---

The block models the gate as follows:

- The gate inputs have infinite resistance and finite or zero capacitance.
- The gate output offers a selection of two models: **Linear** and **Quadratic**. For more information, see “Selecting the Output Model for Logic Blocks”. Use the **Output current-voltage relationship** parameter to specify the output model.
- You can specify propagation delay for both output models. For **Linear** output, the block sets the value of the gate output capacitor such that the resistor-capacitor time constant equals the **Propagation delay** parameter value. For **Quadratic** output, the gate input demand is lagged to approximate the **Propagation delay** parameter value.

The block output voltage depends on the output model selected:

- For **Linear** model, output high is the **High level output voltage** parameter value, and output low is the **Low level output voltage** parameter value.
- For **Quadratic** model, the output voltage for High and Low states is a function of the output current, as explained in “Quadratic Model Output and Parameters”. For zero load current, output high is  $V_{cc}$  (the **Supply voltage** parameter value), and output low is zero volts.

## Basic Assumptions and Limitations

The block does not model the internal individual MOSFET devices that make up the gate (except for the final MOSFET pair if you select the **Quadratic** option for the **Output current-voltage relationship** parameter). This limitation has the following implications:

- The block does not accurately model the gate's response to input noise and inputs that are around the logic threshold voltage.
- The block does not accurately model dynamic response.

Circuits that involve a feedback path around a set of logic gates may require a nonzero propagation delay to be set on one or more gates.

## Parameters

- “Inputs Tab” on page 1-328
- “Outputs Tab” on page 1-328
- “Initial Conditions Tab” on page 1-330

## Inputs Tab

### Low level input voltage

Voltage value below which the block interprets the input voltage as logic LOW. The default value is 2 V.

### High level input voltage

Voltage value above which the block interprets the input voltage as logic HIGH. The default value is 3 V.

### Average input capacitance

Fixed capacitance that approximates the input capacitance for a MOSFET gate. The MOSFET capacitance depends on the applied voltage. When you drive this block with another gate, the **Average input capacitance** produces a rise time similar to that of the MOSFET. You can usually find this capacitance value on a manufacturer datasheet. The default value is 5 pF. Setting this value to zero may result in faster simulation times.

## Outputs Tab

### Output current-voltage relationship

Select the output model, **Linear** or **Quadratic**. The default value is **Linear**.

### Low level output voltage

Voltage value at the output when the output logic level is LOW. The default value is 0 V. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter.

### High level output voltage

Voltage value at the output when the output logic level is HIGH. The default value is 5 V. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter.

### Output resistance

Value of the series output resistor that is used to model the drop in output voltage resulting from the output current. The default value is 25  $\Omega$ . You can derive this value from a datasheet by dividing the high-level output voltage by the maximum low-level output current. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter.

### Supply voltage

Supply voltage value applied to the gate in your circuit. The default value is 5 V. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Measurement voltage

The gate supply voltage for which mask data output resistances and currents are defined. The default value is 5 V. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Logic HIGH output resistance at zero current and at $I_{OH}$

A row vector [  $R_{OH1}$   $R_{OH2}$  ] of two resistance values. The first value  $R_{OH1}$  is the gradient of the output voltage-current relationship when the gate is logic HIGH and there is no output current. The second value  $R_{OH2}$  is the gradient of the output voltage-current relationship when the gate is logic HIGH and the output current is  $I_{OH}$ . The default value is [ 25 250 ]  $\Omega$ . This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Logic HIGH output current $I_{OH}$ when shorted to ground

The resulting current when the gate is in the logic HIGH state, but the load forces the output voltage to zero. The default value is 63 mA. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Logic LOW output resistance at zero current and at $I_{OL}$

A row vector [  $R_{OL1}$   $R_{OL2}$  ] of two resistance values. The first value  $R_{OL1}$  is the gradient of the output voltage-current relationship when the gate is logic LOW and there is no output current. The second value  $R_{OL2}$  is the gradient of the output voltage-current relationship when the gate is logic LOW and the output current is  $I_{OL}$ . The default value is [ 30 800 ]  $\Omega$ . This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Logic LOW output current $I_{OL}$ when shorted to $V_{cc}$

The resulting current when the gate is in the logic LOW state, but the load forces the output voltage to the supply voltage  $V_{cc}$ . The default value is -45 mA. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### **Propagation delay**

Time it takes for the output to swing from LOW to HIGH or HIGH to LOW after the input logic levels change. The default value is 25 ns.

### **Protection diode on resistance**

The gradient of the voltage-current relationship for the protection diodes when forward biased. The default value is 5  $\Omega$ . This parameter is available when you select the Quadratic option for the **Output current-voltage relationship** parameter.

### **Protection diode forward voltage**

The voltage above which the protection diode is turned on. The default value is 0.6 V. This parameter is available when you select the Quadratic option for the **Output current-voltage relationship** parameter.

## **Initial Conditions Tab**

### **Output initial state**

Specify whether the initial output state of the block is High or Low. This parameter is used for both linear and quadratic output states, provided that the **Propagation delay** parameter is greater than zero and the Solver Configuration block does not have the **Start simulation from steady state** option selected. The default value is Low.

## **Ports**

The block has the following ports:

A

Electrical input port

B

Electrical input port

J

Electrical output port



# Comparator

Model a comparator behaviorally



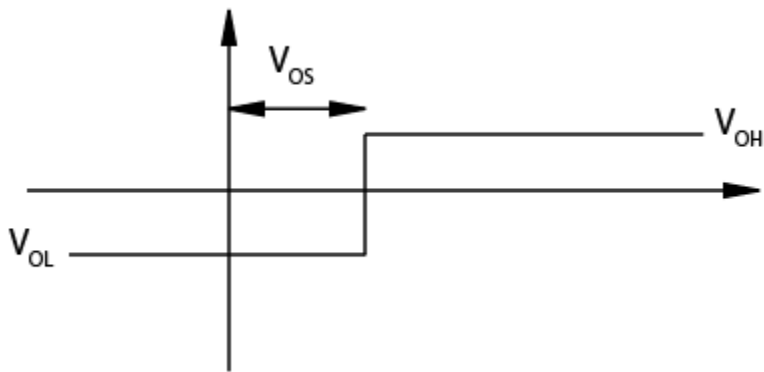
## Library

Integrated Circuits

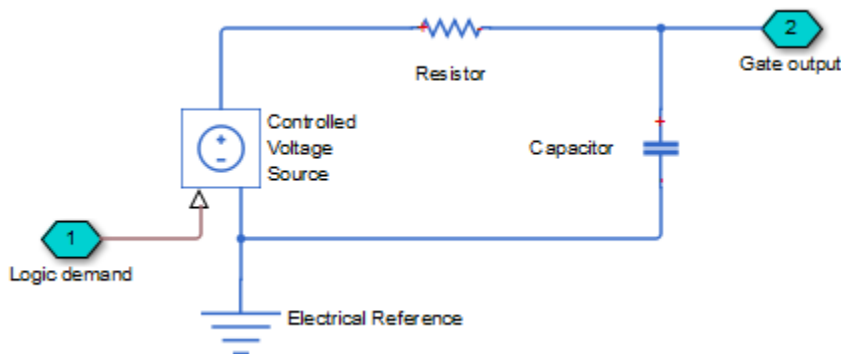
## Description

The Comparator block is an abstracted behavioral model of a comparator integrated circuit. It does not model an internal transistor-level implementation. Therefore, the block runs quickly during simulation but retains the correct I/O behavior. The block models differential inputs electrically as having infinite resistance and a finite or zero capacitance.

The block models the gate output as a voltage source driving a series resistor and a capacitor that connects to ground. The output pin connects to the resistor-capacitor connection node. If the difference in the inputs is greater than the input threshold voltage, then the output is equal to the **High level output voltage** ( $V_{OL}$ ). Otherwise, the output is equal to the **Low level output voltage** ( $V_{OH}$ ).



The output model is shown in the following illustration.



## Basic Assumptions and Limitations

Modeling of the output as a controlled voltage source is representative of a totem-pole or push-pull output stage. To model a device with an open-collector:

- 1 Connect the output pin to the base of an NPN Bipolar Transistor or PNP Bipolar Transistor block.
- 2 Set the **Output resistance** parameter to a suitable value.

## Parameters

- “Inputs Tab” on page 1-333
- “Outputs Tab” on page 1-333

### Inputs Tab

#### Input offset voltage

The voltage which the difference in the input voltages must be greater than so that the comparator gives a logic output 1. The default value is 5 mV.

#### Average input capacitance

You can usually find this capacitance value on a manufacturer datasheet. The default value is 0 pF. Setting this value to zero can result in faster simulation times.

### Outputs Tab

#### Low level output voltage

The steady-state output voltage,  $V_{OL}$ , when the voltage difference across the inputs is less than or equal to the threshold voltage, and the output current is zero. The default value is 0 V.

#### High level output voltage

The steady-state output voltage,  $V_{OH}$ , when the voltage difference across the inputs is greater than the threshold voltage, and the output current is zero. The default value is 5 V.

#### Output resistance

This parameter is the ratio of output voltage drop to output current. Set this parameter to  $(V_{OH} - V_{OH1}) / I_{OH1}$ , where  $V_{OH1}$  is the reduced output high voltage when the output current is  $I_{OH1}$ . The default value is 50  $\Omega$ .

#### Propagation delay

Set this value based on the high-to-low and low-to-high propagation delays. The default value is 0 s.

## Ports

This block has the following ports:

+

Positive electrical input port

-

Negative electrical input port

OUT

Electrical output port

## See Also

CMOS Buffer

# Commutation Diode

Piecewise linear diode with charge dynamics and junction capacitance



## Library

Simscape / Electrical / Power Systems / Semiconductors / Fundamental Components

## Description

The Commutation Diode block augments the Diode block with a model of charge dynamics. For a description of the piecewise linear diode operation that the Commutation Diode block uses, see Diode.

Use the Commutation Diode block in place of the Diode block when you want to specify precisely the charge dynamics of the device as it operates in reverse mode. For example, suppose that your model uses the diode to divert inductive currents from a motor drive or inverter. In this case, precise reverse-mode operation is important and an appropriate time to use the Commutation Diode block.

The Commutation Diode uses a charge model proposed by Lauritzen and Ma [1]. The defining expressions for this charge model are:

$$i = \frac{q_E - q_M}{T_M} \quad (1-1)$$

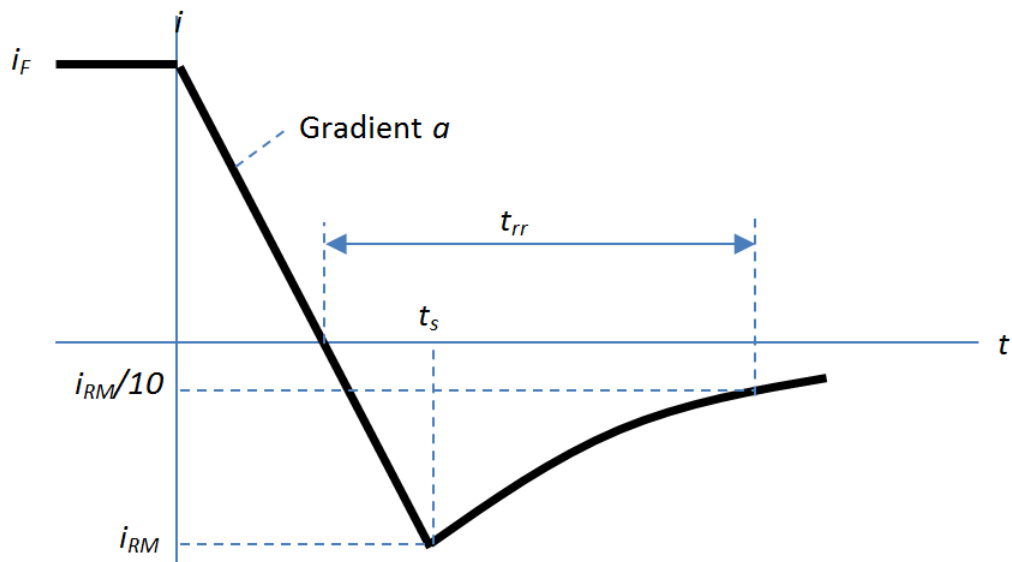
$$\frac{dq_M}{dt} + \frac{q_M}{\tau} - \frac{q_E - q_M}{T_M} = 0 \quad (1-2)$$

$$\begin{aligned} q_E &= (\tau + T_M)(v_D - v_F(1 - RG)) / R & \text{if } v_D > v_F \\ q_E &= (\tau + T_M)Gv_D & \text{if } v_D \leq v_F \end{aligned} \quad (1-3)$$

where:

- $i$  is the diode current.
- $q_E$  is the junction charge.
- $q_M$  is the total stored charge.
- $T_M$  is the transit time.
- $\tau$  is the carrier lifetime.
- $v_D$  is the voltage across the diode.
- $v_F$  is the diode forward voltage.
- $R$  is the diode on resistance.
- $G$  is the diode off conductance.

This graphic shows a typical reverse-mode current characteristic for a diode device.



where:

- $i_{RM}$  is the peak reverse current.
- $i_F$  is the starting forward current when measuring  $i_{RM}$ .

- $a$  is the rate of change of current when measuring  $i_{RM}$ .
- $t_{rr}$  is the reverse recovery time.

On the Charge Dynamics tab of the block, you specify characteristics of your diode device. The block uses these values to calculate the diode charge dynamics expressed in equations 1-1, 1-2, and 1-3.

Data sheets for diodes quote values for peak reverse current for an initial forward current and a steady rate of change of current. The data sheet might also provide values for reverse recovery time and total recovery charge.

## How the Block Calculates $T_M$ and Tau

The block calculates transit time  $T_M$  and carrier lifetime  $\tau$  based on the values you enter on the Charge Dynamics tab of the block dialog box. The block uses  $T_M$  and  $\tau$  to solve the charge dynamics equations 1-1, 1-2, and 1-3.

During initial current drop in reverse mode, the diode is still on, and the rate of change of current is determined by an external test circuit.

Using Equation 1-1,

$$i_F + at = \frac{q_E - q_M}{T_M}. \quad (1-4)$$

Substituting Equation 1-4 into Equation 1-2,

$$\frac{dq_M}{dt} + \frac{q_M}{\tau} = i_F + at. \quad (1-5)$$

Solving Equation 1-5 for  $q_M$ ,

$$q_M = i_F \tau - a\tau^2 + \frac{k}{\exp\left(\frac{t}{\tau}\right)} + at, \quad (1-6)$$

where  $k$  is a constant.

When  $t$  is zero,  $i = i_F$  and  $q_M = \tau i_F$  because the system is in steady state.

Substituting these relationships into Equation 1-6 and solving the equation gives  $k = a\tau^2$ .

Therefore,

$$q_M = i_F \tau + a\tau^2 \left( \frac{1}{\exp\left(\frac{t}{\tau}\right)} - 1 \right) + a\tau t. \quad (1-7)$$

At time  $t = t_s$ , the current is  $i_{RM}$  and the junction charge  $q_E$  is zero.

Substituting these values into Equation 1-1,

$$i_{RM} = \frac{-q_M}{T_M}. \quad (1-8)$$

Rearranging Equation 1-8 to solve for  $q_M$  and substituting the result into Equation 1-7,

$$-T_M i_{RM} = i_F \tau + a\tau^2 \left( \frac{1}{\exp\left(\frac{t_s}{\tau}\right)} - 1 \right) + a\tau t_s. \quad (1-9)$$

Expressing time  $t_s$  in terms of  $i_{RM}$ ,  $i_F$ , and  $a$ ,

$$t_s = \frac{i_{RM} - i_F}{a}. \quad (1-10)$$

Consider the diode recovery, that is, when  $t > t_s$ . The diode is reverse biased, and current and junction charge are effectively zero.

The current is defined by

$$i = i_{RM} \exp\left[-\frac{(t - t_s)}{\tau_{rr}}\right], \quad (1-11)$$

where



$$\frac{1}{\tau_{rr}} = \frac{1}{\tau} + \frac{1}{T_M}. \quad (1-12)$$

The block now relates the expression in Equation 1-12 to the reverse recovery time  $t_{rr}$ .

When  $t = \frac{i_{RM}}{a} + t_{rr}$ , the current is  $\frac{i_{RM}}{10}$ .

Therefore,

$$\exp\left(-\frac{t-t_s}{\tau_{rr}}\right) = 0.1 \quad (1-13)$$

and

$$t_{rr} = \tau_{rr} \log(10) + \frac{i_{RM}}{a}. \quad (1-14)$$

The block uses equations 1-9 and 1-14 to calculate values for  $T_M$  and  $\tau$ . The calculation uses an iterative scheme because of the exponential term in Equation 1-9.

## Alternatives to Specifying $t_{rr}$ Directly

In addition to allowing you to specify reverse recovery time  $t_{rr}$  directly, the block supports two alternative parameterizations. The block can derive  $t_{rr}$  from either of these parameters:

- Reverse recovery time stretch factor  $\lambda$
- Reverse recovery charge  $Q_{rr}$ , when the data sheet specifies this value instead of the reverse recovery time.

The relationship between reverse recovery time stretch factor  $\lambda$  and  $t_{rr}$  is expressed by the equation

$$\lambda = \frac{t_{rr} a}{i_{RM}}.$$

Reverse recovery time must be greater than  $\frac{i_{RM}}{a}$  and a typical value is  $3\left(\frac{i_{RM}}{a}\right)$ .

Therefore, a typical value for  $\lambda$  is 3.  $\lambda$  must be greater than 1.

Reverse recovery charge  $Q_{rr}$  is the integral over time of the reverse current from the point where the current goes negative until it decays back to zero.

The initial charge, to time  $t_s$  (as shown in the figure), is expressed by the equation

$$Q_s = \frac{1}{2}(-i_{RM})\frac{i_{RM}}{a}. \quad (1-15)$$

Integrating Equation 1-11 gives the charge between times  $t_s$  and  $\infty$ . This charge is equal to

$$\tau_{rr}i_{RM}.$$

Therefore, total reverse recovery charge is given by the equation

$$Q_{rr} = -\frac{i_{RM}^2}{2a} + \tau_{rr}i_{RM}. \quad (1-16)$$

Rearranging Equation 1-16 to solve for  $\tau_{rr}$  and substituting the result into Equation 1-14 gives an equation that expresses  $t_{rr}$  in terms of  $Q_{rr}$ :

$$t_{rr} = \left( \frac{Q_{rr}}{i_{RM}} + \frac{i_{RM}}{2a} \right) \log(10) + \frac{i_{RM}}{a}.$$

## Modeling Variants

The block provides a thermal modeling variant. To select a variant, right-click the block in your model. From the context menu, select **Simscape > Block choices**, and then one of these variants:

- **No thermal port** — This variant does not simulate heat generation in the device. This variant is the default.

- **Show thermal port** — This variant contains a thermal port that allows you to model the heat that conduction losses generate. For numerical efficiency, the thermal state does not affect the electrical behavior of the block. The thermal port is hidden by default. When you select a thermal variant of the block, the thermal port appears.

## Ports

+

Electrical conserving port associated with the diode positive terminal

-

Electrical conserving port associated with the diode negative terminal

H

Thermal conserving port. The thermal port is optional and is hidden by default. To enable this port, select a variant that includes a thermal port.

## Parameters

- “Main Tab” on page 1-341
- “Charge Dynamics Tab” on page 1-342

## Main Tab

### Forward voltage

Minimum voltage required across the + and - block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**. The default value is 0.8 V.

### On resistance

Rate of change of voltage versus current above the **Forward voltage**. The default value is 0.001 Ohm.

### Off conductance

Conductance of the reverse-biased diode. The default value is  $1e-5$  1/Ohm.

### **Number of series diodes**

The number of diodes connected in series between the + and - block ports. Each diode has the **Forward voltage**, **On resistance**, and **Off conductance** that you specify. The default value is 1.

### **Number of parallel diodes**

The number of parallel diodes, or number of parallel paths formed by series-connected diodes, between the + and - block ports. Each diode has the **Forward voltage**, **On resistance**, and **Off conductance** that you specify. The default value is 1.

## **Charge Dynamics Tab**

### **Junction capacitance**

Diode junction capacitance. The default value is 50 nF.

### **Peak reverse current, iRM**

Peak reverse current measured by an external test circuit. This value must be less than zero. The default value is -235 A.

### **Initial forward current when measuring iRM**

Initial forward current when measuring peak reverse current. This value must be greater than zero. The default value is 300 A.

### **Rate of change of current when measuring iRM**

Rate of change of current when measuring peak reverse current. This value must be less than zero. The default value is -50 A/ $\mu$ s.

### **Reverse recovery time parameterization**

Determines how you specify reverse recovery time in the block. The default value is Specify reverse recovery time directly.

If you select Specify stretch factor or Specify reverse recovery charge, you specify a value that the block uses to derive the reverse recovery time. For more information on these options, see “Alternatives to Specifying trr Directly” on page 1-339.

### **Reverse recovery time, trr**

Interval between the time when the current initially goes to zero (when the diode turns off) and the time when the current falls to less than 10% of the peak reverse current. The default value is 15  $\mu$ s.

This parameter is visible only if you set **Reverse recovery time parameterization** to Specify reverse recovery time directly.

The value of the **Reverse recovery time, trr** parameter must be greater than the value of the **Peak reverse current, iRM** parameter divided by the value of the **Rate of change of current when measuring iRM** parameter.

### Reverse recovery time stretch factor

Value that the block uses to calculate **Reverse recovery time, trr**. This value must be greater than 1. The default value is 3.

This parameter is visible only if you set **Reverse recovery time parameterization** to Specify stretch factor.

Specifying the stretch factor is an easier way to parameterize the reverse recovery time than specifying the reverse recovery charge. The larger the value of the stretch factor, the longer it takes for the reverse recovery current to dissipate.

### Reverse recovery charge, Qrr

Value that the block uses to calculate **Reverse recovery time, trr**. Use this parameter if the data sheet for your diode device specifies a value for the reverse recovery charge instead of a value for the reverse recovery time.

The reverse recovery charge is the total charge that continues to dissipate when the

diode turns off. The value must be less than  $-\frac{i_{RM}^2}{2a}$ ,

where:

- $i_{RM}$  is the value specified for **Peak reverse current, iRM**.
- $a$  is the value specified for **Rate of change of current when measuring iRM**.

The default value is 1500  $\mu$ As.

The parameter is visible only if you set **Reverse recovery time parameterization** to Specify reverse recovery charge.

## References

- [1] Lauritzen, P.O. & C.L. Ma, "A Simple Diode Model with Reverse Recovery." *IEEE® Transactions on Power Electronics*. Vol. 6, No. 2, 1991, pp. 188-191.

## **See Also**

Diode (Piecewise Linear) | GTO | IGBT | Ideal Semiconductor Switch | MOSFET | Thyristor (Piecewise Linear)

## **Topics**

“Quantifying IGBT Thermal Losses”

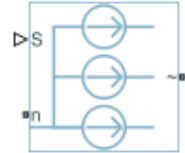
“Simulate Thermal Losses in Semiconductors”

**Introduced in R2013b**

## Controlled Current Source (Three-Phase)

Ideal three-phase controlled current source

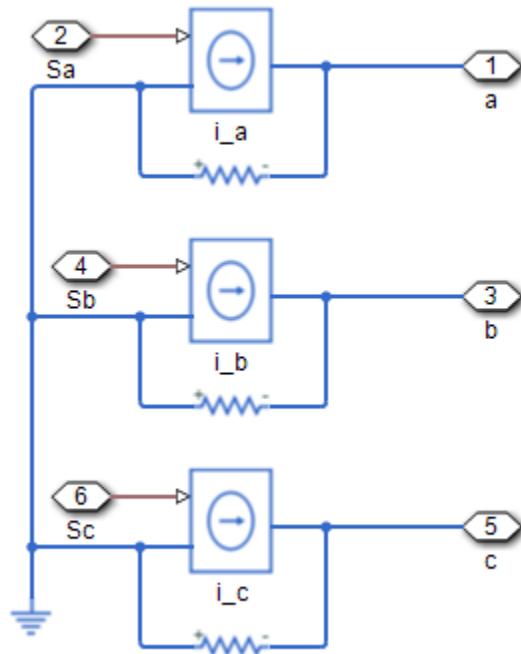
**Library:** Simscape / Electrical / Power Systems / Sources



### Description

The Controlled Current Source (Three-Phase) block represents an ideal three-phase current source that is powerful enough to maintain the specified current through it regardless of the voltage across it. The output currents are  $[i_a \ i_b \ i_c] = S$ , where  $S$  is a vector containing the numerical values presented at the physical signal port.

The figure shows the equivalent circuit for the expanded implementation of the Controlled Current Source (Three-Phase) block.



To use the Controlled Current Source (Three-Phase) block as an abstracted current controller in an electrical drive, connect the conserving ports for the output current directly to the machine.

To access the variant implementation with an expanded, three-phase port, right-click the block and select **Simscape > Block choices**.

## Ports

### Input

#### S — Control

physical signal

Physical signal input port associated with the control signal.



## Conserving

### **n — Neutral**

electrical

Electrical conserving port associated with the neutral phase.

### **~ — Three-phase current**

electrical

Expandable three-phase port.

### **a — a-phase current**

electrical

Electrical conserving port associated with the *a*-phase output current.

### **b — b-phase current**

electrical

Electrical conserving port associated with the *b*-phase output current.

### **c — c-phase current**

electrical

Electrical conserving port associated with the *c*-phase output current.

## See Also

Battery | Current Source (Three-Phase) | DC Current Source | Supercapacitor | Voltage Source (Three-Phase)

## Topics

“Expand and Collapse Three-Phase Ports on a Block”

**Introduced in R2018b**

# Controlled PWM Voltage

Pulse-width modulated voltage source model

**Library:** Simscape / Electrical / Electronics and Mechatronics /  
Actuators & Drivers / Drivers



## Description

The Controlled PWM Voltage block represents a pulse-width modulated (PWM) voltage source. The block has two modeling variants, accessible by right-clicking the block in your block diagram and then selecting the appropriate option from the context menu, under **Simscape > Block choices**:

- **Electrical input ports** — The block calculates the duty cycle based on the reference voltage across its **ref+** and **ref-** ports. This modeling variant is the default.
- **Physical signal input** — Specify the duty cycle value directly by using an input physical signal port.

For the **Electrical input ports** variant of the block, the demanded duty cycle is

$$100 * \frac{V_{ref} - V_{min}}{V_{max} - V_{min}} \textit{percent}$$

where:

- $V_{ref}$  is reference voltage across the **ref+** and **ref-** ports.
- $V_{min}$  is the minimum reference voltage.
- $V_{max}$  is the maximum reference voltage.

The value of the **Output voltage amplitude** parameter determines amplitude of the output voltage.

At time zero, the pulse is initialized as high, unless the **Pulse delay time** parameter is greater than zero, or the demanded duty cycle is zero.

You can use parameters **Pulse delay time** and **Pulse width offset** to add a small turn-on delay and a small turn-off advance. This can be useful when fine-tuning switching times so as to minimize switching losses.

In PWM mode, the block has two options for the type of switching event when moving between output high and output low states:

- **Asynchronous – Best for variable-step solvers** – Asynchronous events are better suited to variable step solvers, because they require fewer simulation steps for the same level of accuracy. In asynchronous mode the PWM switching events generate zero crossings, and therefore switching times are always determined accurately, regardless of the simulation maximum step size.
- **Discrete-time – Best for fixed-step solvers** – Discrete-time events are better suited to fixed-step operation, because then the switching events are always synchronized with the simulation step. Using an asynchronous implementation with fixed-step solvers may sometimes result in events being up to one simulation step late. For more information, see “Simulating with Fixed Time Step – Local and Global Fixed-Step Solvers” (Simscape).

If you use a fixed-step or local solver and the discrete-time switching event type, the following restrictions apply to the **Sample time** parameter value:

- The sample time must be a multiple of the simulation step size.
- The sample time must be small compared to the PWM period, to ensure sufficient resolution.

## Basic Assumptions and Limitations

The model is based on the following assumptions:

- The REF output of this block is floating, it is not tied to the Electrical Reference. One consequence of this is that if you connect the PWM and REF electrical ports directly to the PWM and REF electrical ports of an H-bridge or a gate driver, you must attach an Electrical Reference block to the REF connection line.
- Do not connect the Controlled PWM block directly to a semiconductor gate, because this omits the gate driver output impedance that determines switching dynamics. Use a Gate Driver or a Half-Bridge Driver block to set the gate-source or the gate-emitter voltage.

- Do not use the Controlled PWM block to drive a motor block directly. A PWM motor driver goes open circuit in between pulses. Use the H-Bridge block to drive a motor block.
- When driving a motor via the H-Bridge block, set the **Simulation mode** parameter to **Averaged** to speed up simulations. You must also set the **Simulation mode** parameter of the H-Bridge block to **Averaged** mode. This applies the average of the demanded PWM voltage to the motor. The **Averaged** mode assumes that the impedance of the motor inductive term is small at the **PWM frequency**. To verify this assumption, run the simulation using the **PWM** mode and compare the results to those obtained from using the **Averaged** mode.
- If you are linearizing your model, set the **Simulation mode** parameter to **Averaged** and ensure that you have specified the operating point of the block correctly. You can only linearize the block for inputs corresponding to a duty cycle greater than zero and less than 100 percent.
- When you use this block in **PWM** mode with the **Use local solver option** selected in the Solver Configuration block, set the **Switching event type** parameter to **Discrete –time – Best for fixed-step solvers**. Using the **Asynchronous – Best for variable-step solvers** option in this situation may produce inaccuracies, because simulation with the local solver implies fixed step, and the PWM events will not always coincide precisely with the simulation steps. This results in PWM events sometimes occurring one simulation step late.

## Ports

### Input

**u** — Control signal, unitless

physical signal

Input physical signal that specifies the duty cycle.

### Dependencies

Enabled for the **Physical signal input** variant of the block.

## Conserving

### **ref+ — Positive terminal**

electrical

Positive electrical reference voltage.

#### **Dependencies**

Enabled for the **Electrical input ports** variant of the block.

### **ref- — Negative terminal**

electrical

Negative electrical reference voltage

#### **Dependencies**

Enabled for the **Electrical input ports** variant of the block.

### **PWM — Pulse-width modulated signal**

electrical

Electrical conserving port associated with the pulse-width modulated signal.

### **REF — Floating zero volt reference**

electrical

Electrical conserving port associated with the floating zero volt reference.

## Parameters

### **PWM**

#### **PWM frequency — PWM output signal frequency**

1000 Hz (default)

Frequency of the PWM output signal.

#### **Pulse delay time — Turn-on delay**

0 s (default)

The pulse train does not start until the simulation time is equal to the **Pulse delay time**. You can specify a small value for **Pulse delay time** to fine-tune switching times and ensure that an off-going device is fully off before the on-going device starts to turn on. You can also use larger delay times, for example, if you need the pulse train to start only after a number of cycles. The value you provide must be greater than or equal to zero.

### Dependencies

Enabled when the **Simulation mode** parameter is set to PWM.

### Pulse width offset — Lengthens or shortens the pulse

0 s (default)

The demanded pulse width as defined by the product of the demanded duty cycle and one over the pulse frequency can be offset by the value you provide for **Pulse width offset**. A positive value acts to lengthen the pulse by a fixed amount. A negative value acts to shorten the pulse. You can use this parameter, along with the **Pulse delay time**, to fine-tune switching times so as to minimize switching losses in some circuits.

### Dependencies

Enabled when the **Simulation mode** parameter is set to PWM.

### Minimum pulse width — Minimum length of pulse

0 s (default)

The minimum pulse length, based on the internal clock or defined programmatically, to protect the device being driven. The value you provide must be greater than or equal to zero.

### Dependencies

Enabled when the **Simulation mode** parameter is set to PWM.

### Simulation mode — Select type of output voltage

PWM (default) | Averaged

Select the type of output voltage:

- **PWM** — Produces a pulse-width modulated signal.
- **Averaged** — The output is a constant whose value is equal to the average value of the PWM signal.

**Switching event type – Select type of switching event**

Asynchronous – Best for variable-step solvers (default) | Discrete-time – Best for fixed-step solvers

Select the switching event type when moving between output high and output low states:

- **Asynchronous – Best for variable-step solvers** – This option is more efficient for desktop simulation with variable-step solvers, because it requires fewer simulation steps for the same level of accuracy.
- **Discrete-time – Best for fixed-step solvers** – Use with fixed-step solvers, including the local solver. For more information, see “Simulating with Fixed Time Step – Local and Global Fixed-Step Solvers” (Simscape).

**Dependencies**

Enabled when the **Simulation mode** parameter is set to PWM.

**Sample time – Discrete sampling time**

1e-6 s (default)

The time between updates of the block output state. The sample time must be a multiple of the simulation step size. In order for the PWM control to have sufficient resolution, set the sample time to less than one hundredth of the PWM period. (The PWM period is one over the PWM frequency.)

**Dependencies**

Enabled when the **Switching event type** parameter is set to Discrete-time – Best for fixed-step solvers.

**Input Scaling****Input voltage for 0% duty cycle – Vmin**

0 V (default)

Value of the input voltage at which the PWM signal has a 0% duty cycle.

**Dependencies**

Enabled for the **Electrical input ports** variant of the block.

**Input voltage for 100% duty cycle – Vmax**

5 V (default)

Value of the input voltage at which the PWM signal has a 100% duty cycle.

**Dependencies**

Enabled for the **Electrical input ports** variant of the block.

**Input value for 0% duty cycle — Minimum signal value**

0 (default)

Value of the input signal at which the PWM signal has a 0% duty cycle.

**Dependencies**

Enabled for the **Physical signal input** variant of the block.

**Input value for 100% duty cycle — Maximum signal value**

1 (default)

Value of the input signal at which the PWM signal has a 100% duty cycle

**Dependencies**

Enabled for the **Physical signal input** variant of the block.

## Output Voltage

**Output voltage amplitude — Signal amplitude for high output**

5 V (default)

Amplitude of the PWM signal when the output is high.

## See Also

Gate Driver | H-Bridge | Half-Bridge Driver

**Introduced in R2008b**



# Converter

Connect three-phase AC network to DC network



## Library

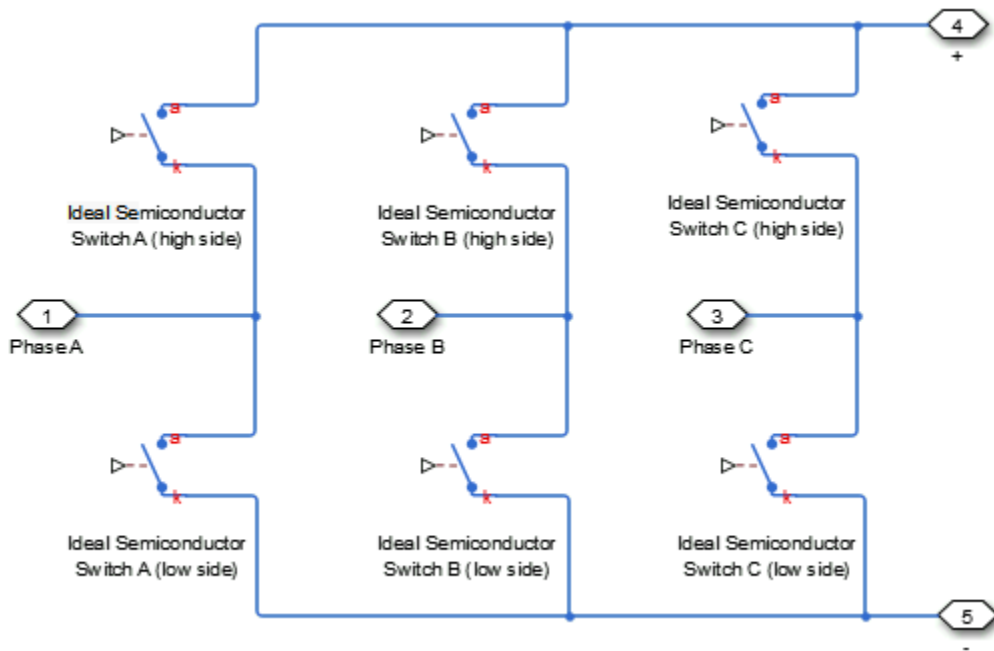
Simscape / Electrical / Power Systems / Semiconductors

## Description

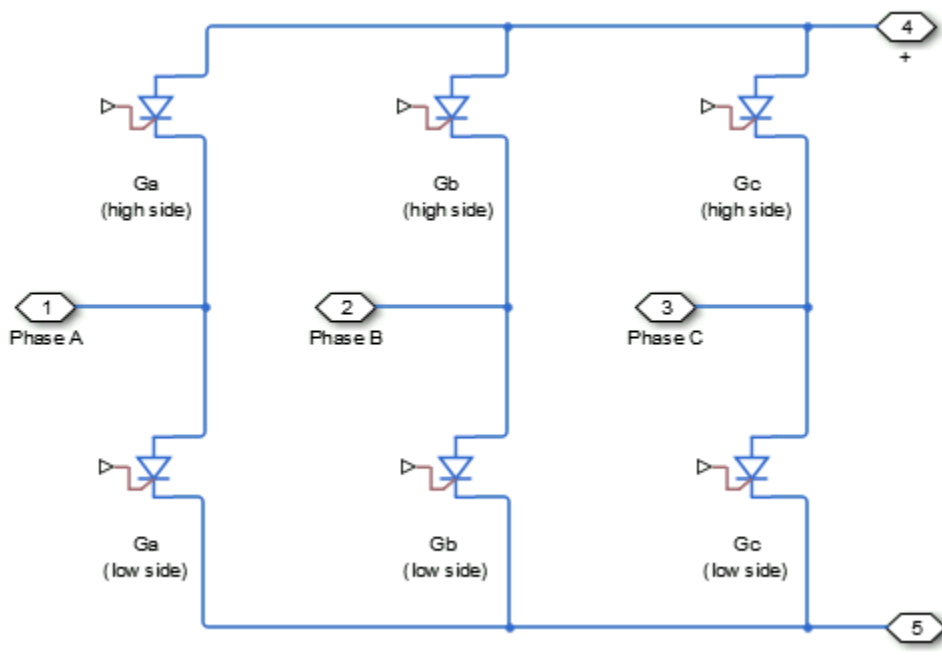
The Converter block models a three-arm converter circuit that connects a three-phase AC network to a DC network.

Each component in the three-arm circuit is the same switching device, which you specify using an option on the Converter block dialog box. The switching devices that you can specify are in the Semiconductors / Fundamental Components library.

The figure shows the equivalent circuit for a converter with fully controlled switching devices (e.g. IGBTs, GTOs).



The figure shows the equivalent circuit for a converter with partially controlled switching devices (e.g. thyristors).



Control the gate ports of the six switching devices via an input to port G on the Converter block:

- 1 Mux all six gate signals into a single vector with a Six-Pulse Gate Multiplexer block.
- 2 Connect the output of the Six-Pulse Gate Multiplexer block to the Converter block G port.

You can specify an integral protection diode for each switching device. An integral diode protects the semiconductor device by providing a conduction path for reverse current. An inductive load can produce a high reverse-voltage spike when the semiconductor device suddenly switches off the voltage supply to the load.

The table shows you how to set the **Integral protection diode** parameter based on your goals.

<b>Goal</b>	<b>Value to Select</b>	<b>Block Behavior</b>
Prioritize simulation speed.	Protection diode with no dynamics	The block includes an integral copy of the Diode block. To parameterise the internal Diode block, use the <b>Protection</b> parameters.
Precisely specify reverse-mode charge dynamics.	Protection diode with charge dynamics	The block includes an integral copy of the Commutation Diode block. To parameterise the internal Commutation Diode block, use the <b>Protection</b> parameters.

You can include a snubber circuit, consisting of a resistor and capacitor connected in series, for each switching device. Snubber circuits protect switching devices against high voltages that inductive loads produce when the device turns off the voltage supply to the load. Snubber circuits also prevent excessive rates of change of current when a switching device turns on.

## Parameters

- “Switching Devices Tab” on page 1-358
- “Protection Diodes Tab” on page 1-362
- “Snubbers Tab” on page 1-364

## Switching Devices Tab

### Switching device

Converter switching device. The default value is `Ideal Semiconductor Switch`.

The switching devices you can select are:

- GTO
- `Ideal Semiconductor Switch`
- IGBT

- MOSFET
- Thyristor

When you select GTO, parameters for the GTO block appear.

### **Additional GTO Parameters**

#### **Forward voltage, $V_f$**

Minimum voltage required across the anode and cathode block ports for the gradient of the device i-v characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**. The default value is 0.8 V.

#### **On-state resistance**

Rate of change of voltage versus current above the forward voltage. The default value is 0.001 Ohm.

#### **Off-state conductance**

Anode-cathode conductance when the device is off. The value must be less than  $1/R$ , where  $R$  is the value of **On-state resistance**. The default value is  $1e-6$  1/Ohm.

#### **Gate trigger voltage, $V_{gt}$**

Gate-cathode voltage threshold. The device turns on when the gate-cathode voltage is above this value. The default value is 1 V.

#### **Gate turn-off voltage, $V_{gt\_off}$**

Gate-cathode voltage threshold. The device turns off when the gate-cathode voltage is below this value. The default value is -1 V.

#### **Holding current**

Current threshold. The device stays on when the current is above this value, even when the gate-cathode voltage falls below the gate trigger voltage. The default value is 1 A.

For more information, see GTO.

When you select **Ideal Semiconductor Switch**, parameters for the **Ideal Semiconductor Switch** block appear.

## **Additional Ideal Semiconductor Switch Parameters**

### **On-state resistance**

Anode-cathode resistance when the device is on. The default value is  $0.001 \text{ } 1/\Omega$ .

### **Off-state conductance**

Anode-cathode conductance when the device is off. The value must be less than  $1/R$ , where  $R$  is the value of **On-state resistance**. The default value is  $1e-6 \text{ } 1/\Omega$ .

### **Threshold voltage, $V_{th}$**

Gate-cathode voltage threshold. The device turns on when the gate-cathode voltage is above this value. The default value is  $6 \text{ V}$ .

For more information, see Ideal Semiconductor Switch.

When you select IGBT, parameters for the IGBT block appear.

## **Additional IGBT Parameters**

### **Forward voltage, $V_f$**

Minimum voltage required across the collector and emitter block ports for the gradient of the diode  $i$ - $v$  characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**. The default value is  $0.8 \text{ V}$ .

### **On-state resistance**

Collector-emitter resistance when the device is on. The default value is  $0.001 \text{ } \Omega$ .

### **Off-state conductance**

Collector-emitter conductance when the device is off. The value must be less than  $1/R$ , where  $R$  is the value of **On-state resistance**. The default value is  $1e-6 \text{ } 1/\Omega$ .

### **Threshold voltage, $V_{th}$**

Gate-emitter voltage at which the device turns on. The default value is  $6 \text{ V}$ .

For more information, see IGBT.

When you select MOSFET, parameters for the MOSFET block appear.

## **Additional MOSFET Parameters**

### **On-state resistance, $R_{DS(on)}$**

Drain-source resistance when the device is on. The default value is 0.001 Ohm.

### **Off-state conductance**

Drain-source conductance when the device is off. The value must be less than  $1/R$ , where  $R$  is the value of **On-state resistance**. The default value is  $1e-6$  1/Ohm.

### **Threshold voltage, $V_{th}$**

Gate-source voltage threshold. The device turns on when the gate-source voltage is above this value. The default value is 6 V.

For more information, see MOSFET.

When you select Thyristor, parameters for the Thyristor block appear.

## **Additional Thyristor Parameters**

### **Forward voltage, $V_f$**

Forward voltage at which the device turns on. The default value is 0.8 V.

### **On-state resistance**

Anode-cathode resistance when the device is on. The default value is 0.001 Ohm.

### **Off-state conductance**

Anode-cathode conductance when the device is off. The value must be less than  $1/R$ , where  $R$  is the value of **On-state resistance**. The default value is  $1e-6$  1/Ohm.

### **Gate trigger voltage, $V_{gt}$**

Gate-cathode voltage threshold. The device turns on when the gate-cathode voltage is above this value. The default value is 1 V.

### **Holding current**

Current threshold. The device stays on when the current is above this value, even when the gate-cathode voltage falls below the gate trigger voltage. The default value is 1 A.

For more information, see Thyristor.

## Protection Diodes Tab

### Integral protection diode

Integral protection diode for each switching device. The default value is **None**.

The diodes you can select are:

- Protection diode with no dynamics
- Protection diode with charge dynamics

When you select **Protection diode with no dynamics**, additional parameters appear.

### Additional Parameters for Protection Diode with No Dynamics

#### Forward voltage

Minimum voltage required across the + and - block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**. The default value is **0.8 V**.

#### On resistance

Rate of change of voltage versus current above the **Forward voltage**. The default value is **0.001 Ohm**.

#### Off conductance

Conductance of the reverse-biased diode. The default value is **1e-5 1/Ohm**.

For more information on these parameters, see **Diode**.

When you select **Protection diode with charge dynamics**, additional parameters appear.

### Additional Parameters for Protection Diode with Charge Dynamics

#### Forward voltage

Minimum voltage required across the + and - block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**. The default value is **0.8 V**.



**On resistance**

Rate of change of voltage versus current above the **Forward voltage**. The default value is 0.001 Ohm.

**Off conductance**

Conductance of the reverse-biased diode. The default value is  $1e-5$  1/Ohm.

**Junction capacitance**

Diode junction capacitance. The default value is 50 nF.

**Peak reverse current, iRM**

Peak reverse current measured by an external test circuit. This value must be less than zero. The default value is -235 A.

**Initial forward current when measuring iRM**

Initial forward current when measuring peak reverse current. This value must be greater than zero. The default value is 300 A.

**Rate of change of current when measuring iRM**

Rate of change of current when measuring peak reverse current. This value must be less than zero. The default value is -50 A/ $\mu$ s.

**Reverse recovery time parameterization**

Determines how you specify reverse recovery time in the block. The default value is Specify reverse recovery time directly.

If you select Specify stretch factor or Specify reverse recovery charge, you specify a value that the block uses to derive the reverse recovery time. For more information on these options, see “Alternatives to Specifying trr Directly” on page 1-339.

**Reverse recovery time, trr**

Interval between the time when the current initially goes to zero (when the diode turns off) and the time when the current falls to less than 10% of the peak reverse current. The default value is 15  $\mu$ s.

This parameter is visible only if you set **Reverse recovery time parameterization** to Specify reverse recovery time directly.

The value of the **Reverse recovery time, trr** parameter must be greater than the value of the **Peak reverse current, iRM** parameter divided by the value of the **Rate of change of current when measuring iRM** parameter.

### Reverse recovery time stretch factor

Value that the block uses to calculate **Reverse recovery time, trr**. This value must be greater than 1. The default value is 3.

This parameter is visible only if you set **Reverse recovery time parameterization** to **Specify stretch factor**.

Specifying the stretch factor is an easier way to parameterize the reverse recovery time than specifying the reverse recovery charge. The larger the value of the stretch factor, the longer it takes for the reverse recovery current to dissipate.

### Reverse recovery charge, Qrr

Value that the block uses to calculate **Reverse recovery time, trr**. Use this parameter if the data sheet for your diode device specifies a value for the reverse recovery charge instead of a value for the reverse recovery time.

The reverse recovery charge is the total charge that continues to dissipate when the

diode turns off. The value must be less than  $-\frac{i_{RM}^2}{2a}$ ,

where:

- $i_{RM}$  is the value specified for **Peak reverse current, iRM**.
- $a$  is the value specified for **Rate of change of current when measuring iRM**.

The default value is 1500  $\mu$ As.

The parameter is visible only if you set **Reverse recovery time parameterization** to **Specify reverse recovery charge**.

For more information on these parameters, see Commutation Diode.

## Snubbers Tab

### Snubber

Snubber for each switching device. The default value is None.

### Snubber resistance

Snubber resistance. This parameter is visible only if you set **Snubber** to RC snubber. The default value is 0.1 Ohm.

### Snubber capacitance

Snubber capacitance. This parameter is visible only if you set **Snubber** to RC snubber. The default value is  $1e-7$  F.

## Ports

The block has the following ports:

G

Vector input port associated with the gate terminals of the switching devices. Connect this port to a Six-Pulse Gate Multiplexer block.

~

Expandable three-phase port

+

Electrical conserving port associated with the DC positive terminal

-

Electrical conserving port associated with the DC negative terminal

## See Also

[Average-Value DC-DC Converter](#) | [Average-Value Inverter](#) | [Average-Value Rectifier](#) | [Bidirectional DC-DC Converter](#) | [Boost Converter](#) | [Buck Converter](#) | [Buck-Boost Converter](#) | [Rectifier](#) | [Six-Pulse Gate Multiplexer](#) | [Three-Level Converter](#)

## Topics

[“Expand and Collapse Three-Phase Ports on a Block”](#)

**Introduced in R2013b**

## Counter

Discrete- or continuous-time counter

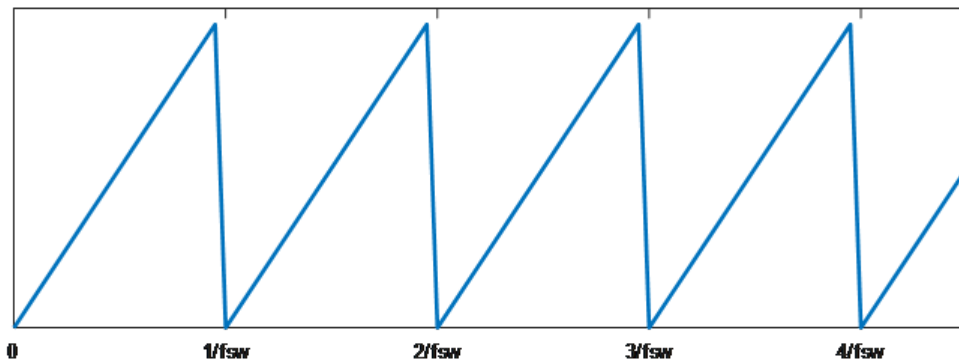
**Library:** Simscape / Electrical / Power Systems / Control / General Control



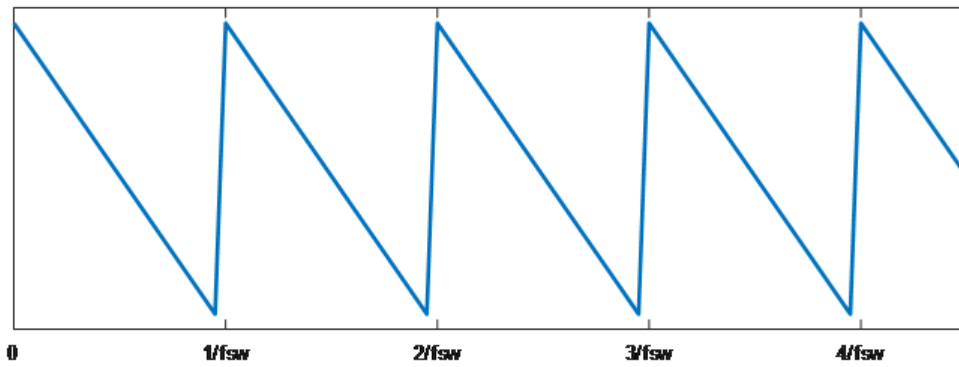
## Description

The Counter block implements an incremental counter in either discrete or continuous time. The counter increments in response to one of these criteria:

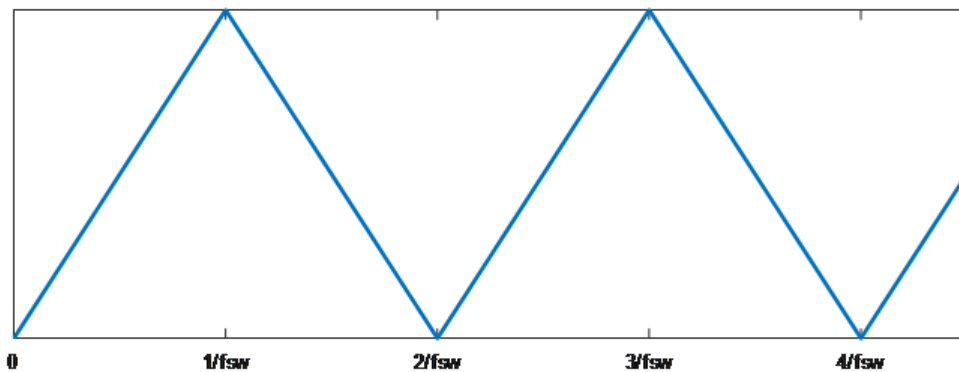
- Up — The input goes up.



- Down — The input goes down.



- Up-Down — The input goes up or down.



## Ports

### Output

**T** — Counter value

scalar

Counter value.

Data Types: single | double

## Parameters

### **Counter type — Counter strategy**

Up (default) | Down | Up-Down

Counter type.

### **Output range — Range**

[0 1] (default) | [-1 1]

Range for the output.

### **Timer period (s) — Period**

0.001 (default) | positive scalar

Timer period, in s.

### **Phase delay (s) — Delay**

0 (default) | 0 or positive scalar

Phase delay, in s. Add a phase delay to change the initial state of the counter.

### **Sample time — Block sample time**

0.0001 (default) | 0 or positive scalar

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For discrete-time simulation, set the sample time to a positive scalar. For continuous-time simulation, set the sample time to 0.

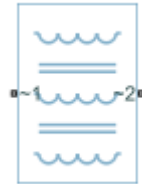
## See Also

**Introduced in R2018b**

# Coupled Lines

Magnetically couple three-phase lines

**Library:** Simscape / Electrical / Power Systems / Passive Devices



## Description

The Coupled Lines block models three magnetically coupled lines. Each line has a self-inductance, series resistance, and parallel conductance. In addition, there is a mutual inductance and mutual resistance between each pair of lines.

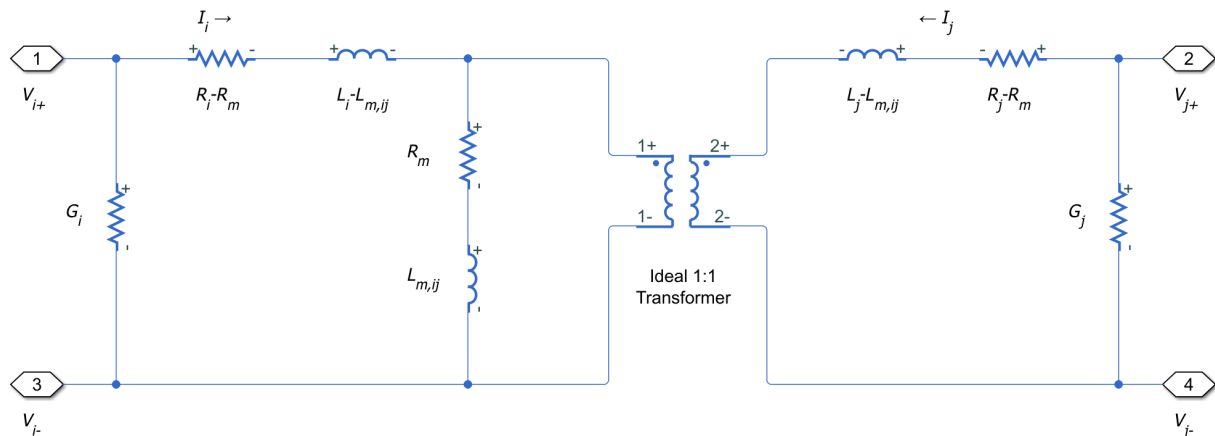
Use this block when the magnetic coupling in a three-phase network is nonnegligible. These effects are most prominent when:

- The lines are parallel and close together.
- The self-inductances of the lines are high.
- The AC frequency of the network is high.

To model magnetic coupling of a single pair of lines, use the Coupled Lines (Pair) block. To model capacitive coupling between the lines, use the Transmission Line block.

## Equivalent Circuit

The equivalent circuit shows the coupling between two arbitrary phases  $i$ , and  $j$ . The block models the magnetic coupling using such an equivalent circuit between each of the three phases  $a$ ,  $b$ , and  $c$ .



Here:

- $R_i$  and  $R_j$  are the series resistances of lines  $i$  and  $j$ , respectively.
- $L_i$  and  $L_j$  are the self-inductances of lines  $i$  and  $j$ , respectively.
- $R_m$  is the mutual resistance between the two lines. You can use this parameter to account for losses in a common return path.
- $L_{m,ij}$  is the mutual inductance between lines  $i$  and  $j$ , respectively.
- $G_i$  and  $G_j$  are the leakage conductances of lines  $i$  and  $j$ , respectively.
- $V_i$  and  $V_j$  are voltage drops across lines  $i$  and  $j$ , respectively.
- $I_i$  and  $I_j$  are the currents through the resistors  $R_i-R_m$  and  $R_j-R_m$ , respectively.

## Equations

The defining equation for this block is:

$$V = \begin{bmatrix} R_a & R_m & R_m \\ R_m & R_b & R_m \\ R_m & R_m & R_c \end{bmatrix} I + \begin{bmatrix} L_a & L_{m,ab} & L_{m,ac} \\ L_{m,ab} & L_b & L_{m,bc} \\ L_{m,ac} & L_{m,bc} & L_c \end{bmatrix} \frac{dI}{dt},$$

where:



$$V = \begin{bmatrix} V_a \\ V_b \\ V_c \end{bmatrix},$$

$$I = \begin{bmatrix} I_a \\ I_b \\ I_c \end{bmatrix}.$$

$I_a$ ,  $I_b$ , and  $I_c$  are, in general, not equal to the currents in lines  $a$ ,  $b$ , and  $c$ . These terminal currents make up the vector:

$$I_{total} = I + \begin{bmatrix} G_a & 0 & 0 \\ 0 & G_b & 0 \\ 0 & 0 & G_c \end{bmatrix} V.$$

## Inductive Coupling

To quantify the strength of the coupling between the two lines, you can use a coupling factor or coefficient of coupling  $k$ . The coupling factor relates the mutual inductance to the line self-inductances:

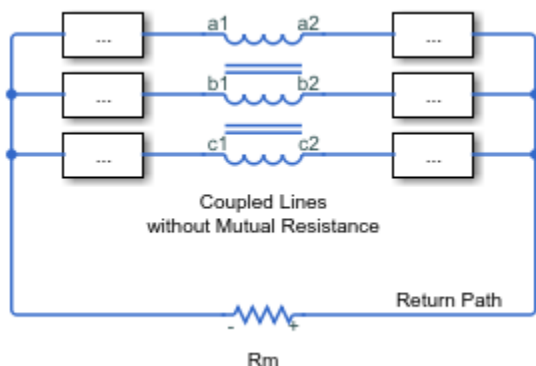
$$L_{m,ij} = k \sqrt{L_i L_j}.$$

This coupling factor must fall in the range  $-1 < k < 1$ , where a negative coupling factor indicates a reversal in orientation of one of the coils. The magnitude of  $k$  indicates:

- $|k| = 0$  — There is no magnetic coupling between the two lines.
- $0 < |k| < 0.5$  — The two lines are loosely coupled and mutual magnetic effects are small.
- $0.5 \leq |k| < 1$  — The two lines are strongly coupled and mutual magnetic effects are large.

## Mutual Resistance

If the three lines share a common return path, you can model the resistance of this return path using the **Mutual resistance** parameter  $R_m$ . This workflow is equivalent to setting the **Mutual resistance** to zero and explicitly modeling the return path resistance  $R_m$ , as shown in this diagram.



If the three lines do not share a common return path, set the mutual resistance parameter to zero and model each of the return resistances explicitly.

## Ports

### Conserving

#### ~1 — Line 1 positive terminal

electrical

Expandable three-phase electrical conserving port associated with the positive terminals of lines *a*, *b*, and *c*. To use this composite port, right-click the block and select **Simscape** > **Block choices** > **Composite three-phase port**.

#### ~2 — Line 1 negative terminal

electrical

Expandable three-phase electrical conserving port associated with the negative terminals of lines *a*, *b*, and *c*. To use this composite port, right-click the block and select **Simscape > Block choices > Composite three-phase port**.

#### **a1 — Line a positive terminal**

electrical

Electrical conserving port associated with the positive terminal of line *a*. To use this expanded port, right-click the block and select **Simscape > Block choices > Expanded three-phase port**.

#### **a2 — Line a negative terminal**

electrical

Electrical conserving port associated with the negative terminal of line *a*. To use this expanded port, right-click the block and select **Simscape > Block choices > Expanded three-phase port**.

#### **b1 — Line b positive terminal**

electrical

Electrical conserving port associated with the positive terminal of line *b*. To use this expanded port, right-click the block and select **Simscape > Block choices > Expanded three-phase port**.

#### **b2 — Line b negative terminal**

electrical

Electrical conserving port associated with the negative terminal of line *b*. To use this expanded port, right-click the block and select **Simscape > Block choices > Expanded three-phase port**.

#### **c1 — Line c positive terminal**

electrical

Electrical conserving port associated with the positive terminal of line *c*. To use this expanded port, right-click the block and select **Simscape > Block choices > Expanded three-phase port**.

#### **c2 — Line c negative terminal**

electrical

Electrical conserving port associated with the negative terminal of line *c*. To use this expanded port, right-click the block and select **Simscape > Block choices > Expanded three-phase port**.

## Parameters

### Main

#### Parameterization — Line impedance parameterization

Balanced impedance (default) | General impedance

Specify how to parameterize the impedance of the three lines:

#### Balanced impedance

Specify the same series resistance, series inductance, and parallel leakage conductance for all lines.

#### General impedance

Specify the series resistance, series inductance, and parallel leakage conductance separately for each line.

#### Line a inductance — Line a self-inductance

1e-3 H (default)

Self-inductance of line *a*. This value must be greater than zero.

#### Dependencies

To enable this parameter, set **Parameterization** to General impedance.

#### Line b inductance — Line b self-inductance

1e-3 H (default)

Self-inductance of line *b*. This value must be greater than zero.

#### Dependencies

To enable this parameter, set **Parameterization** to General impedance.

#### Line c inductance — Line c self-inductance

1e-3 H (default)

Self-inductance of line  $c$ . This value must be greater than zero.

#### Dependencies

To enable this parameter, set **Parameterization** to General impedance.

#### Line inductance — Self-inductance

1e-3 H (default)

Self-inductance of lines  $a$ ,  $b$ , and  $c$ . This value must be greater than zero.

#### Dependencies

To enable this parameter, set **Parameterization** to Balanced impedance.

#### Line a-b mutual inductance — Mutual inductance between a-b

3e-4 H (default)

Mutual inductance between lines  $a$  and  $b$ . If you know the coupling factor, set this value to  $k\sqrt{L_a L_b}$ . To have a physically realizable mutual inductance, this value must satisfy:

$$-\sqrt{L_a L_b} < L_{m,ab} < \sqrt{L_a L_b}.$$

#### Dependencies

To enable this parameter, set **Parameterization** to General impedance.

#### Line b-c mutual inductance — Mutual inductance between b-c

3e-4 H (default)

Mutual inductance between lines  $b$  and  $c$ . If you know the coupling factor, set this value to  $k\sqrt{L_b L_c}$ . To have a physically realizable mutual inductance, this value must satisfy:

$$-\sqrt{L_b L_c} < L_{m,bc} < \sqrt{L_b L_c}.$$

#### Dependencies

To enable this parameter, set **Parameterization** to General impedance.

#### Line a-c mutual inductance — Mutual inductance between a-c

3e-4 H (default)

Mutual inductance between lines  $a$  and  $c$ . If you know the coupling factor, set this value to  $k\sqrt{L_a L_c}$ . To have a physically realizable mutual inductance, this value must satisfy:

$$-\sqrt{L_a L_c} < L_{m,ac} < \sqrt{L_a L_c}.$$

**Dependencies**

To enable this parameter, set **Parameterization** to General impedance.

**Mutual inductance — Mutual inductance**

3e-4 H (default)

Mutual inductance between each pair of lines. If you know the coupling factor, set this value to  $kL$ , where  $L$  is the series inductance of each of the lines. To have a physically realizable mutual inductance, this value must satisfy:

$$-L < L_m < L.$$

**Dependencies**

To enable this parameter, set **Parameterization** to Balanced impedance.

**Resistance****Line a resistance — Line a series resistance**

0.001 Ohm (default)

Series resistance of line  $a$ . This value must be greater than or equal to zero.

**Dependencies**

To enable this parameter, set **Parameterization** to General impedance.

**Line b resistance — Line b series resistance**

0.001 Ohm (default)

Series resistance of line  $b$ . This value must be greater than or equal to zero.

**Dependencies**

To enable this parameter, set **Parameterization** to General impedance.

**Line c resistance — Line c series resistance**

0.001 Ohm (default)

Series resistance of line *c*. This value must be greater than or equal to zero.

**Dependencies**

To enable this parameter, set **Parameterization** to General impedance.

**Line resistance — Series resistance**

0.001 Ohm (default)

Series resistance of lines *a*, *b*, and *c*. This value must be greater than or equal to zero.

**Dependencies**

To enable this parameter, set **Parameterization** to Balanced impedance.

**Mutual resistance — Mutual resistance**

0 Ohm (default)

Mutual resistance between each pair of lines. This value must be greater than or equal to zero. Use this value to account for losses in a common return path.

**Line a leakage conductance — Line a parallel leakage conductance**

1e-9 1/Ohm (default)

Parallel leakage conductance of line *a*. This value must be greater than or equal to zero.

**Dependencies**

To enable this parameter, set **Parameterization** to General impedance.

**Line b leakage conductance — Line b parallel leakage conductance**

1e-9 1/Ohm (default)

Parallel leakage conductance of line *b*. This value must be greater than or equal to zero.

**Dependencies**

To enable this parameter, set **Parameterization** to General impedance.

**Line c leakage conductance — Line c parallel leakage conductance**

1e-9 1/Ohm (default)

Parallel leakage conductance of line *a*. This value must be greater than or equal to zero.

**Dependencies**

To enable this parameter, set **Parameterization** to General impedance.

**Line leakage conductance — Parallel leakage conductance**

1e-9 1/0hm (default)

Parallel leakage conductance of lines *a*, *b*, and *c*. This value must be greater than or equal to zero.

**Dependencies**

To enable this parameter, set **Parameterization** to Balanced impedance.

**See Also**

Coupled Lines (Pair)

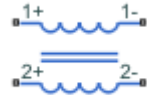
**Introduced in R2018a**



## Coupled Lines (Pair)

Magnetically couple two lines

**Library:** Simscape / Electrical / Power Systems / Passive  
Devices / Fundamental Components



### Description

The Coupled Lines (Pair) block models two magnetically coupled lines. Each line has a self-inductance, series resistance, and parallel conductance. In addition, there is a mutual inductance and mutual resistance between the two lines.

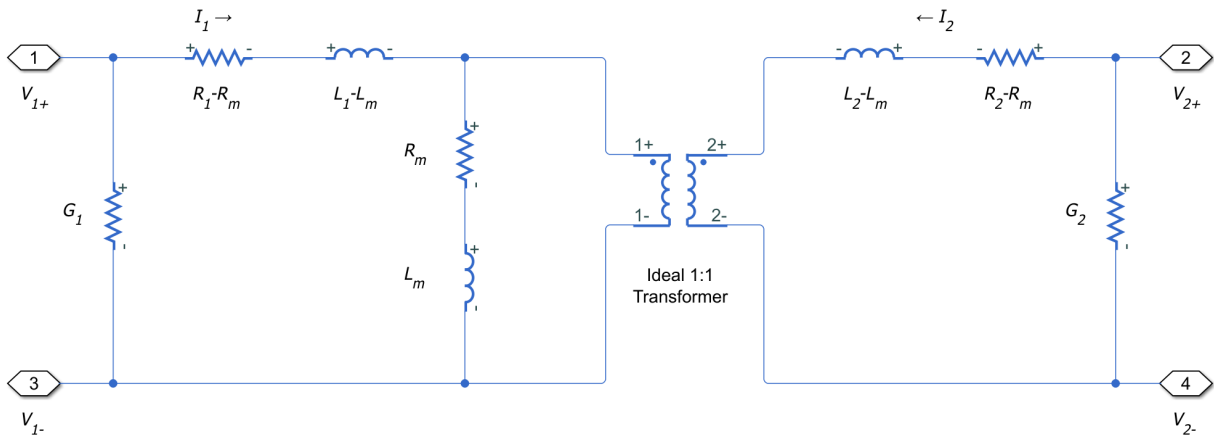
Use this block when the magnetic coupling between the two lines is nonnegligible. These effects are most prominent when:

- The lines are parallel and close together.
- The self-inductances of the lines are high.
- The AC frequency of the network is high.

To model magnetic coupling of a three-phase line, use the Coupled Lines block.

### Equivalent Circuit

The figure shows the equivalent circuit for a pair of coupled lines.



Here:

- $R_1$  and  $R_2$  are the series resistances of lines 1 and 2, respectively.
- $L_1$  and  $L_2$  are the self-inductances of lines 1 and 2, respectively.
- $R_m$  is the mutual resistance between the two lines. You can use this parameter to account for losses in a common return path.
- $L_m$  is the mutual inductance between the two lines.
- $G_1$  and  $G_2$  are the leakage conductances of lines 1 and 2, respectively.
- $V_1$  and  $V_2$  are voltage drops across lines 1 and 2, respectively.
- $I_1$  and  $I_2$  are the currents through the resistors  $R_1-R_m$  and  $R_2-R_m$ , respectively.

## Equations

The defining equation for this block is:

$$V = \begin{bmatrix} R_1 & R_m \\ R_m & R_2 \end{bmatrix} I + \begin{bmatrix} L_1 & L_m \\ L_m & L_2 \end{bmatrix} \frac{dI}{dt},$$

where:

$$V = \begin{bmatrix} V_1 \\ V_2 \end{bmatrix},$$

$$I = \begin{bmatrix} I_1 \\ I_2 \end{bmatrix}.$$

$I_1$  and  $I_2$  are, in general, not equal to the currents in line 1 and line 2. These terminal currents make up the vector:

$$I_{total} = I + \begin{bmatrix} G_1 & 0 \\ 0 & G_2 \end{bmatrix} V.$$

## Inductive Coupling

To quantify the strength of the coupling between the two lines, you can use a coupling factor or coefficient of coupling  $k$ . The coupling factor relates the mutual inductance to the line self-inductances:

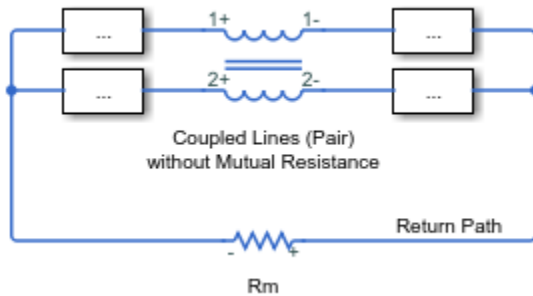
$$L_m = k\sqrt{L_1 L_2}.$$

This coupling factor must fall in the range  $-1 < k < 1$ , where a negative coupling factor indicates a reversal in orientation of one of the coils. The magnitude of  $k$  indicates:

- $|k| = 0$  — There is no magnetic coupling between the two lines.
- $0 < |k| < 0.5$  — The two lines are loosely coupled and mutual magnetic effects are small.
- $0.5 \leq |k| < 1$  — The two lines are strongly coupled and mutual magnetic effects are large.

## Mutual Resistance

If the two lines share a common return path, you can model the resistance of this return path using the **Mutual resistance** parameter. This workflow is equivalent to setting the **Mutual resistance** to zero and explicitly modeling the return path resistance  $R_m$ , as shown in this diagram.



If the two lines do not share a common return path, set the mutual resistance parameter to zero and model each of the return resistances explicitly.

## Ports

### Conserving

#### 1+ — Line 1 positive terminal

electrical

Electrical conserving port associated with the positive terminal of line 1.

#### 1- — Line 1 negative terminal

electrical

Electrical conserving port associated with the negative terminal of line 1.

#### 2+ — Line 2 positive terminal

electrical

Electrical conserving port associated with the positive terminal of line 2.

#### 2- — Line 2 negative terminal

electrical

Electrical conserving port associated with the negative terminal of line 2.

# Parameters

## Parameters

### Parameterization — Line impedance parameterization

Balanced impedance (default) | General impedance

Specify how to parameterize the impedance of the two lines:

#### Balanced impedance

Specify the same series resistance, series inductance, and parallel leakage conductance for both lines.

#### General impedance

Specify the series resistance, series inductance, and parallel leakage conductance separately for each line.

### Line 1 inductance — Line 1 self-inductance

1e-3 H (default)

Self-inductance of line 1. This value must be greater than zero.

#### Dependencies

To enable this parameter, set **Parameterization** to General impedance.

### Line 2 inductance — Line 2 self-inductance

1e-3 H (default)

Self-inductance of line 2. This value must be greater than zero.

#### Dependencies

To enable this parameter, set **Parameterization** to General impedance.

### Line inductance — Self-inductance

1e-3 H (default)

Self-inductance of line 1 and line 2. This value must be greater than zero.

#### Dependencies

To enable this parameter, set **Parameterization** to Balanced impedance.

**Mutual inductance — Mutual inductance between lines**

3e-4 H (default)

Mutual inductance between line 1 and line 2. If you know the coupling factor, set this value to  $k\sqrt{L_1L_2}$ . To have a physically realizable mutual inductance, this value must satisfy:

$$-\sqrt{L_1L_2} < L_m < \sqrt{L_1L_2}.$$

**Line 1 resistance — Line 1 series resistance**

0.001 Ohm (default)

Series resistance of line 1. This value must be greater than or equal to zero.

**Dependencies**

To enable this parameter, set **Parameterization** to General impedance.

**Line 2 resistance — Line 2 series resistance**

0.001 Ohm (default)

Series resistance of line 2. This value must be greater than or equal to zero.

**Dependencies**

To enable this parameter, set **Parameterization** to General impedance.

**Line resistance — Series resistance**

0.001 Ohm (default)

Series resistance of line 1 and line 2. This value must be greater than or equal to zero.

**Dependencies**

To enable this parameter, set **Parameterization** to Balanced impedance.

**Mutual resistance — Mutual resistance between lines**

0 Ohm (default)

Mutual resistance between line 1 and line 2. This value must be greater than or equal to zero. Use this value to account for losses in a common return path.

**Line 1 leakage conductance — Line 1 parallel leakage conductance**

1e-9 1/0hm (default)

Parallel leakage conductance of line 1. This value must be greater than or equal to zero.

**Dependencies**

To enable this parameter, set **Parameterization** to General impedance.

**Line 2 leakage conductance — Line 2 parallel leakage conductance**

1e-9 1/0hm (default)

Parallel leakage conductance of line 2. This value must be greater than or equal to zero.

**Dependencies**

To enable this parameter, set **Parameterization** to General impedance.

**Line leakage conductance — Parallel leakage conductance**

1e-9 1/0hm (default)

Parallel leakage conductance of line 1 and line 2. This value must be greater than or equal to zero.

**Dependencies**

To enable this parameter, set **Parameterization** to Balanced impedance.

**See Also**

Coupled Lines

**Introduced in R2018a**

## Crystal

Model stable resonator

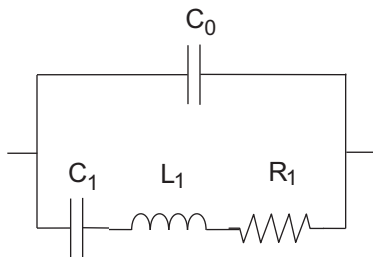


## Library

Passive Devices

## Description

The Crystal block represents the electrical characteristics of a crystal. The following figure shows the equivalent circuit model of the Crystal block.



You specify the equivalent circuit parameters for this model when you set the **Parameterization** parameter to **Equivalent circuit parameters**.

- The capacitor  $C_0$  corresponds to the capacitance you specify in the **Shunt capacitance, C0** parameter.
- The capacitor  $C_1$  corresponds to the capacitance you specify in the **Motional capacitance, C1** parameter.
- The inductor  $L_1$  corresponds to the inductance you specify in the **Motional inductance, L1** parameter.



- The resistor  $R1$  corresponds to the resistance you specify in the **Equivalent series resistance, R1** parameter.

Most datasheets specify crystal frequency rather than inductance, so the block optionally accepts frequency data.

- When you set the **Parameterization** parameter to `Series resonance data`, the block uses the following relationship to calculate  $L1$  from the series resonant frequency:

$$f_s = \frac{1}{2\pi\sqrt{L_1 C_1}}$$

Where  $f_s$  is the **Series resonance, fs** parameter value.

- When you set the **Parameterization** parameter to `Parallel resonance data`, the block uses the following relationship to calculate  $L1$  from the parallel resonant frequency:

$$f_a = \frac{1}{2\pi\sqrt{L_1 C_1 (C_0 + C_L) / (C_1 + C_0 + C_L)}}$$

Where:

- $f_a$  is the **Parallel resonance, fa** parameter value.
- $C_L$  is the **Load capacitance, CL** parameter value.

Some datasheets specify quality factor rather than equivalent series resistance, so the block optionally accepts quality factor data. When you set the **R1 parameterization** parameter to `Quality factor Q`, the block uses the following relationship to calculate  $R1$  from the quality factor:

$$Q = \frac{2\pi f L_1}{R_1}$$

Where  $Q$  is the **Quality factor, Q** parameter value.

---

**Note** The **R1 parameterization** parameter is only visible when you select **Series resonance data** or **Parallel resonance data** for the **Parameterization** parameter.

---

## Basic Assumptions and Limitations

The Crystal block models only the fundamental crystal vibration mode.

## Parameters

### Parameterization

Select one of the following methods for block parameterization:

- **Series resonance data** — Provide series resonant frequency and capacitance data for the crystal. This method is the default.
- **Parallel resonance data** — Provide parallel resonant frequency and capacitance data for the crystal.
- **Equivalent circuit parameters** — Provide electrical parameters for an equivalent circuit model of the crystal.

### Series resonance, fs

Crystal series resonant frequency. This parameter is only visible when you select **Series resonance data** for the **Parameterization** parameter. The default value is 32.764 kHz.

### Parallel resonance, fa

Crystal parallel resonant frequency that corresponds to operating with a parallel load capacitance specified by the **Load capacitance, CL** parameter. This parameter is only visible when you select **Parallel resonance data** for the **Parameterization** parameter. The default value is 32.768 kHz.

### Motional inductance, L1

Inductance that represents the mechanical mass of the crystal. This parameter is only visible when you select **Equivalent circuit parameters** for the **Parameterization** parameter. The default value is 6.742e+03 H.

### R1 parameterization

Select one of the following methods for series resistance parameterization:

- Equivalent series resistance R1 — Provide the resistance value directly. This is the default method.
- Quality factor Q — Provide the quality factor that the block uses to calculate the resistance value.

This parameter is only visible when you select **Series resonance data** or **Parallel resonance data** for the **Parameterization** parameter.

### Quality factor, Q

Crystal quality factor. This parameter is only visible when you make one of the following selections:

- Series resonance data for the **Parameterization** parameter and Quality factor Q for the **R1 parameterization** parameter
- Parallel resonance data for the **Parameterization** parameter and Quality factor Q for the **R1 parameterization** parameter

The default value is 9e+04.

### Equivalent series resistance, R1

Motional damping term. This parameter is only visible when you make one of the following selections:

- Series resonance data for the **Parameterization** parameter and Equivalent series resistance R1 for the **R1 parameterization** parameter
- Parallel resonance data for the **Parameterization** parameter and Equivalent series resistance R1 for the **R1 parameterization** parameter
- Equivalent circuit parameters for the **Parameterization** parameter

The default value is 15 k $\Omega$ .

### Motional capacitance, C1

Capacitance that represents crystal mechanical stiffness under load. The default value is 0.0035 pF.

### Load capacitance, CL

Load capacitance that corresponds to the **Parallel resonance, fa** parameter value. This parameter is only visible when you select **Parallel resonance data** for the **Parameterization** parameter. The default value is 12.5 pF.

**Shunt capacitance, C0**

Electrical capacitance between the two crystal electrical connections. The parameter value must be greater than zero. The default value is 1.6 pF.

**Initial voltage**

The output voltage at the start of the simulation when the output current is zero. The default value is 0 V.

**Ports**

The block has the following ports:

+

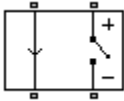
Positive electrical port

-

Negative electrical port

# Current-Controlled Switch

Model current-controlled switch with hysteresis



## Library

SPICE-Compatible Components/Passive Devices

## Description

The Current-Controlled Switch block represents the electrical characteristics of a switch whose state is controlled by the current through the input ports (the controlling current):

- When the controlling current is greater than the sum of the **Threshold current,  $I_T$**  and **Hysteresis current,  $I_H$**  parameter values, the switch is closed and has a resistance equal to the **On resistance,  $R_{ON}$**  parameter value.
- When the controlling current is less than the **Threshold current,  $I_T$**  parameter value minus the **Hysteresis current,  $I_H$**  parameter value, the switch is open and has a resistance equal to the **Off resistance,  $R_{OFF}$**  parameter value.
- When the controlling current is greater than or less than the **Threshold current,  $I_T$**  parameter value by an amount less than or equal to the **Hysteresis current,  $I_H$**  parameter value, the current is in the crossover region and the state of the switch remains unchanged.

## Basic Assumptions and Limitations

The block output resistance model is discontinuous during switching. The discontinuity might cause numerical issues. Try the following actions to resolve the issues:

- Set the **On resistance,  $R_{ON}$**  and **Off resistance,  $R_{OFF}$**  parameter values to keep the ratio  $R_{ON}/R_{OFF}$  as small as possible, and less than  $1e+12$ .

- Increase the **Hysteresis current, IH** parameter value to reduce switch chatter.
- Decrease the **Max step size** parameter value (in the Configuration Parameters block dialog box).

---

**Note** This increases the simulation time.

---

## Parameters

### Threshold current, IT

The current above which the block interprets the controlling current as HIGH. The default value is 0 A.

---

**Note** The controlling current must differ from the threshold current by at least the **Hysteresis current, IH** parameter value to change the state of the switch.

---

### Hysteresis current, IH

The amount by which the controlling current must exceed or fall below the **Threshold current, IT** parameter value to change the state of the switch. The default value is 0 A.

### On resistance, RON

The resistance of the switch when it is closed. The default value is 1  $\Omega$ .

### Off resistance, ROFF

The resistance of the switch when it is open. The default value is 1e+12  $\Omega$ .

### Initial switch state

Select one of the following options for the state of the switch at the start of the simulation:

- On — The switch is initially closed and its resistance value is equal to the **On resistance, RON** parameter value. This is the default option.
- Off — The switch is initially open and its resistance value is equal to the **Off resistance, ROFF** parameter value.

## Ports

The block has the following ports:

- +  
Positive electrical input and output ports.
- Negative electrical input and output ports.

## Current Limiter

Behavioral model of current limiter



## Library

Semiconductor Devices

## Description

The Current Limiter block provides a behavioral model of a current limiter. Use it to represent current limiting as found in power supplies and motor drives, and also to represent components that are used to limit inrush current.

The current limiting acts for both positive and negative currents. For applications where limiting is required in only one direction, you can augment the Current Limiter block with a series diode (blocks any reverse current) or parallel diode (no limiting in the reverse direction).

The block implements current limiting by using a hyperbolic tangent function:

$$i = i_{LIM} \tanh\left(\frac{4v}{v_{LIM}}\right) + g_{LIM}v$$

where:

- $i$  is the current through the component.
- $v$  is the voltage drop across the component.
- $i_{LIM}$  is the current limit.
- $v_{LIM}$  is the approximate voltage drop across the component when the current limit becomes active.



- $g_{LIM}$  is the rate of change of current with voltage drop when on the current limit (limit-state conductance).

When  $v = v_{LIM}$ , then

$$i = i_{LIM} \tanh(4) + g_{LIM}v = 0.9993i_{LIM} + g_{LIM}v$$

Therefore the current is approximately equal to the limit. Choose the value for  $g_{LIM}$  such that  $g_{LIM}v$  is small compared to  $i_{LIM}$  for the maximum expected voltage drop. This term is included in the block equation to improve numerical properties during simulation.

When choosing the value of  $v_{LIM}$ , consider that making it too small will require tight solver tolerances and small step sizes. In practice, current limiters can be implemented using a MOSFET and series source resistor, the gate-source voltage being driven by the series resistor. This implementation does not produce a sharp limit, similar to the tanh curve used in this block. You can use a datasheet plot of current against voltage to pick a suitable value for  $v_{LIM}$ .

## Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal port **H** on the block icon, and adds the **Thermal Port** tab to the block dialog box.

The thermal port model contains a thermal mass. The power dissipated by the current limiter, plus the heat flow into the thermal port, drives the thermal mass differential equation:

$$m \frac{dT}{dt} = P_{loss} + Q_H$$

where:

- $m$  is the thermal mass.
- $T$  is the thermal port temperature.
- $P_{loss}$  is the electrical loss,  $v \cdot i$ .
- $Q_H$  is the heat flow from the external network into the thermal port.

## Parameters

- “Parameters Tab” on page 1-396
- “Thermal Port Tab” on page 1-396

### Parameters Tab

#### Current limit

The maximum current magnitude. The default value is 1 A.

#### Voltage drop when current starts to limit

When the voltage drop is equal to this value, then the current is limited at 0.9993 times the current limit value. The default value is 0.1 V.

#### Limit-state conductance

When the current is limited, this parameter defines the rate of change of current with voltage drop if the current is driven harder onto the limit. The default value is  $1e-3$  1/ $\Omega$ .

### Thermal Port Tab

This tab appears only for blocks with exposed thermal ports. For more information, see “Thermal Port” on page 1-395.

#### Thermal mass

The heat energy required to raise the temperature by one degree. The default value is 100 J/K.

#### Initial temperature

The temperature at the start of simulation. The default value is 25 C.

## Ports

The block has the following ports:

+

Electrical conserving port associated with the current limiter positive terminal

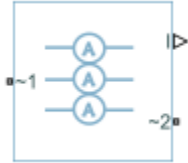
-

Electrical conserving port associated with the current limiter negative terminal

**Introduced in R2015a**

## Current Sensor

Measure phase currents in three-phase system



## Library

Simscape / Electrical / Power Systems / Sensors

## Description

The Current Sensor block represents an ideal three-phase current sensor. The block measures each of the three currents flowing from port ~1 to port ~2 and outputs a single three-element, physical signal vector. Each element of the physical signal output vector is proportional to the current in its respective phase.

## Ports

The block has the following ports:

~1

Expandable three-phase port.

~2

Expandable three-phase port.

I

Three-element physical signal vector output port associated with the phase currents.

## **See Also**

### **Topics**

“Expand and Collapse Three-Phase Ports on a Block”

**Introduced in R2013b**

## Current Source

Simulate current source with DC, AC, and noise components



## Library

Sources

## Description

The Current Source block implements a current source with DC, AC, and noise components. The current flowing through the source from the - terminal to the + terminal is given by:

$$i = i_{DC} + i_{AC} \sin(2\pi ft + \phi) + i_N$$

where:

- $i_{DC}$  is the steady-state DC current component.
- $i_{AC}$  is the amplitude of the AC current component.
- $f$  is the frequency of the AC component.
- $\phi$  is the phase offset of the AC component.
- $i_N$  is the noise current.

You can configure your source as DC-only, AC-only, or a combination of both. By default, both AC and DC components are set to 0. Define the AC/DC current by specifying nonzero parameter values after placing the block in your model.

The noise component is also optional. If you set the **Noise mode** parameter to **Enabled**, then the added noise current is given by:

$$i_N = \sqrt{P_i/2} \frac{N(0,1)}{\sqrt{h}}$$

where:

- $P_i$  is the single-sided noise power spectral density for a 1 ohm load, in  $A^2/Hz$ .
- $N$  is a Gaussian random number with zero mean and standard deviation of one.
- $h$  is the sampling interval.

By default, the **Noise mode** parameter is set to **Disabled**, and the current source generates no thermal noise.

## Noise Options

The block generates Gaussian noise by using the Random Number source in the Simscape Foundation library. You can control the random number seed by setting the **Repeatability** parameter:

- **Not repeatable** — Every time you simulate your model, the block resets the random seed using the MATLAB random number generator:

```
seed = randi(2^32-1);
```

- **Repeatable** — The block automatically generates a seed value and stores it inside the block, to always start the simulation with the same random number. This auto-generated seed value is set when you add a Current Source block from the block library to the model. When you make a new copy of the Current Source block from an existing one in a model, a new seed value is generated. The block sets the value using the MATLAB random number generator command shown above.
- **Specify seed** — If you select this option, the additional **Seed** parameter lets you directly specify the random number seed value.

## Basic Assumptions and Limitations

Simulating with noise enabled slows down simulation. Choose the sample time ( $h$ ) so that noise is generated only at frequencies of interest, and not higher.

## Parameters

- “DC & AC Components Tab” on page 1-402
- “Noise Tab” on page 1-402

### DC & AC Components Tab

#### DC current

The DC component of the output current. The default value is 0 A. Enter a nonzero value to add a DC component to the current source.

#### AC current peak amplitude

Amplitude of the AC component of the output current. The default value is 0 A. Enter a nonzero value to add an AC component to the current source.

#### AC current phase shift

Phase offset of the AC component of the output current. The default value is 0 degrees.

#### AC current frequency

Frequency of the AC component of the output current. The default value is 60 Hz.

### Noise Tab

#### Noise mode

Select the noise option:

- **Disabled** — No noise is produced by the current source. This is the default.
- **Enabled** — The current source generates thermal noise, and the associated parameters become visible on the **Noise** tab.

#### Power spectral density

The single-sided spectrum noise power. Strictly-speaking, this is a density function for the square of the current, commonly thought of as a power into a 1 ohm load, and therefore units are  $A^2/Hz$ . To avoid this unit ambiguity, some datasheets quote noise current as a noise density with units of  $A/\sqrt{Hz}$ . In this case, you should enter the square of the noise density quoted in the datasheet as the parameter value. The default value is 0  $A^2/Hz$ .



### Sample time

Defines the rate at which the noise source is sampled. Choose it to reflect the frequencies of interest in your model. Making the sample time too small will unnecessarily slow down your simulation. The default value is  $1e-3$  s.

### Repeatability

Select the noise control option:

- **Not repeatable** — The random sequence used for noise generation is not repeatable. This is the default.
- **Repeatable** — The random sequence used for noise generation is repeatable, with a system-generated seed.
- **Specify seed** — The random sequence used for noise generation is repeatable, and you control the seed by using the **Seed** parameter.

### Auto-generated seed used for repeatable option

Random number seed stored inside the block to make the random sequence repeatable. This parameter is visible only if you select **Repeatable** for the **Repeatability** parameter. The parameter value is automatically generated using the MATLAB random number generator command. You can modify this parameter value, but it gets overwritten by a new random value if you copy the block to another block in the model. Therefore, if you want to control the seed of the random sequence, use the **Specify seed** option for the **Repeatability** parameter and specify the desired seed value using the **Seed** parameter.

### Seed

Random number seed used by the noise random number generator. This parameter is visible only if you select **Specify seed** for the **Repeatability** parameter. The default value is 0.

## Ports

The block has the following ports:

+

Positive electrical port

-

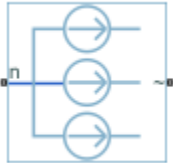
Negative electrical port

## **See Also**

Resistor | Voltage Source

## Current Source (Three-Phase)

Ideal three-phase current source



### Library

Simscape / Electrical / Power Systems / Sources

### Description

The Current Source (Three-Phase) block models an ideal three-phase current source that maintains sinusoidal currents of the specified magnitude through its terminals, independent of the voltage across the source.

The output current is defined by the following equations:

$$I_0 = \sqrt{2}i_{phase\_rms}$$

$$i_a = I_0 \sin(2\pi ft + \varphi)$$

$$i_b = I_0 \sin(2\pi ft + \varphi - 120^\circ)$$

$$i_c = I_0 \sin(2\pi ft + \varphi + 120^\circ),$$

where:

- $I_0$  is the peak phase current.

- $i_{\text{phase\_rms}}$  is the RMS phase current.
- $i_a, i_b, i_c$  are the respective phase currents.
- $f$  is the frequency.
- $\varphi$  is the phase shift.
- $t$  is the time.

The arrow indicates the positive direction of the current flow. The source has a wye configuration, and port n provides a connection to the center of the wye. Port ~ is an expandable three-phase port representing the three phases, a, b, and c.

## Parameters

### Current (phase RMS)

RMS phase current. The default value is  $100/\sqrt{2}$ , or 70.7107, A.

### Phase shift

Phase shift in angular units. The default value is 0 deg.

### Frequency

Current frequency, specified in Hz or units directly convertible to Hz (where Hz is defined as 1/s). For example, kHz and MHz are valid units, but rad/s is not. The default value is 60 Hz.

## Ports

The block has the following ports:

~

Expandable three-phase port

n

Electrical conserving port associated with the center of the wye

## See Also

Battery | Controlled Current Source (Three-Phase) | Supercapacitor | Voltage Source (Three-Phase)

## **Topics**

“Expand and Collapse Three-Phase Ports on a Block”

**Introduced in R2013b**

## DC Current Controller

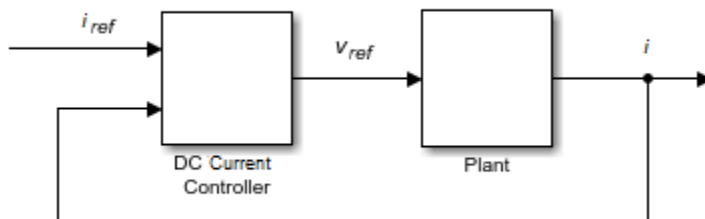
Discrete-time PI-based DC current control

**Library:** Simscape / Electrical / Power Systems / Control / General Machine Control



## Description

The DC Current Controller block implements a discrete-time proportional-integral (PI) DC voltage controller. The block can implement zero cancellation in the feedforward path. To avoid saturation of the integral gain, the block can implement anti-windup gain.



## Equations

The equation that the DC Current Controller block uses to calculate the reference voltage is

$$v_{ref} = \left( K_p + K_i \frac{T_s z}{z-1} \right) (i_{ref} - i),$$

where:

- $v_{ref}$  is the reference voltage.
- $K_p$  is the proportional gain.
- $K_i$  is the integral gain.
- $T_s$  is the sample time.
- $i_{ref}$  is the reference current.
- $i$  is the measured current.

The PI control calculation yields a zero in the closed-loop transfer function. To cancel the zero, the block uses this discrete-time zero-cancellation transfer function:

$$G_{ZC}(z) = \frac{\frac{T_s K_i}{K_p}}{z + \left( \frac{T_s - \frac{K_p}{K_i}}{\frac{K_p}{K_i}} \right)}.$$

To avoid saturation of the integrator output, the block uses an anti-windup mechanism. The integrator gain is then equal to

$$K_i + K_{aw} (v_{ref\_sat} - v_{ref\_unsat}),$$

where:

- $K_{aw}$  is the anti-windup gain.
- $v_{ref\_sat}$  is the saturated reference voltage signal, which the block calculates as

$$v_{ref\_sat} = \min \left( \max (v_{ref\_unsat}, v_{min}), v_{max} \right),$$

where:

- $v_{ref\_unsat}$  is the unsaturated reference voltage signal.
- $v_{min}$  is the lower limit for the output voltage. For positive voltage only,  $v_{min} = 0$ . For positive and negative voltage,  $v_{min} = -v_{max}$
- $v_{max}$  is the upper limit for the output voltage.

## Ports

### Input

#### **iRef — Reference current**

scalar

Desired output current for the plant.

Data Types: single | double

#### **i — Measured current**

scalar

Measured current for the plant.

Data Types: single | double

#### **v<sub>max</sub> — Maximum DC voltage**

scalar

Maximum DC output voltage for the plant.

Data Types: single | double

#### **Reset — External reset**

scalar

External reset signal (rising edge) for the integrator.

Data Types: single | double



## Output

### **vRef — Reference DC voltage**

scalar

Desired DC output voltage for the plant.

Data Types: `single` | `double`

## Parameters

### **Proportional gain — Controller proportional gain, $K_p$**

1 (default) | positive scalar

Proportional gain,  $K_p$ , of the controller.

### **Integral gain — Integral gain, $K_i$**

5 (default) | positive scalar

Integral gain,  $K_i$ , of the controller.

### **Anti-windup gain — Anti-windup gain, $K_{aw}$**

1 (default) | positive scalar

Anti-windup gain,  $K_{aw}$ , of the controller.

### **Voltage limitation — Negative voltage restriction**

Positive and negative voltage (default) | Positive voltage only

Allow for both positive and negative DC voltage output or limit the output to positive DC voltage.

### **Sample time (-1 for inherited) — Block sample time**

-1 (default) | positive scalar

Time, in s, between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

If this block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly using a positive scalar.

## Dependencies

If you set **Sample time (-1 for inherited)** to -1 and select the **Enable zero cancellation** option, the **Discretization sample time** parameter becomes visible.

## Discretization sample time — Sample time for discretization

0.001 (default) | positive scalar

Time, in s, between consecutive discretizations. Discretization is required for zero cancellation.

## Dependencies

This parameter is only visible when both these conditions are met:

- **Sample time** is set to -1.
- **Enable zero cancellation** is selected.

## Enable zero cancellation — Feedforward zero cancellation

off (default) | on

Option to use zero cancellation on the feedforward path.

## Dependencies

If you select the **Enable zero cancellation** option and set **Sample time (-1 for inherited)** to -1, the **Discretization sample time** parameter becomes visible.

## See Also

DC Voltage Controller | DC-DC Voltage Controller | PWM Generator

**Introduced in R2018a**

# DC Current Source

Model constant current source



## Library

SPICE-Compatible Components/Sources

## Description

The DC Current Source block represents a constant current source whose output current value is independent of the voltage across its terminals.

The block uses a small conductance internally to prevent numerical simulation issues. The conductance connects the + and - ports of the device and has a conductance *GMIN*:

- By default, *GMIN* matches the **Minimum conductance GMIN** parameter of the SPICE Environment Parameters block, whose default value is 1e-12.
- To change *GMIN*, add a SPICE Environment Parameters block to your model and set the **Minimum conductance GMIN** parameter to the desired value.

## Parameters

### Constant value, DC

The value of the DC output current. The default value is 0 A.

## Ports

The block has the following ports:

+

Positive electrical voltage.

-

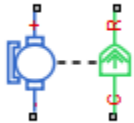
Negative electrical voltage.

## **See Also**

DC Voltage Source

## DC Motor

DC motor model with electrical and torque characteristics

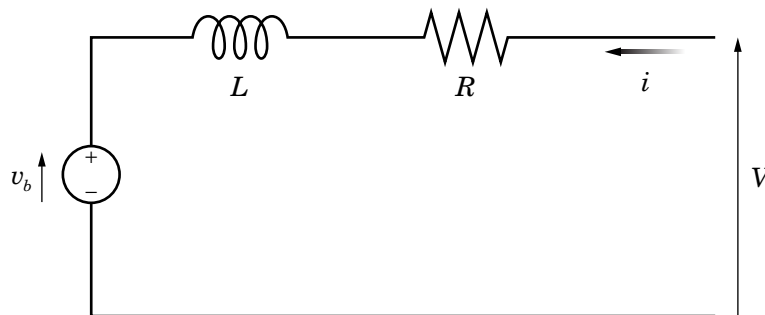


## Library

Rotational Actuators

## Description

The DC Motor block represents the electrical and torque characteristics of a DC motor using the following equivalent circuit model:



You specify the equivalent circuit parameters for this model when you set the **Model parameterization** parameter to **By equivalent circuit parameters**. The resistor  $R$  corresponds to the resistance you specify in the **Armature resistance** parameter. The inductor  $L$  corresponds to the inductance you specify in the **Armature inductance** parameter. The permanent magnets in the motor induce the following back emf  $v_b$  in the armature:

$$v_b = k_v \omega$$

where  $k_v$  is the **Back-emf constant** and  $\omega$  is the angular velocity. The motor produces the following torque, which is proportional to the motor current  $i$ :

$$T_E = k_t i$$

where  $k_t$  is the **Torque constant**. The DC Motor block assumes that there are no electromagnetic losses. This means that mechanical power is equal to the electrical power dissipated by the back emf in the armature. Equating these two terms gives:

$$T_E \omega = v_b i$$

$$k_t i \omega = k_v \omega i$$

$$k_v = k_t$$

As a result, you specify either  $k_v$  or  $k_t$  in the block dialog box.

The torque-speed characteristic for the DC Motor block is related to the parameters in the preceding figure. When you set the **Model parameterization** parameter to **By stall torque & no-load speed** or **By rated power, rated speed & no-load speed**, the block solves for the equivalent circuit parameters as follows:

- 1 For the steady-state torque-speed relationship,  $L$  has no effect.
- 2 Sum the voltages around the loop and rearrange for  $i$ :

$$i = \frac{V - v_b}{R} = \frac{V - k_v \omega}{R}$$

- 3 Substitute this value of  $i$  into the equation for torque:

$$T_E = \frac{k_t}{R} (V - k_v \omega)$$

When you set the **Model parameterization** parameter to **By stall torque & no-load speed**, the block uses the preceding equation to determine values for  $R$  and  $k_t$  (and equivalently  $k_v$ ).

When you set the **Model parameterization** parameter to **By rated power, rated speed & no-load speed**, the block uses the rated speed and power to calculate the rated torque. The block uses the rated torque and no-load speed values in the preceding equation to determine values for  $R$  and  $k_t$ .

The block models motor inertia  $J$  and damping  $\lambda$  for all values of the **Model parameterization** parameter. The resulting torque across the block is:

$$T = \frac{k_t}{R}(V - k_v\omega) - J\dot{\omega} - \lambda\omega$$

It is not always possible to measure rotor damping, and rotor damping is not always provided on a manufacturer datasheet. An alternative is to use the no-load current to infer a value for rotor damping.

For no-load, the electrically-generated mechanical torque must equal the rotor damping torque:

$$k_t i_{no-load} = \lambda \omega_{no-load}$$

where  $i_{no-load}$  is the no-load current. If you select **By no-load current** for the **Rotor damping parameterization** parameter, then this equation is used in addition to the torque-speed equation to determine values for  $\lambda$  and the other equation coefficients.

The value for rotor damping, whether specified directly or in terms of no-load current, is taken into account when determining equivalent circuit parameters for **Model parameterization** options **By stall torque and no-load speed** and **By rated power, rated speed and no-load speed**.

When a positive current flows from the electrical + to - ports, a positive torque acts from the mechanical C to R ports.

## Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal port H on the block icon, and adds the **Temperature Dependence** and **Thermal Port** tabs to the block dialog box.

Use the thermal port to simulate the effects of copper resistance losses that convert electrical power to heat. For more information on using thermal ports and on the **Temperature Dependence** and **Thermal Port** tab parameters, see “Simulating Thermal Effects in Rotational and Translational Actuators”.

## Parameters

- “Electrical Torque Tab” on page 1-418
- “Mechanical Tab” on page 1-420

## Electrical Torque Tab

### Model parameterization

Select one of the following methods for block parameterization:

- **By equivalent circuit parameters** — Provide electrical parameters for an equivalent circuit model of the motor. This is the default method.
- **By stall torque & no-load speed** — Provide torque and speed parameters that the block converts to an equivalent circuit model of the motor.
- **By rated power, rated speed & no-load speed** — Provide power and speed parameters that the block converts to an equivalent circuit model of the motor.

### Armature resistance

Resistance of the conducting portion of the motor. This parameter is only visible when you select **By equivalent circuit parameters** for the **Model parameterization** parameter. The default value is 3.9  $\Omega$ .

### Armature inductance

Inductance of the conducting portion of the motor. If you do not have information about this inductance, set the value of this parameter to a small, nonzero number. The default value is 1.2e-05 H.

### Define back-emf or torque constant

Indicate whether you will specify the motor's back-emf constant or torque constant. When you specify them in SI units, these constants have the same value, so you only specify one or the other in the block dialog box. This parameter is only visible when you select **By equivalent circuit parameters** for the **Model parameterization** parameter. The default value is **Specify back-emf constant**.



**Back-emf constant**

The ratio of the voltage generated by the motor to the speed at which the motor is spinning. The default value is  $7.2e-05$  V/rpm. This parameter is only visible when you select **Specify back-emf constant** for the **Define back-emf or torque constant** parameter.

**Torque constant**

The ratio of the torque generated by the motor to the current delivered to it. This parameter is only visible when you select **Specify torque constant** for the **Define back-emf or torque constant** parameter. The default value is  $6.876e-04$  N\*m/A.

**Stall torque**

The amount of torque generated by the motor when the speed is approximately zero. This parameter is only visible when you select **By stall torque & no-load speed** for the **Model parameterization** parameter. The default value is  $2.4e-04$  N\*m.

**No-load speed**

Speed of the motor when not driving a load. This parameter is only visible when you select **By stall torque & no-load speed** or **By rated power, rated speed & no-load speed** for the **Model parameterization** parameter. The default value is  $1.91e+04$  rpm.

**Rated speed (at rated load)**

Motor speed at the rated mechanical power level. This parameter is only visible when you select **By rated power, rated speed & no-load speed** for the **Model parameterization** parameter. The default value is  $1.5e+04$  rpm.

**Rated load (mechanical power)**

The mechanical power the motor is designed to deliver at the rated speed. This parameter is only visible when you select **By rated power, rated speed & no-load speed** for the **Model parameterization** parameter. The default value is  $0.08$  W.

**Rated DC supply voltage**

The voltage at which the motor is rated to operate. This parameter is only visible when you select **By stall torque & no-load speed** or **By rated power, rated speed & no-load speed** for the **Model parameterization** parameter. The default value is  $1.5$  V.

**Rotor damping parameterization**

Select one of the following methods to specify rotor damping:

- **By damping value** — Specify a value for rotor damping directly, by using the **Rotor damping** parameter on the **Mechanical** tab. This is the default.
- **By no-load current** — The block calculates rotor damping based on the values that you specify for the **No-load current** and **DC supply voltage when measuring no-load current** parameters. If you select this option, the **Rotor damping** parameter is not available on the **Mechanical** tab.

### **No-load current**

Specify the no-load current value, to be used for calculating the rotor damping. This parameter is only visible when you select **By no-load current** for the **Rotor damping parameterization** parameter. The default value is 0 A.

### **DC supply voltage when measuring no-load current**

Specify the DC supply voltage corresponding to the no-load current value, to be used for calculating the rotor damping. This parameter is only visible when you select **By no-load current** for the **Rotor damping parameterization** parameter. The default value is 1.5 V.

## **Mechanical Tab**

### **Rotor inertia**

Resistance of the rotor to change in motor motion. The default value is 0.01 g\*cm<sup>2</sup>. The value can be zero.

### **Rotor damping**

Energy dissipated by the rotor. This parameter is only visible when you select **By damping value** for the **Rotor damping parameterization** parameter on the **Electrical** tab. The default value is 1e-08 N\*m/(rad/s). The value can be zero.

### **Initial rotor speed**

Speed of the rotor at the start of the simulation. The default value is 0 rpm.

## **Ports**

The block has the following ports:

+

Positive electrical input

- Negative electrical input
- C Mechanical rotational conserving port
- R Mechanical rotational conserving port

## Examples

See the following DC motor examples:

- Brushless DC Motor
- Linear Electrical Actuator (Motor Model)
- Linear Electrical Actuator with Control

## References

[1] Bolton, W. Mechatronics: Electronic Control Systems in Mechanical and Electrical Engineering, 3rd edition Pearson Education, 2004.

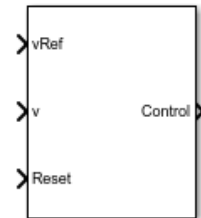
## See Also

Induction Motor | Servomotor | Shunt Motor | Universal Motor

## DC Voltage Controller

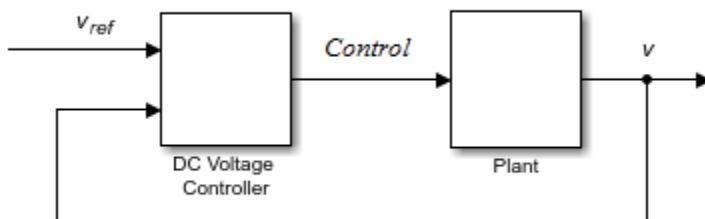
Discrete-time PI-based DC voltage control with feedforward zero cancellation and anti-windup gain

**Library:** Simscape / Electrical / Power Systems / Control / General Machine Control



### Description

The DC Voltage Controller block implements discrete-time PI-based DC voltage control. The block can implement zero cancellation in the feedforward path. To avoid saturation of the integral gain, the block can implement anti-windup gain.



### Equations

The equation that the DC Voltage Controller block uses to calculate the control signal is

$$control = \left( K_p + K_i \frac{T_s z}{z-1} \right) (v_{ref} - v),$$

where:

- $control$  is the control signal, which is expressed as a duty cycle or a current.
- $K_p$  is the proportional gain.
- $K_i$  is the integral gain.
- $T_s$  is the sample time.
- $v_{ref}$  is the reference voltage.
- $v$  is the measured voltage.

The PI control calculation yields a zero in the closed-loop transfer function. To cancel the zero, the block uses this discrete-time zero-cancellation transfer function:

$$G_{ZC}(z) = \frac{\frac{T_s K_i}{K_p}}{z + \left( \frac{T_s - \frac{K_p}{K_i}}{\frac{K_p}{K_i}} \right)}.$$

To avoid saturation of the integrator output, the block uses an anti-windup mechanism. The integrator gain is then equal to

$$K_i + K_{aw} (control_{sat} - control_{unsat}),$$

where:

- $K_{aw}$  is the anti-windup gain.
- $control_{sat}$  is the saturated control signal, which the block calculates as  $control_{sat} = \min(\max(control_{unsat}, control_{min}), control_{max})$ ,

where:

- $control_{unsat}$  is the unsaturated control signal.
- $control_{min}$  is the lower limit for the control signal.
- $v_{max}$  is the upper limit for the control signal.

## Ports

### Input

#### **vRef — Reference DC voltage**

scalar

Desired DC output voltage for the plant.

Data Types: `single` | `double`

#### **v — Measured DC voltage**

scalar

Measured DC output voltage for the plant.

Data Types: `single` | `double`

#### **Reset — External reset**

scalar

External reset signal (rising edge) for the integrator.

Data Types: `single` | `double`

### Output

#### **Control — Control signal**

scalar

Control signal, *control*, expressed as a duty cycle or a current.

Data Types: `single` | `double`

## Parameters

### Proportional gain — Proportional gain, $K_p$

0.1 (default) | positive scalar

Proportional gain,  $K_p$ , of the controller.

### Integral gain — Integral gain, $K_i$

50 (default) | positive scalar

Integral gain,  $K_i$ , of the controller.

### Anti-windup gain — Anti-windup gain, $K_{aw}$

1 (default) | positive scalar

Anti-windup gain,  $K_{aw}$ , of the controller.

### Control action upper limit — Upper limit for the control signal, $control_{max}$

1 (default) | positive scalar

Upper limit for the **Control** output signal. The value must be greater than the value of the **Control action lower limit** parameter.

### Control action lower limit — Lower limit for the control signal, $control_{min}$

0 (default) | non-negative scalar

Lower limit for the **Control** output signal.

### Sample time (-1 for inherited) — Block sample time

-1 (default) | positive scalar

Time, in s, between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

If this block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly using a positive scalar.

### Discretization sample time — Sample time for discretization

0.001 (default) | positive scalar

Time, in s, between consecutive discretizations.

**Time constant voltage filter — DC voltage filter time constant,  $\tau$**   
0.001 (default) | positive scalar

Time constant,  $\tau$ , for the DC voltage filter.

**Dependencies**

The **Time constant voltage filter** parameter is only visible when the **Filter DC voltage** checkbox is selected.

**Enable zero cancellation — Feedforward zero cancellation**  
off (default) | on

Option to use zero cancellation on the feedforward path.

**Dependencies**

If you select the **Enable zero cancellation** option and set **Sample time (-1 for inherited)** to -1, the **Discretization sample time** parameter becomes visible.

**Filter DC voltage — DC voltage filter option**  
on (default) | off

To enable the filter on voltage measurement path, select the checkbox. To disable the filter, clear the check box.

**Dependencies**

The **Time constant voltage filter** parameter is only visible when the **Filter DC voltage** check box is selected.

## See Also

**Blocks**

DC Current Controller | DC-DC Voltage Controller

**Introduced in R2018a**



# DC Voltage Source

Model constant voltage source



## Library

SPICE-Compatible Components/Sources

## Description

The DC Voltage Source block represents a constant voltage source whose output voltage value is independent of the current through the source.

## Parameters

### Constant value, DC

The value of the DC output voltage. The default value is 0 V.

## Ports

The block has the following ports:

+

Positive electrical voltage.

-

Negative electrical voltage.

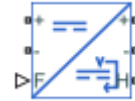
## **See Also**

DC Current Source

# DC-DC Converter

Behavioral model of power converter

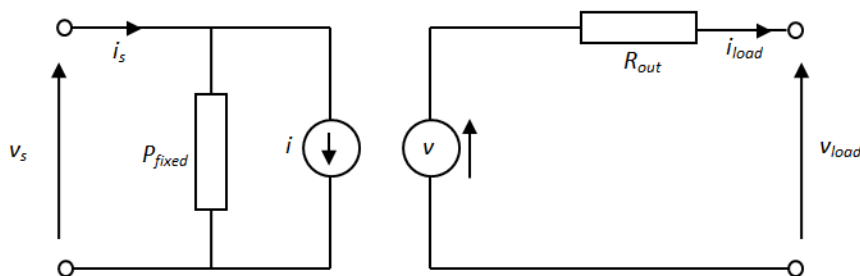
**Library:** Simscape / Electrical / Electronics and Mechatronics / Sources



## Description

The DC-DC Converter block represents a behavioral model of a power converter. This power converter regulates voltage on the load side. To balance input power, output power, and losses, the required amount of power is drawn from the supply side. Alternatively, the converter can support regenerative power flow from load to supply.

This circuit illustrates the converter's behavior.



The  $P_{\text{fixed}}$  component draws a constant power and corresponds to converter losses that are independent of load current. The power drawn is set by the **Fixed converter losses independent of loading** parameter value. The resistor  $R_{\text{out}}$  corresponds to losses that increase with load current, and is determined from the value you specify for the **Percentage efficiency at rated output power** parameter.

The voltage source is defined by the following equation:

$$v = v_{\text{ref}} - i_{\text{load}} D + i_{\text{load}} R_{\text{out}}$$

Where:

- $v_{\text{ref}}$  is the load side voltage set point, as defined by the value you specify for the **Output voltage reference demand** parameter.
- $D$  is the value you specify for the **Output voltage droop with output current** parameter. Having a separate value for droop makes control of how output voltage varies with load independent of load-dependent losses. Instead of specifying  $D$  directly, you can specify the **Percent voltage droop at rated load**.

The current source value  $i$  is calculated so that the power flowing in to the converter equals the sum of the power flowing out plus the converter losses.

To specify the converter behavior when the voltage presented by the load is higher than the converter output voltage reference demand, use the **Power direction** parameter:

- **Unidirectional power flow from supply to regulated side** — Current is blocked by the off-state diode, and the current source current  $i$  is zero. Set the conductance of this diode using the **Diode off-state conductance** parameter.
- **Bidirectional power flow** — Power is transmitted to the supply side, and  $i$  becomes negative.

Optionally, the block can include voltage regulation dynamics. If you select **Specify voltage regulation time constant** for the **Dynamics** parameter, then a first-order lag is added to the equation defining the voltage source value. With the dynamics enabled, a load step change results in a transient change in output voltage, the time constant being defined by the **Voltage regulation time constant** parameter.

## Simulating Faults

You can use the physical signal input port F to simulate both DC supply failure and converter failure. This type of event cannot be simulated by simply disconnecting the DC supply, for example by opening a switch, because the average value model will attempt to increase supply-side current to unrealistic values as supply-side voltage drops.

You control the behavior in response to the physical signal fault input F by the parameters on the **Faults** tab of the block dialog box. With the default parameter settings:

- **Fault condition** is Output open circuit if  $F \geq \text{Fault threshold}$
- **Fault threshold** is 0.5

You can leave the input F unconnected and the converter will work normally.

If a signal is connected to port F, then the block operates according to the parameter settings on the **Faults** tab. For example, if **Fault condition** is **Output open circuit** if  $F \geq \text{Fault threshold}$ , then when the signal at port F rises above the **Fault threshold** value, the converter stops operating. Zero current is taken from the supply side, and zero current is supplied to the load side.

## Modeling thermal effects

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal port **H** on the block icon, and adds the **Thermal Port** tab to the block dialog box.

The block transfers heat generated from electrical losses through a Controlled Heat Flow Rate Source to a Thermal Mass block. The electrical properties of the block do not change with temperature. Specify the thermal properties for this block using the parameters **Thermal mass** and **Initial temperature**.

## Assumptions

- The two electrical networks connected to the supply-side and regulated-side terminals must each have their own Electrical Reference block.
- The supply-side equation defines a power constraint on the product of the voltage ( $v_s$ ) and the current ( $i_s$ ). For simulation, the solver must be able to uniquely determine  $v_s$ . To ensure that the solution is unique, the block implements two assertions:
  - $v_s > 0$  — This assertion ensures that the sign of  $v_s$  is uniquely defined
  - $i_s < i_{\max}$  — This assertion deals with the case when the voltage supply to the block has a series resistance

When there is a series resistance, there are two possible steady-state solutions for  $i_s$  that satisfy the power constraint, the one with the smaller magnitude being the desired one. You should set the value for the **Maximum expected supply-side current** parameter ( $i_{\max}$ ) to be larger than the expected maximum current. This will ensure that when the model is initialized the initial current does not start at the undesired solution.

## Ports

### Conserving

#### **1+ — Input positive terminal**

electrical

Electrical conserving port associated with the positive terminal of the input side.

#### **1- — Input negative terminal**

electrical

Electrical conserving port associated with the negative terminal of the input side.

#### **2+ — Output positive terminal**

electrical

Electrical conserving port associated with the positive terminal of the output side.

#### **2- — Output negative terminal**

electrical

Electrical conserving port associated with the negative terminal of the output side.

#### **F — Fault signal**

physical

Physical signal input port that provides the external fault trigger signal. You can leave this port unconnected if the parameters on the **Faults** tab of the block dialog box are set to their default values.

#### **H — Thermal mass**

thermal

Thermal conserving port that represents the thermal mass. When you expose this port, provide additional parameters to define battery behavior at a second temperature. For more information, see the “Thermal” on page 1-436 parameters.

#### **Dependencies**

To expose this port, right-click the block and select **Simscape > Block choices > Show thermal port**.

# Parameters

## Main

### Output voltage reference demand — Voltage set point

10 V (default)

The set point for the voltage regulator, and the output voltage value when there is no output current.

### Rated output power — Rated power

10 W (default)

Output power for which the percentage efficiency value is given. This parameter is also used to calculate droop,  $D$ , if droop is specified as a percentage.

### Droop parameterization — Droop model

By voltage droop with output current (default) | By percent voltage droop at rated load

Select one of the following methods for droop parameterization:

- By voltage droop with output current — Specify the absolute value of droop,  $D$ . This is the default option.
- By percent voltage droop at rated load — Specify droop,  $D$ , as a percentage at rated load.

### Output voltage droop with output current — Voltage droop at 1 A

0.1 V/A (default)

The number of volts that the output voltage will drop from the set point for an output current of 1 A.

### Dependencies

This parameter is visible only if you select By voltage droop with output current for the **Droop parameterization** parameter.

### Percent voltage droop at rated load — Voltage droop at rated load

2 (default)

The percentage by which voltage drops compared to the nominal output voltage when supplying the rated load.

### **Dependencies**

This parameter is visible only if you select **By percent voltage droop at rated load** for the **Droop parameterization** parameter.

### **Power direction — Power flow direction**

Unidirectional power flow from supply to regulated side (default) |  
Bidirectional power flow

Select one of the following methods for the direction of power conversion:

- **Unidirectional power flow from supply to regulated side** — Most small power regulators are unidirectional. This is the default option.
- **Bidirectional power flow** — Larger power converters can be bidirectional, for example, converters used in electric vehicles to allow regenerative braking.

### **Diode off-state conductance — Unidirectional diode**

1e-8 1/Ohm (default)

Ideal diode incorporated on the output side to prevent current from being forced into the converter in the unidirectional configuration.

### **Maximum expected supply-side current — Maximum supply current**

2 A (default)

Set this value to a value greater than the maximum expected supply-side current in your model. Using twice the expected maximum current is generally sufficient. For more information, see “Assumptions” on page 1-431.

## **Losses**

### **Percentage efficiency at rated output power — Rated efficiency**

80 (default)

Efficiency as defined by 100 times the output load power divided by the input supply power.

### **Fixed converter losses independent of loading — Constant losses**

1 W (default)



Power drawn by the  $P_{\text{fixed}}$  component in the equivalent circuit diagram, which corresponds to converter losses that are independent of load current.

## Dynamics

### Dynamics — Dynamics model

No dynamics (default) | Specify voltage regulation time constant

Specify whether to include voltage regulation dynamics:

- **No dynamics** — Do not consider the voltage regulation dynamics.
- **Specify voltage regulation time constant** — Add a first-order lag to the equation defining the voltage source value. With the dynamics enabled, a load step change results in a transient change in output voltage.

### Voltage regulation time constant — Dynamics time constant

0.02 s (default)

Time constant associated with voltage transients when the load current is stepped.

### Dependencies

This parameter is only visible when you select **Specify voltage regulation time constant** for the **Dynamics** parameter.

### Initial output voltage demand — Initial demand voltage

10 V (default)

Value of  $v_{\text{ref}}$  at time zero. Normally,  $v_{\text{ref}}$  is defined by the **Output voltage reference demand** parameter. However, if you want to initialize the model with no transients when delivering a steady-state load current, you can set the initial  $v_{\text{ref}}$  value by using this parameter, and increase it accordingly to take account of output resistance and droop.

### Dependencies

This parameter is only visible when you select **Specify voltage regulation time constant** for the **Dynamics** parameter.

## Faults

### **Fault condition — Fault model**

Output open circuit if  $F \geq$  Fault threshold (default) | Output open circuit if  $F \leq$  Fault threshold

Selects whether the converter is disabled by a signal that is high or low:

- **Output open circuit if  $F \geq$  Fault threshold** — Converter is disabled if the signal at port **F** rises above the threshold value. This is the default option.
- **Output open circuit if  $F \leq$  Fault threshold** — Converter is disabled if the signal at port **F** falls below the threshold value.

### **Fault threshold — Fault limit**

0.5 (default)

Threshold value used to detect a fault.

## Thermal

### **Thermal mass — Thermal mass associated with the thermal port**

100 J/K (default)

Thermal mass associated with thermal port **H**. It represents the energy required to raise the temperature of the thermal port by one degree.

### **Initial temperature — Initial temperature at the thermal port**

298.15 K (default)

Initial temperature associated with thermal port **H**.

## See Also

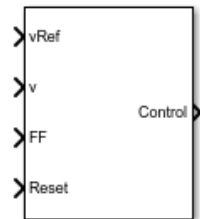
Current Source | Voltage Source

**Introduced in R2012b**

# DC-DC Voltage Controller

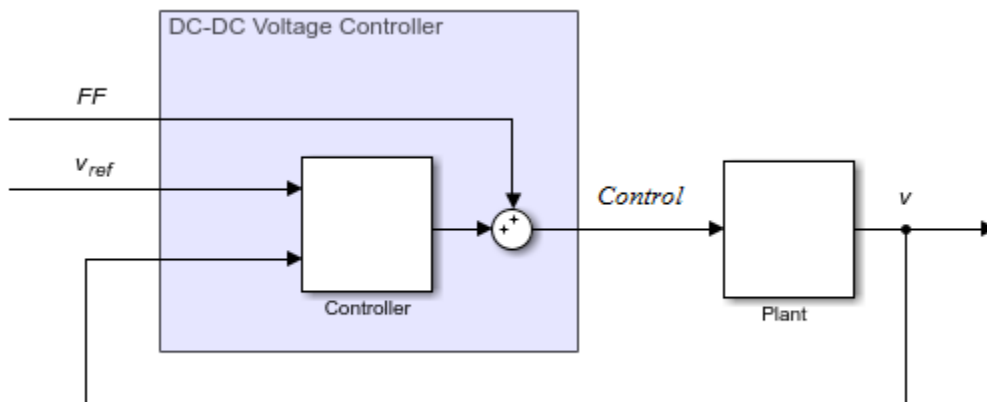
Discrete-time PI-based DC-DC voltage control with feedforward and anti-windup gain

**Library:** Simscape / Electrical / Power Systems / Control / General Machine Control



## Description

The DC-DC Voltage Controller block implements discrete-time proportional-integral (PI) DC-DC voltage control with feedforward,  $FF$ . The feedforward input optimizes the transient response. The block can output a duty cycle or a current control signal. To avoid saturation of the integral gain, the block can implement anti-windup gain.



## Equations

The equation that the DC-DC Voltage Controller block uses to calculate the control signal is

$$control = \left( K_p + K_i \frac{T_s z}{z-1} \right) (v_{ref} - v) + FF,$$

where:

- $control$  is the control signal, expressed as a duty cycle or a current.
- $K_p$  is the proportional gain.
- $K_i$  is the integral gain.
- $T_s$  is the sample time.
- $v_{ref}$  is the reference voltage.
- $v$  is the measured voltage.
- $FF$  is the feedforward input.

To avoid saturation of the integrator output, the block uses an anti-windup mechanism. The integrator gain is then equal to

$$K_i + K_{aw} (control_{sat} - control_{unsat}),$$

where:

- $K_{aw}$  is the anti-windup gain.
- $control_{sat}$  is the saturated control signal, which the block calculates as  $control_{sat} = \min(\max(control_{unsat}, control_{min}), control_{max})$ ,

where:

- $control_{unsat}$  is the unsaturated control signal.
- $control_{min}$  is the lower limit for the control signal.
- $control_{max}$  is the upper limit for the control signal.

## Ports

### Input

**vRef — Reference DC voltage**

scalar

Desired DC output voltage for the plant.

Data Types: `single` | `double`**v — Measured DC voltage**

scalar

Measured DC output voltage for the plant.

Data Types: `single` | `double`**FF — Feedforward**

scalar

Feedforward term.

Data Types: `single` | `double`**Reset — External reset**

scalar

External reset signal (rising edge) for the integrator.

Data Types: `single` | `double`

### Output

**Control — Control signal**

scalar

Control signal, *control*, expressed as a duty cycle or a current.Data Types: `single` | `double`

## Parameters

### **Proportional gain — Proportional gain, $K_p$**

0.1 (default) | positive scalar

Proportional gain,  $K_p$ , of the controller.

### **Integral gain — Integral gain, $K_i$**

50 (default) | positive scalar

Integral gain,  $K_i$ , of the controller.

### **Anti-windup gain — Anti-windup gain, $K_{aw}$**

1 (default) | positive scalar

Anti-windup gain,  $K_{aw}$ , of the controller.

### **Control action upper limit — Upper limit for the control signal, $control_{max}$**

1 (default) | positive scalar

Upper limit for the **Control** output signal. The value must be greater than the value of the **Control action lower limit** parameter.

### **Control action lower limit — Lower limit for the control signal, $control_{min}$**

0 (default) | non-negative scalar

Lower limit for the **Control** output signal.

### **Sample time (-1 for inherited) — Block sample time**

-1 (default) | positive scalar

Time, in s, between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

If this block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly using a positive scalar.

### **Time constant voltage filter — DC voltage filter time constant, $\tau$**

0.001 (default) | positive scalar

Time constant,  $\tau$ , for the DC voltage filter.

**Dependencies**

The **Time constant voltage filter** parameter is only visible when the **Filter DC voltage** checkbox is selected.

**Filter DC voltage — DC voltage filter option**

on (default) | off

To enable the filter on voltage measurement path, select the checkbox. To disable the filter, clear the check box.

**Dependencies**

The **Time constant voltage filter** parameter is only visible when the **Filter DC voltage** check box is selected.

**See Also****Blocks**

DC Current Controller | DC Voltage Controller

**Introduced in R2018a**

## Delta Reference

Reference point for delta-connected network



## Library

Simscape / Electrical / Power Systems / Connections

## Description

In a Simscape Electrical Power Systems model, connect a Delta Reference block to any part of the three-phase system that is connected in a delta winding configuration. The block provides a reference point for the delta winding, representing the center of the line-line vector voltage triangle. The software calculates absolute node voltages relative to the voltage at this reference point.

For example, suppose you model a transmission system that consists of a generator connected in a wye configuration, a wye-delta transformer, a delta-wye transformer, and a load connected in wye. Connect a Delta Reference block to the part of the circuit between the two transformers.

## Ports

The block has the following ports:

~

Expandable three-phase port



## **See Also**

### **Topics**

“Expand and Collapse Three-Phase Ports on a Block”

**Introduced in R2013b**

## Delta1-Delta1-Wye Transformer

Linear nonideal delta1-delta1-wye transformer with three-limb core



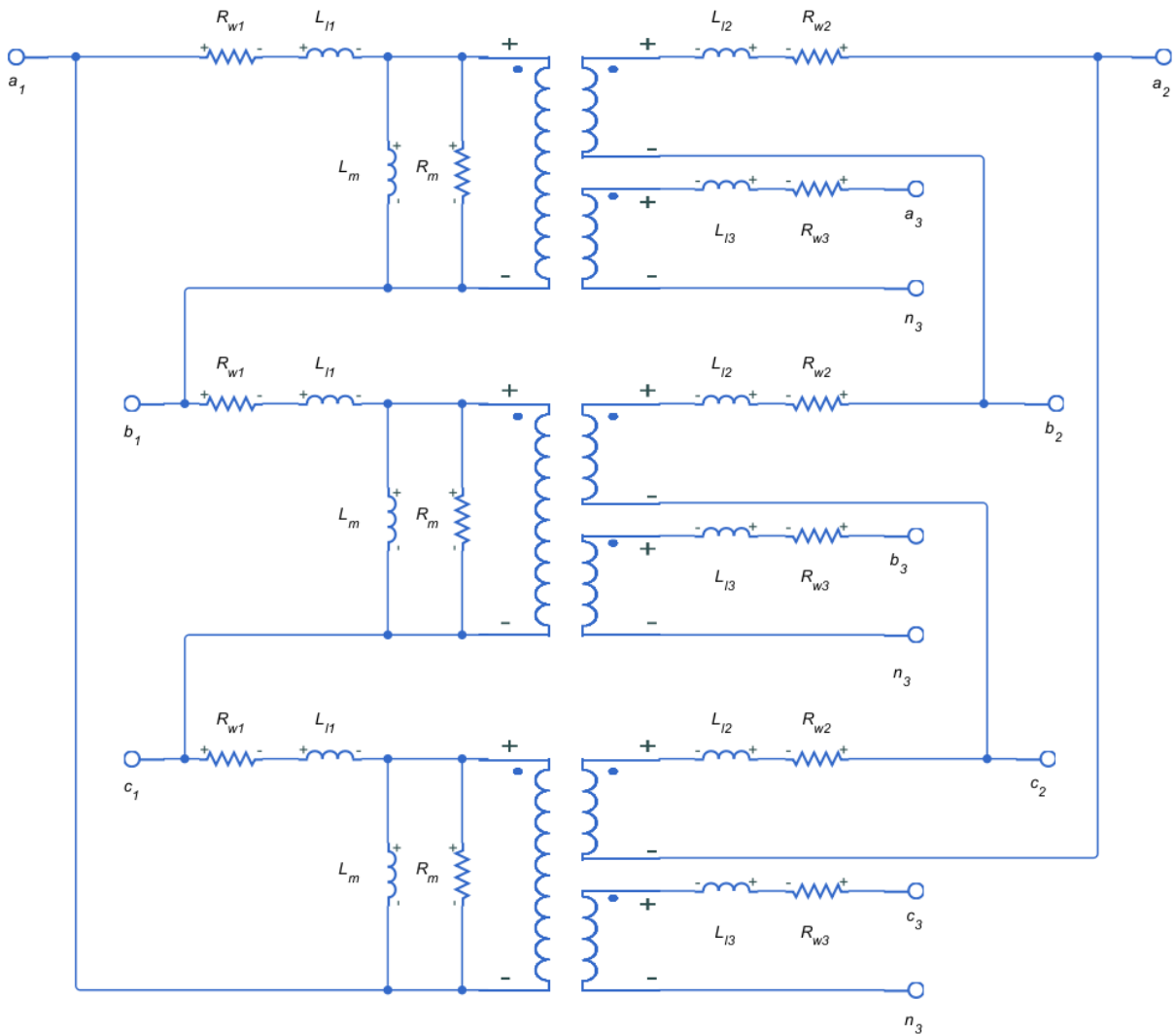
### Library

Simscape / Electrical / Power Systems / Passive Devices / Transformers

### Description

The Delta1-Delta1-Wye Transformer block models a linear, nonideal transformer with a three-limb core, in which the primary windings are configured in a delta connection and there are delta secondary windings and wye secondary windings. The delta voltages lag the wye voltages by 30 degrees, hence the name 1 o'clock delta. The block includes linear winding leakage and linear core magnetization effects.

The figure shows the equivalent circuit diagram for the delta1-delta1-wye transformer.



- $R_{w1}$  is the primary winding resistance.
- $L_{l1}$  is the primary leakage reactance.
- $R_{w2}$  is the delta secondary winding resistance.

- $L_{12}$  is the delta secondary leakage reactance.
- $R_{w3}$  is the wye secondary winding resistance.
- $L_{13}$  is the wye secondary leakage reactance.
- $R_m$  is the shunt magnetizing resistance.
- $L_m$  is the shunt magnetizing reactance.

## Display Options

You can display the transformer per-unit base values in the MATLAB command window using the block context menu. To display the values, right-click the block and select **Electrical > Display Base Values**.

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Parameters

- “Main Tab” on page 1-446
- “Impedances Tab” on page 1-447

### Main Tab

#### Rated apparent power

Apparent power flowing through the transformer when operating at rated capacity.  
The default value is 100e6 V\*A.

#### Primary rated voltage

RMS line voltage applied to the primary winding under normal operating conditions.  
The default value is 4160 V.

**Delta secondary rated voltage**

RMS line voltage applied to the delta secondary winding under normal operating conditions. The default value is 24e3 V.

**Wye secondary rated voltage**

RMS line voltage applied to the wye secondary winding under normal operating conditions. The default value is 24e3 V.

**Rated electrical frequency**

Rated or nominal frequency of the AC network to which the transformer is connected. The default value is 60 Hz.

**Impedances Tab**

Parameters in this tab are expressed in per-unit (pu). For more information, see “Per-Unit System of Units”.

**Primary winding resistance (pu)**

Power loss in the primary winding. The default value is 0.01.

**Primary leakage reactance (pu)**

Magnetic flux loss in the primary winding. The default value is 0.001.

**Delta secondary winding resistance (pu)**

Power loss in the delta secondary winding. The default value is 0.01.

**Delta secondary leakage reactance (pu)**

Magnetic flux loss in the delta secondary winding. The default value is 0.001.

**Wye secondary winding resistance (pu)**

Power loss in the wye secondary winding. The default value is 0.01.

**Wye secondary leakage reactance (pu)**

Magnetic flux loss in the wye secondary winding. The default value is 0.001.

**Shunt magnetizing resistance (pu)**

Magnetic losses in transformer core. The default value is 500.

**Shunt magnetizing reactance (pu)**

Magnetic effects of the transformer core when operating in its linear region. The default value is 500.

## Ports

The block has the following ports:

~1

Expandable three-phase port for primary winding

~2

Expandable three-phase port for delta secondary winding

~3

Expandable three-phase port for wye secondary winding

n3

Electrical conserving port associated with the wye secondary winding neutral point

## See Also

[Delta-Delta Transformer](#) | [Delta11-Delta11-Wye Transformer](#) | [Wye-Delta1 Transformer](#) | [Wye-Delta1-Wye Transformer](#) | [Wye-Delta11 Transformer](#) | [Wye-Delta11-Wye Transformer](#) | [Wye-Wye Transformer](#) | [Zigzag-Delta1-Wye Transformer](#) | [Zigzag-Delta11-Wye Transformer](#)

## Topics

[“Custom Zigzag Transformer”](#)

[“Push-Pull Buck Converter in Continuous Conduction Mode”](#)

[“Push-Pull Buck Converter in Discontinuous Conduction Mode”](#)

**Introduced in R2013b**

# Delta11-Delta11-Wye Transformer

Linear nonideal delta11-delta11-wye transformer with three-limb core



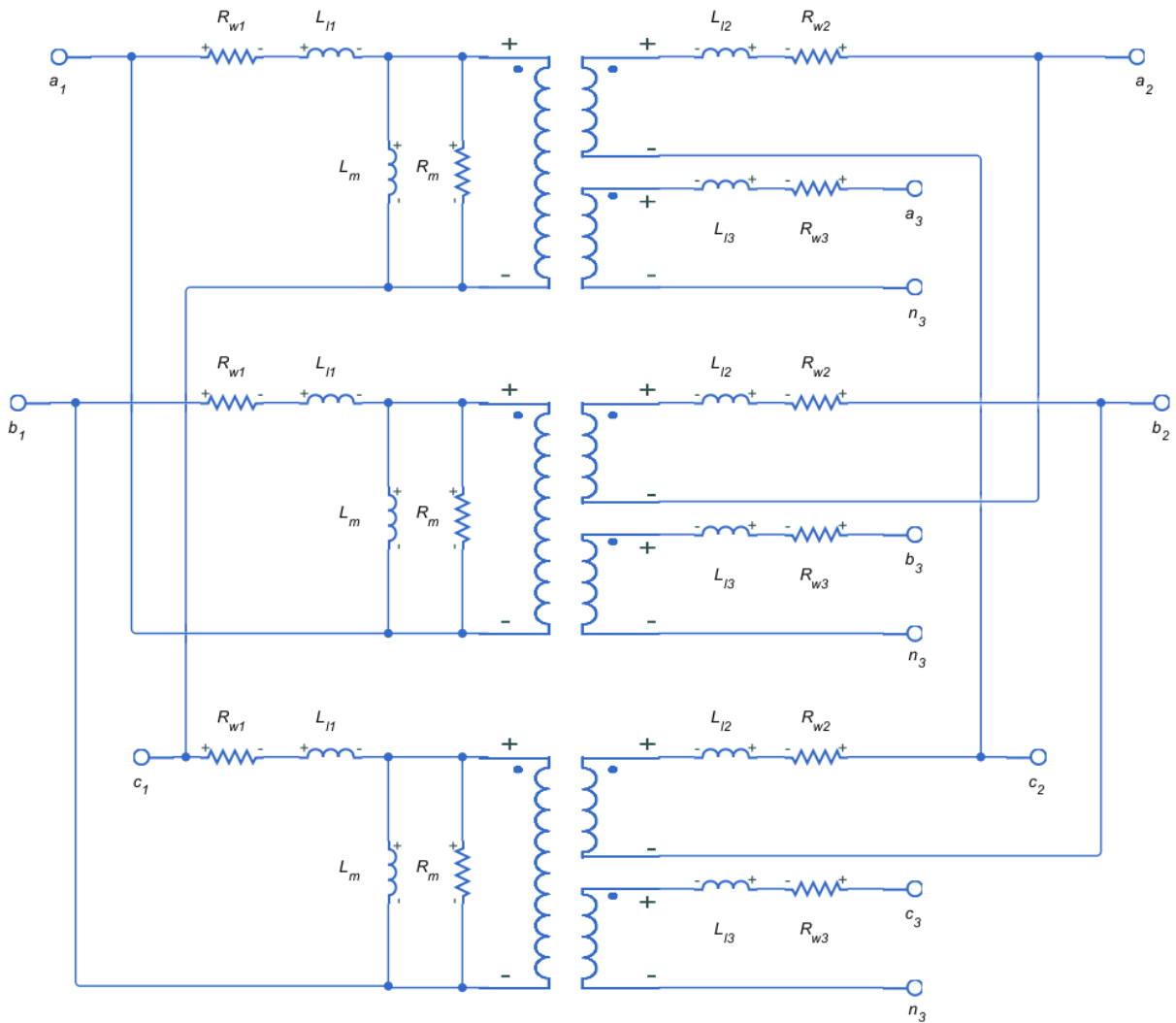
## Library

Simscape / Electrical / Power Systems / Passive Devices / Transformers

## Description

The Delta11-Delta11-Wye Transformer block models a linear, nonideal transformer with a three-limb core, in which the primary windings are configured in a delta connection and there are delta secondary windings and wye secondary windings. The delta voltages lead the wye voltages by 30 degrees, hence the name 11 o'clock delta. The block includes linear winding leakage and linear core magnetization effects.

The figure shows the equivalent circuit diagram for the delta11-delta11-wye transformer.



- $R_{w1}$  is the primary winding resistance.
- $L_{l1}$  is the primary leakage reactance.
- $R_{w2}$  is the delta secondary winding resistance.



- $L_{12}$  is the delta secondary leakage reactance.
- $R_{w3}$  is the wye secondary winding resistance.
- $L_{13}$  is the wye secondary leakage reactance.
- $R_m$  is the shunt magnetizing resistance.
- $L_m$  is the shunt magnetizing reactance.

## Display Options

You can display the transformer per-unit base values in the MATLAB command window using the block context menu. To display the values, right-click the block and select **Electrical > Display Base Values**.

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Parameters

- “Main Tab” on page 1-451
- “Impedances Tab” on page 1-452

### Main Tab

#### Rated apparent power

Apparent power flowing through the transformer when operating at rated capacity.  
The default value is 100e6 V\*A.

#### Primary rated voltage

RMS line voltage applied to the primary winding under normal operating conditions.  
The default value is 4160 V.

**Delta secondary rated voltage**

RMS line voltage applied to the delta secondary winding under normal operating conditions. The default value is 24e3 V.

**Wye secondary rated voltage**

RMS line voltage applied to the wye secondary winding under normal operating conditions. The default value is 24e3 V.

**Rated electrical frequency**

Rated or nominal frequency of the AC network to which the transformer is connected. The default value is 60 Hz.

**Impedances Tab**

Parameters in this tab are expressed in per-unit (pu). For more information, see “Per-Unit System of Units”.

**Primary winding resistance (pu)**

Power loss in the primary winding. The default value is 0.01.

**Primary leakage reactance (pu)**

Magnetic flux loss in the primary winding. The default value is 0.001.

**Delta secondary winding resistance (pu)**

Power loss in the delta secondary winding. The default value is 0.01.

**Delta secondary leakage reactance (pu)**

Magnetic flux loss in the delta secondary winding. The default value is 0.001.

**Wye secondary winding resistance (pu)**

Power loss in the wye secondary winding. The default value is 0.01.

**Wye secondary leakage reactance (pu)**

Magnetic flux loss in the wye secondary winding. The default value is 0.001.

**Shunt magnetizing resistance (pu)**

Magnetic losses in transformer core. The default value is 500.

**Shunt magnetizing reactance (pu)**

Magnetic effects of the transformer core when operating in its linear region. The default value is 500.

## Ports

The block has the following ports:

~1

Expandable three-phase port for primary winding

~2

Expandable three-phase port for delta secondary winding

~3

Expandable three-phase port for wye secondary winding

n3

Electrical conserving port associated with the wye secondary winding neutral point

## See Also

[Delta-Delta Transformer](#) | [Delta1-Delta1-Wye Transformer](#) | [Wye-Delta1 Transformer](#) | [Wye-Delta1-Wye Transformer](#) | [Wye-Delta11 Transformer](#) | [Wye-Delta11-Wye Transformer](#) | [Wye-Wye Transformer](#) | [Zigzag-Delta1-Wye Transformer](#) | [Zigzag-Delta11-Wye Transformer](#)

## Topics

[“Custom Zigzag Transformer”](#)

[“Push-Pull Buck Converter in Continuous Conduction Mode”](#)

[“Push-Pull Buck Converter in Discontinuous Conduction Mode”](#)

**Introduced in R2013b**

## Delta-Connected Load

Three-phase load wired in delta configuration



## Library

Simscape / Electrical / Power Systems / Passive Devices

## Description

The Delta-Connected Load block models a three-phase load wired in a delta configuration. Each limb of the load can include any combination of a resistor (R), capacitor (C), and inductor (L), connected in series or in parallel.

You can specify values for the R, L, and C components directly in terms of resistance, inductance, and capacitance, or by rated powers at a rated voltage and frequency.

- If you parameterize the block directly in terms of R, L, and C values, then for initialization provide a three-element row vector of initial voltages for a capacitor, and a three-element row vector of initial currents for an inductor.
- If you parameterize the block in terms of rated powers, then specify initial conditions in terms of an initial voltage, initial voltage phase, and initial frequency. For example, if the load is connected directly to a three-phase voltage source, then the initial conditions are identical to the source values for RMS line voltage, frequency, and phase shift. To specify zero initial voltage magnitude, set the initial voltage to 0.

For certain combinations of R, L, and C, for some circuit topologies, specify parasitic resistance or conductance values that help the simulation to converge numerically. These parasitic terms ensure that an inductor has a small parallel resistive path and that a

capacitor has a small series resistance. When you parameterize the block in terms of rated powers, the rated power values do not account for these small parasitic terms. The rated powers represent only the R, L, and C values of the load itself.

## Parameters

- “Main Tab” on page 1-455
- “Parasitics Tab” on page 1-456
- “Initial Conditions Tab” on page 1-457

### Main Tab

#### Parameterization

Select one of these values:

- **Specify by rated power** — Specify values for the R, L, and C components by rated powers at a rated voltage and frequency. This is the default.
- **Specify component values directly** — Specify values for the R, L, and C components directly in terms of resistance, inductance, and capacitance.

Switching the **Parameterization** value resets the **Component structure** value. Select the component parameterization option first, and then the component structure. If you later switch the **Parameterization** value, check the **Component structure** value and reselect it, if necessary.

#### Component structure

Select the desired combination of a resistor (R), capacitor (C), and inductor (L), connected in series or in parallel. The default is R, resistor.

#### Rated voltage

Voltage for which load powers are specified. This parameter is visible only when you specify values by rated power. The default value is  $2.4 \times 10^4$  V.

#### Real power

Total real power dissipated by three-phase load when supplied at the rated voltage. This parameter is visible only when you specify values by rated power and select a component structure that includes a resistor. The value must be greater than 0. The default value is 1000 W.

**Rated electrical frequency**

Frequency for which reactive load powers are specified. This parameter is visible only when you specify values by rated power. The default value is 60 Hz.

**Inductive reactive power**

Total inductive reactive power taken by the three-phase load when supplied at the rated voltage. This parameter is visible only when you specify values by rated power and select a component structure that includes an inductor. The value must be greater than 0. The default value is 100 V\*A.

**Capacitive reactive power**

Total capacitive reactive power taken by the three-phase load when supplied at the rated voltage. This parameter is visible only when you specify values by rated power and select a component structure that includes a capacitor. The value must be less than 0. The default value is -100 V\*A.

**Resistance**

The resistance of each of the load limbs. This parameter is visible only when you specify component values directly and select a component structure that includes a resistor. The default value is 1 Ohm.

**Inductance**

Inductance of each of the load limbs. This parameter is visible only when you specify component values directly and select a component structure that includes an inductor. The default value is 0.001 H.

**Capacitance**

Capacitance in each of the load limbs. This parameter is visible only when you specify component values directly and select a component structure that includes a capacitor. The default value is 1e-6 F.

**Parasitics Tab**

**Parasitic series resistance**

Represents small parasitic effects. The parameter value corresponds to the series resistance value added to all instances of capacitors in the load. The default value is 1e-6 Ohm.

**Parasitic parallel conductance**

Represents small parasitic effects. The parameter value corresponds to the parallel conductance value added across all instances of inductors in the load. The default value is 1e-6 1/Ohm.

## Initial Conditions Tab

### Terminal voltage magnitude

Expected initial RMS line voltage at the load. This parameter is visible only when you specify values by rated power. The default value is  $2.4 \times 10^4$  V.

### Terminal voltage angle

Expected initial phase of the voltage at the load. This parameter is visible only when you specify values by rated power. The default value is 0 deg.

### Frequency

Expected initial frequency at the load. This parameter is visible only when you specify values by rated power. The default value is 60 Hz.

### Initial inductor current [ Ia Ib Ic ]

Initial current in the a, b, and c phase inductors, respectively. This parameter is visible only when you specify component values directly and select a component structure that includes an inductor. The default value is [0 0 0] A.

### Initial capacitor voltage [ Va Vb Vc ]

Initial voltage across the a, b, and c phase capacitors, respectively. This parameter is visible only when you specify component values directly and select a component structure that includes a capacitor. The default value is [0 0 0] V.

## Block Parameterization

The following two tables list the block parameters for each **Component structure**, based on the selected **Parameterization** option:

- Specify by rated power
- Specify component values directly

**Specify by Rated Power**

<b>Component Structure</b>	<b>Main Tab</b>	<b>Parasitics Tab</b>	<b>Initial Conditions Tab</b>
R	Rated voltage Real power	None	None
L	Rated voltage Rated electrical frequency Inductive reactive power	Parasitic parallel conductance	Terminal voltage magnitude Terminal voltage angle Frequency
C	Rated voltage Rated electrical frequency Capacitive reactive power	Parasitic series resistance	Terminal voltage magnitude Terminal voltage angle Frequency
Series RL	Rated voltage Rated electrical frequency Real power Inductive reactive power	Parasitic parallel conductance	Terminal voltage magnitude Terminal voltage angle Frequency
Series RC	Rated voltage Rated electrical frequency Real power Capacitive reactive power	None	Terminal voltage magnitude Terminal voltage angle Frequency



<b>Component Structure</b>	<b>Main Tab</b>	<b>Parasitics Tab</b>	<b>Initial Conditions Tab</b>
Series LC	Rated voltage Rated electrical frequency Inductive reactive power Capacitive reactive power	Parasitic parallel conductance	Terminal voltage magnitude Terminal voltage angle Frequency
Series RLC	Rated voltage Rated electrical frequency Real power Inductive reactive power Capacitive reactive power	Parasitic parallel conductance	Terminal voltage magnitude Terminal voltage angle Frequency
Parallel RL	Rated voltage Rated electrical frequency Real power Inductive reactive power	None	Terminal voltage magnitude Terminal voltage angle Frequency

<b>Component Structure</b>	<b>Main Tab</b>	<b>Parasitics Tab</b>	<b>Initial Conditions Tab</b>
Parallel RC	Rated voltage Rated electrical frequency Real power Capacitive reactive power	Parasitic series resistance	Terminal voltage magnitude Terminal voltage angle Frequency
Parallel LC	Rated voltage Rated electrical frequency Inductive reactive power Capacitive reactive power	Parasitic series resistance	Terminal voltage magnitude Terminal voltage angle Frequency
Parallel RLC	Rated voltage Rated electrical frequency Real power Inductive reactive power Capacitive reactive power	Parasitic series resistance	Terminal voltage magnitude Terminal voltage angle Frequency

**Specify Component Values Directly**

<b>Component Structure</b>	<b>Main Tab</b>	<b>Parasitics Tab</b>	<b>Initial Conditions Tab</b>
R	Resistance	None	None
L	Inductance	Parasitic parallel conductance	Initial inductor current [ Ia Ib Ic ]
C	Capacitance	Parasitic series resistance	Initial capacitor voltage [ Va Vb Vc ]
Series RL	Resistance Inductance	Parasitic parallel conductance	Initial inductor current [ Ia Ib Ic ]
Series RC	Resistance Capacitance	None	Initial capacitor voltage [ Va Vb Vc ]
Series LC	Inductance Capacitance	Parasitic parallel conductance	Initial inductor current [ Ia Ib Ic ] Initial capacitor voltage [ Va Vb Vc ]
Series RLC	Resistance Inductance Capacitance	Parasitic parallel conductance	Initial inductor current [ Ia Ib Ic ] Initial capacitor voltage [ Va Vb Vc ]
Parallel RL	Resistance Inductance	None	Initial inductor current [ Ia Ib Ic ]
Parallel RC	Resistance Capacitance	Parasitic series resistance	Initial capacitor voltage [ Va Vb Vc ]
Parallel LC	Inductance Capacitance	Parasitic series resistance	Initial inductor current [ Ia Ib Ic ] Initial capacitor voltage [ Va Vb Vc ]

<b>Component Structure</b>	<b>Main Tab</b>	<b>Parasitics Tab</b>	<b>Initial Conditions Tab</b>
Parallel RLC	Resistance Inductance Capacitance	Parasitic series resistance	Initial inductor current [ Ia Ib Ic ] Initial capacitor voltage [ Va Vb Vc ]

## Ports

The block has one expandable three-phase port, ~.

## See Also

RLC | Wye-Connected Load

## Topics

“Three-Phase Asynchronous Wind Turbine Generator”

“Expand and Collapse Three-Phase Ports on a Block”

**Introduced in R2013b**

# Delta-Delta Transformer

Linear nonideal delta-delta transformer with three-limb core



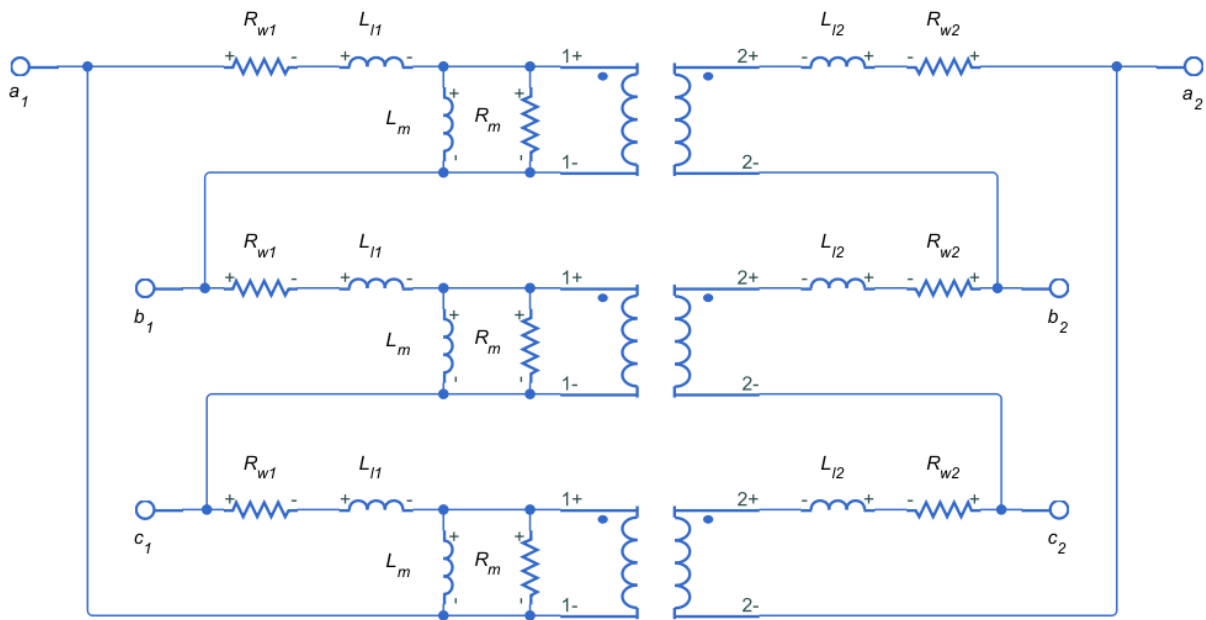
## Library

Simscape / Electrical / Power Systems / Passive Devices / Transformers

## Description

The Delta-Delta Transformer block models a linear, nonideal transformer with a three-limb core, in which both the primary and the secondary windings are configured in a delta connection. The block includes linear winding leakage and linear core magnetization effects.

The figure shows the equivalent circuit diagram for the delta-delta transformer.



- $R_{w1}$  is the primary winding resistance.
- $L_{l1}$  is the primary leakage reactance.
- $R_{w2}$  is the secondary winding resistance.
- $L_{l2}$  is the secondary leakage reactance.
- $R_m$  is the shunt magnetizing resistance.
- $L_m$  is the shunt magnetizing reactance.

## Display Options

You can display the transformer per-unit base values in the MATLAB command window using the block context menu. To display the values, right-click the block and select **Electrical > Display Base Values**.

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Parameters

- “Main Tab” on page 1-465
- “Impedances Tab” on page 1-465

## Main Tab

### Rated apparent power

Apparent power flowing through the transformer when operating at rated capacity.  
The default value is  $100e6$  V\*A.

### Primary rated voltage

RMS line voltage applied to the primary winding under normal operating conditions.  
The default value is  $4160$  V.

### Secondary rated voltage

RMS line voltage applied to the secondary winding under normal operating conditions. The default value is  $24e3$  V.

### Rated electrical frequency

Rated or nominal frequency of the AC network to which the transformer is connected.  
The default value is  $60$  Hz.

## Impedances Tab

Parameters in this tab are expressed in per-unit (pu). For more information, see “Per-Unit System of Units”.

**Primary winding resistance (pu)**

Power loss in the primary winding. The default value is 0.01.

**Primary leakage reactance (pu)**

Magnetic flux loss in the primary winding. The default value is 0.001.

**Secondary winding resistance (pu)**

Power loss in the secondary winding. The default value is 0.01.

**Secondary leakage reactance (pu)**

Magnetic flux loss in the secondary winding. The default value is 0.001.

**Shunt magnetizing resistance (pu)**

Magnetic losses in transformer core. The default value is 500.

**Shunt magnetizing reactance (pu)**

Magnetic effects of the transformer core when operating in its linear region. The default value is 500.

## Ports

The block has the following ports:

~1

Expandable three-phase port for primary winding

~2

Expandable three-phase port for secondary winding

## See Also

Delta1-Delta1-Wye Transformer | Delta11-Delta11-Wye Transformer | Wye-Delta1 Transformer | Wye-Delta1-Wye Transformer | Wye-Delta11 Transformer | Wye-Delta11-Wye Transformer | Wye-Wye Transformer | Zigzag-Delta1-Wye Transformer | Zigzag-Delta11-Wye Transformer

## Topics

“Custom Zigzag Transformer”

“Push-Pull Buck Converter in Continuous Conduction Mode”



“Push-Pull Buck Converter in Discontinuous Conduction Mode”

**Introduced in R2013b**

## Diffusion Resistor

Resistor model including velocity saturation, tolerance, operational limits, fault behavior, and noise

**Library:** Simscape / Electrical / Electronics and Mechatronics / Passive Devices



### Description

The Diffusion Resistor block represents a resistor with velocity saturation, while letting you model the following effects:

- “Tolerances” on page 1-470
- “Operating Limits” on page 1-471
- “Faults” on page 1-471
- “Thermal Noise” on page 1-471
- “Thermal Port” on page 1-472

You can turn these modeling options on and off independently of each other.

In its simplest form, the resistance of the Diffusion Resistor block is:

$$R = R_0 \left( 1 - p_2 - p_3 + p_2 \sqrt{1 + (\theta_2 v_{pn})^2} + p_3 \sqrt[3]{1 + |\theta_3 v_{pn}|^3} \right)$$

where:

- $R_0$  is zero-bias resistance.
- $p_2$  and  $p_3$  are the quadratic and linear voltage coefficients, respectively.
- $\theta_2$  and  $\theta_3$  are inverse voltages for quadratic and linear voltage activation, respectively.
- $v_{pn}$  is applied voltage across the resistor.

At low bias,

$$R \approx R_0 \left( 1 + \frac{p_2 \theta_2^2 v_{pn}^2}{2} \right)$$

and therefore  $p_2$  and  $\theta_2$  determine the low-bias quadratic behavior of the resistor.

At high bias,

$$R \approx R_0 (1 - p_2 - p_3 + |v_{pn}| (p_2 \theta_2 + p_3 \theta_3))$$

and therefore  $p_3$  and  $\theta_3$  impact only the high-bias linear behavior of the resistor.

You can use the voltage-dependence of the resistance to model velocity saturation in a diffused resistor. For sufficiently high voltage,

$$i_{sat} = \frac{1}{R_0 (p_2 \theta_2 + p_3 \theta_3)}$$

where  $i_{sat}$  is saturation current.

## Simplified Parameterization

The simplified parameterization model assumes that the quadratic and linear coefficients are the same. This is one of the recommended assumptions for the `r2_cmc` model, as a reasonable initial guess when performing parameter extraction (see the `r2_cmc` documentation at [https://projects.si2.org/cmc\\_index.php](https://projects.si2.org/cmc_index.php)). With this assumption, it is possible to define two new parameters, **Critical voltage** and **Corner voltage**, which provide a simpler means for parameterizing models:

$$p_2 = p_3 = \frac{v_{co}}{2v_{crit}}$$

$$\theta_2 = \theta_3 = \frac{1}{2v_{co}}$$

where:

- $v_{crit}$  is critical voltage.

- $v_{co}$  is corner voltage.

At high voltage,

$$\frac{dR}{dv_{pn}} \approx \frac{R_0}{v_{crit}}$$

and therefore, critical voltage is the reciprocal of the slope of the increase of  $R/R_0$  with voltage.

With this parameterization, the saturation current is

$$i_{sat} = \frac{v_{crit}}{R_0}$$

## Tolerances

You can apply tolerances to the nominal value you provide for the **Resistance** parameter. Datasheets typically provide a tolerance percentage for a given resistor type. The table shows how the block applies tolerances and calculates resistance based on the selected **Tolerance application** option.

Option	Resistance Value
None – use nominal value	$R_0$
Random tolerance	Uniform distribution: $R_0 \cdot (1 - tol + 2 \cdot tol \cdot rand)$ Gaussian distribution: $R_0 \cdot (1 + tol \cdot randn / nSigma)$
Apply maximum tolerance value	$R_0 \cdot (1 + tol)$
Apply minimum tolerance value	$R_0 \cdot (1 - tol)$

In the table,

- $R_0$  is the **Resistance** parameter value, nominal zero-bias resistance.
- $tol$  is fractional tolerance, **Tolerance (%)** /100.
- $nSigma$  is the value you provide for the **Number of standard deviations for quoted tolerance** parameter.

- `rand` and `randn` are standard MATLAB functions for generating uniform and normal distribution random numbers.

## Operating Limits

You can specify operating limits in terms of power and maximum working voltage. For the thermal variant of the block (see “Thermal Port” on page 1-472), you can also specify operating limits in terms of temperature.

When an operating limit is exceeded, the block can either generate a warning or stop the simulation with an error. For more information, see the “Operating Limits” on page 1-477 parameters section.

## Faults

The Diffusion Resistor block allows you to model an electrical fault as an instantaneous change in resistance. The block can trigger fault events:

- At a specific time
- When a current limit is exceeded for longer than a specific time interval

You can enable or disable these trigger mechanisms separately, or use them together if more than one trigger mechanism is required in a simulation. When more than one mechanism is enabled, the first mechanism to trigger the fault takes precedence. In other words, component fails no more than once per simulation.

When the resistor fails, its resistance is changed to the value you specify for the **Faulted zero-voltage resistance** parameter. You can also choose whether to issue an assertion when a fault occurs, by using the **Reporting when a fault occurs** parameter. The assertion can take the form of a warning or an error. By default, the block does not issue an assertion.

## Thermal Noise

The Diffusion Resistor block can generate thermal noise current. If you set the **Noise mode** parameter to **Enabled**, then the block includes a noise current source connected in parallel to the diffusion resistor.

If the sampling time is  $h$ , then the thermal noise is given by:

$$i_N = \sqrt{2kT/R} \frac{N(0,1)}{\sqrt{h}}$$

where:

- $k$  is the Boltzmann constant, 1.3806504e-23 J/K.
- $T$  is temperature.
- $R$  is resistance.
- $N$  is a Gaussian random number with zero mean and standard deviation of one.
- $2kT/R$  is the double-sided thermal noise power distribution (the single-sided equivalent is  $4kT/R$ ).

The block generates Gaussian noise by using the Random Number source in the Simscape Foundation library. You can control the random number seed by setting the **Repeatability** parameter:

- **Not repeatable** — Every time you simulate your model, the block resets the random seed using the MATLAB random number generator:  

```
seed = randi(2^32-1);
```
- **Repeatable** — The block automatically generates a seed value and stores it inside the block, to always start the simulation with the same random number. This auto-generated seed value is set when you add a Diffusion Resistor block from the block library to the model. When you make a new copy of the Diffusion Resistor block from an existing one in a model, a new seed value is generated. The block sets the value using the MATLAB random number generator command shown above.
- **Specify seed** — If you select this option, the additional **Seed** parameter lets you directly specify the random number seed value.

## Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal port H on the block icon, and adds the **Thermal** tab and the **Variables** tab to the block dialog box.

Use the **Thermal** tab to specify how the resistance value changes with temperature and to set the thermal mass. Use the **Variables** tab to set the initial temperature target.

For the thermal variant, the defining equation for the resistance is augmented with additional temperature scaling:

$$R = R_0 \left( 1 + T_{C1}^{eff} \Delta T + T_{C2}^{eff} (\Delta T)^2 \right) \left( 1 - p_2 - p_3 + p_2 \sqrt{1 + (\theta_2 v_{pn})^2} + p_3 \sqrt[3]{1 + |\theta_3 v_{pn}|^3} \right)$$

where  $T_{C1}^{eff}$  and  $T_{C2}^{eff}$  are the linear and quadratic temperature scaling coefficients, respectively.

$$\Delta T = T_{sim} - T_{meas}$$

where:

- $T_{sim}$  is simulation temperature.
- $T_{meas}$  is measurement temperature.

With the thermal port exposed, the generated noise uses the temperature at the thermal port when determining the instantaneous noise value. Exposing the thermal port also extends the options on the **Operating Limits** tab as follows:

- The **Power rating** parameter becomes temperature dependent. You define a temperature up to which the full power rating is available, plus a higher temperature for which the power rating is reduced to zero. It is assumed that the power rating decreases linearly with temperature between these two values.
- An additional parameter, **Operating temperature range, [Tmin Tmax]**, lets you define the valid temperature range for block operation.

## Variables

Use the **Variables** section of the block interface to set the priority and initial target values for the block variables prior to simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

This section appears only for the blocks with exposed thermal port. The **Temperature** variable lets you specify a high-priority target for the temperature at the start of simulation.

## Basic Assumptions and Limitations

Simulating with noise enabled slows down simulation. Choose the sample time ( $h$ ) so that noise is generated only at frequencies of interest, and not higher.

## Ports

### Conserving

#### + — Positive terminal

electrical

Electrical conserving port associated with the resistor positive terminal.

#### - — Negative terminal

electrical

Electrical conserving port associated with the resistor negative terminal.

#### H — Resistor thermal mass

thermal

Thermal conserving port that represents the resistor thermal mass.

#### Dependencies

Enabled for the thermal variant of the block. For more information, see “Thermal Port” on page 1-472.

## Parameters

### Main

#### Resistance — Nominal zero-bias resistance

1  $\Omega$  (default)

The zero-bias resistance, used as the nominal resistance value. Resistance value must be greater than zero. For the thermal variant of the block, this is the zero-bias resistance at a



temperature equal to the **Measurement temperature** parameter in the “Thermal” on page 1-483 section.

### **Tolerance (%) — Resistor tolerance, in percent**

5 (default)

The resistor tolerance as defined on the manufacturer datasheet.

### **Tolerance application — Select how to apply tolerance during simulation**

None — use nominal value (default) | Random tolerance | Apply maximum tolerance value | Apply minimum tolerance value

Select how to apply tolerance during simulation:

- **None — use nominal value** — The block does not apply tolerance, uses the nominal resistance value. This is the default.
- **Random tolerance** — The block applies random offset to the resistance value, within the tolerance value limit. You can choose Uniform or Gaussian distribution for calculating the random number by using the **Tolerance distribution** parameter.
- **Apply maximum tolerance value** — The resistance is increased by the specified tolerance percent value.
- **Apply minimum tolerance value** — The resistance is decreased by the specified tolerance percent value.

### **Tolerance distribution — Select the distribution type**

Uniform (default) | Gaussian

Select the distribution type for random tolerance:

- **Uniform** — Uniform distribution
- **Gaussian** — Gaussian distribution

### **Dependencies**

Enabled when the **Tolerance application** parameter is set to **Random tolerance**.

### **Number of standard deviations for quoted tolerance — Used for calculating the Gaussian random number**

4 (default)

Number of standard deviations for calculating the Gaussian random number.

### **Dependencies**

Enabled when the **Tolerance distribution** parameter is set to Gaussian.

### **Parameterization — Select parameterization method**

Simplified (default) | Advanced

Select how to apply tolerance during simulation:

- **Simplified** — Assume that the quadratic and linear coefficients are the same, and define block behavior using the **Critical voltage** and **Corner voltage** parameters.
- **Advanced** — Explicitly specify values for the quadratic and linear voltage coefficients and for the inverse voltages for quadratic and linear voltage activation.

### **Critical voltage — Critical voltage for saturation**

4 V (default)

Critical voltage for the saturation mechanism. You can determine this parameter value by taking the reciprocal of the slope of the increase of  $R/R_0$  with voltage.

### **Dependencies**

Enabled when the **Parameterization** parameter is set to Simplified.

### **Corner voltage — Voltage at which the resistance increase starts to occur**

2 V (default)

Corner voltage, at which the resistance increase starts to occur. The **Corner voltage** must be less than the **Critical voltage**.

### **Dependencies**

Enabled when the **Parameterization** parameter is set to Simplified.

### **Quadratic voltage coefficient — Coefficient $p_2$**

0.25 (default)

Coefficient  $p_2$  from the defining equation.

### **Dependencies**

Enabled when the **Parameterization** parameter is set to Advanced.

**Inverse voltage for quadratic voltage activation – Coefficient  $\theta_2$** 

0.25 1/V (default)

Coefficient  $\theta_2$  from the defining equation.**Dependencies**Enabled when the **Parameterization** parameter is set to Advanced.**Linear voltage coefficient – Coefficient  $p_3$** 

0.25 (default)

Coefficient  $p_3$  from the defining equation.**Dependencies**Enabled when the **Parameterization** parameter is set to Advanced.**Inverse voltage for linear voltage activation – Coefficient  $\theta_3$** 

0.25 1/V (default)

Coefficient  $\theta_3$  from the defining equation.**Dependencies**Enabled when the **Parameterization** parameter is set to Advanced.**Operating Limits****Enable operating limits – Select Yes to enable reporting when the operational limits are exceeded**

No (default) | Yes

Select Yes to enable reporting when the operational limits are exceeded. The associated parameters in the **Operating Limits** section become visible to let you select the reporting method and specify the operating limits in terms of power and maximum working voltage. Parameters that specify operating limits in terms of temperature are visible only for blocks with exposed thermal port (see “Thermal Port” on page 1-472). The default value is No.

**Reporting when operating limits exceeded – Select the reporting method**

Warn (default) | Error

Select what happens when an operating limit is exceeded:

- Warn — The block issues a warning.
- Error — Simulation stops with an error.

### **Dependencies**

Enabled when the **Enable operating limits** parameter is set to Yes.

### **Maximum working voltage — Maximum voltage allowed for normal block operation**

100 V (default)

Maximum voltage magnitude allowed for normal block operation.

### **Dependencies**

Enabled when the **Enable operating limits** parameter is set to Yes.

### **Power rating — Maximum power allowed for normal block operation**

1 W (default)

Maximum power allowed for normal block operation.

If you expose the thermal port of the block, this parameter becomes temperature dependent. The value you specify for the **Power rating** parameter applies up to the temperature specified by the **Temperature below which full power rating is available** parameter value. Then the power rating decreases linearly with temperature, until it becomes 0 at temperature specified by the **Temperature above which power rating is reduced to zero** parameter value.

### **Dependencies**

Enabled when the **Enable operating limits** parameter is set to Yes.

### **Temperature below which full power rating is available — Maximum temperature where full power rating still applies**

70 °C (default)

Maximum temperature where full power rating, specified by the **Power rating** parameter value, still applies.

### **Dependencies**

Enabled for the thermal variant of the block. For more information, see “Thermal Port” on page 1-1315.

**Temperature above which power rating is reduced to zero – Temperature where power rating becomes 0**

155 °C (default)

Temperature where power rating becomes 0. Above this temperature, the simulation always issues an assertion regardless of dissipated power. This parameter value must be higher than **Temperature below which full power rating is available**.

**Dependencies**

Enabled for the thermal variant of the block. For more information, see “Thermal Port” on page 1-1315.

**Operating temperature range, [Tmin Tmax] – Minimum and maximum temperature values allowed for normal block operation**

[-50 150] °C (default)

A row vector of length 2 specifying minimum and maximum temperature values allowed for normal block operation. The first element is the lowest allowable operating temperature, and the second element is the largest allowable operating temperature.

**Dependencies**

Enabled for the thermal variant of the block. For more information, see “Thermal Port” on page 1-1315.

**Faults****Enable faults – Select Yes to enable faults modeling**

No (default) | Yes

Select **Yes** to enable faults modeling. The associated parameters in the **Faults** section become visible to let you select the reporting method and specify the trigger mechanism (temporal or behavioral). You can enable these trigger mechanisms separately or use them together.

**Reporting when a fault occurs – Choose whether to issue an assertion when a fault occurs**

None (default) | Warn | Error

Choose whether to issue an assertion when a fault occurs:

- **None** — The block does not issue an assertion.
- **Warn** — The block issues a warning.
- **Error** — Simulation stops with an error.

### Dependencies

Enabled when the **Enable faults** parameter is set to Yes.

### Faulted zero-voltage resistance — Resistance when block is in faulted state

inf  $\Omega$  (default)

Zero-voltage resistance between the + and - ports when the block is in the faulted state.

### Dependencies

Enabled when the **Enable faults** parameter is set to Yes.

### Enable temporal fault trigger — Select Yes to enable time-based fault triggering

No (default) | Yes

Select Yes to enable time-based fault triggering. You can enable the temporal and behavioral trigger mechanisms separately or use them together.

### Dependencies

Enabled when the **Enable faults** parameter is set to Yes.

### Simulation time for fault event — Time before entering faulted state

1 s (default)

Set the simulation time at which you want the block to enter the faulted state.

### Dependencies

Enabled when the **Enable temporal fault trigger** parameter is set to Yes.

### Enable behavioral fault trigger — Select Yes to enable behavioral fault triggering

No (default) | Yes

Select Yes to enable behavioral fault triggering. You can enable the temporal and behavioral trigger mechanisms separately or use them together.

### Dependencies

Enabled when the **Enable faults** parameter is set to Yes.

### Maximum permissible current – Current threshold to fault transition

1 A (default)

Specify the maximum permissible current value. If the current exceeds this value for longer than the **Time to fail when exceeding maximum permissible current** parameter value, then the block enters the faulted state.

### Dependencies

Enabled when the **Enable behavioral fault trigger** parameter is set to Yes.

### Time to fail when exceeding maximum permissible current – Maximum length of time the current exceeds the threshold

1 s (default)

Set the maximum length of time that the current can exceed the maximum permissible value without triggering the fault.

### Dependencies

Enabled when the **Enable behavioral fault trigger** parameter is set to Yes.

## Noise

### Noise mode – Select whether to model thermal noise current

Disabled (default) | Enabled

Select whether to model thermal noise current:

- **Disabled** — No noise is produced by the resistor.
- **Enabled** — Resistor generates thermal noise current, and the associated parameters become visible in the **Noise** section.

### Sample time – Rate at which the noise source is sampled

1e-3 s (default)

Defines the rate at which the noise source is sampled. Choose it to reflect the frequencies of interest in your model. Making the sample time too small will unnecessarily slow down your simulation.

### **Dependencies**

Enabled when the **Noise mode** parameter is set to Enabled.

### **Repeatability — Select the noise control option**

Not repeatable (default) | Repeatable | Specify seed

Select the noise control option:

- **Not repeatable** — The random sequence used for noise generation is not repeatable.
- **Repeatable** — The random sequence used for noise generation is repeatable, with a system-generated seed.
- **Specify seed** — The random sequence used for noise generation is repeatable, and you control the seed by using the **Seed** parameter.

### **Dependencies**

Enabled when the **Noise mode** parameter is set to Enabled.

### **Auto-generated seed used for repeatable option — Auto-generated random number seed**

random real number

Random number seed stored inside the block to make the random sequence repeatable. The parameter value is automatically generated using the MATLAB random number generator command. You can modify this parameter value, but it gets overwritten by a new random value if you copy the block to another block in the model. Therefore, if you want to control the seed of the random sequence, use the **Specify seed** option for the **Repeatability** parameter and specify the desired seed value using the **Seed** parameter.

### **Dependencies**

Enabled when the **Repeatability** parameter is set to Repeatable.

### **Seed — Random number seed**

0 (default)

Seed used by the noise random number generator.

### **Dependencies**

Enabled when the **Repeatability** parameter is set to **Specify seed**.



**Device simulation temperature — Temperature of resistor at the start of the simulation**

25 °C (default)

The temperature of the resistor at the start of the simulation.

**Dependencies**

Enabled when the **Noise mode** parameter is set to Enabled.

For blocks with an exposed thermal port, this parameter is disabled. Instead, use the **Variables** tab to set the initial temperature target. For more information, see “Variables” on page 1-473.

**Thermal**

This section appears only for blocks with exposed thermal port. For more information, see “Thermal Port” on page 1-472.

**Resistance linear temperature coefficient — Specifies how the resistance value changes with temperature**

0 1/K (default)

The coefficient  $T_{C1}^{eff}$  in the equation that describes resistance as a function of temperature. See “Thermal Port” on page 1-472 for details.

**Resistance quadratic temperature coefficient — Specifies how the resistance value changes with temperature**0 1/K<sup>2</sup> (default)

The coefficient  $T_{C2}^{eff}$  in the equation that describes resistance as a function of temperature. See “Thermal Port” on page 1-472 for details.

**Measurement temperature — Temperature corresponding to nominal resistance**

25 °C (default)

The temperature  $T_0$ , for which the nominal resistance  $R$  is specified.

**Thermal mass — Thermal mass associated with port H**

100 J/K (default)

Thermal mass associated with the thermal port H. It represents the energy required to raise the temperature of the thermal port by one degree.

## **See Also**

Fault | Resistor

**Introduced in R2017b**

# Diode

Diode model; piecewise linear, piecewise linear zener, or exponential diode



## Library

Semiconductor Devices

## Description

The Diode block represents one of the following types of diodes:

- “Piecewise Linear” on page 1-485
- “Piecewise Linear Zener” on page 1-485
- “Exponential” on page 1-486

### Piecewise Linear

The piecewise linear diode model is the same model found in the Simscape Diode block, with the addition of a fixed junction capacitance. If the diode forward voltage exceeds the value specified in the **Forward voltage** parameter, the diode behaves as a linear resistor with the resistance specified in the **On resistance** parameter. Otherwise, the diode behaves as a linear resistor with the small conductance specified in the **Off conductance** parameter. Zero voltage across the diode results in zero current flowing.

### Piecewise Linear Zener

The piecewise linear zener diode model behaves like the piecewise linear diode model for bias voltages above  $-V_z$ , where  $V_z$  is the **Reverse breakdown voltage  $V_z$**  parameter value. For voltages less than  $-V_z$ , the diode behaves as a linear resistor with the low Zener resistance specified in the **Zener resistance  $R_z$**  parameter. This diode model also includes a fixed junction capacitance.

---

**Note** The **Reverse breakdown voltage Vz** parameter is defined as a positive number. The p-n voltage at breakdown is  $-Vz$ , which is negative.

---

## Exponential

The exponential diode model provides the following relationship between the diode current  $I$  and the diode voltage  $V$ :

$$I = IS \cdot \left( e^{\frac{qV}{NkT_{m1}}} - 1 \right) \quad V > -BV$$

$$I = -IS \cdot \left( e^{\frac{-q(V+Vz)}{kT_{m1}}} - e^{\frac{qV}{NkT_{m1}}} \right) \quad V \leq -BV$$

where:

- $q$  is the elementary charge on an electron (1.602176e-19 Coulombs).
- $k$  is the Boltzmann constant (1.3806503e-23 J/K).
- $BV$  is the **Reverse breakdown voltage BV** parameter value.
- $N$  is the emission coefficient.
- $IS$  is the saturation current.
- $T_{m1}$  is the temperature at which the diode parameters are specified, as defined by the **Measurement temperature** parameter value.

When  $(qV / NkT_{m1}) > 80$ , the block replaces  $e^{\frac{qV}{NkT_{m1}}}$  with  $(qV / NkT_{m1} - 79)e^{80}$ , which matches the gradient of the diode current at  $(qV / NkT_{m1}) = 80$  and extrapolates linearly.

When  $(qV / NkT_{m1}) < -79$ , the block replaces  $e^{\frac{qV}{NkT_{m1}}}$  with  $(qV / NkT_{m1} + 80)e^{-79}$ , which also matches the gradient and extrapolates linearly. Typical electrical circuits do not reach these extreme values. The block provides this linear extrapolation to help convergence when solving for the constraints during simulation.

When you select Use parameters  $IS$  and  $N$  for the **Parameterization** parameter, you specify the diode in terms of the **Saturation current IS** and **Emission coefficient N**

parameters. When you select Use I-V curve data points for the **Parameterization** parameter, you specify two voltage and current measurement points on the diode I-V curve and the block derives the  $IS$  and  $N$  values. The block then calculates  $IS$  and  $N$  as follows:

- $$N = ((V_1 - V_2) / V_t) / (\log(I_1) - \log(I_2))$$
- $$IS = (I_1 / (\exp(V_1 / (NV_t)) - 1) + I_2 / (\exp(V_2 / (NV_t)) - 1)) / 2$$

where:

- $V_t = kT_{m1} / q$ .
- $V_1$  and  $V_2$  are the values in the **Voltages [V1 V2]** vector.
- $I_1$  and  $I_2$  are the values in the **Currents [I1 I2]** vector.

When you select Use an I-V data point and  $IS$  for the **Parameterization** parameter, then the block calculates  $N$  as follows:

$$N = V_1 / \left( V_t \log \left( \frac{I_1}{IS} + 1 \right) \right)$$

When you select Use an I-V data point and  $N$  for the **Parameterization** parameter, then the block calculates  $IS$  as follows:

$$IS = I_1 / (\exp(V_1 / (NV_t)) - 1)$$

The exponential diode model provides the option to include a junction capacitance:

- When you select Include fixed or zero junction capacitance for the **Junction capacitance** parameter, the capacitance is fixed.
- When you select Use parameters  $CJO$ ,  $VJ$ ,  $M$  &  $FC$  for the **Junction capacitance** parameter, the block uses the coefficients  $CJO$ ,  $VJ$ ,  $M$ , and  $FC$  to calculate a junction capacitance that depends on the junction voltage.
- When you select Use C-V curve data points for the **Junction capacitance** parameter, the block uses three capacitance values on the C-V capacitance curve to estimate  $CJO$ ,  $VJ$ , and  $M$  and uses these values with the specified value of  $FC$  to calculate a junction capacitance that depends on the junction voltage. The block calculates  $CJO$ ,  $VJ$ , and  $M$  as follows:

- $CJ0 = C_1((V_{R2} - V_{R1}) / (V_{R2} - V_{R1}(C_2 / C_1)^{-1/M}))^M$
- $VJ = -(-V_{R2}(C_1 / C_2)^{-1/M} + V_{R1}) / (1 - (C_1 / C_2)^{-1/M})$
- $M = \log(C_3 / C_2) / \log(V_{R2} / V_{R3})$

where:

- $V_{R1}$ ,  $V_{R2}$ , and  $V_{R3}$  are the values in the **Reverse bias voltages [VR1 VR2 VR3]** vector.
- $C_1$ ,  $C_2$ , and  $C_3$  are the values in the **Corresponding capacitances [C1 C2 C3]** vector.

It is not possible to estimate  $FC$  reliably from tabulated data, so you must specify its value using the **Capacitance coefficient FC** parameter. In the absence of suitable data for this parameter, use a typical value of 0.5.

The reverse bias voltages (defined as positive values) should satisfy  $V_{R3} > V_{R2} > V_{R1}$ . This means that the capacitances should satisfy  $C_1 > C_2 > C_3$  as reverse bias widens the depletion region and hence reduces capacitance. Violating these inequalities results in an error. Voltages  $V_{R2}$  and  $V_{R3}$  should be well away from the Junction potential  $VJ$ . Voltage  $V_{R1}$  should be less than the Junction potential  $VJ$ , with a typical value for  $V_{R1}$  being 0.1 V.

The voltage-dependent junction is defined in terms of the capacitor charge storage  $Q_j$  as:

- For  $V < FC \cdot VJ$ :

$$Q_j = CJ0 \cdot (VJ / (M - 1)) \cdot ((1 - V / VJ)^{1-M} - 1)$$

- For  $V \geq FC \cdot VJ$ :

$$Q_j = CJ0 \cdot F_1 + (CJ0 / F_2) \cdot (F_3 \cdot (V - FC \cdot VJ) + 0.5(M / VJ) \cdot (V^2 - (FC \cdot VJ)^2))$$

where:

- $F_1 = (VJ / (1 - M)) \cdot (1 - (1 - FC)^{1-M})$
- $F_2 = (1 - FC)^{1+M}$

- $F_3 = 1 - FC \cdot (1 + M)$

These equations are the same as used in [2 on page 1-501], except that the temperature dependence of  $VJ$  and  $FC$  is not modeled. This model does not include the diffusion capacitance term that affects performance for high frequency switching applications.

For applications such as commutation diodes it can be important to model diode charge dynamics. When a forward-biased diode has a reverse voltage applied across it, it takes time for the charge to dissipate and hence for the diode to turn off. The time taken for the diode to turn off is captured primarily by the transit time parameter. Once the diode is off, any remaining charge then dissipates, the rate at which this happens being determined by the carrier lifetime.

The Diode block uses the model of Lauritzen and Ma [3 on page 1-501] to capture these effects. The three defining equations are:

$$I = \frac{q_E - q_M}{TT}$$

$$\frac{dq_M}{dt} + \frac{q_M}{\tau} - \frac{q_E - q_M}{TT} = 0$$

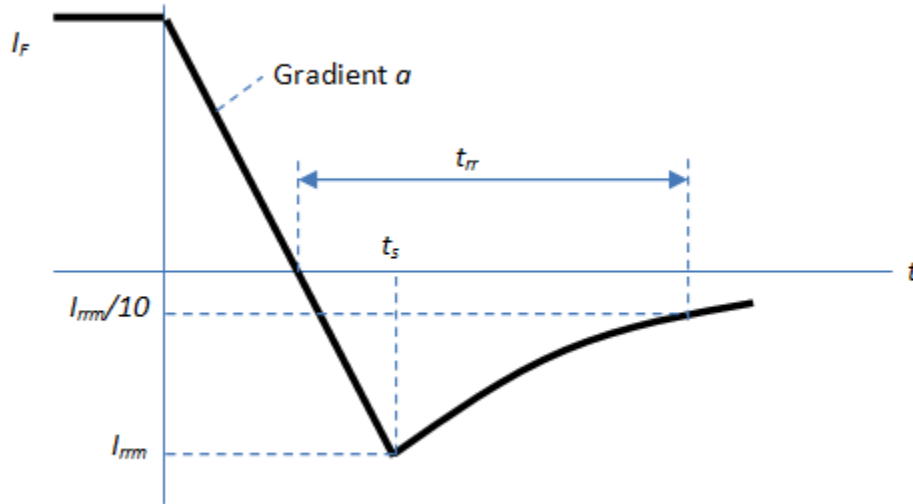
$$q_E = (\tau + TT) IS \left( \exp\left(\frac{V}{N \cdot V_t}\right) - 1 \right)$$

where:

- $I$  is the diode current.
- $V$  is the diode voltage.
- $N$  is the emission coefficient.
- $q_E$  is the junction charge.
- $q_M$  is the total stored charge.
- $TT$  is the transit time.
- $\tau$  is the carrier lifetime.

Datasheets do not typically provide values for  $TT$  and  $\tau$ . Therefore the Diode block provides an alternative parameterization in terms of **Peak reverse current,  $I_{rrm}$**  and

**Reverse recovery time,  $t_{rr}$ .** Equivalent values for  $TT$  and  $\tau$  are calculated from these values, plus information on the initial forward current and rate of change of current used in the test circuit when measuring  $I_{rrm}$  and  $t_{rr}$ . The test circuit can consist of a series voltage source, resistor, inductor and the diode. The polarity of the voltage source is switched so as to move the diode from forward conduction to reverse biased. The following figure shows an idealized diode current response.



The value of the series resistor and applied voltage value determine the initial current  $I_F$ . The value of the series inductance and the applied reverse voltage value determine the current gradient,  $a$ .

The precise values of peak reverse current and reverse recovery time depend on the test circuit used. Also, junction capacitance has some effect on the current recovery characteristic. However, a junction capacitor value that dominates the response is physically unrealistic.

Only the exponential diode supports modeling of the diode charge dynamics. If you select the Exponential for the **Diode model** parameter, then the **Capacitance** tab contains an additional parameter called **Charge dynamics**. Select between the three options:

- Do not model charge dynamics



- Use peak reverse current and reverse recovery time
- Use transit time and carrier lifetime

The default behavior for the Diode is that dependence on temperature is not modeled, and the device is simulated at the temperature for which you provide block parameters. The Exponential diode model contains several options for modeling the dependence of the diode current-voltage relationship on the temperature during simulation. Temperature dependence of the junction capacitance is not modeled, this being a much smaller effect.

When including temperature dependence, the diode defining equation remains the same. The measurement temperature value,  $T_{m1}$ , is replaced with the simulation temperature,  $T_s$ . The saturation current,  $IS$ , becomes a function of temperature according to the following equation:

$$IS_{T_s} = IS_{T_{m1}} \cdot (T_s / T_{m1})^{XTI/N} \cdot \exp\left(-\frac{EG}{NkT_s} (1 - T_s / T_{m1})\right)$$

where:

- $T_{m1}$  is the temperature at which the diode parameters are specified, as defined by the **Measurement temperature** parameter value.
- $T_s$  is the simulation temperature.
- $IS_{T_{m1}}$  is the saturation current at measurement temperature.
- $IS_{T_s}$  is the saturation current at simulation temperature. This is the saturation current value used in the standard diode equation when temperature dependence is modeled.
- $EG$  is the energy gap for the semiconductor type measured in Joules. The value for silicon is usually taken to be 1.11 eV, where 1 eV is 1.602e-19 Joules.
- $XTI$  is the saturation current temperature exponent. This is usually set to 3.0 for pn-junction diodes, and 2.0 for Schottky barrier diodes.
- $N$  is the emission coefficient.
- $k$  is the Boltzmann constant (1.3806503e-23 J/K).

Appropriate values for  $XTI$  and  $EG$  depend on the type of diode and the semiconductor material used. Default values for particular material types and diode types capture approximate behavior with temperature. The block provides default values for common types of diode.

In practice, the values of  $XTI$  and  $EG$  need tuning to model the exact behavior of a particular diode. Some manufacturers quote these tuned values in a SPICE Netlist, and you can read off the appropriate values. Otherwise you can determine improved estimates for  $EG$  by using a datasheet-defined current-voltage data point at a higher temperature. The block provides a parameterization option for this. It also gives the option of specifying the saturation current at a higher temperature  $IS_{Tm2}$  directly.

You can also tune the values of  $XTI$  and  $EG$  yourself, to match lab data for your particular device. You can use Simulink Design Optimization™ software to help tune the values for  $XTI$  and  $EG$ .

---

**Caution** Device temperature behavior is also dependent on the emission coefficient. An inappropriate value for the emission coefficient can give incorrect temperature dependence, because saturation current is a function of the ratio of  $EG$  to  $N$ .

---

If defining a finite reverse breakdown voltage  $BV$ , then the value of the reverse breakdown voltage is modulated by the reverse breakdown temperature coefficient  $TCV$  (specified using the **Reverse breakdown voltage temperature coefficient, dBV/dT** parameter):

$$BV_{T_s} = BV_{T_{m1}} - TCV \cdot (T_s - T_{m1})$$

## Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal port H on the block icon, and adds the **Thermal Port** tab to the block dialog box.

Use the thermal port to simulate the effects of generated heat and device temperature. For more information on using thermal ports and on the **Thermal Port** tab parameters, see “Simulating Thermal Effects in Semiconductors”.

## Variables

Use the **Variables** section of the block interface to set the priority and initial target values for the block variables prior to simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

## Basic Assumptions and Limitations

The Exponential diode model has the following limitations:

- When you select **Use I-V curve data points** for the **Parameterization** parameter, choose a pair of voltages near the diode turn-on voltage. Typically, this is in the range from 0.05 to 1 Volt. Using values outside of this region may lead to numerical issues and poor estimates for  $I_S$  and  $N$ .
- The block does not account for temperature-dependent effects on the junction capacitance.
- You may need to use nonzero ohmic resistance and junction capacitance values to prevent numerical simulation issues, but the simulation may run faster with these values set to zero.

## Parameters

- “Main” on page 1-493
- “Breakdown” on page 1-496
- “Capacitance” on page 1-496
- “Temperature Dependence” on page 1-499

## Main

### Diode model

Select one of the following diode models:

- **Piecewise Linear (Foundation Library)** — Use a piecewise linear model for the diode, as described in “Piecewise Linear” on page 1-485. This is the default method.
- **Piecewise Linear Zener** — Use a piecewise linear model with reverse breakdown characteristics for the diode, as described in “Piecewise Linear Zener” on page 1-485.
- **Exponential** — Use a standard exponential model for the diode, as described in “Exponential” on page 1-486.

## Forward voltage

Minimum voltage that needs to be applied for the diode to become forward-biased. This parameter is only visible when you select **Piecewise Linear (Foundation Library)** or **Piecewise Linear Zener** for the **Diode model** parameter. The default value is 0.6 V.

## On resistance

The resistance of the diode when it is forward biased. This parameter is only visible when you select **Piecewise Linear (Foundation Library)** or **Piecewise Linear Zener** for the **Diode model** parameter. The default value is 0.3  $\Omega$ .

## Off conductance

The conductance of the diode when it is reverse biased. This parameter is only visible when you select **Piecewise Linear (Foundation Library)** or **Piecewise Linear Zener** for the **Diode model** parameter. The default value is  $1e-08$  1/ $\Omega$ .

## Parameterization

Select one of the following methods for model parameterization:

- Use two I-V curve data points — Specify measured data at two points on the diode I-V curve. This is the default method.
- Use parameters *IS* and *N* — Specify saturation current and emission coefficient.
- Use an I-V data point and *IS* — Specify measured data at a single point on the diode I-V curve in combination with the saturation current.
- Use an I-V data point and *N* — Specify measured data at a single point on the diode I-V curve in combination with the emission coefficient.

This parameter is only visible when you select **Exponential** for the **Diode model** parameter.

## Currents [I1 I2]

A vector of the current values at the two points on the diode I-V curve that the block uses to calculate *IS* and *N*. This parameter is only visible when you select **Exponential** for the **Diode model** parameter and **Use two I-V curve data points** for the **Parameterization** parameter. The default value is [ 0.0137 0.545 ] A.

## Voltages [V1 V2]

A vector of the voltage values at the two points on the diode I-V curve that the block uses to calculate *IS* and *N*. This parameter is only visible when you select

Exponential for the **Diode model** parameter and Use two I-V curve data points for the **Parameterization** parameter. The default value is [ 0.6 0.7 ] V.

### **Current I1**

A current value at the point on the diode I-V curve that the block uses for calculations. This parameter is only visible when you select Exponential for the **Diode model** parameter and either Use an I-V data point and IS or Use an I-V data point and N for the **Parameterization** parameter. Depending on the **Parameterization** value, the block uses this parameter to calculate either *N* or *IS*. The default value is 0.07 A.

### **Voltage V1**

A voltage value at the point on the diode I-V curve that the block uses for calculations. This parameter is only visible when you select Exponential for the **Diode model** parameter and either Use an I-V data point and IS or Use an I-V data point and N for the **Parameterization** parameter. Depending on the **Parameterization** value, the block uses this parameter to calculate either *N* or *IS*. The default value is 0.7 V.

### **Saturation current, IS**

The magnitude of the current that the ideal diode equation approaches asymptotically for very large reverse bias levels. This parameter is only visible when you select Exponential for the **Diode model** parameter and either Use parameters IS and N or Use an I-V data point and IS for the **Parameterization** parameter. The default value is 1e-14 A.

### **Emission coefficient, N**

The diode emission coefficient or ideality factor. This parameter is only visible when you select Exponential for the **Diode model** parameter and either Use parameters IS and N or Use an I-V data point and N for the **Parameterization** parameter. The default value is 1.

### **Ohmic resistance, RS**

The series diode connection resistance. This parameter is only visible when you select Exponential for the **Diode model** parameter. The default value is 0.01  $\Omega$ .

### **Measurement temperature**

The temperature  $T_{m1}$  at which IS or the I-V curve was measured. This parameter is only visible when you select Exponential for the **Diode model** parameter. The default value is 25 °C.

## Breakdown

This section is not applicable for Piecewise Linear diode models.

### Zener resistance $R_z$

The resistance of the diode when the voltage is less than the **Reverse breakdown voltage  $V_z$**  value. This parameter is only visible when you select Piecewise Linear Zener for the **Diode model** parameter. The default value is  $0.3 \Omega$ .

### Reverse breakdown voltage $V_z$

The reverse voltage below which the diode resistance changes to the **Zener resistance  $R_z$**  value. This parameter is only visible when you select Piecewise Linear Zener for the **Diode model** parameter. The default value is 50 V.

### Reverse breakdown voltage $BV$

The reverse voltage below which to model the rapid increase in conductance that occurs at diode breakdown. This parameter is only visible when you select Exponential for the **Diode model** parameter. The default value is Inf V, which effectively omits reverse breakdown from the model.

## Capacitance

### Junction capacitance

- When you select Piecewise Linear (Foundation Library) or Piecewise Linear Zener for the **Diode model** parameter, the **Junction capacitance** parameter is the fixed junction capacitance value. The default value is 5 pF.
- When you select Exponential for the **Diode model** parameter, the **Junction capacitance** parameter lets you select one of the following options for modeling the junction capacitance:
  - Include fixed or zero junction capacitance — Model the junction capacitance as a fixed value.
  - Use C-V curve data points — Specify measured data at three points on the diode C-V curve.
  - Use parameters  $CJ_0$ ,  $VJ$ ,  $M$  &  $FC$  — Specify zero-bias junction capacitance, junction potential, grading coefficient, and forward-bias depletion capacitance coefficient.

### Zero-bias junction capacitance **CJ0**

The value of the capacitance placed in parallel with the exponential diode term. This parameter is only visible when you select **Exponential** for the **Diode model** parameter and **Include fixed or zero junction capacitance** or **Use parameters CJ0, VJ, M & FC** for the **Junction capacitance** parameter. The default value is 5 pF.

### Reverse bias voltages [**VR1 VR2 VR3**]

A vector of the reverse bias voltage values at the three points on the diode C-V curve that the block uses to calculate *CJ0*, *VJ*, and *M*. This parameter is only visible when you select **Exponential** for the **Diode model** parameter and **Use C-V curve data points** for the **Junction capacitance** parameter. The default value is [ 0.1 10 100 ] V.

### Corresponding capacitances [**C1 C2 C3**]

A vector of the capacitance values at the three points on the diode C-V curve that the block uses to calculate *CJ0*, *VJ*, and *M*. This parameter is only visible when you select **Exponential** for the **Diode model** parameter and **Use C-V curve data points** for the **Junction capacitance** parameter. The default value is [ 3.5 1 0.4 ] pF.

### Junction potential **VJ**

The junction potential. This parameter is only visible when you select **Exponential** for the **Diode model** parameter and **Use parameters CJ0, VJ, M & FC** for the **Junction capacitance** parameter. The default value is 1 V.

### Grading coefficient **M**

The grading coefficient. This parameter is only visible when you select **Exponential** for the **Diode model** parameter and **Use parameters CJ0, VJ, M & FC** for the **Junction capacitance** parameter. The default value is 0.5.

### Capacitance coefficient **FC**

Fitting coefficient that quantifies the decrease of the depletion capacitance with applied voltage. This parameter is only visible when you select **Exponential** for the **Diode model** parameter and **Use C-V curve data points** or **Use parameters CJ0, VJ, M & FC** for the **Junction capacitance** parameter. The default value is 0.5.

### Charge model

Select one of the following methods for charge dynamics parameterization:

- **Do not model charge dynamics** — Do not include charge dynamics modeling. This is the default method.

- Use peak reverse current and reverse recovery time — Model charge dynamics by providing values for peak reverse current,  $I_{rrm}$ , and reverse recovery time,  $t_{rr}$ , plus information on the initial forward current and rate of change of current used in the test circuit when measuring  $I_{rrm}$  and  $t_{rr}$ . Use this option if the manufacturer datasheet does not provide values for transit time,  $TT$ , and carrier lifetime,  $\tau$ .
- Use transit time and carrier lifetime — Model charge dynamics by providing values for transit time,  $TT$ , and carrier lifetime,  $\tau$ .

### **Peak reverse current, $I_{rrm}$**

The peak reverse current measured in a test circuit. This parameter is only visible when you select **Exponential** for the **Diode model** parameter and **Use peak reverse current and reverse recovery time** for the **Charge model** parameter. The default value is 7.15 A.

### **Starting forward current when measuring $I_{rrm}$**

The initial forward current when measuring peak reverse current. This parameter is only visible when you select **Exponential** for the **Diode model** parameter and **Use peak reverse current and reverse recovery time** for the **Charge model** parameter. The default value is 4 A.

### **Rate of change of current when measuring $I_{rrm}$**

The rate of change of current when measuring peak reverse current. This parameter is only visible when you select **Exponential** for the **Diode model** parameter and **Use peak reverse current and reverse recovery time** for the **Charge model** parameter. The default value is -750 A/us.

### **Reverse recovery time, $t_{rr}$**

The time between the point where the current initially goes to zero when the diode turns off, and the point where the current falls to less than ten percent of the peak reverse current. This parameter is only visible when you select **Exponential** for the **Diode model** parameter and **Use peak reverse current and reverse recovery time** for the **Charge model** parameter. The default value is 115 ns.

### **Transit time, $TT$**

A measure of how long it takes carriers to cross the diode junction. This parameter is only visible when you select **Exponential** for the **Diode model** parameter and **Use transit time and carrier lifetime** for the **Charge model** parameter. The default value is 50 ns.



### Carrier lifetime, tau

A measure of how long it takes for the carriers to dissipate once the diode is no longer conducting. This parameter is only visible when you select **Exponential** for the **Diode model** parameter and **Use transit time and carrier lifetime** for the **Charge model** parameter. The default value is 100 ns.

## Temperature Dependence

This section is applicable for Exponential diode models only.

### Parameterization

Select one of the following methods for temperature dependence parameterization:

- **None – Simulate at parameter measurement temperature –** Temperature dependence is not modeled, or the model is simulated at the measurement temperature  $T_{m1}$  (as specified by the **Measurement temperature** parameter on the **Main** tab). This is the default method.
- **Use an I-V data point at second measurement temperature –** If you select this option, you specify a second measurement temperature  $T_{m2}$ , and the current and voltage values at this temperature. The model uses these values, along with the parameter values at the first measurement temperature  $T_{m1}$ , to calculate the energy gap value.
- **Specify saturation current at second measurement temperature –** If you select this option, you specify a second measurement temperature  $T_{m2}$ , and saturation current value at this temperature. The model uses these values, along with the parameter values at the first measurement temperature  $T_{m1}$ , to calculate the energy gap value.
- **Specify the energy gap, EG –** Specify the energy gap value directly.

### Current I1 at second measurement temperature

Specify the diode current  $I1$  value when the voltage is  $V1$  at the second measurement temperature. This parameter is only visible when you select **Use an I-V data point at second measurement temperature** for the **Parameterization** parameter. The default value is 0.245 A.

### Voltage V1 at second measurement temperature

Specify the diode voltage  $V1$  value when the current is  $I1$  at the second measurement temperature. This parameter is only visible when you select **Use an I-V data**

point at second measurement temperature for the **Parameterization** parameter. The default value is 0.5 V.

#### **Saturation current, IS, at second measurement temperature**

Specify the saturation current *IS* value at the second measurement temperature. This parameter is only visible when you select Specify saturation current at second measurement temperature for the **Parameterization** parameter. The default value is 1.25e-7 A.

#### **Second measurement temperature**

Specify the value for the second measurement temperature. This parameter is only visible when you select either Use an I-V data point at second measurement temperature or Specify saturation current at second measurement temperature for the **Parameterization** parameter. The default value is 125 °C.

#### **Energy gap parameterization**

This parameter is only visible when you select Specify the energy gap, EG for the **Parameterization** parameter. It lets you select a value for the energy gap from a list of predetermined options, or specify a custom value:

- Use nominal value for silicon (EG=1.11eV) — This is the default.
- Use nominal value for 4H-SiC silicon carbide (EG=3.23eV)
- Use nominal value for 6H-SiC silicon carbide (EG=3.00eV)
- Use nominal value for germanium (EG=0.67eV)
- Use nominal value for gallium arsenide (EG=1.43eV)
- Use nominal value for selenium (EG=1.74eV)
- Use nominal value for Schottky barrier diodes (EG=0.69eV)
- Specify a custom value — If you select this option, the **Energy gap, EG** parameter appears in the dialog box, to let you specify a custom value for *EG*.

#### **Energy gap, EG**

Specify a custom value for the energy gap, *EG*. This parameter is only visible when you select Specify a custom value for the **Energy gap parameterization** parameter. The default value is 1.11 eV.

#### **Saturation current temperature exponent parameterization**

Select one of the following options to specify the saturation current temperature exponent value:

- Use nominal value for pn-junction diode ( $X_{TI}=3$ ) — This is the default.
- Use nominal value for Schottky barrier diode ( $X_{TI}=2$ )
- Specify a custom value — If you select this option, the **Saturation current temperature exponent,  $X_{TI}$**  parameter appears in the dialog box, to let you specify a custom value for  $X_{TI}$ .

### Saturation current temperature exponent, $X_{TI}$

Specify a custom value for the saturation current temperature exponent,  $X_{TI}$ . This parameter is only visible when you select Specify a custom value for the **Saturation current temperature exponent parameterization** parameter. The default value is 3.

### Reverse breakdown voltage temperature coefficient, $dBV/dT$

This coefficient modulates the reverse breakdown voltage  $BV$ . If you define the reverse breakdown voltage  $BV$  as a positive quantity, a positive value for  $TCV$  implies that the magnitude of the reverse breakdown voltage decreases with temperature. The default value is 0 V/K.

### Device simulation temperature

Specify the value for the temperature  $T_s$ , at which the device is to be simulated. The default value is 25 °C.

## Ports

The block has the following ports:

+

Electrical conserving port associated with the diode positive terminal

-

Electrical conserving port associated with the diode negative terminal

## References

- [1] MH. Ahmed and P.J. Spreadbury. *Analogue and digital electronics for engineers*. 2nd Edition, Cambridge University Press, 1984.

- [2] G. Massobrio and P. Antognetti. *Semiconductor Device Modeling with SPICE*. 2nd Edition, McGraw-Hill, 1993.
- [3] Lauritzen, P.O. and C.L. Ma. "A Simple Diode Model with Reverse Recovery." *IEEE Transactions on Power Electronics*. Vol. 6, No. 2, April 1991.

# Diode (Piecewise Linear)

Piecewise linear diode



## Library

Simscape / Electrical / Power Systems / Semiconductors / Fundamental Components

## Description

The Diode (Piecewise Linear) block models a piecewise linear diode.

If the voltage across the diode exceeds the value specified in the block **Forward voltage** parameter, then the diode behaves like a linear resistor plus a series voltage source. The value of the **On resistance** parameter specifies the resistance of the linear resistor.

If the voltage across the diode is less than the forward voltage, the diode behaves like a linear resistor with low conductance specified by the value of the **Off conductance** parameter.

When forward biased, the series voltage source is given by

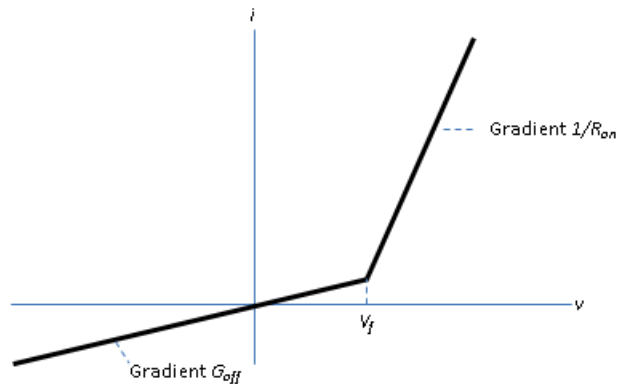
$$V = V_f(1 - R_{on}G_{off}),$$

where:

- $V$  is the voltage supplied by the series voltage source.
- $V_f$  is the forward voltage.
- $R_{on}$  is the on resistance.
- $G_{off}$  is the off conductance.

The  $R_{on} * G_{off}$  term ensures that the diode current is exactly zero when the voltage across it is zero.

The figure shows a typical I-V characteristic for a diode device.



## Modeling Variants

The block provides a thermal modeling variant. To select a variant, right-click the block in your model. From the context menu, select **Simscape > Block choices**, and then one of these variants:

- **No thermal port** — This variant does not simulate heat generation in the device. This variant is the default.
- **Show thermal port** — This variant contains a thermal port that allows you to model the heat that conduction losses generate. For numerical efficiency, the thermal state does not affect the electrical behavior of the block. The thermal port is hidden by default. When you select a thermal variant of the block, the thermal port appears.

## Ports

+

Electrical conserving port associated with the diode positive terminal

-

Electrical conserving port associated with the diode negative terminal

H

Thermal conserving port. The thermal port is optional and is hidden by default. To enable this port, select a variant that includes a thermal port.

## Parameters

### Forward voltage

Minimum voltage required across the + and - block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**. The default value is 0.8 V.

### On resistance

Rate of change of voltage versus current above the **Forward voltage**. The default value is 0.001 Ohm.

### Off conductance

Conductance of the reverse-biased diode. The default value is  $1e-5$  1/Ohm.

### Number of series diodes

The number of diodes connected in series between the + and - block ports. Each diode has the **Forward voltage**, **On resistance**, and **Off conductance** that you specify. The default value is 1.

### Number of parallel diodes

The number of parallel diodes, or number of parallel paths formed by series-connected diodes, between the + and - block ports. Each diode has the **Forward voltage**, **On resistance**, and **Off conductance** that you specify. The default value is 1.

## See Also

Commutation Diode | GTO | IGBT | Ideal Semiconductor Switch | MOSFET | Thyristor (Piecewise Linear)

## Topics

“Quantifying IGBT Thermal Losses”

“Simulate Thermal Losses in Semiconductors”

**Introduced in R2013b**



# Discrete PI Controller

Discrete-time PI controller with external anti-windup input

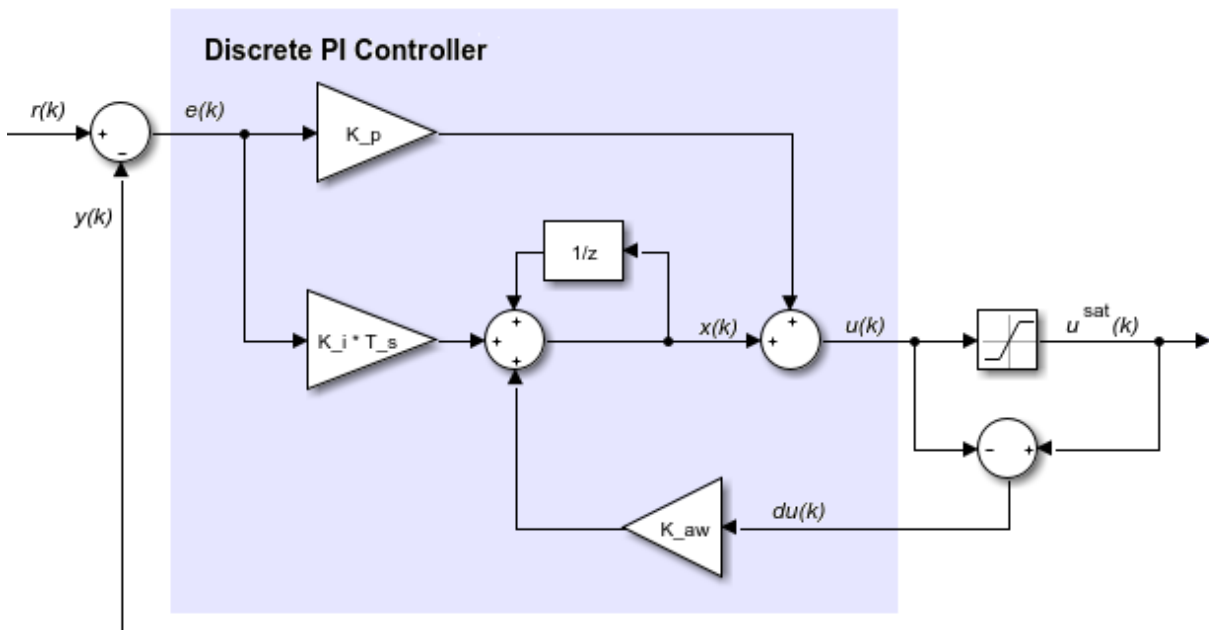
**Library:** Simscape / Electrical / Power Systems / Control / General Control



## Description

The Discrete PI Controller block implements discrete PI control with external anti-windup input.

This diagram is the equivalent circuit for the controller with external anti-windup input.



## Equations

The Discrete PI Controller block calculates the control signal using the backward Euler discretization method:

$$u(k) = \left[ K_p + (K_i + du(k)K_{aw}) \frac{T_s z}{z-1} \right] e(k),$$

where

- $u$  is the control signal.
- $K_p$  is the proportional gain coefficient.
- $K_i$  is the integral gain coefficient.
- $K_{aw}$  is the anti-windup gain coefficient.
- $T_s$  is the sampling period.
- $e$  is the error signal.

To prevent excessive overshoot, the block can use back calculation to implement an external anti-windup mechanism. It inputs  $du(k)$ , the difference between the saturated control signal,  $u^{sat}(k)$ , and the calculated unsaturated control signal,  $u(k)$ . It then multiplies the difference by the anti-windup coefficient and adds the amplified signal from the integral gain.

## Ports

### Input

#### **e — Error signal**

scalar

Error signal,  $e(k)$ , obtained as the difference between the reference,  $r(k)$ , and measurement,  $y(k)$ , signals.

Data Types: `single` | `double`

#### **du — Control signal saturation**

scalar

Difference,  $du(k)$ , between the saturated  $u^{sat}(k)$  and the unsaturated control signals,  $u(k)$ . If  $du(k)$  is zero, the anti-windup is disabled.

Description

Data Types: single | double

### **Reset — Integrator gain reset**

scalar

External reset (rising edge) signal for the integrator.

Data Types: single | double

## **Output**

### **u — Control signal**

scalar

Control signal,  $u(k)$ .

Data Types: single | double

## **Parameters**

### **Proportional gain — $K_p$**

1 (default) | positive scalar

Proportional gain,  $K_p$ , of the PI controller.

### **Integral gain — $K_i$**

1 (default) | positive scalar

Integral gain,  $K_i$ , of the PI controller.

### **Anti-windup gain — $K_{aw}$**

1 (default) | positive scalar

Anti-windup gain,  $K_{aw}$ , of the PI controller.

### **Integrator initial condition — Initial integrator value**

0 (default) | scalar

Value of the integrator at simulation start time.

## **Sample time (-1 for inherited) — Sampling interval**

-1 (default) | default value or a positive number

Time interval between samples. If the block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

## **References**

[1] Åström, K. and T. Häggglund. *Advanced PID Control*. Research Triangle Park, NC: ISA, 2005.

## **See Also**

### **Blocks**

Discrete PI Controller with Integral Anti-Windup

**Introduced in R2017b**

# Discrete PI Controller with Integral Anti-Windup

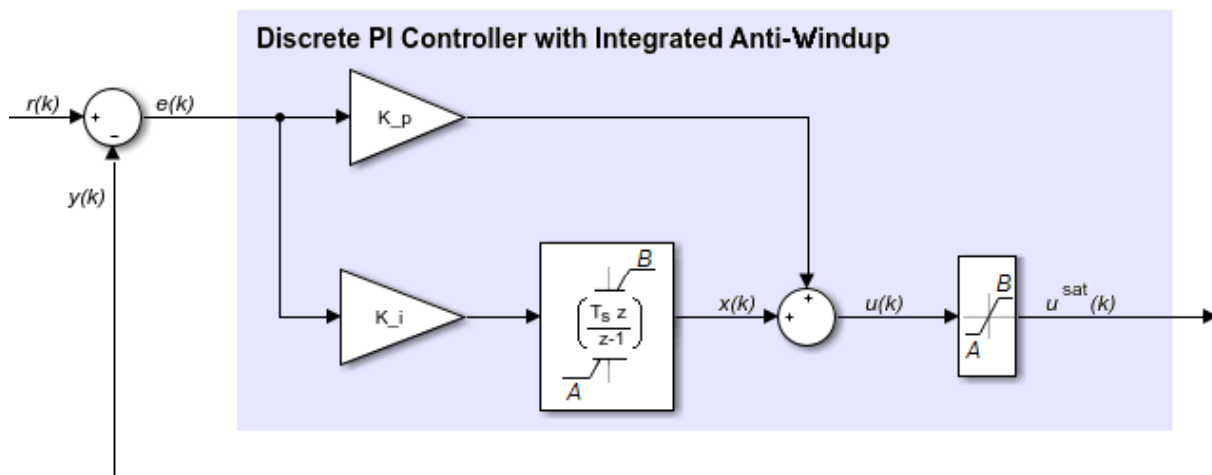
Discrete-time PI control with integral anti-windup

**Library:** Simscape / Electrical / Power Systems / Control / General Control



## Description

The Discrete PI Controller with Integral Anti-Windup block implements discrete PI control with internal anti-windup. The figure shows the equivalent circuit for the controller with internal anti-windup.



## Equations

The block calculates the control signal using the backward Euler discretization method:

$$u(k) = \text{sat} \left( K_p e(k) + \text{sat} \left( K_i \frac{T_s z}{z-1} e(k), A, B \right), A, B \right),$$

$$\text{sat}(x, A, B) = \min(\max(x, A), B),$$

where:

- $u$  is the control signal.
- $K_p$  is the proportional gain coefficient.
- $e$  is the error signal.
- $K_i$  is the integral gain coefficient.
- $T_s$  is the sampling period.
- $A$  is the lower limit for saturation.
- $B$  is the upper limit for saturation.

## Ports

### Input

#### **e — Error signal**

scalar

Error signal,  $e(k)$ , obtained as the difference between the reference,  $r(k)$ , and measurement  $y(k)$  signals.

Data Types: `single` | `double`

#### **Reset — Integrator gain reset**

scalar

External reset (rising edge) signal for the integrator.

Data Types: `single` | `double`

## Output

### **u — Control signal**

scalar

Control signal,  $u(k)$ .

Data Types: `single` | `double`

## Parameters

### **Proportional gain — $K_p$**

1 (default) | positive scalar

Proportional gain,  $K_p$ , of the PI controller.

### **Integral gain — $K_i$**

1 (default) | positive scalar

Integral gain,  $K_i$ , of the PI controller.

### **Upper saturation limit — $B$**

5 (default) | scalar greater than the value of the **Lower saturation limit** parameter

Upper limit,  $B$ , of the output for the PI controller.

### **Lower saturation limit — $A$**

-5 (default) | scalar

Upper limit,  $A$ , of the output for the PI controller.

### **Integrator initial condition — Initial integrator value**

0 (default) | scalar

Value of the integrator at simulation start time.

### **Sample time (-1 for inherited) — Block sample time**

-1 (default) | 0 | positive scalar

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For inherited discrete-time operation, specify -1. For discrete-time operation, specify a positive integer. For continuous-time operation, specify 0.

If this block is in a masked subsystem, or other variant subsystem that allows you to switch between continuous operation and discrete operation, promote the sample time parameter. Promoting the sample time parameter ensures correct switching between the continuous and discrete implementations of the block. For more information, see “Promote Parameter to Mask” (Simulink).

## References

[1] *IEEE Recommended Practice for Excitation System Models for Power System Stability Studies*. IEEE Std 421.5/D39. Piscataway, NJ: IEEE-SA, 2015.

## See Also

### Blocks

Discrete PI Controller

### Topics

“Promote Parameter to Mask” (Simulink)

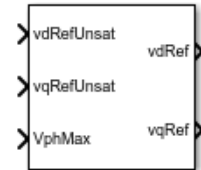
**Introduced in R2017b**



## d-q Voltage Limiter

Limit voltage in the rotor direct-quadrature reference frame

**Library:** Simscape / Electrical / Power Systems / Control / Protection

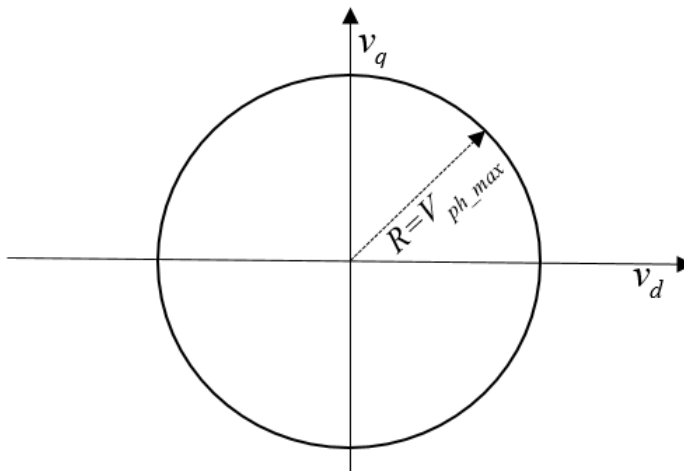


### Description

The d-q Voltage Limiter block implements a voltage limiter in the rotor direct-quadrature ( $d$ - $q$ ) reference frame.

### Electrical Defining Equations

The figure shows the circle that limits the  $d$ - $q$  voltage vector.



That is,

$$\sqrt{v_d^2 + v_q^2} \leq V_{ph\_max}$$

where:

- $v_d$  is the  $d$ -axis voltage.
- $v_q$  is the  $q$ -axis voltage.
- $V_{ph\_max}$  is the maximum phase voltage.

Three cases of voltage limiting are possible:

- $d$ -axis prioritization
- $q$ -axis prioritization
- $d$ - $q$  equivalence

If one axis is prioritized over the other axis, the constrained or saturated voltages are defined as

$$v_1^{sat} = \min\left(\max\left(v_1^{unsat}, -V_{ph\_max}\right), V_{ph\_max}\right)$$

and

$$v_2^{sat} = \min\left(\max\left(v_2^{unsat}, -V_{2\_max}\right), V_{2\_max}\right),$$

where:

- $$v_{2\_max} = \sqrt{\left(V_{ph\_max}\right)^2 - \left(v_1^{sat}\right)^2}$$
- $v_1$  is voltage of the prioritized axis.
- $v_2$  is voltage of the nonprioritized axis.

If neither axis is prioritized, the constrained voltages are defined as

$$v_d^{sat} = \min\left(\max\left(v_d^{unsat}, -V_{d\_max}\right), V_{d\_max}\right)$$

and

$$v_q^{sat} = \min\left(\max\left(v_q^{unsat}, -V_{q\_max}\right), V_{q\_max}\right),$$

where:

- $$V_{d\_max} = \frac{V_{ph\_max} |v_d^{unsat}|}{\sqrt{(v_d^{unsat})^2 + (v_q^{unsat})^2}}$$
- $$V_{q\_max} = \frac{V_{ph\_max} |v_q^{unsat}|}{\sqrt{(v_d^{unsat})^2 + (v_q^{unsat})^2}}$$

## Ports

### Input

**vdRefUnsatsat** —  $v_d^{unsat}$

scalar

Unsaturated direct-axis reference voltage.

Example: Example

Data Types: single | double

**VqRefUnsatsat** —  $v_q^{unsat}$

scalar

Unsaturated quadrature-axis reference voltage.

Example: Example

Data Types: single | double

**VphMax** —  $V_{ph\_max}$

scalar

Maximum phase voltage.

Data Types: single | double

## Output

**vdRef** —  $v_d^{sat}$

scalar

Saturated direct-axis reference voltage.

Data Types: single | double

**vqRef** —  $v_q^{sat}$

scalar

Saturated quadrature-axis reference voltage.

Example: Example

Data Types: single | double

## Parameters

**Axis prioritization — Prioritize the *d*- or *q* axis**

Q-axis (default) | D-axis | D-Q equivalence

Prioritize the direct-axis, the quadrature-axis, or neither axis.

**Sample time (-1 for inherited) — Sampling interval**

-1 (default) | default value or a positive number

Time interval between samples. If the block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

## See Also

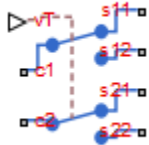
**Simscape Blocks**

Voltage Source

**Introduced in R2017b**

## DPDT Switch

Model double-pole double-throw switch



## Library

Passive Devices/Switches

## Description

The DPDT Switch block models a double-pole double-throw switch:

- When the switch is closed, ports c1 and c2 are connected to ports s12 and s22, respectively.
- When the switch is open, ports c1 and c2 are connected to ports s11 and s21, respectively.

Closed connections are modeled by a resistor with value equal to the **Closed resistance** parameter value. Open connections are modeled by a resistor with value equal to the reciprocal of the **Open conductance** parameter value.

If the **Threshold width** parameter is set to zero, the switch is closed if the voltage presented at the vT control port exceeds the value of the **Threshold** parameter.

If the **Threshold width** parameter is greater than zero, then switch conductance  $G$  varies smoothly between off-state and on-state values:

$$G = \frac{x}{R_{closed}} + (1 - x)G_{open}$$

$$\lambda = \frac{vT - \text{Threshold}}{\text{Threshold width}}$$

$$x = \begin{cases} 0 & \text{for } \lambda \leq 0 \\ 3\lambda^2 - 2\lambda^3 & \text{for } 0 < \lambda < 1 \\ 1 & \text{for } \lambda \geq 1 \end{cases}$$

The block uses the function  $3\lambda^2 - 2\lambda^3$  because its derivative is zero for  $\lambda = 0$  and  $\lambda = 1$ .

Defining a small positive **Threshold width** can help solver convergence in some models, particularly if the control port signal  $vT$  varies continuously as a function of other network variables. However, defining a nonzero threshold width precludes the solver making use of switched linear optimizations. Therefore, if the rest of your network is switched linear, MathWorks recommends that you set **Threshold width** to zero.

Optionally, you can add a delay between the point at which the voltage at  $vT$  passes the threshold and the switch opening or closing. To enable the delay, on the **Dynamics** tab, set the **Model dynamics** parameter to Model turn-on and turn-off times.

## Parameters

- “Main Tab” on page 1-521
- “Dynamics Tab” on page 1-522

### Main Tab

#### Closed resistance

Resistance between the c and s electrical ports when the switch is closed. The value must be greater than zero. The default value is 0.01  $\Omega$ .

#### Open conductance

Conductance between the c and s electrical ports when the switch is open. The value must be greater than zero. The default value is 1e-6 S.

#### Threshold

The threshold voltage for the control physical signal input  $vT$  above which the switch will turn on. The default value is 0.5 V.

## Threshold width

The minimum increase in the control signal  $v_T$  above the threshold value that will move the switch from fully open to fully closed. The default value is 0 V.

## Dynamics Tab

### Model dynamics

Select whether the block models a switching delay:

- **No dynamics** — Do not model the delay. This is the default option.
- **Model turn-on and turn-off times** — Use additional parameters to model a delay between the point at which the voltage at  $v_T$  passes the threshold and the switch opening or closing.

### Turn-on delay

Time between the input voltage exceeding the threshold voltage and the switch closing. This parameter is only visible when you select **Model turn-on and turn-off times** for the **Model dynamics** parameter. The value must be greater than zero. The default value is 1e-3 seconds.

### Turn-off delay

Time between the input voltage falling below the threshold voltage and the switch opening. This parameter is only visible when you select **Model turn-on and turn-off times** for the **Model dynamics** parameter. The value must be greater than zero. The default value is 1e-3 seconds.

### Initial input value, $v_T$

The value of the physical signal input  $v_T$  at time zero. This value is used to initialize the delayed control voltage parameter internally. This parameter is only visible when you select **Model turn-on and turn-off times** for the **Model dynamics** parameter. The default value is 0 V.

## Ports

This block has the following ports:

$v_T$

Physical signal that opens and closes the switch



c1, c2, s11, s12, s21, s22

Electrical conserving ports

## **See Also**

[DPST Switch](#) | [SPDT Switch](#) | [SPST Switch](#) | [Switch](#)

## DPST Switch

Model double-pole single-throw switch



## Library

Passive Devices/Switches

## Description

The DPST Switch block models a double-pole single-throw switch. When the switch is closed, ports c1 and c2 are connected to ports s1 and s2, respectively.

Closed connections are modeled by a resistor with value equal to the **Closed resistance** parameter value. Open connections are modeled by a resistor with value equal to the reciprocal of the **Open conductance** parameter value.

If the **Threshold width** parameter is set to zero, the switch is closed if the voltage presented at the vT control port exceeds the value of the **Threshold** parameter.

If the **Threshold width** parameter is greater than zero, then switch conductance  $G$  varies smoothly between off-state and on-state values:

$$G = \frac{x}{R_{closed}} + (1 - x)G_{open}$$

$$\lambda = \frac{vT - \text{Threshold}}{\text{Threshold width}}$$

$$x = \begin{cases} 0 & \text{for } \lambda \leq 0 \\ 3\lambda^2 - 2\lambda^3 & \text{for } 0 < \lambda < 1 \\ 1 & \text{for } \lambda \geq 1 \end{cases}$$

The block uses the function  $3\lambda^2 - 2\lambda^3$  because its derivative is zero for  $\lambda = 0$  and  $\lambda = 1$ .

Defining a small positive **Threshold width** can help solver convergence in some models, particularly if the control port signal  $vT$  varies continuously as a function of other network variables. However, defining a nonzero threshold width precludes the solver making use of switched linear optimizations. Therefore, if the rest of your network is switched linear, MathWorks recommends that you set **Threshold width** to zero.

Optionally, you can add a delay between the point at which the voltage at  $vT$  passes the threshold and the switch opening or closing. To enable the delay, on the **Dynamics** tab, set the **Model dynamics** parameter to `Model turn-on and turn-off times`.

## Parameters

- “Main Tab” on page 1-525
- “Dynamics Tab” on page 1-526

### Main Tab

#### Closed resistance

Resistance between the c and s electrical ports when the switch is closed. The value must be greater than zero. The default value is  $0.01 \Omega$ .

#### Open conductance

Conductance between the c and s electrical ports when the switch is open. The value must be greater than zero. The default value is  $1e-6 \text{ S}$ .

#### Threshold

The threshold voltage for the control physical signal input  $vT$  above which the switch will turn on. The default value is  $0.5 \text{ V}$ .

#### Threshold width

The minimum increase in the control signal  $vT$  above the threshold value that will move the switch from fully open to fully closed. The default value is  $0 \text{ V}$ .

## Dynamics Tab

### Model dynamics

Select whether the block models a switching delay:

- **No dynamics** — Do not model the delay. This is the default option.
- **Model turn-on and turn-off times** — Use additional parameters to model a delay between the point at which the voltage at  $v_T$  passes the threshold and the switch opening or closing.

### Turn-on delay

Time between the input voltage exceeding the threshold voltage and the switch closing. This parameter is only visible when you select **Model turn-on and turn-off times** for the **Model dynamics** parameter. The value must be greater than zero. The default value is  $1e-3$  seconds.

### Turn-off delay

Time between the input voltage falling below the threshold voltage and the switch opening. This parameter is only visible when you select **Model turn-on and turn-off times** for the **Model dynamics** parameter. The value must be greater than zero. The default value is  $1e-3$  seconds.

### Initial input value, $v_T$

The value of the physical signal input  $v_T$  at time zero. This value is used to initialize the delayed control voltage parameter internally. This parameter is only visible when you select **Model turn-on and turn-off times** for the **Model dynamics** parameter. The default value is 0 V.

## Ports

This block has the following ports:

$v_T$

Physical signal that opens and closes the switch

$c_1$ ,  $c_2$ ,  $s_1$ ,  $s_2$

Electrical conserving ports

## **See Also**

[DPDT Switch](#) | [SPDT Switch](#) | [SPST Switch](#) | [Switch](#)

## Eddy Current

Model eddy current losses

**Library:** Simscape / Electrical / Electronics and Mechatronics /  
Passive Devices  
Simscape / Electrical / Power Systems / Passive  
Devices / Fundamental Components



### Description

The Eddy Current block models the effect of eddy currents by generating a magnetomotive force (MMF) that opposes changes in the magnetic flux. In addition, the block models parasitic effects using a series reluctance and a parallel leakage permeance.

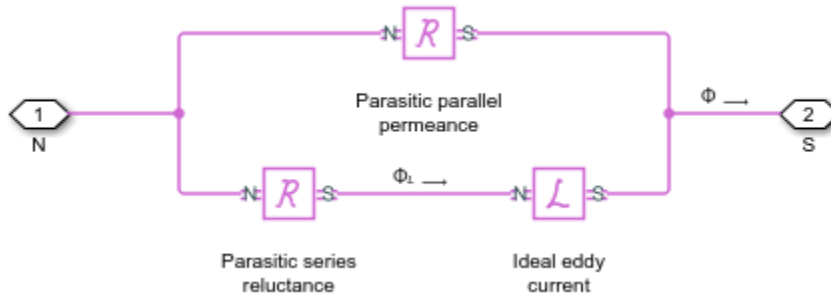
Eddy currents in magnetic core materials are caused by time-varying magnetic fields. These changing field densities induce voltage potentials within the material, causing current to flow in closed loops. Such currents are usually undesired and cause heating of the magnetic material.

Use this block to add eddy current losses in the magnetic domain to a custom transformer or other magnetic component.

The eddy current component is sometimes referred to as magnetic inductance because it is the magnetic-domain analog to an inductor in the electrical domain.

### Equations and Equivalent Circuit

This is the equivalent magnetic circuit for the block, including the eddy current path (bottom) and parallel leakage path (top).



In the diagram:

- $\Phi$  is the total flux at the terminals. This flux is the summation of the eddy current loop and the parallel leakage path.
- $\Phi_L$  is the flux through the eddy current loop.

The block calculates the terminal MMF,  $F$ , and flux,  $\Phi$ , as:

$$F = G_{\text{eddy}} \frac{d\Phi_L}{dt} + R \Phi_L$$

$$\Phi = F P + \Phi_L$$

where:

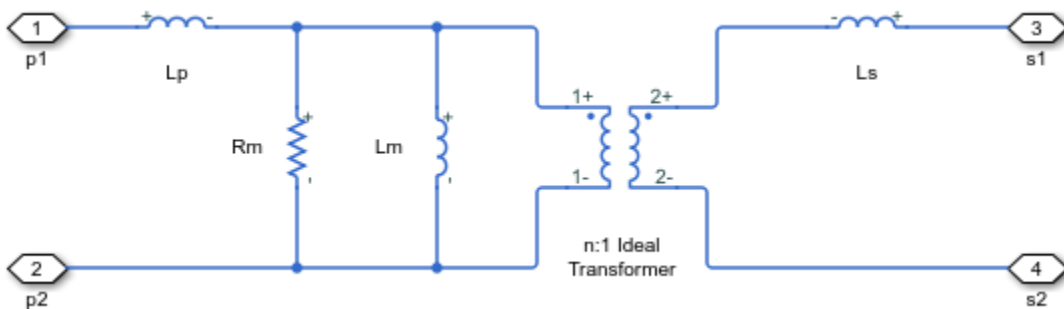
- $G_{\text{eddy}}$  is the conductance of the eddy current loop.
- $R$  is the parasitic series reluctance of the eddy current path.
- $P$  is the parallel permeance.

Because the parasitic series reluctance and parallel permeance are lossless, the total dissipated power over the block is:

$$P_{\text{diss}} = G_{\text{eddy}} \left( \frac{d\Phi_L}{dt} \right)^2$$

## Relating Transformer Electrical and Magnetic Models

In the electrical domain, eddy current losses are modeled using a parallel resistance across the primary winding. This is the equivalent circuit for a nonideal two-winding transformer.

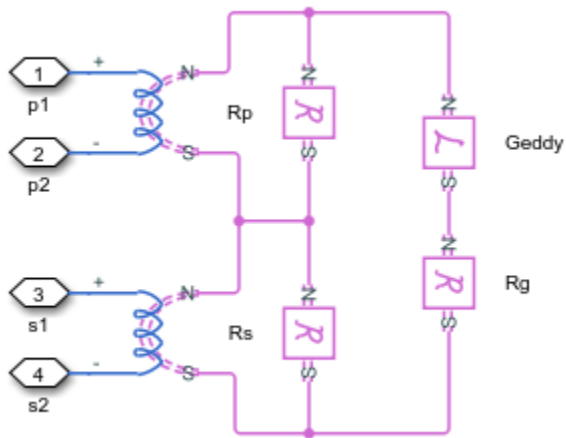


In the diagram:

- $L_p$  and  $L_s$  are the self-inductances of the primary and secondary windings, respectively.
- $L_m$  is the mutual inductance between the two windings.
- $R_m$  is the mutual resistance between the two windings, caused by the eddy current losses.

This two-winding transformer can similarly be represented in the magnetic domain. This is the equivalent circuit.





In the diagram:

- $R_p$  and  $R_s$  are the reluctances associated with the primary and secondary windings, respectively.
- $R_g$  is the reluctance associated with the magnetic coupling of the two windings.
- $n$  is the turns ratio between the two windings.

The electrical- and magnetic-domain circuits are equivalent if:

$$R_g = \frac{n^2}{L_m}$$

$$R_p = \frac{n^2}{L_p}$$

$$R_s = \frac{1}{L_s}$$

$$G_{\text{eddy}} = \frac{n^2}{R_m}$$

You can use these relationships to calculate the equivalent two-winding transformer properties for one domain from the other.

## Ports

### Conserving

#### **N — North terminal**

magnetic

Magnetic conserving port associated with the north terminal of the block.

#### **S — South terminal**

magnetic

Magnetic conserving port associated with the south terminal of the block.

## Parameters

#### **Conductance of eddy current loop — Eddy current conductance**

1 1/0hm (default) | positive number

Conductance,  $G_{\text{eddy}}$ , of the eddy current loop. This component is the magnetic-domain analog of inductance in the electrical domain.

#### **Parasitic series reluctance — Series reluctance**

1e-3 1/H (default) | nonnegative number

Parasitic reluctance,  $R$ , of the eddy current loop.

#### **Parasitic parallel permeance — Parallel leakage permeance**

1e-6 H (default) | nonnegative number

Parasitic permeance,  $\mathbb{P}$ , of the parallel leakage path. To aid simulation convergence in some circuit topologies, set this parameter to a small value.

## References

- [1] Brown, A. D., J. N. Ross, and K. G. Nichols. "Time-domain simulation of mixed nonlinear magnetic and electronic systems." *IEEE Transactions on Magnetics*. Vol. 37, Number 1, 2001, pp. 522-532.

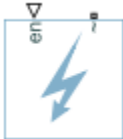
## See Also

Mutual Inductor | Nonlinear Transformer

**Introduced in R2018a**

## Enabled Fault

Signal-enabled single-phase, two-phase, or three-phase grounded or ungrounded fault



## Library

Simscape / Electrical / Power Systems / Passive Devices / Faults

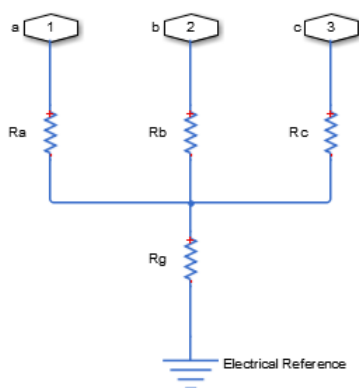
## Description

The Enabled Fault models any permutation of a single-phase, two-phase, or three-phase grounded or ungrounded fault. You specify the fault activation threshold using the block **Threshold** parameter. An external control signal *en* enables the fault. The fault is active when *en* is greater than the threshold. The fault is inactive when *en* is less than or equal to the threshold.

You can set the Enabled Fault block to represent any of these permutations:

- Single-phase-to-ground fault (a-g, b-g, or c-g)
- Two-phase fault (a-b, b-c, or c-a)
- Two-phase-to-ground fault (a-b-g, b-c-g, or c-a-g)
- Three-phase fault (a-b-c)
- Three-phase-to-ground fault (a-b-c-g)

The figure shows the equivalent circuit diagram for the Enabled Fault block.



You can determine the resistance in the equivalent circuit using the equations in the table.

Fault type	Value of $R_a$	Value of $R_b$	Value of $R_c$	Value of $R_g$
None / inactive	$\frac{1}{G_{pn}}$	$\frac{1}{G_{pn}}$	$\frac{1}{G_{pn}}$	Infinity / open circuit
a-g	$R_{pn}$	$\frac{1}{G_{pn}}$	$\frac{1}{G_{pn}}$	$R_{ng}$
b-g	$\frac{1}{G_{pn}}$	$R_{pn}$	$\frac{1}{G_{pn}}$	$R_{ng}$
c-g	$\frac{1}{G_{pn}}$	$\frac{1}{G_{pn}}$	$R_{pn}$	$R_{ng}$
a-b	$R_{pn}$	$R_{pn}$	$\frac{1}{G_{pn}}$	Infinity / open circuit
b-c	$\frac{1}{G_{pn}}$	$R_{pn}$	$R_{pn}$	Infinity / open circuit

Fault type	Value of $R_a$	Value of $R_b$	Value of $R_c$	Value of $R_g$
c-a	$R_{pn}$	$\frac{1}{G_{pn}}$	$R_{pn}$	Infinity / open circuit
a-b-g	$R_{pn}$	$R_{pn}$	$\frac{1}{G_{pn}}$	$R_{ng}$
b-c-g	$\frac{1}{G_{pn}}$	$R_{pn}$	$R_{pn}$	$R_{ng}$
c-a-g	$R_{pn}$	$\frac{1}{G_{pn}}$	$R_{pn}$	$R_{ng}$
a-b-c	$R_{pn}$	$R_{pn}$	$R_{pn}$	Infinity / open circuit
a-b-c-g	$R_{pn}$	$R_{pn}$	$R_{pn}$	$R_{ng}$

where:

- $R_a$  is the resistance between the a-phase and the neutral point of a wye connection.
- $R_b$  is the resistance between the b-phase and the neutral point of a wye connection.
- $R_c$  is the resistance between the c-phase and the neutral point of a wye connection.
- $R_g$  is the resistance between the neutral point of a wye connection and electrical reference.
- $R_{pn}$  is the value of the **Faulted phase-neutral resistance** parameter.
- $R_{ng}$  is the value of the **Faulted neutral-ground resistance** parameter.
- $G_{pn}$  is the value of the **Unfaulted phase-neutral conductance** parameter.

## Parameters

- “Main Tab” on page 1-537
- “Parasitics Tab” on page 1-538

## Main Tab

### Fault type

Select one of the following:

- None — Specifies that the fault is not active. This is the default value.
- Single-phase to ground (a-g)
- Single-phase to ground (b-g)
- Single-phase to ground (c-g)
- Two-phase (a-b)
- Two-phase (b-c)
- Two-phase (c-a)
- Two-phase to ground (a-b-g)
- Two-phase to ground (b-c-g)
- Two-phase to ground (c-a-g)
- Three-phase (a-b-c)
- Three-phase to ground (a-b-c-g)

### Faulted phase-neutral resistance

Resistance between the phase connection and the neutral point when the fault is active. This parameter is visible if the **Fault type** parameter is set to anything other than None. The default value is  $1e-3$  Ohm.

### Faulted neutral-ground resistance

Resistance between the neutral point and the electrical reference when fault is active. This parameter is visible if the **Fault type** parameter is set to any fault which includes a ground connection. The default value is  $1e-3$  Ohm.

### Threshold

Threshold for activating the fault. If the input *en* is above the value for the **Threshold** parameter, then the fault is active. If the input *en* is equal to or less than the value for the **Threshold** parameter, then the fault is not active. This parameter is visible if the **Fault type** parameter is set to anything other than None. The default value is 0.

## Parasitics Tab

### Unfaulted phase-neutral conductance

Conductance between the phase connections and the neutral point when a phase is not involved in the fault. The default value is  $1e-6$  1/Ohm.

## Ports

The block has the following ports:

~

Expandable three-phase port for connecting the fault to the system

en

Physical signal scalar control input port for enabling the fault

## See Also

Time-Based Fault

**Introduced in R2014a**



# Exponential Current Source

Model exponential pulse current source



## Library

SPICE-Compatible Components/Sources

## Description

The Exponential Current Source block represents a current source whose output current value is an exponential pulse as a function of time and is independent of the voltage across the terminals of the source. The following equations describe the current through the source as a function of time:

$$I_{out} (0 \leq Time \leq TDR) = I1$$

$$I_{out} (TDR < Time \leq TDF) = I1 + (I2 - I1) * (1 - e^{-(Time-TDR)/TR})$$

$$I_{out} (TDF < Time) = I1 + (I2 - I1) * (e^{-(Time-TDF)/TF} - e^{-(Time-TDR)/TR})$$

where:

- $I1$  is the **Initial value**, **I1** parameter value.
- $I2$  is the **Pulse value**, **I2** parameter value.
- $TDR$  is the **Rise delay time**, **TDR** parameter value.
- $TR$  is the **Rise time**, **TR** parameter value.
- $TDF$  is the **Fall delay time**, **TDF** parameter value.
- $TF$  is the **Fall time**, **TF** parameter value.

The block uses a small conductance internally to prevent numerical simulation issues. The conductance connects the + and - ports of the device and has a conductance *GMIN*:

- By default, *GMIN* matches the **Minimum conductance GMIN** parameter of the SPICE Environment Parameters block, whose default value is  $1e-12$ .
- To change *GMIN*, add a SPICE Environment Parameters block to your model and set the **Minimum conductance GMIN** parameter to the desired value.

## Parameters

### Initial value, I1

The value of the output current at time zero. The default value is 0 A.

### Pulse value, I2

The asymptotic value of the output current when the output is high. The default value is 0 A.

### Rise delay time, TDR

The rise time delay. The default value is 0 s.

### Rise time, TR

The time it takes the output current to rise from the **Initial Value, I1** value to the **Pulse Value, I2** value. The default value is  $1e-09$  s. The value must be greater than 0.

### Fall delay time, TDR

The fall time delay. The default value is 0 s, which differs from the SPICE default value.

### Fall time, TF

The time it takes the output current to fall from the **Pulse value, I2** value to the **Initial value, I1** value. The default value is  $1e-09$  s. The value must be greater than 0.

## Ports

The block has the following ports:

+

Positive electrical voltage.

-

Negative electrical voltage.

## **See Also**

Exponential Voltage Source

## Exponential Voltage Source

Model exponential pulse voltage source



### Library

SPICE-Compatible Components/Sources

### Description

The Exponential Voltage Source block represents a voltage source whose output voltage value is an exponential pulse as a function of time and is independent of the current through the source. The following equations describe the output current as a function of time:

$$V_{out} (0 \leq Time \leq TDR) = V1$$

$$V_{out} (TDR < Time \leq TDF) = V1 + (V2 - V1) * (1 - e^{-(Time-TDR)/TR})$$

$$V_{out} (TDF < Time) = V1 + (V2 - V1) * (e^{-(Time-TDF)/TF} - e^{-(Time-TDR)/TR})$$

where:

- $V1$  is the **Initial value**, **V1** parameter value.
- $V2$  is the **Pulse value**, **V2** parameter value.
- $TDR$  is the **Rise delay time**, **TDR** parameter value.
- $TR$  is the **Rise time**, **TR** parameter value.
- $TDF$  is the **Fall delay time**, **TDF** parameter value.
- $TF$  is the **Fall time**, **TF** parameter value.

## Parameters

### Initial value, V1

The value of the output voltage at time zero. The default value is 0 V.

### Pulse value, V2

The asymptotic value of the output voltage when the output is high. The default value is 0 V.

### Rise delay time, TDR

The rise time delay. The default value is 0 s.

### Rise time, TR

The time it takes the output voltage to rise from the **Initial value, I1** value to the **Pulse value, I2** value. The default value is 1e-09 s. The value must be greater than 0.

### Fall delay time, TDR

The fall time delay. The default value is 0 s.

### Fall Time, TF

The time it takes the output voltage to fall from the **Pulse value, I2** value to the **Initial value, I1** value. The default value is 1e-09 s. The value must be greater than 0.

## Ports

The block has the following ports:

+

Positive electrical voltage.

-

Negative electrical voltage.

## See Also

Exponential Current Source

## Fault

Model electrical fault



## Library

Passive Devices

## Description

The Fault block allows you to represent an electrical fault as an instantaneous change in resistance. You can use it to replicate both open-circuit and short-circuit fault behaviors. The block can trigger fault events:

- At a specific time
- When a predefined voltage range or current range is exceeded
- When an external trigger signal goes high or low

Optionally, the external trigger option also permits the fault to be reset when the trigger signal reverts. You can enable or disable all three trigger mechanisms separately, or use them together if more than one trigger mechanism is required in a simulation.

When no fault is triggered, the resistance between the two electrical ports is defined by the **Unfaulted resistance** parameter value. The default value for this parameter is `inf` ohms, i.e. the ports are open-circuit. When a fault is triggered, the block changes the resistance between the two electrical ports to the **Faulted resistance** value. The default value for this parameter is `1e-3` ohms, i.e. the ports are short-circuited.

You can choose whether to issue an assertion when a fault occurs, by using the **Reporting when a fault occurs** parameter. The assertion can take the form of a warning or an error. By default, the block does not issue an assertion.

The physical output X represents the fault state; it is 1 if the block is faulted, and 0 otherwise. The physical signal input F is the external fault trigger signal and is used only if **Enable external fault trigger** is set to Yes.

## Parameters

- “Main Tab” on page 1-545
- “Temporal Trigger Tab” on page 1-545
- “Behavioral Trigger Tab” on page 1-546
- “External Trigger Tab” on page 1-546

### Main Tab

#### Unfaulted resistance

Resistance between the + and - ports when there is no fault. The default value is  $\text{inf } \Omega$ .

#### Faulted resistance

Resistance between the + and - ports when the block is in the faulted state. The default value is  $1\text{e-}3 \Omega$ .

#### Reporting when a fault occurs

Choose whether to issue an assertion when a fault occurs:

- None — The block does not issue an assertion. This is the default.
- Warn — The block issues a warning.
- Error — Simulation stops with an error.

### Temporal Trigger Tab

#### Enable temporal fault trigger

Select Yes to enable time-based fault triggering. The default value is No.

#### Simulation time for fault event

Set the simulation time at which you want the block to enter the fault state. This parameter is visible only if the **Enable temporal fault trigger** parameter is set to Yes. The default value is 1 s.

## Behavioral Trigger Tab

### Enable behavioral fault trigger

Select **Yes** to enable behavioral fault triggering. The default value is **No**.

### Permissible voltage range

Specify a vector of length 2 that defines the permissible voltage range. If the voltage exceeds this range for longer than the **Time to fail when exceeding voltage range** parameter value, then the block enters the fault state. This parameter is visible only if the **Enable behavioral fault trigger** parameter is set to **Yes**. The default value is [-100, 100] V.

### Time to fail when exceeding voltage range

Set the maximum length of time that the voltage can exceed the permissible voltage range without triggering the fault. This parameter is visible only if the **Enable behavioral fault trigger** parameter is set to **Yes**. The default value is 1 s.

### Permissible current range

Specify a vector of length 2 that defines the permissible current range. If the current exceeds this range for longer than the **Time to fail when exceeding current range** parameter value, then the block enters the fault state. This parameter is visible only if the **Enable behavioral fault trigger** parameter is set to **Yes**. The default value is [-1, 1] A.

### Time to fail when exceeding current range

Set the maximum length of time that the current can exceed the permissible current range without triggering the fault. This parameter is visible only if the **Enable behavioral fault trigger** parameter is set to **Yes**. The default value is 1 s.

## External Trigger Tab

### Enable external fault trigger

Select **Yes** to enable external fault triggering. The physical signal input F provides the external fault trigger signal. The default value is **No**.

### External fault trigger

Choose the fault condition:

- **Faulted if  $F \geq \text{Fault threshold}$**  — The fault occurs when the external signal value becomes greater than, or equal to, the **Fault threshold** parameter value. This is the default.



- **Faulted if  $F \leq \text{Fault threshold}$**  — The fault occurs when the external signal value becomes less than, or equal to, the **Fault threshold** parameter value.

This parameter is visible only if the **Enable external fault trigger** parameter is set to Yes.

### **Fault threshold**

The threshold value that triggers the fault when the external signal crosses it in the direction, specified by the fault condition. This parameter is visible only if the **Enable external fault trigger** parameter is set to Yes. The default value is 0.5.

### **Fault resets when fault trigger reverts**

Select Yes to have the fault reset when the trigger signal reverts. The default value is No. This parameter is visible only if the **Enable external fault trigger** parameter is set to Yes.

## **Ports**

The block has the following ports:

+

Positive electrical port.

-

Negative electrical port.

F

Physical signal input port that provides the external fault trigger signal.

X

Physical signal output port that indicates the fault state. Outputs 1 if the block is faulted, and 0 otherwise.

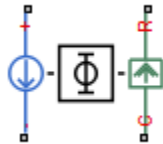
## **See Also**

Resistor

**Introduced in R2015b**

# FEM-Parameterized Linear Actuator

Model linear actuator defined in terms of magnetic flux



## Library

Translational Actuators

## Description

The FEM-Parameterized Linear Actuator block implements a model of a linear actuator defined in terms of magnetic flux. Use this block to model custom solenoids and linear motors where magnetic flux depends on both distance and current. You parameterize the block using data from a third-party Finite Element Magnetic (FEM) package.

The block has two options for the electrical equation. The first, **Define in terms of  $d\Phi(i, x)/dx$  and  $d\Phi(i, x)/di$** , defines the current in terms of partial derivatives of the magnetic flux ( $\Phi$ ) with respect to distance ( $x$ ) and current ( $i$ ), the equations for which are:

$$\frac{di}{dt} = \left( v - iR - \frac{\partial \Phi}{\partial x} \frac{dx}{dt} \right) / \frac{\partial \Phi}{\partial i}$$

The second option, **Define in terms of  $\Phi(i, x)$** , defines the voltage across the component directly in terms of the flux, the equation for which is:

$$v = iR + \frac{d}{dt} \Phi(x, i)$$

Numerically, defining the electrical equation in terms of flux partial derivatives is better because the back-emf is piecewise continuous. If using the flux directly, using a finer grid size for current and position will improve results, as will selecting cubic or spline interpolation.

In both cases, you have an option to either directly specify the force as a function of current and position, by using the **Force matrix,  $F(i,x)$**  parameter, or have the block automatically calculate the force matrix.

If entering the electromagnetic force data directly, you can either use data supplied by the finite element magnetic package (which you used to determine the flux) or calculate the force from the flux with following equation:

$$F = \int_0^i \frac{\partial \Phi(x,i)}{\partial x} di$$

See the Finite Element Parameterized Solenoid example model and its initialization file `elec_fem_solenoid_ini.m` for an example of how to implement this type of integration in MATLAB.

Alternatively, the block can automatically calculate the force matrix from the flux information that you provide. To select this option, set the **Calculate force matrix?** parameter to **Yes**. The force matrix calculation occurs at model initialization based on current block flux linkage information. The force is calculated by numerically integrating the rate of change of flux linkage with respect to position over current, according to the preceding equation. If the **Electrical model** parameter is set to **Define in terms of  $\Phi(i,x)$** , then the block must first estimate the **Flux partial derivative wrt displacement,  $d\Phi(i,x)/dx$**  parameter value from the flux linkage data. When doing this, the block uses the interpolation method specified by the **Interpolation method** parameter. Typically, the **Smooth** option is most accurate, but the **Linear** option is most robust.

You can define  $\Phi$  and its partial derivatives for just positive, or positive and negative currents. If defining for just positive currents, then the block assumes that  $\Phi(-i,x) = -\Phi(i,x)$ . Therefore, if the current vector is positive only:

- The first current value must be zero.
- The flux corresponding to zero current must be zero.
- The partial derivative of flux with respect to displacement must be zero for zero current.

To model a linear motor with a repeated flux pattern, set the **Flux dependence on displacement** parameter to *Cyclic*. When selecting this option, the force and flux (or force and flux partial derivatives depending on the option chosen) must have identical first and last columns.

## Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal port H on the block icon, and adds the **Temperature Dependence** and **Thermal Port** tabs to the block dialog box.

Use the thermal port to simulate the effects of copper resistance losses that convert electrical power to heat. For more information on using thermal ports and on the **Temperature Dependence** and **Thermal Port** tab parameters, see “Simulating Thermal Effects in Rotational and Translational Actuators”.

## Basic Assumptions and Limitations

This block has the following limitations:

- It is imperative that you supply a consistent set of force and flux data. There is no checking that the force matrix is consistent with the flux data.
- When driving the FEM-Parameterized Linear Actuator block via a series inductor, you may need to include a parallel conductance in the inductor component.

## Parameters

- “Magnetic Force Tab” on page 1-550
- “Mechanical Tab” on page 1-554

## Magnetic Force Tab

### Electrical model

Select one of the following parameterization options, based on the underlying electrical model:

- Define in terms of  $d\Phi(i,x)/dx$  and  $d\Phi(i,x)/di$  — Define the current through the block in terms of partial derivatives of the magnetic flux with respect to distance and current. This is the default method.
- Define in terms of  $\Phi(i,x)$  — Define the voltage across the block terminals directly in terms of the flux.

### Current vector, $i$

Specify a vector of monotonically increasing current values corresponding to your force-flux data. If you specify positive currents only, the first element must be zero. The default value is [ 0 0.1 0.2 0.3 0.4 0.5 0.6 0.7 0.8 0.9 1 ] A.

### Displacement vector, $x$

Specify a vector of monotonically increasing displacement values corresponding to your force-flux data. The default value is [ 0 0.05 0.1 0.15 0.2 ] m/m.

### Flux partial derivative wrt current, $d\Phi(i,x)/di$

Specify a matrix of the flux partial derivatives with respect to current. This parameter is visible only if **Electrical model** is set to Define in terms of  $d\Phi(i,x)/dx$  and  $d\Phi(i,x)/di$ . The default value, in Wb/A, is:

```
[ 0.104 0.098 0.091 0.085 0.078;
0.095 0.089 0.084 0.079 0.073;
0.085 0.081 0.077 0.073 0.069;
0.076 0.073 0.07 0.067 0.064;
0.067 0.065 0.063 0.061 0.06;
0.057 0.057 0.056 0.056 0.055;
0.048 0.049 0.049 0.05 0.05;
0.038 0.04 0.042 0.044 0.046;
0.029 0.032 0.035 0.038 0.041;
0.02 0.024 0.028 0.033 0.037;
0.01 0.016 0.021 0.027 0.032 ]
```

### Flux partial derivative wrt displacement, $d\Phi(i,x)/dx$

Specify a matrix of the flux partial derivatives with respect to displacement. This parameter is visible only if **Electrical model** is set to Define in terms of  $d\Phi(i,x)/dx$  and  $d\Phi(i,x)/di$ . The default value, in Wb/m, is:

```
[ 0 0 0 0 0;
-11.94 -10.57 -9.19 -7.81 -6.43;
-21.17 -19.92 -18.67 -17.42 -16.16;
-27.99 -26.87 -25.75 -24.62 -23.5;
-32.42 -31.43 -30.43 -29.43 -28.44;
-34.46 -33.59 -32.72 -31.85 -30.98;
```

```
-34.09 -33.35 -32.61 -31.87 -31.12;  
-31.33 -30.72 -30.1 -29.49 -28.87;  
-26.17 -25.68 -25.2 -24.71 -24.22;  
-18.62 -18.26 -17.9 -17.54 -17.18;  
-8.66 -8.43 -8.2 -7.97 -7.73 ]
```

**Flux linkage matrix, Phi(i,x)**

Specify a matrix of the total flux linkage, that is, flux times the number of turns. This parameter is visible only if **Electrical model** is set to Define in terms of Phi(i,x). The default value, in Wb, is:

```
[ 0 0 0 0 0;  
0.0085 0.0079 0.0075 0.0071 0.0067;  
0.0171 0.016 0.0151 0.0143 0.0137;  
0.0254 0.0239 0.0226 0.0215 0.0206;  
0.033 0.0312 0.0297 0.0283 0.0271;  
0.0396 0.0377 0.036 0.0345 0.0331;  
0.0452 0.0433 0.0415 0.0399 0.0384;  
0.0495 0.0478 0.0461 0.0446 0.0431;  
0.0526 0.0512 0.0498 0.0485 0.0472;  
0.0545 0.0537 0.0528 0.0519 0.0508;  
0.0554 0.0553 0.0551 0.0548 0.0542 ]
```

**Calculate force matrix?**

Specify the way of providing the electromagnetic force data:

- No — specify directly — Enter the electromagnetic force data directly, by using the **Force matrix, F(i,x)** parameter. This is the default option.
- Yes — The block calculates the force from the flux linkage information, as a function of current and displacement.

**Force matrix, F(i,x)**

Specify a matrix of the electromagnetic force applied to the plunger or moving part. This parameter is visible only if **Calculate force matrix?** is set to No — specify directly. The default value, in N, is:

```
[ 0 0 0 0 0;  
-0.6 -0.5 -0.4 -0.3 -0.3;  
-2.3 -2 -1.7 -1.4 -1.2;  
-4.9 -4.3 -3.7 -3.2 -2.7;  
-8.3 -7.3 -6.4 -5.5 -4.7;  
-12.2 -10.7 -9.4 -8.2 -7.2;  
-16.2 -14.4 -12.7 -11.3 -10;
```

```
-20 -17.9 -15.9 -14.3 -12.9;
-23.3 -20.9 -18.8 -17.1 -15.7;
-25.7 -23.1 -21.1 -19.4 -18.2;
-26.5 -24.1 -22.2 -20.9 -20.1 ]
```

### Flux dependence on displacement

Specify the flux pattern:

- **Unique** — No flux pattern present. This is the default option.
- **Cyclic** — Select this option to model a linear motor with a repeated flux pattern. The force and flux (or force and flux partial derivatives, depending on the **Electrical model** option chosen) must have identical first and last columns.

### Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- **Linear** — Select this option to get the best performance.
- **Smooth** — Select this option to produce a continuous surface with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (2D) block reference page.

### Extrapolation method

Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- **Linear** — Select this option to produce a surface with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.
- **Nearest** — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (2D) block reference page.

This parameter is not available if you set the **Flux dependence on displacement** parameter to **Cyclic**.

### Winding resistance

Total resistance of the electrical winding. The default value is 14 Ohm.

## **Mechanical Tab**

### **Damping**

Linear damping. The default value is 1 N/(m/s). The value can be zero.

### **Plunger mass**

Mass of the moving part, which corresponds to mechanical translational port R. The default value is 0.05 kg. The value can be zero.

### **Minimum stroke**

The stroke at which the lower mechanical end stop is applied. The default value is 0. The value can be -Inf.

### **Maximum stroke**

The stroke at which the upper mechanical end stop is applied. The default value is 0.2 mm. The value can be Inf.

### **Initial plunger position**

Position of the plunger at the start of the simulation. The default value is 0 mm.

### **Initial plunger velocity**

Speed of the plunger at the start of the simulation. The default value is 0 mm/s.

### **Contact stiffness**

Contact stiffness between plunger and end stops. The default value is 1e8 N/m.

### **Contact damping**

Contact damping between plunger and end stops. The default value is 1e4 N/(m/s).

## **Ports**

This block has the following ports:

+

Positive electrical conserving port

-

Negative electrical conserving port

C

Mechanical translational conserving port connected to the actuator case



R

Mechanical translational conserving port connected to the plunger

## Examples

The Solenoid Parameterized with FEM Data example illustrates the use and parameterization options of this block.

## See Also

FEM-Parameterized Rotary Actuator | Solenoid

## FEM-Parameterized PMSM

Permanent magnet synchronous motor defined in terms of magnetic flux linkage



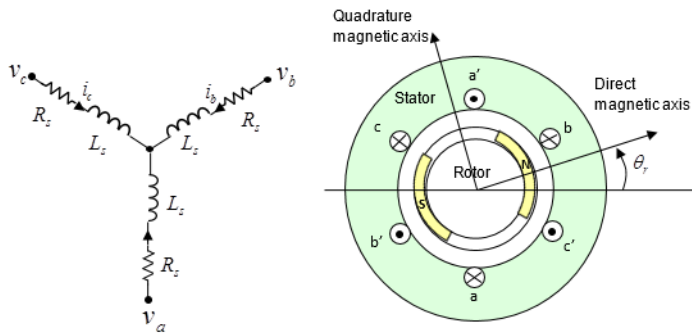
## Library

Rotational Actuators

## Description

The FEM-Parameterized PMSM block implements a model of a permanent magnet synchronous motor (PMSM) defined in terms of magnetic flux linkage. You parameterize the block by providing tabulated data of motor magnetic flux as a function of current and rotor angle. This is the way third-party Finite Element Magnetic (FEM) packages usually export flux information. Because of the tabulated form, the flux can vary in a nonlinear way on both rotor angle and current. You can therefore use this block to model PMSM with trapezoidal back-emf profile, sometimes called brushless DC motor, as well as regular PMSM.

The figure shows the equivalent circuit for a wye-connected PMSM. The rotor angle is zero when the permanent magnet flux aligns with the A-phase magnetic axis.



In practice, the flux linking each of the three windings depends on all three currents and rotor angle. Tabulating flux as a function of four independent variables might lead to simulation inefficiency and significant memory requirements to manage the data. The block, therefore, lets you select between the following parameterization methods for flux and torque:

- **2-D partial derivative data** — 2-D table lookup, with options to tabulate in terms of current and rotor angle, or in terms of  $d$ -axis and  $q$ -axis currents. The first option assumes constant mutual inductance and supports nonsinusoidal back emf profiles. The second option assumes a sinusoidal back emf and captures saturation effects for interior PMSMs (IPMSMs).
- **3-D partial derivative data** — 3-D table lookup, based on direct current, quadrature current, and rotor angle. You provide the flux lookup data for the  $a$  phase. The block uses Park transform to map the three stator winding currents to direct and quadrature currents. This method reduces the data complexity, as compared to the 4-D table lookup, and therefore results in improved simulation performance.
- **4-D partial derivative data** — 4-D table lookup, based on the three stator winding currents and the rotor angle. You provide the flux lookup data for the  $a$  phase. This model has the best fidelity of the three, but also is the most costly in terms of simulation performance and memory requirements.
- **3-D flux linkage data** — 3-D table lookup, based on the flux linkage data. You can provide the flux linkage data in a variety of formats. The block uses Park transform to map the three stator winding currents to direct and quadrature currents. This method reduces the data complexity, as compared to the 4-D table lookup, and therefore results in improved simulation performance.

By default, all of the block variants implement a wye-wound configuration for the stator windings. The 3-D flux linkage data variant also supports a delta-wound configuration, selectable using the **Winding type** parameter. When in the delta-wound configuration, the *a* phase is connected between ports **a** and **b**, the *b* phase between ports **b** and **c** and the *c* phase between ports **c** and **a**.

To access these parameterization methods, right-click the block in your model, select **Simscape > Block choices**, and then select the desired block variant, with or without thermal ports. By default, the thermal ports are not exposed. For more information, see “Thermal Ports” on page 1-568.

## 2-D Data Model with Constant Mutual Inductance

In this 2-D flux data model, the flux linking each winding is assumed to depend nonlinearly only on the current in that same winding, plus the rotor angle. In practice, this is a reasonable assumption for many permanent magnet synchronous motors; however, it is less accurate for switched reluctance motors. Given this assumption, the fluxes in the three windings are:

$$\begin{bmatrix} \phi_a \\ \phi_b \\ \phi_c \end{bmatrix} = \begin{bmatrix} 0 & -M_s & -M_s \\ -M_s & 0 & -M_s \\ -M_s & -M_s & 0 \end{bmatrix} \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix} + \begin{bmatrix} \phi(i_a, \theta_r) \\ \phi(i_b, \theta_r - 2\pi / (3N)) \\ \phi(i_c, \theta_r - 4\pi / (3N)) \end{bmatrix}$$

where  $\phi(\theta_r, i_a)$  is the flux linkage for the A-phase winding as a function of rotor angle and A-phase current.  $\theta_r = 0$  corresponds to the rotor *d*-axis aligning with the A-phase positive magnetic flux direction.  $M_s$  is the stator-stator mutual inductance.

For improved numerical performance, the equations implemented in the block actually work with the partial derivatives of flux linkage with respect to current,  $\partial\phi(i, \theta_r) / \partial i$ , and rotor angle,  $\partial\phi(i, \theta_r) / \partial\theta_r$ , rather than the flux directly. If your FEM package does not export these partial derivatives, you can determine them using a MATLAB script. See the Solenoid Parameterized with FEM Data example model and its supporting MATLAB script for an example of how to do this.

The electrical equations for the block, defined in terms of flux partial derivatives, are:

$$\begin{aligned}
 v_a &= \frac{\partial \phi}{\partial i_a} \frac{di_a}{dt} + \frac{\partial \phi}{\partial \theta_r} \frac{d\theta_r}{dt} - M_s \left( \frac{di_b}{dt} + \frac{di_c}{dt} \right) + R_s i_a \\
 v_b &= \frac{\partial \phi}{\partial i_b} \frac{di_b}{dt} + \frac{\partial \phi}{\partial \theta_r} \frac{d\theta_r}{dt} - M_s \left( \frac{di_a}{dt} + \frac{di_c}{dt} \right) + R_s i_b \\
 v_c &= \frac{\partial \phi}{\partial i_c} \frac{di_c}{dt} + \frac{\partial \phi}{\partial \theta_r} \frac{d\theta_r}{dt} - M_s \left( \frac{di_a}{dt} + \frac{di_b}{dt} \right) + R_s i_c
 \end{aligned}$$

where

- $v_a, v_b, v_c$  are the voltages applied to the A, B, and C stator windings.
- $i_a, i_b, i_c$  are the stator currents in each of the three windings.
- $R_s$  is the resistance of each of the stator windings.
- $M_s$  is the stator-stator mutual inductance.
- $\partial \phi / \partial i_a, \partial \phi / \partial i_b, \partial \phi / \partial i_c$  are the partial derivatives of flux linkage with respect to stator current in each of the three windings.
- $\partial \phi / \partial \theta_r$  is the partial derivative of flux linkage with respect to rotor angle.

The block can automatically calculate the torque matrix from the flux information that you provide. Alternatively, you can set the **Calculate torque matrix?** parameter to **No** and directly specify the torque as a function of current and rotor angle. See the FEM-Parameterized Rotary Actuator block reference page for more information.

## 2-D Data Model with Sinusoidal Back EMF

In this 2-D flux data model, the flux linking each winding is assumed to depend nonlinearly on all stator winding currents, plus it is assumed that the permanent magnet flux linkage is sinusoidal. Interior magnet PMSMs (or IPMSMs) usually fit this assumption well. The equations are:

$$\begin{bmatrix} \phi_d \\ \phi_q \end{bmatrix} = \begin{bmatrix} L_d(i_d, i_q) & \\ & L_q(i_d, i_q) \end{bmatrix} \begin{bmatrix} i_d \\ i_q \end{bmatrix} + \begin{bmatrix} \phi_m(i_d, i_q) \\ 0 \end{bmatrix}$$

$$T = \frac{3}{2} N \left( i_q \left( i_d L_d(i_d, i_q) + \phi_m(i_d, i_q) \right) - i_d i_q L_q(i_d, i_q) \right)$$

where

- $i_d$  and  $i_q$  are the  $d$ -axis and  $q$ -axis currents, respectively.
- $\phi_d$  and  $\phi_q$  are the  $d$ -axis and  $q$ -axis flux linkages, respectively.
- $\phi_m$  is the permanent magnet flux linkage.
- $L_d$  and  $L_q$  are the  $d$ -axis and  $q$ -axis inductances, respectively. They are assumed to depend on the  $d$ -axis and  $q$ -axis currents.
- $N$  is the number of pole pairs.
- $T$  is the electrical torque.

### 3-D Partial Derivative Data Model Using Park Transform

Working with four-dimensional data has both a simulation performance cost and a memory cost. To reduce the table dimension to three-dimensional, the 3-D data model uses Park transform to map the three currents to direct and quadrature currents:

$$\begin{bmatrix} i_d \\ i_q \end{bmatrix} = \frac{2}{3} \begin{bmatrix} \cos\theta_e & \cos\left(\theta_e - \frac{2\pi}{3}\right) & \cos\left(\theta_e + \frac{2\pi}{3}\right) \\ -\sin\theta_e & -\sin\left(\theta_e - \frac{2\pi}{3}\right) & -\sin\left(\theta_e + \frac{2\pi}{3}\right) \end{bmatrix} \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix}$$

In the general case, Park transform maps to direct, quadrature, and zero-sequence currents. However, the zero-sequence current is typically small under normal operating conditions. Therefore, the model neglects the dependence of the flux linkage terms on zero-sequence current, and determines the flux linkage in terms of just direct and quadrature currents plus rotor angle. The flux equation for the 3-D data model is:

$$\begin{bmatrix} \phi_a \\ \phi_b \\ \phi_c \end{bmatrix} = \begin{bmatrix} \phi(i_d, i_q, \theta_r) \\ \phi(i_d, i_q, \theta_r - 2\pi / (3N)) \\ \phi(i_d, i_q, \theta_r - 4\pi / (3N)) \end{bmatrix}$$

The electrical equations for the block are also defined in terms of flux partial derivatives, similar to the 4-D data model. You can calculate 3-D flux linkage partial derivative data from 4-D flux linkage data using `elec_calculateFluxPartialDerivatives`.

## 4-D Partial Derivative Data Model

The flux linking each of the windings is a function of the current in that winding, the currents in the other two windings, and the rotor angle. For full accuracy, the 4-D flux data model assumes that the flux linkage is a function of the three currents and the rotor angle, therefore performing four-dimensional table lookups. The flux equation is:

$$\begin{bmatrix} \phi_a \\ \phi_b \\ \phi_c \end{bmatrix} = \begin{bmatrix} \phi(i_a, i_b, i_c, \theta_r) \\ \phi(i_b, i_c, i_a, \theta_r - 2\pi / (3N)) \\ \phi(i_c, i_a, i_b, \theta_r - 4\pi / (3N)) \end{bmatrix}$$

where

- $\phi_a, \phi_b, \phi_c$  are the flux linkages for the A, B, and C stator windings.
- $i_a, i_b, i_c$  are the stator currents in each of the three windings.
- $\theta_r$  is the rotor angle.  $\theta_r = 0$  corresponds to the case where the permanent magnet flux is aligned with the A-phase stator winding flux.
- $N$  is the number of pole pairs.

Flux linkage data is assumed cyclic with  $\theta_r$ . If, for example, the motor has six pole pairs, then the range for the data is  $0 \leq \theta_r \leq 60^\circ$ . You must provide data both at 0 and 60 degrees, and because the data is cyclic, the flux linkage partial derivatives must be the same at these two end points.

The torque equation is:

$$\tau = T(i_a, i_b, i_c, \theta_r)$$

The 4-D data model does not have an option for the block to determine torque from flux linkage. Because of the increased numerical overhead in the 4-D case, it is better to precalculate the torque just once, rather than calculate it every time you run the simulation.

For improved numerical performance, the equations implemented in the block actually work with the partial derivatives of flux linkage with respect to the three currents and the rotor angle, rather than the flux directly. If your FEM package does not export these partial derivatives, you can determine them using `elec_calculateFluxPartialDerivatives`.

The electrical equations for the block, defined in terms of flux partial derivatives, are:

$$v_a = \frac{\partial \phi_a}{\partial i_a} \frac{di_a}{dt} + \frac{\partial \phi_a}{\partial i_b} \frac{di_b}{dt} + \frac{\partial \phi_a}{\partial i_c} \frac{di_c}{dt} + \frac{\partial \phi_a}{\partial \theta_r} \frac{d\theta_r}{dt} + R_s i_a$$

$$v_b = \frac{\partial \phi_b}{\partial i_a} \frac{di_a}{dt} + \frac{\partial \phi_b}{\partial i_b} \frac{di_b}{dt} + \frac{\partial \phi_b}{\partial i_c} \frac{di_c}{dt} + \frac{\partial \phi_b}{\partial \theta_r} \frac{d\theta_r}{dt} + R_s i_b$$

$$v_c = \frac{\partial \phi_c}{\partial i_a} \frac{di_a}{dt} + \frac{\partial \phi_c}{\partial i_b} \frac{di_b}{dt} + \frac{\partial \phi_c}{\partial i_c} \frac{di_c}{dt} + \frac{\partial \phi_c}{\partial \theta_r} \frac{d\theta_r}{dt} + R_s i_c$$

where

- $v_a$ ,  $v_b$ ,  $v_c$  are the voltages applied to the A, B, and C stator windings.
- $i_a$ ,  $i_b$ ,  $i_c$  are the stator currents in each of the three windings.
- $R_s$  is the resistance of each of the stator windings.

### 3-D Flux Linkage Data Models

The 3-D flux linkage data options let you work with raw flux linkage data exported from your finite-element (FE) motor design tool. This is in contrast to the 3-D partial derivative data options, for which you need to determine the partial derivatives. You can provide flux linkage data in a variety of formats, to support different FE tool conventions:

- Tabulate DQ-axes flux linkage data or A-phase flux linkage data — Some tools support working with flux linkage resolved into direct (D) and quadrature (Q) axes. An advantage of this approach is that data for rotor angles in the range 0 to 360/N/3 degrees is required (where  $N$  is the number of pole pairs). Other tools work directly with A-, B-, and C-phase flux linkages, and for this you can import just the A-phase flux linkage, for which the rotor angle range must be in the range 0 to 360/N degrees. The implicit assumption of importing just the A-phase data is that the B and C phase data is the same except shifted in phase.
- Tabulate using cartesian or polar current coordinates — Cartesian tabulation implies the flux linkage is tabulated in terms of D-axis current and Q-axis current (plus rotor angle). Alternatively, polar tabulation involves tabulating flux linkages in terms of current magnitude, current advance angle relative to the Q-axis, and rotor angle. The advantage of polar coordinates is that it more naturally reflects the permitted operating currents, thereby avoiding unused table data points.

These conventions result in four **Flux linkage data format** parameterization options:



- D and Q axes flux linkages as a function of D-axis current ( $i_D$ ), Q-axis current ( $i_Q$ ), and rotor angle ( $\theta$ )
- D and Q axes flux linkages as a function of peak current magnitude ( $I$ ), current advance angle ( $B$ ), and rotor angle ( $\theta$ )
- A-phase flux linkage as a function of D-axis current ( $i_D$ ), Q-axis current ( $i_Q$ ), and rotor angle ( $\theta$ )
- A-phase flux linkage as a function of peak current magnitude ( $I$ ), current advance angle ( $B$ ), and rotor angle ( $\theta$ )

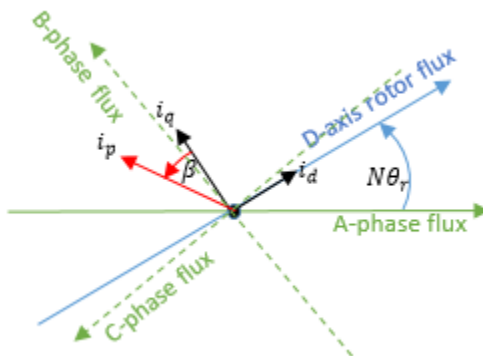
Besides selecting the flux linkage data format used by your FE tool, you have to select the version of Park transform used by the tool. The four conventions are described below and correspond to the four options for the **Park's convention for tabulated data** drop-down menu.

---

**Note** When looking at logged values for D- and Q-axis currents, keep in mind that for each of these options, the format is converted, as needed, so that internally the FEM-Parameterized PMSM block consistently uses Option 1.

---

This is the Park's convention used internally by Simscape Electrical motor and machine blocks. All other options are converted into this format.



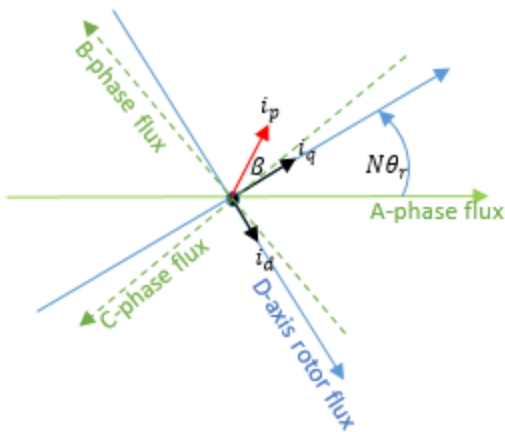
- $N$ : number of pole pairs
- $\theta_r$ : rotor angle

- $i_d, i_q$ : D-axis and Q-axis currents
- $i_p$ : Current magnitude =  $\sqrt{i_d^2 + i_q^2}$
- $\beta$ : Current advance angle =  $\tan^{-1}(-i_d / i_q)$

Corresponding Park transform is

$$\begin{bmatrix} i_d \\ i_q \\ i_0 \end{bmatrix} = \frac{2}{3} \begin{bmatrix} \cos(N\theta_r) & \cos\left(N\theta_r - \frac{2\pi}{3}\right) & \cos\left(N\theta_r + \frac{2\pi}{3}\right) \\ -\sin(N\theta_r) & -\sin\left(N\theta_r - \frac{2\pi}{3}\right) & -\sin\left(N\theta_r + \frac{2\pi}{3}\right) \\ \frac{1}{2} & \frac{1}{2} & \frac{1}{2} \end{bmatrix} \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix}$$

where  $i_a, i_b,$  and  $i_c$  are the A-phase, B-phase, and C-phase currents, respectively.



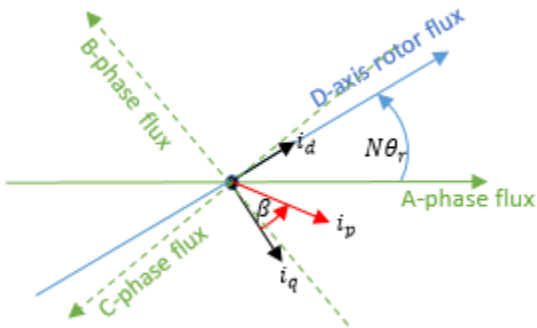
- $N$ : number of pole pairs
- $\theta_r$ : rotor angle
- $i_d, i_q$ : D-axis and Q-axis currents

- $i_p$ : Current magnitude =  $\sqrt{i_d^2 + i_q^2}$
- $\beta$ : Current advance angle =  $\tan^{-1}(-i_d / i_q)$

Corresponding Park transform is

$$\begin{bmatrix} i_d \\ i_q \\ i_0 \end{bmatrix} = \frac{2}{3} \begin{bmatrix} \sin(N\theta_r) & \sin\left(N\theta_r - \frac{2\pi}{3}\right) & \sin\left(N\theta_r + \frac{2\pi}{3}\right) \\ \cos(N\theta_r) & \cos\left(N\theta_r - \frac{2\pi}{3}\right) & \cos\left(N\theta_r + \frac{2\pi}{3}\right) \\ \frac{1}{2} & \frac{1}{2} & \frac{1}{2} \end{bmatrix} \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix}$$

where  $i_a$ ,  $i_b$ , and  $i_c$  are the A-phase, B-phase, and C-phase currents, respectively.

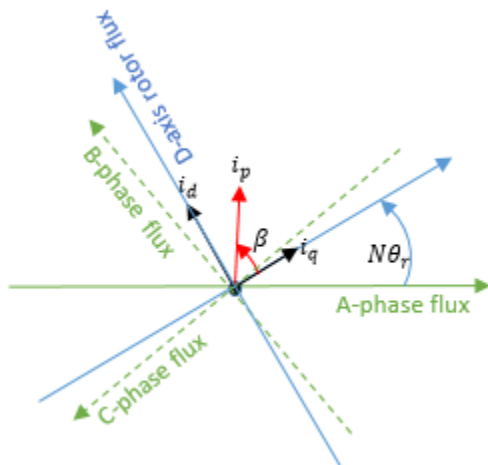


- $N$ : number of pole pairs
- $\theta_r$ : rotor angle
- $i_d, i_q$ : D-axis and Q-axis currents
- $i_p$ : Current magnitude =  $\sqrt{i_d^2 + i_q^2}$
- $\beta$ : Current advance angle =  $\tan^{-1}(-i_d / i_q)$

Corresponding Park transform is

$$\begin{bmatrix} i_d \\ i_q \\ i_0 \end{bmatrix} = \frac{2}{3} \begin{bmatrix} \cos(N\theta_r) & \cos\left(N\theta_r - \frac{2\pi}{3}\right) & \cos\left(N\theta_r + \frac{2\pi}{3}\right) \\ \sin(N\theta_r) & \sin\left(N\theta_r - \frac{2\pi}{3}\right) & \sin\left(N\theta_r + \frac{2\pi}{3}\right) \\ \frac{1}{2} & \frac{1}{2} & \frac{1}{2} \end{bmatrix} \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix}$$

where  $i_a$ ,  $i_b$ , and  $i_c$  are the A-phase, B-phase, and C-phase currents, respectively.



- $N$ : number of pole pairs
- $\theta_r$ : rotor angle
- $i_d, i_q$ : D-axis and Q-axis currents
- $i_p$ : Current magnitude =  $\sqrt{i_d^2 + i_q^2}$
- $\beta$ : Current advance angle =  $\tan^{-1}(-i_d / i_q)$

Corresponding Park transform is

$$\begin{bmatrix} i_d \\ i_q \\ i_0 \end{bmatrix} = \frac{2}{3} \begin{bmatrix} -\sin(N\theta_r) & -\sin\left(N\theta_r - \frac{2\pi}{3}\right) & -\sin\left(N\theta_r + \frac{2\pi}{3}\right) \\ \cos(N\theta_r) & \cos\left(N\theta_r - \frac{2\pi}{3}\right) & \cos\left(N\theta_r + \frac{2\pi}{3}\right) \\ \frac{1}{2} & \frac{1}{2} & \frac{1}{2} \end{bmatrix} \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix}$$

where  $i_a$ ,  $i_b$ , and  $i_c$  are the A-phase, B-phase, and C-phase currents, respectively.

## Calculating Iron Losses

Regardless of the parameterization methods for flux and torque, all block variants use the same iron losses model, which is based on the work of Mellor [1]. Iron losses are divided into two terms, one representing the main magnetizing path, and the other representing the cross-tooth tip path that becomes active during field weakened operation.

The term representing the main magnetizing path depends on the induced RMS stator

voltage,  $V_{m_{rms}}$  :

$$P_{OC}(V_{m_{rms}}) = \frac{a_h}{k} V_{m_{rms}} + \frac{a_j}{k^2} V_{m_{rms}}^2 + \frac{a_{ex}}{k^{1.5}} V_{m_{rms}}^{1.5}$$

This is the dominant term during no-load operation.  $k$  is the back emf constant relating

RMS volts per Hz. It is defined as  $k = V_{m_{rms}}/f$ , where  $f$  is the electrical frequency. The first term on the right-hand side is the magnetic hysteresis loss, the second is the eddy current loss and the third is the excess loss. The three coefficients appearing on the numerators are derived from the values that you provide for the open-circuit hysteresis, eddy, and excess losses.

The term representing the cross-tooth tip path becomes important when a demagnetizing field is set up and can be determined from a finite element analysis short-circuit test. It

depends on the RMS emf associated with the cross-tooth tip flux,  $V_{d_{rms}}^*$  :

$$P_{SC}(V_{d_{rms}}^*) = \frac{b_h}{k} V_{d_{rms}}^* + \frac{b_j}{k^2} V_{d_{rms}}^{*2} + \frac{b_{ex}}{k^{1.5}} V_{d_{rms}}^{*1.5}$$

The three numerator terms are derived from the values you provide for the short-circuit hysteresis, eddy, and excess losses.

## Thermal Ports

The block has four optional thermal ports, one for each of the three windings and one for the rotor. These ports are hidden by default. To expose the thermal ports, right-click the block in your model, select **Simscape > Block choices**, and then select the desired block variant with thermal ports: **2-D data | Show thermal port**, **3-D A-phase data | Show thermal port**, **4-D A-phase data | Show thermal port**, or **3-D DQ data | Show thermal port**. This action displays the thermal ports on the block icon, and adds the **Temperature Dependence** and **Thermal Port** tabs to the block dialog box. These tabs are described further on this reference page.

Use the thermal ports to simulate the effects of copper resistance and iron losses that convert electrical power to heat. For more information on using thermal ports in actuator blocks, see “Simulating Thermal Effects in Rotational and Translational Actuators”.

## Basic Assumptions and Limitations

This block has the following limitations:

- For the 2-D data model, the stator-stator mutual inductance, defined by the **Stator mutual inductance,  $M_s$**  parameter value, is constant during simulation and does not vary with rotor angle. This means that the block is suitable for modeling most PMSM and brushless DC motors, but not switched reluctance motors.
- The 3-D and 4-D data models assume symmetry, so that the flux linkage dependency on currents and rotor angle for windings B and C can be determined from that for winding A.
- For the 3-D flux linkage data model, you do not provide information about zero-sequence inductance. As a result, the block does not expose its neutral port and machine currents are always balanced.
- For the 4-D data model, consider memory requirements when fixing the independent parameter values (three currents and rotor angles). The linear interpolation option uses less memory, but the smooth interpolation option is more accurate for a given independent parameter spacing.
- The iron losses model assumes sinusoidal currents.

## Parameters

### Electrical (2-D Partial Derivative Data Variant)

This configuration of the **Electrical** tab corresponds to the 2-D Partial Derivative Data block variants, with or without thermal ports. If you are using the 3-D Partial Derivative Data, 4-D Partial Derivative Data, or 3-D Flux Linkage Data variant of the block, see “Electrical (3-D Partial Derivative Data Variant)” on page 1-572, “Electrical (4-D Partial Derivative Data Variant)” on page 1-574, or “Electrical (3-D Flux Linkage Data Variant)” on page 1-576 respectively.

#### Parameterization

Select the parameterization method:

- **Assume constant mutual inductance - tabulate with phase current and rotor angle** — This method assumes that the flux linking each winding depends nonlinearly only on the current in that same winding, plus the rotor angle.
- **Assume sinusoidal back emf - tabulate with d- and q-axis currents** — This method assumes that the flux linking each winding depends nonlinearly on all stator winding currents. It also assumes that the permanent magnet flux linkage is sinusoidal. This option is usually a good fit for interior magnet PMSMs (or IPMSMs).

#### Current vector, $i$

Vector of currents corresponding to the provided flux linkage partial derivatives. The current vector must be two-sided (have positive and negative values). The default value is  $[-2, 0, 2]$  A.

This parameter is visible only if **Parameterization** is set to **Assume constant mutual inductance - tabulate with phase current and rotor angle**.

#### Rotor angle vector, $\theta$

Vector of rotor angles corresponding to the provided flux linkage partial derivatives. The vector must start at zero. This value corresponds to the angle where the A-phase magnetic flux aligns with the rotor permanent magnetic peak flux direction (the direct-axis, or  $d$ -axis). The last value,  $\theta_{\max}$ , must be the rotor angle where the flux linkage pattern peaks again. Therefore, the number of pole pairs is  $360/\theta_{\max}$  if  $\theta_{\max}$  is expressed in degrees. The default value is  $[0, 20, 40, 60]$  deg, which corresponds to a 6 pole-pair motor.

This parameter is visible only if **Parameterization** is set to Assume constant mutual inductance - tabulate with phase current and rotor angle.

**Flux linkage partial derivative wrt current,  $d\Phi(i,\theta)/di$** 

Matrix of the flux linkage partial derivatives with respect to current, defined as a function of current vector and rotor angle vector. Flux linkage is the flux multiplied by the number of winding turns. The default value is  $0.0002 \cdot \text{ones}(3,4)$  Wb/A, which corresponds to the special case where stator inductance does not depend on stator current or on rotor angle.

This parameter is visible only if **Parameterization** is set to Assume constant mutual inductance - tabulate with phase current and rotor angle.

**Flux linkage partial derivative wrt angle,  $d\Phi(i,\theta)/d\theta$** 

Matrix of the flux linkage partial derivatives with respect to rotor angle, defined as a function of current vector and rotor angle vector. Flux linkage is the flux multiplied by the number of winding turns. The default value is  $[0, -0.16, 0.16, 0; 0, -0.16, 0.16, 0; 0, -0.16, 0.16, 0]$  Wb/rad, which corresponds to the special case where stator inductance does not depend on stator current.

This parameter is visible only if **Parameterization** is set to Assume constant mutual inductance - tabulate with phase current and rotor angle.

**Direct axis current vector,  $id$** 

Vector of  $d$ -axis currents corresponding to the provided inductances. The current vector must be two-sided (have positive and negative values). The default value is  $[-200, 0, 200]$  A.

This parameter is visible only if **Parameterization** is set to Assume sinusoidal back emf - tabulate with  $d$ - and  $q$ -axis currents.

**Quadrature axis current vector,  $iq$** 

Vector of  $q$ -axis currents corresponding to the provided inductances. The current vector must be two-sided (have positive and negative values). The default value is  $[-200, 0, 200]$  A.

This parameter is visible only if **Parameterization** is set to Assume sinusoidal back emf - tabulate with  $d$ - and  $q$ -axis currents.

**Ld matrix,  $Ld(id,iq)$** 

Matrix of the  $d$ -axis inductances with respect to current, defined as a function of  $d$ -axis and  $q$ -axis current vectors. The default value is  $0.0002 \cdot \text{ones}(3,3)$  H.



This parameter is visible only if **Parameterization** is set to Assume sinusoidal back emf - tabulate with d- and q-axis currents.

### **Lq matrix, Lq(id,iq)**

Matrix of the  $q$ -axis inductances with respect to current, defined as a function of  $d$ -axis and  $q$ -axis current vectors. The default value is  $0.0002 \times \text{ones}(3,3)$  H.

This parameter is visible only if **Parameterization** is set to Assume sinusoidal back emf - tabulate with d- and q-axis currents.

### **Permanent magnet flux linkage, PM(id,iq)**

Matrix of the permanent magnet flux linkages with respect to current, defined as a function of  $d$ -axis and  $q$ -axis current vectors. Flux linkage is the flux multiplied by the number of winding turns. The default value is  $0.1 \times \text{ones}(3,3)$  Wb.

This parameter is visible only if **Parameterization** is set to Assume sinusoidal back emf - tabulate with d- and q-axis currents.

### **Number of pole pairs**

Number of the permanent magnet motor pole pairs. The default value is 6.

This parameter is visible only if **Parameterization** is set to Assume sinusoidal back emf - tabulate with d- and q-axis currents.

### **Calculate torque matrix?**

Specify the way of providing the electromagnetic torque data:

- Yes — The block calculates the torque from the flux linkage information, as a function of current and rotor angle. This is the default option.
- No – specify directly — Enter the electromagnetic torque data directly, by using the **Torque matrix, T(i,theta)** parameter.

This parameter is visible only if **Parameterization** is set to Assume constant mutual inductance - tabulate with phase current and rotor angle. If **Parameterization** is set to Assume sinusoidal back emf - tabulate with d- and q-axis currents, the equation for torque is explicit in terms of the provided matrices.

### **Torque matrix, T(i,theta)**

Specify a matrix of the electromagnetic torque applied to the rotor, as a function of current and rotor angle. This parameter is visible only if **Calculate torque matrix?** is set to No – specify directly. The default value is  $[0, 0.3, -0.3, 0; 0, 0, 0; 0, 0, 0; 0, -0.3, 0.3, 0]$  N\*m.

### **Interpolation method**

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- **Linear** — Uses an extension of linear algorithm for multidimensional interpolation. Select this option to get the best performance.
- **Smooth** — Uses a modified Akima interpolation algorithm. Select this option to produce a continuous surface with continuous first-order derivatives.

For more information on interpolation methods, see the PS Lookup Table (2D) block reference page.

### **Stator resistance per phase, Rs**

Resistance of each of the stator windings. The default value is 0.013 Ohm.

### **Stator mutual inductance, Ms**

Stator-stator mutual inductance, which is assumed to be independent of both current and rotor angle. The default value is 0.00002 H.

This parameter is visible only if **Parameterization** is set to Assume constant mutual inductance - tabulate with phase current and rotor angle.

## **Electrical (3-D Partial Derivative Data Variant)**

This configuration of the **Electrical** tab corresponds to the 3-D Partial Derivative Data block variants, with or without thermal ports. If you are using the 2-D Partial Derivative Data, 4-D Partial Derivative Data, or 3-D Flux Linkage Data variant of the block, see “Electrical (2-D Partial Derivative Data Variant)” on page 1-569, “Electrical (4-D Partial Derivative Data Variant)” on page 1-574, or “Electrical (3-D Flux Linkage Data Variant)” on page 1-576 respectively.

### **Direct-axis current vector, iD**

Vector of direct-axis currents corresponding to the provided flux linkage partial derivatives. The current vector must be two-sided (have positive and negative values). The default value is [-200, 0, 200] A.

### **Quadrature-axis current vector, iQ**

Vector of quadrature-axis currents corresponding to the provided flux linkage partial derivatives. The current vector must be two-sided (have positive and negative values). The default value is [-200, 0, 200] A.

**Rotor angle vector, theta**

Vector of rotor angles corresponding to the provided flux linkage partial derivatives. The vector must start at zero. This value corresponds to the angle where the A-phase magnetic flux aligns with the rotor permanent magnetic peak flux direction (the direct-axis, or  $d$ -axis). The last value,  $\theta_{\max}$ , must be the rotor angle where the flux linkage pattern peaks again. Therefore, the number of pole pairs is  $360/\theta_{\max}$  if  $\theta_{\max}$  is expressed in degrees. The default value is [0, 20, 40, 60] deg, which corresponds to a 6 pole-pair motor.

**A-phase flux linkage partial derivative wrt iA, dPhiA(iD,iQ,theta)/diA**

Matrix of the A-phase flux linkage partial derivatives with respect to current in winding A, defined as a function of the two current vectors and the rotor angle vector. Flux linkage is the flux multiplied by the number of winding turns. The default value is zeros(3, 3, 4) Wb/A.

**A-phase flux linkage partial derivative wrt iB, dPhiA(iD,iQ,theta)/diB**

Matrix of the A-phase flux linkage partial derivatives with respect to current in winding B, defined as a function of the two current vectors and the rotor angle vector. Flux linkage is the flux multiplied by the number of winding turns. The default value is zeros(3, 3, 4) Wb/A.

**A-phase flux linkage partial derivative wrt iC, dPhiA(iD,iQ,theta)/diC**

Matrix of the A-phase flux linkage partial derivatives with respect to current in winding C, defined as a function of the two current vectors and the rotor angle vector. Flux linkage is the flux multiplied by the number of winding turns. The default value is zeros(3, 3, 4) Wb/A.

**A-phase flux linkage partial derivative wrt angle, dPhiA(iD,iQ,theta)/dtheta**

Matrix of the A-phase flux linkage partial derivatives with respect to rotor angle, defined as a function of the two current vectors and the rotor angle vector. Flux linkage is the flux multiplied by the number of winding turns. The default value is zeros(3, 3, 4) Wb/rad.

**Torque matrix, T(iD,iQ,theta)**

Specify a matrix of the electromagnetic torque applied to the rotor, as a function of the two currents and the rotor angle. The default value is zeros(3, 3, 4) N\*m.

**Interpolation method**

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- **Linear** — Uses an extension of linear algorithm for multidimensional interpolation. Select this option to get the best performance.

- **Smooth** — Uses a modified Akima interpolation algorithm. Select this option to produce a continuous surface with continuous first-order derivatives.

For more information on interpolation methods, see the PS Lookup Table (3D) block reference page.

**Stator resistance per phase,  $R_s$** 

Resistance of each of the stator windings. The default value is 0.013 Ohm.

**Electrical (4-D Partial Derivative Data Variant)**

This configuration of the **Electrical** tab corresponds to the 4-D Partial Derivative Data block variants, with or without thermal ports. If you are using the 2-D Partial Derivative Data, 3-D Partial Derivative Data, or 3-D Flux Linkage Data variant of the block, see “Electrical (2-D Partial Derivative Data Variant)” on page 1-569, “Electrical (3-D Partial Derivative Data Variant)” on page 1-572, or “Electrical (3-D Flux Linkage Data Variant)” on page 1-576 respectively.

**A-phase current vector,  $i_A$** 

Vector of A-phase currents corresponding to the provided flux linkage partial derivatives. The current vector must be two-sided (have positive and negative values). The default value is [-200, 0, 200] A.

**B-phase current vector,  $i_B$** 

Vector of B-phase currents corresponding to the provided flux linkage partial derivatives. The current vector must be two-sided (have positive and negative values). The default value is [-200, 0, 200] A.

**C-phase current vector,  $i_C$** 

Vector of C-phase currents corresponding to the provided flux linkage partial derivatives. The current vector must be two-sided (have positive and negative values). The default value is [-200, 0, 200] A.

**Rotor angle vector,  $\theta$** 

Vector of rotor angles corresponding to the provided flux linkage partial derivatives. The vector must start at zero. This value corresponds to the angle where the A-phase magnetic flux aligns with the rotor permanent magnetic peak flux direction (the direct-axis, or  $d$ -axis). The last value,  $\theta_{\max}$ , must be the rotor angle where the flux linkage pattern peaks again. Therefore, the number of pole pairs is  $360/\theta_{\max}$  if  $\theta_{\max}$  is expressed in degrees. The default value is [0, 20, 40, 60] deg, which corresponds to a 6 pole-pair motor.

**A-phase flux linkage partial derivative wrt  $i_A$ ,  $d\Phi_A(i_A, i_B, i_C, \theta)/di_A$** 

Matrix of the A-phase flux linkage partial derivatives with respect to current in winding A, defined as a function of the three current vectors and the rotor angle vector. Flux linkage is the flux multiplied by the number of winding turns. The default value is `zeros(3, 3, 3, 4)` Wb/A.

**A-phase flux linkage partial derivative wrt  $i_B$ ,  $d\Phi_A(i_A, i_B, i_C, \theta)/di_B$** 

Matrix of the A-phase flux linkage partial derivatives with respect to current in winding B, defined as a function of the three current vectors and the rotor angle vector. Flux linkage is the flux multiplied by the number of winding turns. The default value is `zeros(3, 3, 3, 4)` Wb/A.

**A-phase flux linkage partial derivative wrt  $i_C$ ,  $d\Phi_A(i_A, i_B, i_C, \theta)/di_C$** 

Matrix of the A-phase flux linkage partial derivatives with respect to current in winding C, defined as a function of the three current vectors and the rotor angle vector. Flux linkage is the flux multiplied by the number of winding turns. The default value is `zeros(3, 3, 3, 4)` Wb/A.

**A-phase flux linkage partial derivative wrt angle,  $d\Phi_A(i_A, i_B, i_C, \theta)/d\theta$** 

Matrix of the A-phase flux linkage partial derivatives with respect to rotor angle, defined as a function of three current vectors and the rotor angle vector. Flux linkage is the flux multiplied by the number of winding turns. The default value is `zeros(3, 3, 3, 4)` Wb/rad.

**Torque matrix,  $T(i_A, i_B, i_C, \theta)$** 

Specify a matrix of the electromagnetic torque applied to the rotor, as a function of the three currents and the rotor angle. The default value is `zeros(3, 3, 3, 4)` N\*m.

**Interpolation method**

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- **Linear** — Uses an extension of linear algorithm for multidimensional interpolation. Select this option to get the best performance.
- **Smooth** — Uses a modified Akima interpolation algorithm. Select this option to produce a continuous surface with continuous first-order derivatives.

For more information on interpolation methods, see the PS Lookup Table (4D) block reference page.

### **Stator resistance per phase, Rs**

Resistance of each of the stator windings. The default value is 0.013 Ohm.

## **Electrical (3-D Flux Linkage Data Variant)**

This configuration of the **Electrical** tab corresponds to the 3-D Flux Linkage Data block variants, with or without thermal ports. If you are using the 2-D Partial Derivative Data, 3-D Partial Derivative Data, or 4-D Partial Derivative Data variant of the block, see “Electrical (2-D Partial Derivative Data Variant)” on page 1-569, “Electrical (3-D Partial Derivative Data Variant)” on page 1-572, or “Electrical (4-D Partial Derivative Data Variant)” on page 1-574, respectively.

### **Flux linkage data format**

Select the flux linkage data format used by your FE tool:

- D and Q axes flux linkages as a function of D-axis current ( $i_D$ ), Q-axis current ( $i_Q$ ), and rotor angle ( $\theta$ )
- D and Q axes flux linkages as a function of peak current magnitude ( $I$ ), current advance angle ( $B$ ), and rotor angle ( $\theta$ )
- A-phase flux linkage as a function of D-axis current ( $i_D$ ), Q-axis current ( $i_Q$ ), and rotor angle ( $\theta$ )
- A-phase flux linkage as a function of peak current magnitude ( $I$ ), current advance angle ( $B$ ), and rotor angle ( $\theta$ )

The default value is D and Q axes flux linkages as a function of D-axis current ( $i_D$ ), Q-axis current ( $i_Q$ ), and rotor angle ( $\theta$ ).

### **Winding type**

Select the configuration for the stator windings:

- **Wye-wound** — The stator windings are wye-wound. This is the default value.
- **Delta-wound** — The stator windings are delta-wound. The *a*-phase is connected between ports **a** and **b**, the *b*-phase between ports **b** and **c** and the *c*-phase between ports **c** and **a**.

### **Number of pole pairs**

Number of the permanent magnet motor pole pairs. The default value is 4.

### Park's convention for tabulated data

Select the order and reference angle for the Park transform mapping the given  $dq$  data to the three windings.

- Q leads D, rotor angle measured from A-phase to D-axis — Quadrature-direct transformation with angle measured with respect to  $d$  axis.
- Q leads D, rotor angle measured from A-phase to Q-axis — Quadrature-direct transformation with angle measured with respect to  $q$  axis.
- D leads Q, rotor angle measured from A-phase to D-axis — Direct-quadrature transformation with angle measured with respect to  $d$  axis.
- D leads Q, rotor angle measured from A-phase to Q-axis — Direct-quadrature transformation with angle measured with respect to  $q$  axis.

The default value is D leads Q, rotor angle measured from A-phase to D-axis.

### Direct-axis current vector, $iD$

Vector of direct-axis currents at which the flux linkage is tabulated. The current vector must be two-sided (have positive and negative values). The default value is  $[-200, 0, 200]$  A.

This parameter is visible only if **Flux linkage data format** is set to D and Q axes flux linkages as a function of D-axis current ( $iD$ ), Q-axis current ( $iQ$ ), and rotor angle ( $\theta$ ) or A-phase flux linkage as a function of D-axis current ( $iD$ ), Q-axis current ( $iQ$ ), and rotor angle ( $\theta$ ).

### Quadrature-axis current vector, $iQ$

Vector of quadrature-axis currents at which the flux linkage is tabulated. The current vector must be two-sided (have positive and negative values). The default value is  $[-200, 0, 200]$  A.

This parameter is visible only if **Flux linkage data format** is set to D and Q axes flux linkages as a function of D-axis current ( $iD$ ), Q-axis current ( $iQ$ ), and rotor angle ( $\theta$ ) or A-phase flux linkage as a function of D-axis current ( $iD$ ), Q-axis current ( $iQ$ ), and rotor angle ( $\theta$ ).

### Peak current magnitude vector, $I$

Row vector of current magnitudes at which the flux linkage is tabulated. The first element must be zero. The adjacent current value should be small relative to the

current values at which magnetic saturation begins to occur. This is because derived flux partial derivatives are ill-defined at zero current, and so are calculated at this first nonzero current instead. The default value is [0, 100, 200] A.

This parameter is visible only if **Flux linkage data format** is set to D and Q axes flux linkages as a function of peak current magnitude (I), current advance angle (B), and rotor angle (theta) or A-phase flux linkage as a function of peak current magnitude (I), current advance angle (B), and rotor angle (theta).

### **Current advance angle, B**

Row vector of current advance angle values at which the flux linkage is tabulated. Current advance angle is defined as the angle by which the current leads the quadrature (Q) axis. The angle range must be -180 to 180 degrees. The default value is [-180, -90, 0, 90, 180] degrees.

This parameter is visible only if **Flux linkage data format** is set to D and Q axes flux linkages as a function of peak current magnitude (I), current advance angle (B), and rotor angle (theta) or A-phase flux linkage as a function of peak current magnitude (I), current advance angle (B), and rotor angle (theta).

### **Rotor angle vector, theta**

Vector of rotor angles at which the flux linkage is tabulated. The vector must start at zero. This value corresponds to the angle where the A-phase magnetic flux aligns with the rotor permanent magnetic peak flux direction (the direct-axis, or *d*-axis). The last value,  $\theta_{\max}$ , must be the rotor angle where the flux linkage pattern peaks again. Therefore, the number of pole pairs is  $360/\theta_{\max}$  if  $\theta_{\max}$  is expressed in degrees. The default value is [[0, 5, 10, 15, 20, 25, 30] deg, which corresponds to a 4 pole-pair motor.

If **Flux linkage data format** is D and Q axes flux linkages as a function of D-axis current (iD), Q-axis current (iQ), and rotor angle (theta) or D and Q axes flux linkages as a function of peak current magnitude (I), current advance angle (B), and rotor angle (theta) (that is, if you tabulate D and Q flux linkage data), then the rotor angle vector must have four or more points and a range from 0 to  $120/N$  degrees, where *N* is the number of pole pairs. If **Flux linkage data format** is A-phase flux linkage as a function of D-axis current (iD), Q-axis current (iQ), and rotor angle (theta) or A-phase flux linkage as a function of peak current magnitude (I), current advance angle (B), and rotor angle (theta)



(that is, if you tabulate A-phase flux linkage data), then the rotor angle vector must have  $3n+1$  points, where  $n \geq 2$ , and the range must be from 0 to  $360/N$  degrees.

#### **D-axis flux linkage, $F_d(iD, iQ, \theta)$**

Matrix of the  $d$ -axis flux linkage, defined as a function of the  $dq$  currents, and the rotor angle vector. Flux linkage is the flux multiplied by the number of winding turns. The default value is `zeros(3, 3, 7)` Wb.

If your flux data is given in a different order, you can use the `permute` function to reorder it. For an example of this reordering, see the associated MATLAB script in “Import IPMSM Flux Linkage Data from ANSYS Maxwell”.

This parameter is visible only if **Flux linkage data format** is set to D and Q axes flux linkages as a function of D-axis current ( $iD$ ), Q-axis current ( $iQ$ ), and rotor angle ( $\theta$ ).

#### **Q-axis flux linkage, $F_q(iD, iQ, \theta)$**

Matrix of the  $q$ -axis flux linkage, defined as a function of the  $dq$  currents, and the rotor angle vector. Flux linkage is the flux multiplied by the number of winding turns. The default value is `zeros(3, 3, 7)` Wb.

If your flux data is given in a different order, you can use the `permute` function to reorder it. For an example of this reordering, see the associated MATLAB script in “Import IPMSM Flux Linkage Data from ANSYS Maxwell”.

This parameter is visible only if **Flux linkage data format** is set to D and Q axes flux linkages as a function of D-axis current ( $iD$ ), Q-axis current ( $iQ$ ), and rotor angle ( $\theta$ ).

#### **D-axis flux linkage, $F_d(I, B, \theta)$**

3-D matrix of  $d$ -axis flux linkage values as a function of **Peak current magnitude vector, I**, **Current advance angle, B**, and **Rotor angle vector,  $\theta$** . The default value is `zeros(3, 5, 7)` Wb.

If your flux data is given in a different order, you can use the `permute` function to reorder it. For an example of this reordering, see the associated MATLAB script in “Import IPMSM Flux Linkage Data from ANSYS Maxwell”.

This parameter is visible only if **Flux linkage data format** is set to D and Q axes flux linkages as a function of peak current magnitude ( $I$ ), current advance angle ( $B$ ), and rotor angle ( $\theta$ ).

**Q-axis flux linkage,  $F_q(I,B,\theta)$** 

3-D matrix of  $q$ -axis flux linkage values as a function of **Peak current magnitude vector,  $I$** , **Current advance angle,  $B$** , and **Rotor angle vector,  $\theta$** . The default value is `zeros(3, 5, 7)` Wb.

If your flux data is given in a different order, you can use the `permute` function to reorder it. For an example of this reordering, see the associated MATLAB script in “Import IPMSM Flux Linkage Data from ANSYS Maxwell”.

This parameter is visible only if **Flux linkage data format** is set to D and Q axes flux linkages as a function of peak current magnitude ( $I$ ), current advance angle ( $B$ ), and rotor angle ( $\theta$ ).

**A-phase flux linkage,  $F(iD,iQ,\theta)$** 

3-D matrix of A-phase flux linkage values, as a function of the  $dq$  currents and the rotor angle. The default value is `zeros(3, 3, 7)` Wb.

If your flux data is given in a different order, you can use the `permute` function to reorder it. For an example of this reordering, see the associated MATLAB script in “Import IPMSM Flux Linkage Data from ANSYS Maxwell”.

This parameter is visible only if **Flux linkage data format** is set to A-phase flux linkage as a function of D-axis current ( $iD$ ), Q-axis current ( $iQ$ ), and rotor angle ( $\theta$ ).

**A-phase flux linkage,  $F(I,B,\theta)$** 

3-D matrix of A-phase flux linkage values, as a function of **Peak current magnitude vector,  $I$** , **Current advance angle,  $B$** , and **Rotor angle vector,  $\theta$** . The default value is `zeros(3, 5, 7)` Wb.

If your flux data is given in a different order, you can use the `permute` function to reorder it. For an example of this reordering, see the associated MATLAB script in “Import IPMSM Flux Linkage Data from ANSYS Maxwell”.

This parameter is visible only if **Flux linkage data format** is set to A-phase flux linkage as a function of peak current magnitude ( $I$ ), current advance angle ( $B$ ), and rotor angle ( $\theta$ ).

**Torque matrix,  $T(iD,iQ,\theta)$** 

3-D matrix of the electromagnetic torque applied to the rotor, as a function of the  $dq$  currents and the rotor angle. The default value is `zeros(3, 3, 7)` N\*m.

If your flux data is given in a different order, you can use the `permute` function to reorder it. For an example of this reordering, see the associated MATLAB script in “Import IPMSM Flux Linkage Data from ANSYS Maxwell”.

This parameter is visible only if **Flux linkage data format** is set to D and Q axes flux linkages as a function of D-axis current ( $i_D$ ), Q-axis current ( $i_Q$ ), and rotor angle ( $\theta$ ) or A-phase flux linkage as a function of D-axis current ( $i_D$ ), Q-axis current ( $i_Q$ ), and rotor angle ( $\theta$ ).

### **Torque matrix, $T(I,B,\theta)$**

3-D matrix of the electromagnetic torque applied to the rotor, as a function of **Peak current magnitude vector,  $I$** , **Current advance angle,  $B$** , and **Rotor angle vector,  $\theta$** . The default value is zeros(3, 5, 7) N\*m.

If your flux data is given in a different order, you can use the `permute` function to reorder it. For an example of this reordering, see the associated MATLAB script in “Import IPMSM Flux Linkage Data from ANSYS Maxwell”.

This parameter is visible only if **Flux linkage data format** is set to D and Q axes flux linkages as a function of peak current magnitude ( $I$ ), current advance angle ( $B$ ), and rotor angle ( $\theta$ ) or A-phase flux linkage as a function of peak current magnitude ( $I$ ), current advance angle ( $B$ ), and rotor angle ( $\theta$ ).

### **Interpolation method**

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- **Linear** — Uses an extension of linear algorithm for multidimensional interpolation. Select this option to get the best performance.
- **Smooth** — Uses a modified Akima interpolation algorithm. Select this option to produce a continuous surface with continuous first-order derivatives.

For more information on interpolation methods, see the PS Lookup Table (4D) block reference page.

### **Stator resistance per phase, $R_s$**

Resistance of each of the stator windings. The default value is 0.013 Ohm.

## Iron Losses

### Open-circuit iron losses, [P\_hysteresis P\_eddy P\_excess]

Row vector, of length 3, of the open-circuit iron losses due to hysteresis, Eddy, and excess losses, respectively, at the frequency specified by **Electrical frequency at which losses determined**. The default value is [0.0, 0.0, 0.0] W.

### Short-circuit iron losses, [P\_hysteresis P\_eddy P\_excess]

Row vector, of length 3, of the short-circuit iron losses due to hysteresis, Eddy, and excess losses, respectively, at the frequency specified by **Electrical frequency at which losses determined**. The default value is [0.0, 0.0, 0.0] W.

### Electrical frequency at which losses determined

Electrical frequency at which the open-circuit and short-circuit iron losses were measured. The default value is 60 Hz.

### Short-circuit RMS current for short-circuit iron losses

The resulting short-circuit RMS phase current when measuring the short-circuit losses. The default value is 95 A.

## Mechanical

### Rotor inertia

Inertia of the rotor attached to mechanical translational port R. The default value is 0.01 kg\*m<sup>2</sup>. The value can be zero.

### Rotor damping

Rotary damping. The default value is 0 N\*m/(rad/s).

## Temperature Dependence

This tab appears only for blocks with exposed thermal ports. For more information, see “Thermal Ports” on page 1-568.

### Measurement temperature

The temperature for which motor parameters are quoted. The default value is 298.15 K.

### Resistance temperature coefficient

Coefficient  $\alpha$  in the equation relating resistance to temperature, as described in “Thermal Model for Actuator Blocks”. The default value, 3.93e-3 1/K, is for copper.

**Permanent magnet flux temperature coefficient**

The fractional rate of change of permanent magnet flux density with temperature. It is used to linearly reduce the torque and the induced back EMF as temperature rises. The value is usually negative. The default value is  $-0.001$ .

**Thermal Port**

This tab appears only for blocks with exposed thermal ports. For more information, see “Thermal Ports” on page 1-568.

**Thermal mass for each stator winding**

The thermal mass value for the A, B, and C windings. The thermal mass is the energy required to raise the temperature by one degree. The default value is 100 J/K.

**Initial stator winding temperatures**

A 1-by-3 row vector defining the temperature of the A, B, and C thermal ports at the start of simulation. The default value is [298.15, 298.15, 298.15] K.

**Rotor thermal mass**

The thermal mass of the rotor, that is, the energy required to raise the temperature of the rotor by one degree. The default value is 200 J/K.

**Rotor initial temperature**

The temperature of the rotor at the start of simulation. The default value is 298.15 K.

**Percentage of main flux path iron losses associated with the rotor**

The percentage of the main flux path iron losses associated with the magnetic path through the rotor. It determines how much of the iron loss heating is attributed to the rotor thermal port HR, and how much is attributed to the three winding thermal ports HA, HB, and HC. The default value is 90%.

**Percentage of cross-tooth flux path iron losses associated with the rotor**

The percentage of the cross-tooth flux path iron losses associated with the magnetic path through the rotor. It determines how much of the iron loss heating is attributed to the rotor thermal port HR, and how much is attributed to the three winding thermal ports HA, HB, and HC. The default value is 30%.

**Ports**

This block has the following ports:

a

A-phase electrical connection.

b

B-phase electrical connection.

c

C-phase electrical connection.

n

Electrical conserving port associated with the neutral phase.

C

Mechanical rotational conserving port connected to the motor case.

R

Mechanical rotational conserving port connected to the rotor.

HA

Winding A thermal port. For more information, see “Thermal Ports” on page 1-568.

HB

Winding B thermal port. For more information, see “Thermal Ports” on page 1-568.

HC

Winding C thermal port. For more information, see “Thermal Ports” on page 1-568.

HR

Rotor thermal port. For more information, see “Thermal Ports” on page 1-568.

## References

- [1] Mellor, P.H., R. Wrobel, and D. Holliday. “A computationally efficient iron loss model for brushless AC machines that caters for rated flux and field weakened operation.” *IEEE Electric Machines and Drives Conference*. May 2009.

## See Also

FEM-Parameterized Linear Actuator | FEM-Parameterized Rotary Actuator |  
elec\_calculateFluxPartialDerivatives

## **Topics**

“Import IPMSM Flux Linkage Data from ANSYS Maxwell”

HEV PMSM Drive Test Harness

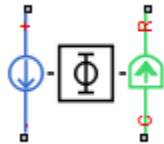
PMSM Iron Losses

PMSM with Thermal Model

**Introduced in R2015b**

# FEM-Parameterized Rotary Actuator

Model rotary actuator defined in terms of magnetic flux



## Library

Rotational Actuators

## Description

The FEM-Parameterized Rotary Actuator block implements a model of a rotary actuator defined in terms of magnetic flux. Use this block to model custom rotary actuators and motors where magnetic flux depends on both rotor angle and current. You parameterize the block using data from a third-party Finite Element Magnetic (FEM) package.

The block has two options for the electrical equation. The first, **Define in terms of  $d\Phi(i, \theta)/d\theta$  and  $d\Phi(i, \theta)/di$** , defines the current in terms of partial derivatives of the magnetic flux ( $\Phi$ ) with respect to rotor angle ( $\theta$ ) and current ( $i$ ), the equations for which are:

$$\frac{di}{dt} = \left( v - iR - \frac{\partial \Phi}{\partial \theta} \frac{d\theta}{dt} \right) / \frac{\partial \Phi}{\partial i}$$

The second option, **Define in terms of  $\Phi(i, \theta)$** , defines the voltage across the component directly in terms of the flux, the equation for which is:

$$v = iR + \frac{d}{dt} \Phi(\theta, i)$$



Numerically, defining the electrical equation in terms of flux partial derivatives is better because the back-emf is piecewise continuous. If using the flux directly, using a finer grid size for current and position will improve results, as will selecting cubic or spline interpolation.

In both cases, you have an option to either directly specify the torque as a function of current and rotor angle, by using the **Torque matrix, T(i,theta)** parameter, or have the block automatically calculate the torque matrix.

If entering the electromagnetic torque data directly, you can either use data supplied by the finite element magnetic package (which you used to determine the flux) or calculate the torque from the flux with following equation:

$$T = \int_0^i \frac{\partial \Phi(\theta, i)}{\partial \theta} di$$

See the Finite Element Parameterized Solenoid example model and its initialization file `elec_fem_solenoid_ini.m` for an example of how to implement this type of integration in MATLAB.

Alternatively, the block can automatically calculate the torque matrix from the flux information that you provide. To select this option, set the **Calculate torque matrix?** parameter to **Yes**. The torque matrix calculation occurs at model initialization based on current block flux linkage information. The torque is calculated by numerically integrating the rate of change of flux linkage with respect to angle over current, according to the preceding equation. If the **Electrical model** parameter is set to **Define in terms of Phi(i,theta)**, then the block must first estimate the **Flux partial derivative wrt angle, Phi(i,theta)/dtheta** parameter value from the flux linkage data. When doing this, the block uses the interpolation method specified by the **Interpolation method** parameter. Typically, the **Smooth** option is most accurate, but the **Linear** option is most robust.

You can define  $\Phi$  and its partial derivatives for just positive, or positive and negative currents. If defining for just positive currents, then the block assumes that  $\Phi(-i,x) = -\Phi(i,x)$ . Therefore, if the current vector is positive only:

- The first current value must be zero.
- The flux corresponding to zero current must be zero.
- The partial derivative of flux with respect to rotor angle must be zero for zero current.

To model a rotary motor with a repeated flux pattern, set the **Flux dependence on displacement** parameter to *Cyclic*. When selecting this option, the torque and flux (or torque and flux partial derivatives depending on the option chosen) must have identical first and last columns.

## Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal port H on the block icon, and adds the **Temperature Dependence** and **Thermal Port** tabs to the block dialog box.

Use the thermal port to simulate the effects of copper resistance losses that convert electrical power to heat. For more information on using thermal ports and on the **Temperature Dependence** and **Thermal Port** tab parameters, see “Simulating Thermal Effects in Rotational and Translational Actuators”.

## Basic Assumptions and Limitations

This block has the following limitations:

- It is imperative that you supply a consistent set of torque and flux data. There is no checking that the torque matrix is consistent with the flux data.
- When driving the FEM-Parameterized Rotary Actuator block via a series inductor, you may need to include a parallel conductance in the inductor component.

## Parameters

- “Magnetic Force Tab” on page 1-588
- “Mechanical Tab” on page 1-591

## Magnetic Force Tab

### Electrical model

Select one of the following parameterization options, based on the underlying electrical model:

- Define in terms of  $d\Phi(i, \theta)/d\theta$  and  $d\Phi(i, \theta)/di$  — Define the current through the block in terms of partial derivatives of the magnetic flux with respect to rotor angle and current. This is the default method.
- Define in terms of  $\Phi(i, \theta)$  — Define the voltage across the block terminals directly in terms of the flux.

### Current vector, $i$

Specify a vector of monotonically increasing current values corresponding to your torque-flux data. If you specify positive currents only, the first element must be zero. The default value is [ 0 0.2 0.4 0.6 0.8 1 ] A.

### Angle vector, $\theta$

Specify a vector of monotonically increasing rotor angle values corresponding to your torque-flux data. The default value is [ 0 10 20 30 40 50 60 70 80 90 100 110 120 130 140 150 160 170 180 ] deg.

### Flux partial derivative wrt current, $d\Phi(i, \theta)/di$

Specify a matrix of the flux partial derivatives with respect to current. This parameter is visible only if **Electrical model** is set to **Define in terms of  $d\Phi(i, \theta)/d\theta$  and  $d\Phi(i, \theta)/di$** . The default value, in Wb/A, is:

```
[ 0.002 0.0024 0.0035 0.0052 0.0074 0.0096 0.0118 0.0135 0.0146 ...
  0.015 0.0146 0.0135 0.0118 0.0096 0.0074 0.0052 0.0035 0.0024 0.002;
0.002 0.0024 0.0035 0.0052 0.0074 0.0096 0.0118 0.0135 0.0146 ...
  0.015 0.0146 0.0135 0.0118 0.0096 0.0074 0.0052 0.0035 0.0024 0.002;
0.002 0.0024 0.0035 0.0052 0.0074 0.0096 0.0118 0.0135 0.0146 ...
  0.015 0.0146 0.0135 0.0118 0.0096 0.0074 0.0052 0.0035 0.0024 0.002;
0.002 0.0024 0.0035 0.0052 0.0074 0.0096 0.0118 0.0135 0.0146 ...
  0.015 0.0146 0.0135 0.0118 0.0096 0.0074 0.0052 0.0035 0.0024 0.002;
0.002 0.0024 0.0035 0.0052 0.0074 0.0096 0.0118 0.0135 0.0146 ...
  0.015 0.0146 0.0135 0.0118 0.0096 0.0074 0.0052 0.0035 0.0024 0.002; ]
```

### Flux partial derivative wrt angle, $d\Phi(i, \theta)/d\theta$

Specify a matrix of the flux partial derivatives with respect to rotor angle. This parameter is visible only if **Electrical model** is set to **Define in terms of  $d\Phi(i, \theta)/d\theta$  and  $d\Phi(i, \theta)/di$** . The default value, in Wb/rad, is:

```
[ 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0;
0 9e-4 0.0017 0.0023 0.0026 0.0026 0.0023 0.0017 9e-4 ...
  0 -9e-4 -0.0017 -0.0023 -0.0026 -0.0026 -0.0023 -0.0017 -9e-4 0;
0 0.0018 0.0033 0.0045 0.0051 0.0051 0.0045 0.0033 0.0018 ...
  0 -0.0018 -0.0033 -0.0045 -0.0051 -0.0051 -0.0045 -0.0033 -0.0018 0;
0 0.0027 0.005 0.0068 0.0077 0.0077 0.0068 0.005 0.0027 ...
  0 -0.0027 -0.005 -0.0068 -0.0077 -0.0077 -0.0068 -0.005 -0.0027 0;
0 0.0036 0.0067 0.009 0.0102 0.0102 0.009 0.0067 0.0036 ...
  0 -0.0036 -0.0067 -0.009 -0.0102 -0.0102 -0.009 -0.0067 -0.0036 0;
```

```
0 0.0044 0.0084 0.0113 0.0128 0.0128 0.0113 0.0084 0.0044 ...
0 -0.0044 -0.0084 -0.0113 -0.0128 -0.0128 -0.0113 -0.0084 -0.0044 0 ]
```

### Flux linkage matrix, Phi(i,theta)

Specify a matrix of the total flux linkage, that is, flux times the number of turns. This parameter is visible only if **Electrical model** is set to Define in terms of Phi(i, theta). The default value, in Wb, is:

```
[ 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0;
4e-4 4.8e-4 7e-4 0.00105 0.00147 0.00193 0.00235 0.0027 0.00292 ...
0.003 0.00292 0.0027 0.00235 0.00193 0.00147 0.00105 7e-4 4.8e-4 4e-4;
8e-4 9.6e-4 0.00141 0.0021 0.00295 0.00385 0.0047 0.00539 0.00584 ...
0.006 0.00584 0.00539 0.0047 0.00385 0.00295 0.0021 0.00141 9.6e-4 8e-4;
0.0012 0.00144 0.00211 0.00315 0.00442 0.00578 0.00705 0.00809 0.00876 ...
0.009 0.00876 0.00809 0.00705 0.00578 0.00442 0.00315 0.00211 0.00144 0.0012;
0.0016 0.00191 0.00282 0.0042 0.0059 0.0077 0.0094 0.01078 0.01169 ...
0.012 0.01169 0.01078 0.0094 0.0077 0.0059 0.0042 0.00282 0.00191 0.0016;
0.002 0.00239 0.00352 0.00525 0.00737 0.00963 0.01175 0.01348 0.01461 ...
0.015 0.01461 0.01348 0.01175 0.00963 0.00737 0.00525 0.00352 0.00239 0.002 ]
```

### Calculate torque matrix?

Specify the way of providing the electromagnetic torque data:

- No — specify directly — Enter the electromagnetic torque data directly, by using the **Torque matrix, T(i,theta)** parameter. This is the default option.
- Yes — The block calculates the torque from the flux linkage information, as a function of current and rotor angle.

### Torque matrix, T(i,theta)

Specify a matrix of the electromagnetic torque applied to the rotor. This parameter is visible only if **Calculate torque matrix?** is set to No — specify directly. The default value, in mN\*m, is:

```
[ 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0;
0 0.0889 0.1671 0.2252 0.2561 0.2561 0.2252 0.1671 0.0889 ...
0 -0.0889 -0.1671 -0.2252 -0.2561 -0.2561 -0.2252 -0.1671 -0.0889 0;
0 0.3557 0.6685 0.9007 1.0242 1.0242 0.9007 0.6685 0.3557 ...
0 -0.3557 -0.6685 -0.9007 -1.0242 -1.0242 -0.9007 -0.6685 -0.3557 0;
0 0.8003 1.5041 2.0265 2.3045 2.3045 2.0265 1.5041 0.8003 ...
0 -0.8003 -1.5041 -2.0265 -2.3045 -2.3045 -2.0265 -1.5041 -0.8003 0;
0 1.4228 2.674 3.6027 4.0968 4.0968 3.6027 2.674 1.4228 ...
0 -1.4228 -2.674 -3.6027 -4.0968 -4.0968 -3.6027 -2.674 -1.4228 0;
0 2.2231 4.1781 5.6292 6.4013 6.4013 5.6292 4.1781 2.2231 ...
0 -2.2231 -4.1781 -5.6292 -6.4013 -6.4013 -5.6292 -4.1781 -2.2231 0 ]
```

### Flux dependence on displacement

Specify the flux pattern:

- Unique — No flux pattern present. This is the default option.

- **Cyclic** — Select this option to model a rotary motor with a repeated flux pattern. The torque and flux (or torque and flux partial derivatives, depending on the **Electrical model** option chosen) must have identical first and last columns.

### Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- **Linear** — Select this option to get the best performance.
- **Smooth** — Select this option to produce a continuous surface with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (2D) block reference page.

### Extrapolation method

Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- **Linear** — Select this option to produce a surface with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.
- **Nearest** — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (2D) block reference page.

This parameter is not available if you set the **Flux dependence on displacement** parameter to **Cyclic**.

### Winding resistance

Total resistance of the electrical winding. The default value is 10 Ohm.

## Mechanical Tab

### Damping

Rotary damping. The default value is  $1e-4$  N\*m/(rad/s). The value can be zero.

### **Rotor inertia**

Inertia of the rotor attached to mechanical translational port R. The default value is  $5e-5 \text{ kg}\cdot\text{m}^2$ . The value can be zero.

### **Minimum rotor angle**

The rotor angle at which the lower mechanical end stop is applied. The default value is  $-\text{Inf}$ .

### **Maximum rotor angle**

The rotor angle at which the upper mechanical end stop is applied. The default value is  $\text{Inf}$ .

### **Initial rotor position**

Position of the rotor at the start of the simulation. The default value is  $0 \text{ deg}$ .

### **Initial rotor velocity**

Angular velocity of the rotor at the start of the simulation. The default value is  $0 \text{ deg/s}$ .

### **Contact stiffness**

Contact stiffness between rotor and end stops. The default value is  $1e8 \text{ N}\cdot\text{m}/\text{rad}$ .

### **Contact damping**

Contact damping between rotor and end stops. The default value is  $1e4 \text{ N}\cdot\text{m}/(\text{rad/s})$ .

## **Ports**

This block has the following ports:

+

Positive electrical conserving port

-

Negative electrical conserving port

C

Mechanical rotational conserving port connected to the actuator case

R

Mechanical rotational conserving port connected to the rotor

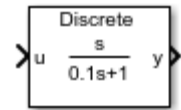
## **See Also**

FEM-Parameterized Linear Actuator | FEM-Parameterized PMSM | Solenoid

## Filtered Derivative (Discrete or Continuous)

Discrete-time or continuous-time filtered derivative

**Library:** Simscape / Electrical / Power Systems / Control /  
General Control



### Description

The Filtered Derivative (Discrete or Continuous) block implements a filtered derivative in conformance with IEEE 421.5-2016<sup>[1]</sup>.

You can switch between continuous and discrete implementations of the derivative using the **Sample time** parameter.

### Equations

To configure the filtered derivative for continuous time, set the **Sample time** property to 0. This representation is equivalent to the continuous transfer function:

$$G(s) = \frac{Ks}{Ts + 1},$$

where:

- $K$  is the gain.
- $T$  is the time constant.

From the preceding transfer function, the derivative defining equations are:

$$\begin{cases} \dot{x}(t) = \frac{1}{T}(Ku(t) - x(t)) \\ y(t) = \frac{1}{T}(Ku(t) - x(t)) \end{cases} \quad x(0) = u_0, \quad y(0) = 0,$$



where:

- $u$  is the block input.
- $x$  is the state.
- $y$  is the block output.
- $t$  is the simulation time.
- $u_0$  is the initial input to the block.

To configure the filtered derivative for discrete time, set the **Sample time** property to a positive, nonzero value, or to -1 to inherit the sample time from an upstream block. The discrete representation is equivalent to the transfer function:

$$\left(\frac{K}{T}\right) \frac{z-1}{z+T_s/T-1},$$

where:

- $K$  is the gain.
- $T$  is the time constant.
- $T_s$  is the sample time.

From the discrete transfer function, the derivative equations are defined using the forward Euler method:

$$\begin{cases} x(n+1) = \left(1 - \frac{T_s}{T}\right)x(n) + \left(\frac{T_s}{T}\right)u(n) \\ y(n) = \frac{K}{T}(u(n) - x(n)) \end{cases} \quad x(0) = u_0, y(0) = 0,$$

where:

- $u$  is the block input.
- $x$  is the block state.
- $y$  is the block output.
- $n$  is the simulation time step.
- $u_0$  is the initial input to the block.

## Initial Conditions

The block sets the state initial condition to the initial input, making the initial output zero.

## Limiting the Output

Limit the filtered derivative output by setting the **Upper saturation limit** and **Lower saturation limit** parameters to finite values.

Unlike other common blocks given in IEEE 421.5-2016, there is no difference between the windup and anti-windup saturation methods for the filtered derivative. The output can respond immediately to a reversal of the input sign when the output is saturated.

## Ports

### Input

#### **u — Derivative input**

vector

Filtered derivative input signal. The block uses the input initial value to determine the state initial value.

Data Types: `single` | `double`

### Output

#### **y — Derivative output**

vector

Filtered derivative output signal.

Data Types: `single` | `double`

## Parameters

#### **Gain — Derivative gain**

1 (default) | positive number

Filtered derivative gain.

**Time constant — Derivative time constant**

0.1 (default) | positive number

Filtered derivative time constant. For acceptable accuracy, set this value at least 10 times greater than the **Sample time**.

**Upper saturation limit — Output upper limit**

inf (default) | real number

Filtered derivative upper output limit. Set this to inf for an unsaturated upper limit.

**Lower saturation limit — Output lower limit**

-inf (default) | real number

Filtered derivative lower output limit. Set this to -inf for an unsaturated lower limit.

**Minimum sample time to time constant ratio — Discrete ratio**

10 (default) | real number

Minimum acceptable sample time to time constant ratio. As the sample time approaches the time constant, the accuracy of the block decreases. Use this parameter to set the tolerance of this ratio.

**Sample time (-1 for inherited) — Block sample time**

-1 (default) | positive number

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For inherited discrete-time operation, specify -1. For discrete-time operation, specify a positive integer. For continuous-time operation, specify 0.

For acceptable accuracy, set this value at least 10 times smaller than the **Time constant** parameter.

If this block is in a masked subsystem, or other variant subsystem that allows either continuous and discrete operation, promote the sample time parameter. Promoting the sample time parameter ensures correct switching between the continuous and discrete implementations of the block. For more information, see “Promote Parameter to Mask” (Simulink).

## References

- [1] IEEE. 2016. *IEEE Recommended Practice for Excitation System Models for Power System Stability Studies*. IEEE Std 421.5-2016. Piscataway, NJ: IEEE-SA, 2016.

## See Also

### Blocks

Integrator (Discrete or Continuous) | Integrator with Wrapped State (Discrete or Continuous) | Lead-Lag (Discrete or Continuous) | Low-Pass Filter (Discrete or Continuous) | Washout (Discrete or Continuous)

**Introduced in R2017b**

# Finite-Gain Op-Amp

Gain-limited operational amplifier model with optional noise

**Library:** Simscape / Electrical / Electronics and Mechatronics /  
Integrated Circuits



## Description

The Finite-Gain Op-Amp block models a gain-limited operational amplifier. If the voltages at the positive and negative ports are  $V_p$  and  $V_m$ , respectively, the output voltage is:

$$V_{out} = A(V_p - V_m) - I_{out} * R_{out}$$

where:

- $A$  is the gain.
- $R_{out}$  is the output resistance.
- $I_{out}$  is the output current.

The input current is:

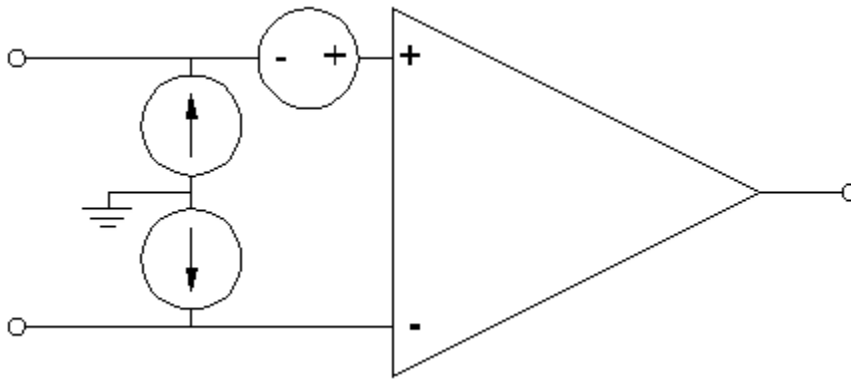
$$\frac{V_p - V_m}{R_{in}}$$

where  $R_{in}$  is the input resistance.

The output voltage is limited by the minimum and maximum output values you specify in the block dialog box.

## Thermal Noise

The Finite-Gain Op-Amp block can generate thermal noise. If you set the **Noise mode** parameter to **Enabled**, then the equivalent circuit for the block includes a noise current source attached to each of the inputs, and a noise voltage source attached to the noninverting input. These three noise sources are independent and uncorrelated.



The block generates noise voltage and current according to:

$$v_{noise} = \frac{v_{density}}{\sqrt{2h}} N(0,1)$$

$$i_{noise} = \frac{i_{density}}{\sqrt{2h}} N(0,1)$$

where:

- $v_{noise}$  is the noise voltage.
- $v_{density}$  is the single-sided, spectral amplitude density of the voltage noise.
- $i_{noise}$  is the noise current at an input.
- $i_{density}$  is the single-sided, spectral amplitude density of the current noise applied to that input.
- $h$  is the sampling time.
- $N$  is a Gaussian random number with zero mean and standard deviation of one.

The block generates Gaussian noise by using the Random Number source in the Simscape Foundation library. You can control the random number seed by setting the **Repeatability** parameter:

- **Not repeatable** — Every time you simulate your model, the block resets the random seed using the MATLAB random number generator:

```
seed = randi(2^32-1);
```

- **Repeatable** — The block automatically generates a seed value and stores it inside the block, to always start the simulation with the same random number. This auto-generated seed value is set when you add a Finite-Gain Op-Amp block from the block library to the model. When you make a new copy of the Finite-Gain Op-Amp block from an existing one in a model, a new seed value is generated. The block sets the value using the MATLAB random number generator command shown above.
- **Specify seed** — If you select this option, additional parameters let you directly specify the random number seed values for input voltage, noninverting input current, and inverting input current.

## Ports

### Conserving

#### + — Positive electrical voltage

electrical

Electrical conserving port associated with the op-amp noninverting input.

#### - — Negative electrical voltage

electrical

Electrical conserving port associated with the op-amp inverting input.

#### out — Output voltage

electrical

Electrical conserving port associated with the op-amp output. The port name is hidden on the block icon, but you can see it in simulation data logs.

## Parameters

### Main

#### Gain, A — Open-loop gain

1000 (default)

The open-loop gain of the operational amplifier.

**Input resistance,  $R_{in}$  — Resistance at block input**

1e6  $\Omega$  (default)

The resistance at the input of the operational amplifier that the block uses to calculate the input current.

**Output resistance,  $R_{out}$  — Resistance at block output**

100  $\Omega$  (default)

The resistance at the output of the operational amplifier that the block uses to calculate the drop in output voltage due to output current.

**Minimum output,  $V_{min}$  — Output voltage lower limit**

-15 V (default)

The lower limit on the operational amplifier output voltage.

**Maximum output,  $V_{max}$  — Output voltage upper limit**

15 V (default)

The upper limit on the operational amplifier output voltage.

## Noise

**Noise mode — Select whether op-amp generates thermal noise**

Disabled (default) | Enabled

Select whether to model thermal noise effects:

- **Disabled** — Op-amp does not generate thermal noise.
- **Enabled** — Op-amp generates thermal noise voltage and current, and the associated parameters become visible in the **Noise** section.

**Input noise voltage density — Density of voltage noise**

30e-9 V/Hz<sup>0.5</sup> (default)

Single-sided, spectral amplitude density of the voltage noise applied to the noninverting input.



**Dependencies**

Enabled when the **Noise mode** parameter is set to Enabled.

**Noise current parameterization – Select whether noise current density is different or the same for both inputs**

Apply same density function to both inputs (default) | Apply different density function to each input

Select whether current density values applied to block inputs are different or the same.

**Dependencies**

Enabled when the **Noise mode** parameter is set to Enabled.

**Input noise current density – Current noise at both inputs**

0.5e-12 A/Hz<sup>0.5</sup> (default)

Single-sided, spectral amplitude density of the current noise applied to both inputs. Note that even though the density function is the same for both inputs, the actual noise current also depends on the random number seed. If the seeds used for the random number generation are different, then the actual noise currents at the inputs are also different.

**Dependencies**

Enabled when the **Noise current parameterization** parameter is set to Apply same density function to both inputs.

**Noninverting input noise current density – Current noise at noninverting input**

0.5e-12 A/Hz<sup>0.5</sup> (default)

Single-sided, spectral amplitude density of the current noise applied to noninverting input.

**Dependencies**

Enabled when the **Noise current parameterization** parameter is set to Apply different density function to each input.

**Inverting input noise current density – Current noise at inverting input**

0.5e-12 A/Hz<sup>0.5</sup> (default)

Single-sided, spectral amplitude density of the current noise applied to inverting input.

### **Dependencies**

Enabled when the **Noise current parameterization** parameter is set to Apply different density function to each input.

### **Sample time — Rate at which the noise source is sampled**

1e-3 s (default)

Defines the rate at which the noise source is sampled. Choose it to reflect the frequencies of interest in your model. Making the sample time too small will unnecessarily slow down your simulation.

### **Dependencies**

Enabled when the **Noise mode** parameter is set to Enabled.

### **Repeatability — Select the noise control option**

Not repeatable (default) | Repeatable | Specify seed

Select the noise control option:

- **Not repeatable** — The random sequence used for noise generation is not repeatable.
- **Repeatable** — The random sequence used for noise generation is repeatable, with a system-generated seed.
- **Specify seed** — The random sequence used for noise generation is repeatable, and you control the seed by using the seed parameters. You specify seed values separately for input noise voltage, noninverting input noise current, and inverting input noise current.

### **Dependencies**

Enabled when the **Noise mode** parameter is set to Enabled.

### **Input noise voltage auto-generated seed used for repeatable option — Auto-generated random number seed for voltage noise**

random real number

Random number seed stored inside the block to make the random sequence repeatable. The parameter value is automatically generated using the MATLAB random number generator command. You can modify this parameter value, but it gets overwritten by a new random value if you copy the block to another block in the model. Therefore, if you

want to control the seed of the random sequence, use the `Specify seed` option for the **Repeatability** parameter and specify the desired seed value using the **Input noise voltage seed** parameter.

#### Dependencies

Enabled when the **Repeatability** parameter is set to Repeatable.

#### **Noninverting input noise current auto-generated seed used for repeatable option — Auto-generated random number seed for current noise at noninverting input**

random real number

Random number seed stored inside the block to make the random sequence repeatable. The parameter value is automatically generated using the MATLAB random number generator command. You can modify this parameter value, but it gets overwritten by a new random value if you copy the block to another block in the model. Therefore, if you want to control the seed of the random sequence, use the `Specify seed` option for the **Repeatability** parameter and specify the desired seed value using the **Noninverting input noise current seed** parameter.

#### Dependencies

Enabled when the **Repeatability** parameter is set to Repeatable.

#### **Inverting input noise current auto-generated seed used for repeatable option — Auto-generated random number seed for current noise at inverting input**

random real number

Random number seed stored inside the block to make the random sequence repeatable. The parameter value is automatically generated using the MATLAB random number generator command. You can modify this parameter value, but it gets overwritten by a new random value if you copy the block to another block in the model. Therefore, if you want to control the seed of the random sequence, use the `Specify seed` option for the **Repeatability** parameter and specify the desired seed value using the **Inverting input noise current seed** parameter.

#### Dependencies

Enabled when the **Repeatability** parameter is set to Repeatable.

#### **Input noise voltage seed — Random number seed for voltage noise**

0 (default)

Seed used by the noise random number generator.

**Dependencies**

Enabled when the **Repeatability** parameter is set to Specify seed.

**Noninverting input noise current seed — Random number seed for current noise at noninverting input**

0 (default)

Seed used by the noise random number generator.

**Dependencies**

Enabled when the **Repeatability** parameter is set to Specify seed.

**Inverting input noise current seed — Random number seed for current noise at inverting input**

0 (default)

Seed used by the noise random number generator.

**Dependencies**

Enabled when the **Repeatability** parameter is set to Specify seed.

## See Also

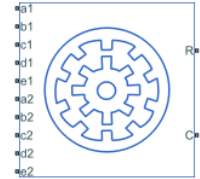
[Band-Limited Op-Amp](#) | [Fully Differential Op-Amp](#) | [Op-Amp](#)

**Introduced in R2008b**

# Five-Phase Switched Reluctance Machine

Five-phase switched reluctance machine (SRM)

**Library:** Simscape / Electrical / Power Systems / Machines / Reluctance



## Description

The Five-Phase Switched Reluctance Machine block represents a five-phase switched reluctance machine (SRM). The diagram shows the motor construction.



## Equations

The rotor stroke angle for a five-phase machine is

$$\theta_{st} = \frac{2\pi}{5N_r}$$

where:

- $\theta_{st}$  is the stroke angle.

- $N_r$  is the number of rotor poles.

The torque production capability,  $\beta$ , of one rotor pole is

$$\beta = \frac{2\pi}{N_r}.$$

The mathematical model for a switched reluctance machine (SRM) is highly nonlinear due to influence of the magnetic saturation on the flux linkage-to-angle curve,  $\lambda(\theta_{ph})$ . The phase voltage equation for an SRM is

$$v_{ph} = R_s i_{ph} + \frac{d\lambda_{ph}(i_{ph}, \theta_{ph})}{dt}$$

where:

- $v_{ph}$  is the voltage per phase.
- $R_s$  is the stator resistance per phase.
- $i_{ph}$  is the current per phase.
- $\lambda_{ph}$  is the flux linkage per phase.
- $\theta_{ph}$  is the angle per phase.

Rewriting the phase voltage equation in terms of partial derivatives yields this equation:

$$v_{ph} = R_s i_{ph} + \frac{\partial \lambda_{ph}}{\partial i_{ph}} \frac{di_{ph}}{dt} + \frac{\partial \lambda_{ph}}{\partial \theta_{ph}} \frac{d\theta_{ph}}{dt}.$$

Transient inductance is defined as

$$L_t(i_{ph}, \theta_{ph}) = \frac{\partial \lambda_{ph}(i_{ph}, \theta_{ph})}{\partial i_{ph}},$$

or more simply as

$$\frac{\partial \lambda_{ph}}{\partial i_{ph}}.$$

Back electromotive force is defined as

$$E_{ph} = \frac{\partial \lambda_{ph}}{\partial \theta_{ph}} \omega_r.$$

Substituting these terms into the rewritten voltage equation yields this voltage equation:

$$v_{ph} = R_s i_{ph} + L_t(i_{ph}, \theta_{ph}) \frac{di_{ph}}{dt} + E_{ph}.$$

Applying the co-energy formula to equations for torque,

$$T_{ph} = \frac{\partial W(\theta_{ph})}{\partial \theta_r},$$

and energy,

$$W(i_{ph}, \theta_{ph}) = \int_0^{i_{ph}} \lambda_{ph}(i_{ph}, \theta_{ph}) di_{ph}.$$

yields an integral equation that defines the instantaneous torque per phase, that is,

$$T_{ph}(i_{ph}, \theta_{ph}) = \int_0^{i_{ph}} \frac{\partial \lambda_{ph}(i_{ph}, \theta_{ph})}{\partial \theta_{ph}} di_{ph}.$$

Integrating over the phases gives this equation, which defines the total instantaneous torque for a five-phase SRM

$$T = \sum_{j=1}^5 T_{ph}(j).$$

The equation for motion is

$$J \frac{d\omega}{dt} = T - T_L - B_m \omega$$

where:

- $J$  is the rotor inertia.
- $\omega$  is the mechanical rotational speed.
- $T$  is the rotor torque. For the Switched Reluctance Machine block, torque flows from the machine case (block conserving port **C**) to the machine rotor (block conserving port **R**).
- $T_L$  is the load torque.
- $J$  is the rotor inertia.
- $B_m$  is the rotor damping.

For high-fidelity modeling and control development, use empirical data and finite element calculation to determine the flux linkage curve in terms of current and angle, that is,

$$\lambda_{ph}(i_{ph}, \theta_{ph}).$$

For low-fidelity modeling, you can also approximate the curve using analytical techniques. One such technique [2] uses this exponential function:

$$\lambda_{ph}(i_{ph}, \theta_{ph}) = \lambda_{sat} \left( 1 - e^{-i_{ph} f(\theta_{ph})} \right),$$

where:

- $\lambda_{sat}$  is the saturated flux linkage.
- $f(\theta_r)$  is obtained by Fourier expansion.

For the Fourier expansion, use the first two even terms of this equation:

$$f(\theta_{ph}) = a + b \cos(N_r \theta_{ph})$$

where  $a > b$ ,

$$a = \frac{L_{\min} + L_{\max}}{2\lambda_{sat}},$$

and



$$b = \frac{L_{max} - L_{min}}{2\lambda_{sat}}.$$

## Assumptions

A zero rotor angle corresponds to a rotor pole that is aligned perfectly with the  $a$ -phase, that is, peak flux.

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Ports

### Conserving

#### **R — Machine rotor**

mechanical rotational

Mechanical rotational conserving port associated with the machine rotor.

Data Types: double

#### **C — Machine case**

mechanical rotational

Mechanical rotational conserving port associated with the machine case.

Data Types: double

#### **a1 — a-phase positive supply**

electrical

Electrical positive supply for phase-*a*.

Data Types: double

**b1 — b-phase positive supply**

electrical

Electrical positive supply for phase-*b*.

Data Types: double

**c1 — c-phase positive supply**

electrical

Electrical positive supply for phase-*c*.

Data Types: double

**d1 — d-phase positive supply**

electrical

Electrical positive supply for phase-*d*.

Data Types: double

**e1 — e-phase positive supply**

electrical

Electrical positive supply for phase-*e*.

Data Types: double

**a2 — a-phase negative supply**

electrical

Electrical negative supply for phase-*a*.

Data Types: double

**b2 — b-phase negative supply**

electrical

Electrical negative supply for phase-*b*.

Data Types: double

**c2 — c-phase negative supply**

electrical

Electrical negative supply for phase-c.

Data Types: double

**d2 — d-phase negative supply**

electrical

Electrical negative supply for phase-d.

Data Types: double

**e2 — e-phase negative supply**

electrical

Electrical negative supply for phase-e.

Data Types: double

## Parameters

### Main

**Number of rotor poles — Rotor pole**

8 (default) | positive integer

Number of pole pairs on the rotor.

**Stator resistance per phase — Resistance**

3 0hm (default) | positive scalar

Per-phase resistance of each of the stator windings.

**Stator parameterization — Parameterization method**

Specify saturated flux linkage (default) | Specify flux characteristic

Method for parameterizing the stator.

**Dependencies**

Selecting Specify saturated flux linkage enables these parameters:

- **Saturated flux linkage**
- **Aligned inductance**
- **Unaligned inductance**

Selecting `Specify flux characteristic` enables these parameters:

- **Current vector,  $i$**
- **Angle vector,  $\theta$**
- **Flux linkage matrix,  $\Phi(i,\theta)$**

### **Saturated flux linkage — Flux linkage**

0.43 Wb (default) | positive scalar

Saturated flux linkage per phase.

#### **Dependencies**

To enable this parameter, set **Stator parameterization** to `Specify saturated flux linkage`.

### **Aligned inductance — Inductance**

0.0046 H (default) | positive scalar

The value of this parameter must be greater than the value of the **Unaligned inductance** parameter.

#### **Dependencies**

To enable this parameter, set **Stator parameterization** to `Specify saturated flux linkage`.

### **Unaligned inductance — Inductance**

6.7e-4 H (default) | positive scalar

The value of this parameter must be less than the value of the **Aligned inductance** parameter.

#### **Dependencies**

To enable this parameter, set **Stator parameterization** to `Specify saturated flux linkage`.

**Current vector,  $\mathbf{i}$  – Current**

[0, 50, 100] A (default) | positive vector

Current vector used to identify the flux linkage curve family.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify flux characteristic.

**Angle vector,  $\theta$  – Angle**

[0, 22.5, 45] deg (default) | positive vector

Angle vector used to identify the flux linkage curve family.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify flux characteristic.

**Flux linkage matrix,  $\Phi(\mathbf{i}, \theta)$  – Flux**

[0, 0, 0; .37, .06, .37; .43, .1, .43] Wb (default) | scalar

Flux linkage matrix that defines the flux linkage curve family.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify flux characteristic.

**Mechanical****Rotor inertia – Inertia**

0.01 kg\*m<sup>2</sup> (default) | positive scalar

Inertia of the rotor attached to mechanical translational port **R**.

**Rotor Damping – Damping**

0 N\*m/(rad/s) (default) | scalar

Rotary damping.

## References

- [1] Boldea, I. and S. A. Nasar. *Electric Drives*. 2nd Ed. New York: CRC Press, 2005.
- [2] Ilić-Spong, M., R. Marino, S. Peresada, and D. Taylor. "Feedback linearizing control of switched reluctance motors." *IEEE Transactions on Automatic Control*. Vol. 32, Number 5, 1987, pp. 371-379.

## See Also

Four-Phase Switched Reluctance Machine | Switched Reluctance Machine | Synchronous Reluctance Machine

**Introduced in R2018b**

# Floating Neutral

Floating neutral point for phases of three-phase system



## Library

Simscape / Electrical / Power Systems / Connections

## Description

The Floating Neutral block connects the individual phases of a three-phase system to form a floating neutral point.

---

**Note** If you want to create a neutral point that you can connect to other blocks, use the Neutral Port block. If you want to create a neutral point that is connected to ground, use the Grounded Neutral block.

---

## Parameters

### Parasitic ground conductance

Parasitic conductance to ground. A nonzero value is required for the simulation of some circuit topologies.

The default value is  $1e-12$  1/Ohm.

## Ports

The block has the following ports:

~

Expandable three-phase port

## **See Also**

### **Simscape Blocks**

Grounded Neutral | Neutral Port | Open Circuit

### **Topics**

“Asynchronous Machine Direct Torque Control”

“Asynchronous Machine Scalar Control”

“Electric Engine Dyno”

**Introduced in R2013b**



# Foster Thermal Model

Heat transfer through a semiconductor module

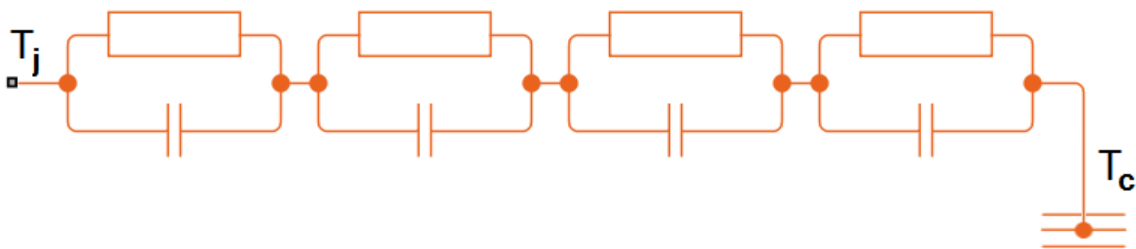


## Library

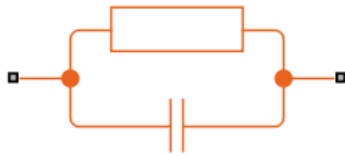
Simscape / Electrical / Power Systems / Semiconductors / Fundamental Components / Thermal

## Description

The Foster Thermal Model block represents heat transfer through a semiconductor module. The figure shows an equivalent circuit for a fourth-order Foster Thermal Model block.  $T_j$  is the junction temperature and  $T_c$  is the base plate temperature.



A Foster thermal model contains one or more instances of Foster thermal model elements. The figure shows an equivalent circuit for a Foster thermal model element.



The number of thermal elements is equal to the order of representation. For a first order model, use scalar block parameters. For an  $n$ th order model, use row vectors of length  $n$ . Other terms that describe a Foster thermal model are:

- Partial fraction circuit
- Pi model

The defining equations for a first-order Foster thermal model element are:

$$C_{thermal} = \frac{\tau}{R_{thermal}}$$

and

$$Q_{AB} = \frac{T_{AB}}{R_{thermal}} + C_{thermal} \frac{dT_{AB}}{dt},$$

where:

- $C_{thermal}$  is the thermal capacity.
- $\tau$  is the thermal time constant.
- $R_{thermal}$  is the thermal resistance.
- $Q_{AB}$  is the heat flow through the material.
- $T_{AB}$  is the temperature difference between the material layers.

## Parameters

### Thermal resistance data

Thermal resistance values,  $R_{thermal}$ , of the semiconductor module, specified as a vector. The default value is [ 0.0016 0.0043 0.0013 0.0014 ] K/W.

### Thermal time constant data

Thermal time constant values,  $\tau$ , of the semiconductor module, specified as a vector.  
The default value is [ 0.0068 0.064 0.32 2 ] s.

## Ports

The block has the following ports:

A

Thermal conserving port associated with the semiconductor junction.

B

Thermal conserving port associated with the base plate junction.

## References

[1] Schütze, T. *AN2008-03: Thermal equivalent circuit models*. Application Note. V1.0.  
Germany: Infineon Technologies AG, 2008.

## See Also

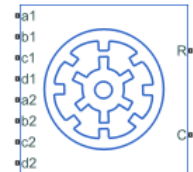
“Quantifying IGBT Thermal Losses” | Cauer Thermal Model Element | Thermal Resistor

**Introduced in R2016a**

## Four-Phase Switched Reluctance Machine

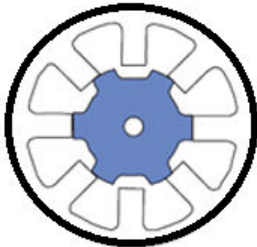
Four-phase switched reluctance machine (SRM)

**Library:** Simscape / Electrical / Power Systems / Machines / Reluctance



### Description

The Four-Phase Switched Reluctance Machine block represents a four-phase switched reluctance machine (SRM). The diagram shows the motor construction.



### Equations

The rotor stroke angle for a four-phase machine is

$$\theta_{st} = \frac{2\pi}{4N_r},$$

where:

- $\theta_{st}$  is the stoke angle.

- $N_r$  is the number of rotor poles.

The torque production capability,  $\beta$ , of one rotor pole is

$$\beta = \frac{2\pi}{N_r}.$$

The mathematical model for a switched reluctance machine (SRM) is highly nonlinear due to influence of the magnetic saturation on the flux linkage-to-angle curve,  $\lambda(\theta_{ph})$ . The phase voltage equation for an SRM is

$$v_{ph} = R_s i_{ph} + \frac{d\lambda_{ph}(i_{ph}, \theta_{ph})}{dt}$$

where:

- $v_{ph}$  is the voltage per phase.
- $R_s$  is the stator resistance per phase.
- $i_{ph}$  is the current per phase.
- $\lambda_{ph}$  is the flux linkage per phase.
- $\theta_{ph}$  is the angle per phase.

Rewriting the phase voltage equation in terms of partial derivatives yields this equation:

$$v_{ph} = R_s i_{ph} + \frac{\partial \lambda_{ph}}{\partial i_{ph}} \frac{di_{ph}}{dt} + \frac{\partial \lambda_{ph}}{\partial \theta_{ph}} \frac{d\theta_{ph}}{dt}.$$

Transient inductance is defined as

$$L_t(i_{ph}, \theta_{ph}) = \frac{\partial \lambda_{ph}(i_{ph}, \theta_{ph})}{\partial i_{ph}},$$

or more simply as

$$\frac{\partial \lambda_{ph}}{\partial i_{ph}}.$$

Back electromotive force is defined as

$$E_{ph} = \frac{\partial \lambda_{ph}}{\partial \theta_{ph}} \omega_r.$$

Substituting these terms into the rewritten voltage equation yields this voltage equation:

$$v_{ph} = R_s i_{ph} + L_t(i_{ph}, \theta_{ph}) \frac{di_{ph}}{dt} + E_{ph}.$$

Applying the co-energy formula to equations for torque,

$$T_{ph} = \frac{\partial W(\theta_{ph})}{\partial \theta_r},$$

and energy,

$$W(i_{ph}, \theta_{ph}) = \int_0^{i_{ph}} \lambda_{ph}(i_{ph}, \theta_{ph}) di_{ph}.$$

yields an integral equation that defines the instantaneous torque per phase, that is,

$$T_{ph}(i_{ph}, \theta_{ph}) = \int_0^{i_{ph}} \frac{\partial \lambda_{ph}(i_{ph}, \theta_{ph})}{\partial \theta_{ph}} di_{ph}.$$

Integrating over the phases gives this equation, which defines the total instantaneous torque for a four-phase SRM:

$$T = \sum_{j=1}^4 T_{ph}(j)$$

The equation for motion is

$$J \frac{d\omega}{dt} = T - T_L - B_m \omega$$

where:

- $J$  is the rotor inertia.
- $\omega$  is the mechanical rotational speed.
- $T$  is the rotor torque. For the Switched Reluctance Machine block, torque flows from the machine case (block conserving port **C**) to the machine rotor (block conserving port **R**).
- $T_L$  is the load torque.
- $J$  is the rotor inertia.
- $B_m$  is the rotor damping.

For high-fidelity modeling and control development, use empirical data and finite element calculation to determine the flux linkage curve in terms of current and angle, that is,

$$\lambda_{ph}(i_{ph}, \theta_{ph}).$$

For low-fidelity modeling, you can also approximate the curve using analytical techniques. One such technique<sup>[2]</sup>, uses this exponential function:

$$\lambda_{ph}(i_{ph}, \theta_{ph}) = \lambda_{sat} \left( 1 - e^{-i_{ph} f(\theta_{ph})} \right),$$

where:

- $\lambda_{sat}$  is the saturated flux linkage.
- $f(\theta_r)$  is obtained by Fourier expansion.

For the Fourier expansion, use the first two even terms of this equation:

$$f(\theta_{ph}) = a + b \cos(N_r \theta_{ph})$$

where  $a > b$ ,

$$a = \frac{L_{\min} + L_{\max}}{2\lambda_{sat}},$$

and

$$b = \frac{L_{max} - L_{min}}{2\lambda_{sat}}.$$

## Assumptions

A zero rotor angle corresponds to a rotor pole that is aligned perfectly with the  $a$ -phase, that is, peak flux.

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Ports

### Conserving

#### **R — Machine rotor**

mechanical rotational

Mechanical rotational conserving port associated with the machine rotor.

Data Types: double

#### **C — Machine case**

mechanical rotational

Mechanical rotational conserving port associated with the machine case.

Data Types: double

#### **a1 — a-phase positive supply**

electrical



Electrical positive supply for phase-*a*.

Data Types: double

**b1 — *b*-phase positive supply**

electrical

Electrical positive supply for phase-*b*.

Data Types: double

**c1 — *c*-phase positive supply**

electrical

Electrical positive supply for phase-*c*.

Data Types: double

**d1 — *d*-phase positive supply**

electrical

Electrical positive supply for phase-*d*.

Data Types: double

**a2 — *a*-phase negative supply**

electrical

Electrical negative supply for phase-*a*.

Data Types: double

**b2 — *b*-phase negative supply**

electrical

Electrical negative supply for phase-*b*.

Data Types: double

**c2 — *c*-phase negative supply**

electrical

Electrical negative supply for phase-*c*.

Data Types: double

## **d2 — *d*-phase negative supply**

electrical

Electrical negative supply for phase-*d*.

Data Types: double

## **Parameters**

### **Main**

#### **Number of rotor poles — Rotor pole**

6 (default) | positive integer

Number of pole pairs on the rotor.

#### **Stator resistance per phase — Resistance**

3 0hm (default) | positive scalar

Per-phase resistance of each of the stator windings.

#### **Stator parameterization — Parameterization method**

Specify saturated flux linkage (default) | Specify flux characteristic

Method for parameterizing the stator.

### **Dependencies**

Selecting Specify saturated flux linkage enables these parameters:

- **Saturated flux linkage**
- **Aligned inductance**
- **Unaligned inductance**

Selecting Specify flux characteristic enables these parameters:

- **Current vector, *i***
- **Angle vector, *theta***
- **Flux linkage matrix,  $\Phi(i,theta)$**

**Saturated flux linkage — Flux linkage**

0.43 Wb (default) | positive scalar

Saturated flux linkage per phase.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify saturated flux linkage.

**Aligned inductance — Inductance**

0.0046 H (default) | positive scalar

The value of this parameter must be greater than the value of the **Unaligned inductance** parameter.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify saturated flux linkage.

**Unaligned inductance — Inductance**

6.7e-4 H (default) | positive scalar

The value of this parameter must be less than the value of the **Aligned inductance** parameter.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify saturated flux linkage.

**Current vector,  $i$  — Current**

[0, 50, 100] A (default) | positive vector

Current vector used to identify the flux linkage curve family.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify flux characteristic.

**Angle vector,  $\theta$  — Angle**

[0, 30, 60] deg (default) | positive vector

Angle vector used to identify the flux linkage curve family.

### Dependencies

To enable this parameter, set **Stator parameterization** to Specify flux characteristic.

### Flux linkage matrix, $\Phi(i, \theta)$ — Flux

[0, 0, 0; .37, .06, .37; .43, .1, .43] Wb (default) | scalar

Flux linkage matrix that defines the flux linkage curve family.

### Dependencies

To enable this parameter, set **Stator parameterization** to Specify flux characteristic.

## Mechanical

### Rotor inertia — Inertia

0.01 kg\*m<sup>2</sup> (default) | positive scalar

Inertia of the rotor attached to mechanical translational port **R**.

### Rotor Damping — Damping

0 N\*m/(rad/s) (default) | scalar

Rotary damping.

## References

[1] Boldea, I. and S. A. Nasar. *Electric Drives*. 2nd Ed. New York: CRC Press, 2005.

[2] Ilić-Spong, M., R. Marino, S. Peresada, and D. Taylor. "Feedback linearizing control of switched reluctance motors." *IEEE Transactions on Automatic Control*. Vol. 32, Number 5, 1987, pp. 371-379.

## See Also

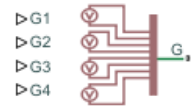
Five-Phase Switched Reluctance Machine | Switched Reluctance Machine | Synchronous Reluctance Machine

**Introduced in R2018b**

## Four-Pulse Gate Multiplexer

Combine four gate-pulse voltage signals into a single vector

**Library:** Simscape / Electrical / Power Systems /  
Semiconductors



### Description

The Four-Pulse Gate Multiplexer block multiplexes four separate voltage signals into a single vector. The vectorized signal can control the gates of four switching devices in a converter, such as a Four-Quadrant Chopper block.

### Model

Each of two model variants for the Four-Pulse Gate Multiplexer block corresponds to a **Block choice** option. To access the block choices, in the model window, right-click the block, and then use either of these methods:

- From the context menu, select **Simscape > Block choices**.
- On the Simulink Editor menu bar, select **View > Property Inspector**. In the **Property Inspector** window, click the value of the **Block choice**.

The model variants are:

- PS ports — Four-pulse gate multiplexer with physical signal ports. Select this default option to control switching device gates in a converter block using Simulink gate-control voltage signals. To multiplex and connect Simulink signals to the gate-control inport of a converter block:
  - 1 Convert each voltage signal using a Simulink-PS Converter block.
  - 2 Multiplex the converted gate signals into a single vector using the multiplexor block.
  - 3 Connect the vector signal to the **G** port of the converter.

- Electrical ports — Four-pulse gate multiplexer with electrical conserving ports. To control switching device gates in a converter block using Simscape Electrical Electronics and Mechatronics blocks, select this option. The electrical ports include pairs of electrical connections. Each pair corresponds to the gate and cathode of a switching device in the connected converter block.

## Ports

### Input

#### **G1 — Gate-control voltage signal 1**

physical signal

Physical signal port associated with the gate terminal of the first switching device in a connected converter block.

#### **Dependencies**

This port only appears for the PS ports block choice.

Data Types: double

#### **G2 — Gate-control voltage signal 2**

physical signal

Physical signal port associated with the gate terminal of the second switching device in a connected converter block.

#### **Dependencies**

This port only appears for the PS ports block choice.

Data Types: double

#### **G3 — Gate-control voltage signal 3**

physical signal

Physical signal port associated with the gate terminal of the third switching device in a connected converter block.

#### **Dependencies**

This port only appears for the PS ports block choice.

Data Types: double

**G4 — Gate-control voltage signal 4**

physical signal

Physical signal port associated with the gate terminal of the fourth switching device in a connected converter block.

**Dependencies**

This port only appears for the PS ports block choice.

Data Types: double

**Conserving**

**G1 — Gate-control voltage signal 1**

electrical

Electrical conserving port associated with the gate terminal of the first switching device in a connected converter block.

**Dependencies**

This port only appears for the Electrical ports block choice.

**a — A-phase AC reference point**

electrical

Electrical conserving port associated with the A-phase for the high-side switching device.

**Dependencies**

This port only appears for the Electrical ports block choice.

**G2 — Gate-control voltage signal 2**

electrical

Electrical conserving port associated with the gate terminal of the second switching device in a connected converter block.

**Dependencies**

This port only appears for the Electrical ports block choice.



**b — B-phase AC reference point**

electrical

Electrical conserving port associated with the *B*-phase for the high-side switching device.

**Dependencies**

This port only appears for the `Electrical ports` block choice.

**G3 — Gate-control voltage signal 3**

electrical

Electrical conserving port associated with the gate terminal of the third switching device in a connected converter block.

**Dependencies**

This port only appears for the `Electrical ports` block choice.

**G4 — Gate-control voltage signal 4**

electrical

Electrical conserving port associated with the gate terminal of the fourth switching device in a connected converter block.

**Dependencies**

This port only appears for the `Electrical ports` block choice.

**L — DC reference point**

electrical

Electrical conserving port associated with the DC negative connection for the low-side switching device.

**Dependencies**

This port only appears for the `Electrical ports` block choice.

**See Also**

Four-Quadrant Chopper | Six-Pulse Gate Multiplexer | Twelve-Pulse Gate Multiplexer | Two-Pulse Gate Multiplexer

## **Topics**

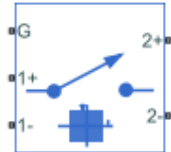
Switch Between Physical Signal and Electrical Ports

**Introduced in R2018a**

# Four-Quadrant Chopper

Four-quadrant controlled DC-DC chopper

**Library:** Simscape / Electrical / Power Systems / Semiconductors

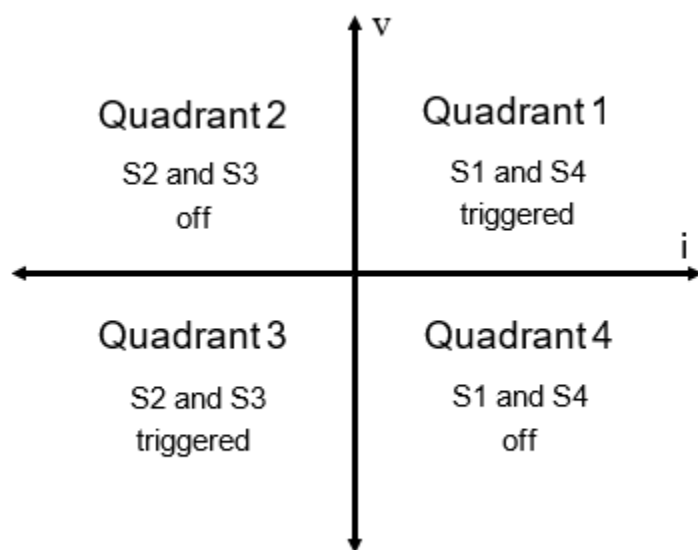
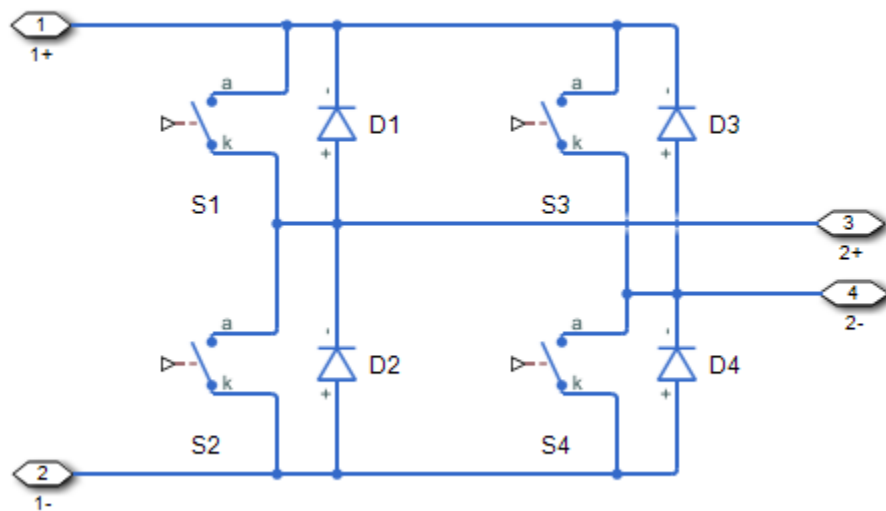


## Description

The Four-Quadrant Chopper block represents a four-quadrant controlled chopper for converting a fixed DC input to a variable DC output. The block contains two bridge arms. Each bridge arm each has two switching devices. Options for the type of switching devices are:

- GTO — Gate turn-off thyristor. For information on the I-V characteristic of the device, see GTO.
- Ideal semiconductor switch — For information on the I-V characteristic of the device, see Ideal Semiconductor Switch.
- IGBT — Insulated-gate bipolar transistor. For information on the I-V characteristic of the device, see IGBT.
- MOSFET — N-channel metal-oxide-semiconductor field-effect transistor. For information on the I-V characteristic of the device, see MOSFET.
- Thyristor — For information on the I-V characteristic of the device, see Thyristor.

The figures show the equivalent circuit and the operation for the block.



## Protection

The block contains an integral protection diode for each switching device. The integral diode protects the semiconductor device by providing a conduction path for reverse current. An inductive load can produce a high reverse-voltage spike when the semiconductor device suddenly switches off the voltage supply to the load.

To configure the internal protection diode block, use the **Protection Diode** parameters. This table shows how to set the **Model dynamics** parameter based on your goals.

Goals	Value to Select	Integral Protection Diode
Prioritize simulation speed.	Protection diode with no dynamics	The Diode block
Prioritize model fidelity by precisely specifying reverse-mode charge dynamics.	Protection diode with charge dynamics	The Commutation Diode block

You can also include a snubber circuit for each switching device. Snubber circuits contain a series-connected resistor and capacitor. They protect switching devices against high voltages that inductive loads produce when the device turns off the voltage supply to the load. Snubber circuits also prevent excessive rates of current change when a switching device turns on.

To include and configure a snubber circuit for each switching device, use the **Snubbers** parameters.

## Gate Control

To connect Simulink gate-control voltage signals to the gate ports of the internal switching devices:

- 1 Convert each voltage signal using a Simulink-PS Converter block.
- 2 Multiplex the converted gate signals into a single vector using a Four-Pulse Gate Multiplexer block.
- 3 Connect the vector signal to the **G** port.

## Ports

### Conserving

#### **G — Switching device gate control**

electrical | vector

Electrical conserving port associated with the gate terminals of the switching devices.

Data Types: double

#### **1+ — Positive DC voltage 1**

electrical | scalar

Electrical conserving port associated with the positive terminal of the first DC voltage.

Data Types: double

#### **1- — Negative DC voltage 1**

electrical | scalar

Electrical conserving port associated with the negative terminal of the first DC voltage.

Data Types: double

#### **2+ — Positive DC voltage 2**

electrical | scalar

Electrical conserving port associated with the positive terminal of the second DC voltage.

Data Types: double

#### **2- — Negative DC voltage 2**

electrical | scalar

Electrical conserving port associated with the negative terminal of the second DC voltage.

Data Types: double

## Parameters

### Switching Devices

This table shows how the visibility of **Switching Devices** parameters depends on the **Switching device** that you select. To learn how to read the table, see “Parameter Dependencies” on page A-2.

#### Switching Devices Parameter Dependencies

Parameters and Options				
Switching device				
<i>Ideal Semiconductor Switch</i>	<i>GTO</i>	<i>IGBT</i>	<i>MOSFET</i>	<i>Thyristor</i>
On-state resistance	Forward voltage	Forward voltage	Drain-source on resistance	Forward voltage
Off-state conductance	On-state resistance	On-state resistance	Off-state conductance	On-state resistance
Threshold voltage	Off-state conductance	Off-state conductance	Threshold voltage	Off-state conductance
	Gate trigger voltage, Vgt	Threshold voltage		Gate trigger voltage, Vgt
	Gate turn-off voltage, Vgt_off			Gate turn-off voltage, Vgt_off
	Holding current			Holding current

#### Switching device – Switch type

*Ideal Semiconductor Switch* (default) | *GTO* | *IGBT* | *MOSFET* | *Thyristor*

Switching device type for the converter.

#### Dependencies

See the Switching Devices Parameter Dependencies table.

#### Forward voltage – Voltage

0.8 Ohm (default) | scalar

For the different switching device types, the **Forward voltage** is taken as:

- GTO — Minimum voltage required across the anode and cathode block ports for the gradient of the device I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**
- IGBT — Minimum voltage required across the collector and emitter block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**
- Thyristor — Minimum voltage required for the device to turn on

### Dependencies

See the Switching Devices Parameter Dependencies table.

### On-state resistance — Resistance

0.001 Ohm (default) | scalar

For the different switching device types, the **On-state resistance** is taken as:

- GTO — Rate of change of voltage versus current above the forward voltage
- Ideal semiconductor switch — Anode-cathode resistance when the device is on
- IGBT — Collector-emitter resistance when the device is on
- Thyristor — Anode-cathode resistance when the device is on

### Dependencies

See the Switching Devices Parameter Dependencies table.

### Drain-source on resistance — Resistance

0.001 Ohm (default) | scalar

Resistance between the drain and the source, which also depends on the gate-to-source voltage.

### Dependencies

See the Switching Devices Parameter Dependencies table.

### Off-state conductance — Conductance

1e-5 1/Ohm (default) | scalar



Conductance when the device is off. The value must be less than  $1/R$ , where  $R$  is the value of **On-state resistance**.

For the different switching device types, the **On-state resistance** is taken as:

- GTO — Anode-cathode conductance
- Ideal semiconductor switch — Anode-cathode conductance
- IGBT — Collector-emitter conductance
- MOSFET — Drain-source conductance
- Thyristor — Anode-cathode conductance

#### **Dependencies**

See the Switching Devices Parameter Dependencies table.

#### **Threshold voltage — Voltage threshold**

6 V (default) | scalar

Gate voltage threshold. The device turns on when the gate voltage is above this value. For the different switching device types, the device voltage of interest is:

- Ideal semiconductor switch — Gate-emitter voltage
- IGBT — Gate-cathode voltage
- MOSFET — Gate-source voltage

#### **Dependencies**

See the Switching Devices Parameter Dependencies table.

#### **Gate trigger voltage, $V_{gt}$ — Voltage threshold**

1 V (default) | scalar

Gate-cathode voltage threshold. The device turns on when the gate-cathode voltage is above this value.

#### **Dependencies**

See the Switching Devices Parameter Dependencies table.

#### **Gate turn-off voltage, $V_{gt\_off}$ — Voltage threshold**

-1 V (default) | scalar

Gate-cathode voltage threshold. The device turns off when the gate-cathode voltage is below this value.

### **Dependencies**

See the Switching Devices Parameter Dependencies table.

### **Holding current — Current threshold**

1 A (default) | scalar

Gate current threshold. The device stays on when the current is above this value, even when the gate-cathode voltage falls below the gate trigger voltage.

### **Dependencies**

See the Switching Devices Parameter Dependencies table.

## **Protection Diode**

The visibility of **Protection Diode** parameters depends on how you configure the protection diode **Model dynamics** and **Reverse recovery time parameterization** parameters. To learn how to read this table, see “Parameter Dependencies” on page A-2.

## Protection Diode Parameter Dependencies

Parameters and Options			
<b>Model dynamics</b>			
<i>Protection diode with no dynamics</i>	<i>Protection diode with charge dynamics</i>		
Forward voltage	Forward voltage		
On resistance	On resistance		
Off conductance	Off conductance		
	Junction capacitance		
	Peak reverse current, iRM		
	Initial forward current when measuring iRM		
	Rate of change of current when measuring iRM		
	<b>Reverse recovery time parameterization</b>		
	<i>Specify stretch factor</i>	<i>Specify reverse recovery time directly</i>	<i>Specify reverse recovery charge</i>
	Reverse recovery time stretch factor	Reverse recovery time, trr	Reverse recovery charge, Qrr

### Model dynamics — Diode model

Protection diode with no dynamics (default) | Protection diode with charge dynamics

Diode type. The options are:

- Diode with no dynamics — Select this option to prioritize simulation speed using the Diode block.
- Diode with charge dynamics — Select this option to prioritize model fidelity in terms of reverse mode charge dynamics using the Commutation Diode block.

### Dependencies

See the Protection Diode Parameter Dependencies table.

### Forward voltage — Voltage

0.8 V (default) | scalar

Minimum voltage required across the positive and negative block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**.

**On resistance — Resistance**

0.001 Ohm (default) | scalar

Rate of change of voltage versus current above the **Forward voltage**.

**Off conductance — Conductance**

1e-5 1/Ohm (default) | scalar

Conductance of the reverse-biased diode.

**Junction capacitance — Capacitance**

50 nF (default) | scalar

Diode junction capacitance.

**Dependencies**

See the Protection Diode Parameter Dependencies table.

**Peak reverse current,  $i_{RM}$  — Current**

-235 A (default) | scalar less than 0

Peak reverse current measured by an external test circuit.

**Dependencies**

See the Protection Diode Parameter Dependencies table.

**Initial forward current when measuring  $i_{RM}$  — Current**

300 A (default) | scalar greater than 0

Initial forward current when measuring peak reverse current. This value must be greater than zero.

**Dependencies**

See the Protection Diode Parameter Dependencies table.

**Rate of change of current when measuring  $i_{RM}$  — Current change rate**

-50 A/us (default) | scalar

Rate of change of current when measuring peak reverse current.

**Dependencies**

See the Protection Diode Parameter Dependencies table.

**Reverse recovery time parameterization – Recovery-time model**

Specify stretch factor (default) | Specify reverse recovery time directly  
| Specify reverse recovery charge

Model for parameterizing the recovery time. When you select **Specify stretch factor** or **Specify reverse recovery charge**, you can specify a value that the block uses to derive the reverse recovery time. For more information on these options, see “Alternatives to Specifying trr Directly” on page 1-339.

**Dependencies**

See the Protection Diode Parameter Dependencies table.

**Reverse recovery time stretch factor – Stretch factor**

3 (default) | scalar greater than 1

Value that the block uses to calculate **Reverse recovery time, trr**. Specifying the stretch factor is an easier way to parameterize the reverse recovery time than specifying the reverse recovery charge. The larger the value of the stretch factor, the longer it takes for the reverse recovery current to dissipate.

**Dependencies**

See the Protection Diode Parameter Dependencies table.

**Reverse recovery time, trr – Time**

15 us (default) | scalar

Interval between the time when the current initially goes to zero (when the diode turns off) and the time when the current falls to less than 10 percent of the peak reverse current.

The value of the **Reverse recovery time, trr** parameter must be greater than the value of the **Peak reverse current, iRM** parameter divided by the value of the **Rate of change of current when measuring iRM** parameter.

**Dependencies**

See the Protection Diode Parameter Dependencies table.

**Reverse recovery charge, Qrr — Charge**

1500 s\*uA (default) | scalar

Value that the block uses to calculate **Reverse recovery time, trr**. Use this parameter if the data sheet for your diode device specifies a value for the reverse recovery charge instead of a value for the reverse recovery time.

The reverse recovery charge is the total charge that continues to dissipate when the

diode turns off. The value must be less than  $-\frac{i_{RM}^2}{2a}$ ,

where:

- $i_{RM}$  is the value specified for **Peak reverse current, iRM**.
- $a$  is the value specified for **Rate of change of current when measuring iRM**.

**Dependencies**

See the Protection Diode Parameter Dependencies table.

**Snubbers**

The table summarizes the **Snubbers** parameter dependencies. To learn how to read the table, see “Parameter Dependencies” on page A-2.

**Snubbers Parameter Dependencies**

<b>Snubbers Parameter Dependencies</b>	
<b>Snubber</b>	
<i>None</i>	<i>RC Snubber</i>
	Snubber resistance
	Snubber capacitance

**Snubber — Snubber model**

None (default) | RC snubber

Switching device snubber.

**Dependencies**

See the Snubbers Parameter Dependencies table.

**Snubber resistance — Resistance**

0.1 Ohm (default) | scalar

Resistance of the switching device snubber.

**Dependencies**

See the Snubbers Parameter Dependencies table.

**Snubber capacitance — Capacitance**

1e-7 (default) | F | scalar

Capacitance of the switching device snubber.

**Dependencies**

See the Snubbers Parameter Dependencies table.

**References**

- [1] Trzynadlowski, A. M. *Introduction to Modern Power Electronics, 2nd Edition*. Hoboken, NJ: John Wiley & Sons Inc., 2010.

**See Also**

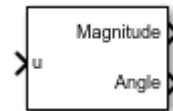
Average-Value Chopper | Four-Pulse Gate Multiplexer | One-Quadrant Chopper | Two-Quadrant Chopper

**Introduced in R2018a**

# Fourier Analysis

Discrete or continuous time Fourier analysis

**Library:** Simscape / Electrical / Power Systems / Control /  
General Control



## Description

This block performs a Fourier analysis on the input signal in either discrete or continuous time.

## Equations

A periodic function  $x(t)$  can be decomposed to an infinite sum of sine and cosine functions as

$$x(t) = \frac{a_0}{2} + \sum_{n=1}^{\infty} [a_n \cos(nt) + b_n \sin(nt)]$$

where:

- $a_0$  is the DC component.
- $a_n$  and  $b_n$  are constant Fourier coefficients.
- $n$  is the harmonic number.

The coefficients  $a_n$  and  $b_n$  are defined as

$$a_n = \frac{2}{T_0} \int_{t_0}^{t_0+T_0} x(t) \cos(n\Omega_0 t) dt, \quad n = 0, 1, \dots$$



$$b_n = \frac{2}{T_0} \int_{t_0}^{t_0+T_0} x(t) \sin(n\Omega_0 t) dt, \quad n = 1, 2, \dots$$

$$\Omega_0 = \frac{2\pi}{T_0}$$

$$T_0 = \frac{1}{f}$$

where  $f$  is the fundamental frequency.

The magnitude and angle corresponding to the harmonic number are defined as:

$$|X_n| = \sqrt{a_n^2 + b_n^2}$$

$$\theta_n = -\tan^{-1} \left[ \frac{b_n}{a_n} \right]$$

## Ports

### Input

#### **u** — Fourier analysis input

scalar | vector

Input signal to be analyzed. The input can be a single signal or multiple multiplexed signals. Input signals can be AC currents or voltages in an electrical system.

Data Types: single | double

### Output

#### **Magnitude** — Signal magnitude

scalar | vector

Signal magnitude corresponding to the harmonic number.

Data Types: `single` | `double`

**Angle — Signal angle**

`scalar` | `vector`

Signal angle corresponding to the harmonic number.

Data Types: `single` | `double`

## Parameters

**Fundamental frequency (Hz) — Signal fundamental frequency**

`60` (default) | `positive scalar` | `vector with positive values`

Fundamental frequency of the signal, in Hz. If you specify the fundamental frequency using a vector, it must match the input vector dimensions.

**Harmonic numbers — Signal harmonic numbers**

`[1 2]` (default) | `vector with elements  $\geq 0$`

Specify signal harmonic numbers. Vector elements must be greater than or equal to 0.

**Initial magnitude — Initial signal magnitude**

`1` (default) | `vector with elements  $\geq 0$`

Specify the initial magnitude of the signal. Vector elements must be greater than or equal to 0.

**Initial phase (rad) — Initial signal phase**

`0` (default) | `vector`

Initial phase angle of the signal, in rad.

**Sample time (-1 for inherited) — Block sample time**

`-1` (default) | `0` | `positive scalar`

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For inherited discrete-time operation, specify -1. For discrete-time operation, specify a positive integer. For continuous-time operation, specify 0.

If this block is in a masked subsystem, or other variant subsystem that allows you to switch between continuous operation and discrete operation, promote the sample time parameter. Promoting the sample time parameter ensures correct switching between the continuous and discrete implementations of the block. For more information, see “Promote Parameter to Mask” (Simulink).

## See Also

### Blocks

Moving Average

**Introduced in R2018b**

## Fully Differential Op-Amp

Model operational amplifier with fully differential output, that is, not referenced to ground



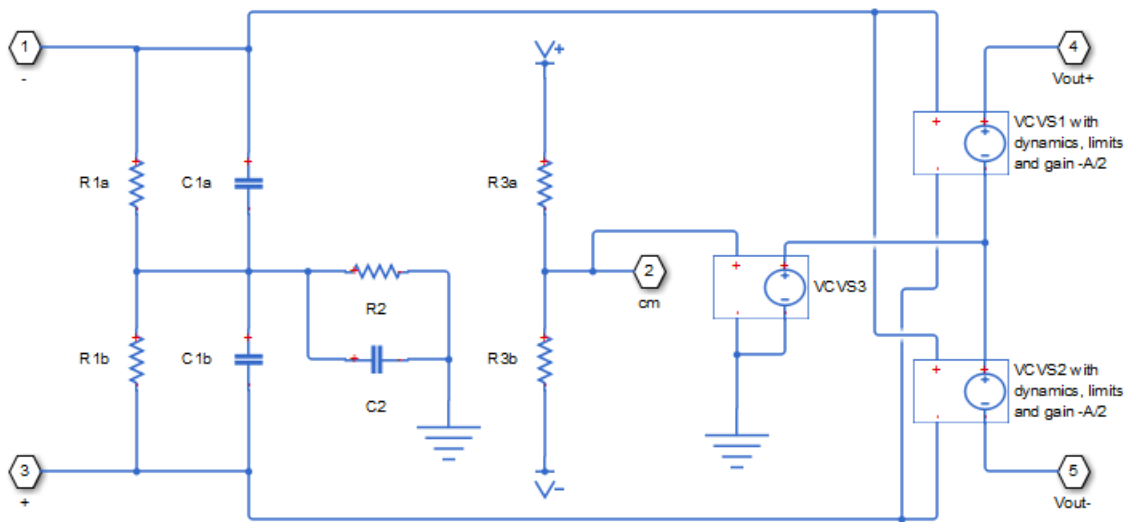
## Library

Integrated Circuits

## Description

The Fully Differential Op-Amp block models a fully differential operational amplifier. Differential signal transmission is better than single-ended transmission due to reduced susceptibility to external noise sources. Applications include data acquisition where inputs are differential, for example, sigma-delta converters.

The following diagram shows the internal representation of the amplifier.



Parameters for the circuit components are derived from the block parameters that you provide. The gain of the two voltage-controlled voltage sources (VCVS1 and VCVS2) is set to half of the differential gain value. Similarly the slew rate of each of the voltage sources is set to half of the differential maximum slew rate value. The voltages of the two output ports  $V_{out+}$  and  $V_{out-}$  are both limited to be within the minimum and maximum output voltages that you specify.

The output voltage for zero differential input voltage is controlled by the common-mode port,  $cm$ . If no current is drawn from the  $cm$  port by the external circuit, then the output voltage is set to be the average of the positive and negative supply voltages by the resistor ladder of  $R3a$  and  $R3b$ . Note that the negative supply voltage can be zero, which corresponds to operation when a split supply is not available. The values for the minimum and maximum output voltages that you provide must be consistent with the values for the supply voltages that you provide. So, for example, the maximum output high voltage will be less than the positive supply voltage, the difference corresponding to the number of p-n junction voltage drops in the circuit.

---

**Note** Physical Network block diagrams do not allow unconnected Conserving ports. If you want to leave pin  $cm$  open-circuit, connect it to an Open Circuit block from the Simscape Foundation library.

---

## Basic Assumptions and Limitations

This block provides a behavioral model of a fully differential operational amplifier. It does not represent nonlinear effects, such as variation in gain with output voltage amplitude, and the nonlinear nature of the output voltage-current relationship for large load currents.

## Parameters

- “Gain Tab” on page 1-656
- “Input Impedance Tab” on page 1-656
- “Output Limits Tab” on page 1-657
- “Output Bias Tab” on page 1-657
- “Initial Conditions Tab” on page 1-658

## Gain Tab

### Differential gain

The gain applied to a voltage difference between the + and - inputs. The default value is 1000.

### Bandwidth

The frequency at which the differential voltage gain drops by 3 dB from its dc value. The default value is 1.5 GHz.

## Input Impedance Tab

### Differential input resistance

The input resistance seen by a voltage source applied across the + and - inputs. The default value is 1.3 MOhm.

### Differential input capacitance

The input capacitance seen by a current source applied across the + and - inputs. The default value is 1.8 pF.

**Common-mode input resistance**

The input resistance seen by a voltage source applied between ground and the + input, or between ground and the - input. The default value is 1 MOhm.

**Common-mode input capacitance**

The input capacitance seen by a current source applied between ground and the + input, or between ground and the - input. The default value is 2.3 pF.

**Output Limits Tab****Output resistance**

The output resistance of either of the outputs with respect to the common-mode voltage reference. Differential output resistance is therefore twice the value of the output resistance R<sub>out</sub>. The default value is 1 Ohm.

**Minimum output voltage low**

The minimum output voltage for either of the two output pins with respect to ground. The default value is -1.4 V.

**Maximum output voltage low**

The maximum output voltage for either of the two output pins with respect to ground. The default value is 1.4 V.

**Differential maximum slew rate**

The maximum slew rate of the differential output voltage. The default value is 5000 V/ $\mu$ s.

**Output Bias Tab****Common-mode port input resistance**

The input resistance seen by a voltage source applied between ground and the common mode port. The default value is 23 kOhm.

**Negative supply voltage**

The value of the negative supply voltage connected to common-mode bias resistor R3b (see diagram). The default value is -5 V.

**Positive supply voltage**

The value of the positive supply voltage connected to common-mode bias resistor R3a (see diagram). The default value is 5 V.

## Initial Conditions Tab

### Initial differential output voltage

The initial differential voltage across the two outputs if the output current is zero. The default value is 0 V.

## Ports

The block has the following ports:

+

Noninverting input.

-

Inverting input.

cm

Common-mode port. If you want to leave this pin open-circuit, connect a voltage sensor between the cm port and a reference port.

Vout+

Noninverting output.

Vout -

Inverting output.

## See Also

[Band-Limited Op-Amp](#) | [Finite-Gain Op-Amp](#) | [Op-Amp](#)



# Fuse

Model fuse that protects against excessive current

**Library:** Simscape / Electrical / Electronics and Mechatronics /  
Passive Devices



## Description

The Fuse block breaks the circuit, to protect it against excessive current. The block provides the following modeling options:

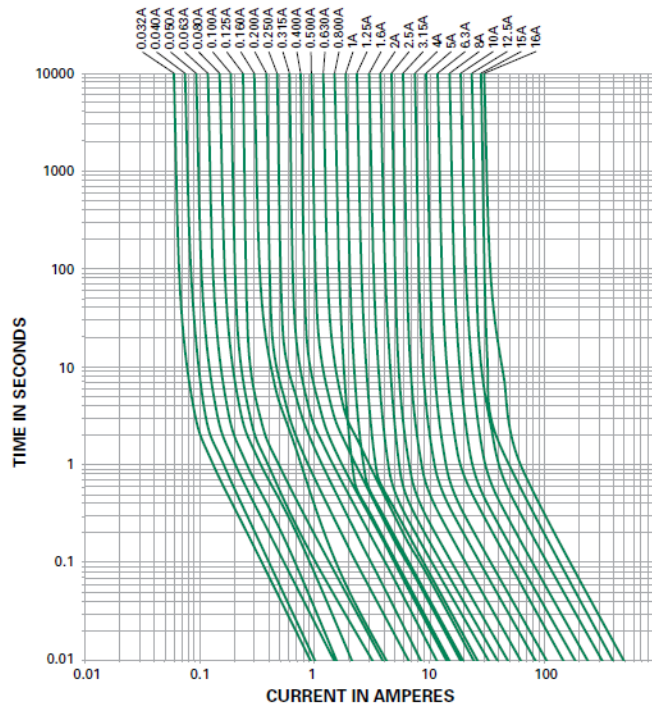
- **Opening time is independent of current** — The fuse opens when the current through the fuse exceeds the threshold current continuously for longer than the specified amount of time. This option assumes that the time to open is independent of the current. It also implicitly assumes that heat in the fuse is instantaneously dissipated once the current drops below a certain threshold.
- **Opening time depends on  $I^2 \cdot t$**  — Makes fuse opening dependent on the energy dissipated in the fuse. The nominal melting value ( $I^2 \cdot t$ ) is usually available on manufacturer datasheets. If you select this option, the fuse blows when:

$$\int_{t_{\text{over}}}^t i^2 dt \geq I^2 t$$

where  $t_{\text{over}}$  is the time when the current crosses over the threshold current,  $i$  is instantaneous current through the fuse, and  $I^2 t$  is the nominal melting value from the datasheet. Use this option for high-current range of operation.

- **Opening time is tabulated** — Makes fuse opening dependent on the average time-current tabulated data. This data indicates the time that it takes for the fuse to open given a certain current, and is available on manufacturer datasheets as time-current curves.

For example, these are the time-current curves from a Littelfuse® 218 Series fuse datasheet.



Select the appropriate curve and enter the corresponding current and time values into the **Tabulated currents** and **Tabulated opening time** parameter vectors, respectively. Use this option for small currents.

## Ports

### Conserving

+ — Positive terminal  
electrical

Electrical conserving port associated with the fuse positive terminal.

- — Negative terminal  
electrical

Electrical conserving port associated with the fuse negative terminal.

## Parameters

### Parameterization — Model dependence of opening time on current

Opening time is independent of current (default) | Opening time depends on  $I^2 \cdot t$  | Opening time is tabulated

Select whether opening time depends on current:

- **Opening time is independent of current** — The fuse opens when the current through the fuse exceeds the threshold current continuously for longer than the specified amount of time.
- **Opening time depends on  $I^2 \cdot t$**  — Makes fuse opening dependent on the energy dissipated in the fuse. Use this option for high-current range of operation.
- **Opening time is tabulated** — Makes fuse opening dependent on the average time-current tabulated data. Use this option for small currents.

### Rated current, $I_{\text{rated}}$ — Nominal operating current

1 A (default)

The fuse rating from the datasheet, or nominal operating current.

### Dependencies

Enabled when the **Parameterization** parameter is set to **Opening time is independent of current** or **Opening time depends on  $I^2 \cdot t$** .

### Ratio of minimum melting current to rated current, $I_{\text{melt}}/I_{\text{rated}}$ — Defined threshold current for melting

1 (default)

Minimum melting current defines the fuse operating threshold. If the current through the fuse does not exceed this threshold, the fuse stays in low-ohmic state. With the default parameter value, 1, the threshold current is the same as the rated current. In most cases, this parameter value should be greater than 1. You can also set it to a value less than 1, for situations where the fuse may blow at a current level below the rated current (for example, due to high temperature, or due to repetitive pulses). The parameter value must be greater than 0.

### **Dependencies**

Enabled when the **Parameterization** parameter is set to `Opening time is independent of current` or `Opening time depends on  $I^2 \cdot t$` .

### **Time to fuse — Time needed for the fuse to blow**

0 s (default)

The amount of time needed for the fuse to blow. If the current through the fuse exceeds the threshold current continuously for longer than this amount of time, the fuse enters the high-ohmic state and breaks the circuit. If the current falls below the threshold before the time elapsed, the fuse stays in the low-ohmic state. If the current exceeds the threshold again, the time counter restarts. With the default parameter value, 0, the fuse blows immediately as soon as the current exceeds the minimum melting current. The parameter value must be greater than or equal to 0.

### **Dependencies**

Enabled when the **Parameterization** parameter is set to `Opening time is independent of current`.

### **Nominal melting, $I^2 \cdot t$ — Dissipated energy needed for the fuse to blow**

1 s\*A<sup>2</sup> (default)

The amount of energy dissipated in the fuse that is needed for the fuse to blow. This is the nominal melting, or  $I^2t$ , parameter from the datasheet. The value must be greater than or equal to 0.

### **Dependencies**

Enabled when the **Parameterization** parameter is set to `Opening time depends on  $I^2 \cdot t$` .

### **Tabulated currents — Currents data from the time-current curves**

[1.9, 2, 2.5, 30] A (default)

The tabulated current values from a time-current curve on the manufacturer datasheet. Every element in this vector must be greater than 0, and the vector must be strictly ascending.

### **Dependencies**

Enabled when the **Parameterization** parameter is set to `Opening time is tabulated`.

**Tabulated opening times — Times data from the time-current curves**

[10000, 1, 0.3, 0.001] s (default)

The tabulated time values from a time-current curve on the manufacturer datasheet. Every element in this vector must be greater than or equal to 0, and the vector must be strictly descending. The vector must be of the same size as the **Tabulated currents** vector.

**Dependencies**

Enabled when the **Parameterization** parameter is set to `Opening time is tabulated`.

**Fuse resistance R — Resistance of the fuse**

0.01  $\Omega$  (default)

The fuse resistance in low-ohmic state. The parameter value must be greater than 0.

**Open-circuit conductance G — Open-circuit fuse conductance**

1e-8 1/ $\Omega$  (default)

The open-circuit fuse conductance in high-ohmic state, when the fuse has blown. The parameter value must be greater than 0.

## See Also

**Introduced in R2008b**

## Gate Driver

Abstracted representation of gate driver integrated circuit

**Library:** Simscape / Electrical / Electronics and Mechatronics /  
Actuators & Drivers / Drivers



### Description

The Gate Driver block provides an abstracted representation of a gate driver integrated circuit. The block models input hysteresis, propagation delay, and turn-on/turn-off dynamics. Unless modeling a gate driver circuit explicitly, always use this block or the Half-Bridge Driver block to set gate-source voltage on a MOSFET block or gate-emitter voltage on an IGBT block. Do not connect a controlled voltage source directly to a semiconductor gate, because this omits the gate driver output impedance that determines switching dynamics.

The Gate Driver block has two modeling variants, accessible by right-clicking the block in your block diagram and then selecting the appropriate option from the context menu, under **Simscape > Block choices**:

- **Physical signal input** — The driver output state is controlled by a physical signal input *u*. Use this variant if all of your controller, including PWM waveform generation, is determined by Simulink blocks. This modeling variant is the default.
- **Electrical input ports** — The driver output state is controlled by two electrical input connections, PWM and REF. Use this variant if your model has upstream analog components, such as the Controlled PWM Voltage source.

When the input rises above the logic 1 input level, the transition of the output state from off to on is initiated after a delay equal to the turn-on propagation delay. The demanded output voltage across the G and S ports steps in value from the off-state output voltage to the on-state output voltage, but the actual output voltage is set by the RC time constant associated with the **On-state gate drive resistance** value and the total load capacitance. Similarly, when the input falls below the logic 0 input value, the transition of the output state from on to off is initiated after a delay equal to the turn-off propagation delay and with dynamics now set by the **Off-state gate drive resistance** value.

## Faults

You can insert a fault into the output of the gate driver at a specified simulation time, to make the connected semiconductor device either permanently off or permanently on. Use this feature to represent a failed semiconductor device as failed at open-circuit or at normal on-state conditions.

## Ports

### Input

#### **u — Control signal, unitless**

physical signal

Input physical signal that specifies the input control value.

#### **Dependencies**

Enabled for the **Physical signal input** variant of the block.

### Conserving

#### **PWM — Pulse-width modulated signal**

electrical

Electrical conserving port associated with the pulse-width modulated signal.

#### **Dependencies**

Enabled for the **Electrical input ports** variant of the block.

#### **REF — Floating zero volt reference**

electrical

Electrical conserving port associated with the floating zero volt reference.

#### **Dependencies**

Enabled for the **Electrical input ports** variant of the block.

## **G — Gate connection**

electrical

Electrical conserving port associated with the gate. Connect this port to the gate of a MOSFET or IGBT block.

## **S — Source or emitter connection**

electrical

Electrical conserving port associated with the source or emitter. Connect this port to the source of a MOSFET block or the emitter of an IGBT block.

# Parameters

## **Input Logic**

### **Logic 1 input value — Signal value for logic 1**

0.7 (default)

Value of the input signal corresponding to the logic 1 level.

#### **Dependencies**

Enabled for the **Physical signal input** variant of the block.

### **Logic 0 input value — Signal value for logic 0**

0.3 (default)

Value of the input signal corresponding to the logic 0 level.

#### **Dependencies**

Enabled for the **Physical signal input** variant of the block.

### **Logic 1 input voltage — Voltage value for logic 1**

2.0 V (default)

Value of the input voltage corresponding to the logic 1 level.

#### **Dependencies**

Enabled for the **Electrical input ports** variant of the block.



**Logic 0 input voltage — Voltage value for logic 0**

0.8 V (default)

Value of the input voltage corresponding to the logic 0 level.

**Dependencies**

Enabled for the **Electrical input ports** variant of the block.

**Outputs****On-state gate-source voltage — Demanded output voltage for on state**

15 V (default)

Demanded output voltage when the driver is in on state.

**Off-state gate-source voltage — Demanded output voltage for off state**

0 V (default)

Demanded output voltage when the driver is in off state.

**Timing****Propagation delay (logic 0->logic 1) — Turn-on propagation delay**

50 ns (default)

When the input rises above the logic 1 input level, the transition of the output state from off to on is initiated after a delay equal to the turn-on propagation delay.

**Propagation delay (logic 1->logic 0) — Turn-off propagation delay**

50 ns (default)

When the input falls below the logic 0 input value, the transition of the output state from on to off is initiated after a delay equal to the turn-off propagation delay.

**Dynamics****Parameterization — Select driver parameterization**

Output impedance (default) | Rise and fall times

Select the type of driver parameterization:

- **Output impedance** — Specify on-state and off-state gate drive resistances.
- **Rise and fall times** — Specify rise time, fall time, and load capacitance.

### **On-state gate drive resistance — Gate drive resistance for on state**

2  $\Omega$  (default)

Gate drive resistance when the driver is in on state.

#### **Dependencies**

Enabled when the **Parameterization** parameter is set to **Output impedance**.

### **Off-state gate drive resistance — Gate drive resistance for off state**

2  $\Omega$  (default)

Gate drive resistance when the driver is in off state.

#### **Dependencies**

Enabled when the **Parameterization** parameter is set to **Output impedance**.

### **Rise time — Driver rise time**

20 ns (default)

Driver rise time from 10% to 90%.

#### **Dependencies**

Enabled when the **Parameterization** parameter is set to **Rise and fall times**.

### **Fall time — Driver fall time**

20 ns (default)

Driver fall time from 90% to 10%.

#### **Dependencies**

Enabled when the **Parameterization** parameter is set to **Rise and fall times**.

### **Load capacitance for rise and fall times — Driver load capacitance**

10 nF (default)

Driver load capacitance.

**Dependencies**

Enabled when the **Parameterization** parameter is set to Rise and fall times.

**Faults****Enable faults — Select Yes to enable faults modeling**

No (default) | Yes

Select Yes to enable faults modeling. The associated parameters in the **Faults** section become visible to let you specify time to fail and the failure mode.

**Simulation time for fault event — Time before entering faulted state**

1 s (default)

Set the simulation time at which you want the block to enter the faulted state.

**Dependencies**

Enabled when the **Enable faults** parameter is set to Yes.

**Failure mode — State of driver after failure**

Fail off (default) | Fail on

Select whether driver fails by making the connected semiconductor device permanently turned off or on.

**Dependencies**

Enabled when the **Enable faults** parameter is set to Yes.

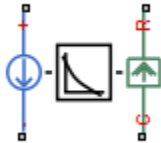
**See Also**

Controlled PWM Voltage | Half-Bridge Driver | N-Channel IGBT | N-Channel MOSFET | P-Channel MOSFET

**Introduced in R2017b**

## Generic Linear Actuator

Model generic linear actuator driven from DC voltage source or PWM driver



## Library

Translational Actuators

## Description

The Generic Linear Actuator block implements a model of a generic linear actuator designed to be driven from a DC voltage source or a PWM driver. Define force-speed characteristics in terms of tabulated values for powering the motor at the rated voltage. This functionality enables you to model a motor without referencing an equivalent circuit.

The motor or actuator architecture determines the way in which electrical losses depend on force. For example, a DC motor has losses that are proportional to the square of the current. As force is proportional to current, losses are also proportional to mechanical force. Most motors have an electrical loss term that is proportional to the square of mechanical force. The Generic Linear Actuator block calculates this loss term using the **Motor efficiency (percent)** and **Speed at which efficiency is measured** parameters that you provide.

Some motors also have a loss term that is independent of force. An example is a shunt motor where the field winding draws a constant current regardless of load. The **Force-independent electrical losses** parameter accounts for this effect.

The motor efficiency is the mechanical power divided by the sum of the mechanical power and both electrical loss terms. The block assumes that the speed at which the motor efficiency is defined is in the motoring quadrant and, therefore, positive.

You can operate the block in the reverse direction by changing the sign of the voltage applied. The H-Bridge block, for example, reverses motor direction if the voltage at the REV port is greater than the **Reverse threshold voltage** parameter. However, if you are using the block in reverse, specify the force-speed data for forward operation:

- Positive forces and positive speeds in the motoring quadrant.
- Positive force and negative speeds in the generating counterclockwise quadrant.
- Negative force and positive speed in the generating clockwise quadrant.

## Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal port H on the block icon, and adds the **Temperature Dependence** and **Thermal Port** tabs to the block dialog box.

Use the thermal port to simulate the effects of copper resistance losses that convert electrical power to heat. For more information on using thermal ports and on the **Temperature Dependence** and **Thermal Port** tab parameters, see “Simulating Thermal Effects in Rotational and Translational Actuators”.

## Basic Assumptions and Limitations

This block has the following limitations:

- The force-speed curve data corresponds only to the rated voltage, so the block produces accurate results only when driven by plus or minus the rated voltage.
- The block requires you to provide force-speed data for the full range over which you use the actuator. To use the actuator in the generating and braking regions, provide additional data outside of the normal motoring region.
- Model behavior is sensitive to force-speed data. For example, no-load speed is correctly defined and finite only when the data crosses the speed axis.
- To drive the block from the H-Bridge block:
  - Do not place any other blocks between the H-Bridge and the Generic Linear Actuator blocks.

- In the H-Bridge block dialog box, set the **Freewheeling mode** to **Via one semiconductor switch and one freewheeling diode**. Selecting **Via two freewheeling diodes** does not set the bridge output voltage to zero when the PWM input signal is low.
- In the H-Bridge, Generic Linear Actuator, and Controlled PWM Voltage block dialog boxes, ensure that the **Simulation mode** is the same for all three blocks.

## Parameters

- “Electrical Force Tab” on page 1-672
- “Mechanical Tab” on page 1-673

### Electrical Force Tab

#### Speed values

Specify a vector of speeds, including their units, for your force-speed data. The default value is [ -15 -10 -5 0 5 10 15 20 25 30 ] m/s.

#### Force values

Specify a vector of forces, including their units, for your force-speed data. The default value is [ 4 3.5 3 2.5 2 1.5 1 0.5 0 -0.5 ] N.

#### Rated voltage

Indicate the voltage for which the device you are modeling is rated. The default value is 12 V.

#### Motor efficiency (percent)

Efficiency that the block uses to calculate force-dependent electrical losses. The default value is 70.

#### Speed at which efficiency is measured

Speed that the block uses to calculate force-dependent electrical losses. The default value is 20 m/s.

#### Force-independent electrical losses

Fixed electrical loss associated with the actuator when the force is zero. The default value is 2 W.

#### Simulation mode

If you set the **Simulation mode** parameter to **PWM**, apply a PWM waveform switching between zero and rated volts to the block electrical terminals. The current drawn

from the electrical supply is equal to the amount required to deliver the mechanical power and to compensate for electrical losses. If the applied voltage exceeds the rated voltage, the resultant force scales proportionately. However, applying anything other than the rated voltage can provide unrepresentative results. PWM is the default setting.

If you set the **Simulation mode** parameter to **Averaged**, the force generated in response to an applied voltage  $V_{av}$  is

$$\frac{V_{av}}{V_{rated}} \times F(v)$$

where  $F(v)$  is the force value at speed  $v$ . The current drawn from the supply is such that the product of the current and  $V_{av}$  is equal to the average power that is consumed.

## Mechanical Tab

### Plunger mass

Mass of the moving part of the motor. The default value is 0.1 kg. The value can be zero.

### Linear damping

Linear damping. The default value is 1e-05 N/(m/s). The value can be zero.

### Initial plunger speed

Speed of the plunger at the start of the simulation. The default value is 0 m/s.

## Ports

This block has the following ports:

+

Positive electrical conserving port

-

Negative electrical conserving port

C

Mechanical translational conserving port connected to the actuator case

R

Mechanical translational conserving port connected to the plunger

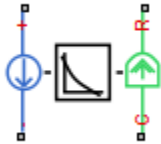
## **See Also**

Generic Rotary Actuator | H-Bridge



# Generic Rotary Actuator

Model generic rotary actuator driven from DC voltage source or PWM driver



## Library

Rotational Actuators

## Description

The Generic Rotary Actuator block implements a model of a generic rotary actuator designed to be driven from a DC voltage source or PWM driver. You define torque-speed characteristics in terms of tabulated values for powering the motor at the rated voltage. This functionality allows you to model a motor without referencing an equivalent circuit.

The motor or actuator architecture determines the way in which electrical losses depend on torque. For example, a DC motor has losses that are proportional to the square of the current. As torque is proportional to current, losses are also proportional to mechanical torque. Most motors have an electrical loss term that is proportional to the square of mechanical torque. The Generic Rotary Actuator block calculates this loss term using the **Motor efficiency (percent)** and **Speed at which efficiency is measured** parameters that you provide.

Some motors also have a loss term that is independent of torque. An example is a shunt motor where the field winding draws a constant current regardless of load. The **Torque-independent electrical losses** parameter accounts for this effect.

The motor efficiency is the mechanical power divided by the sum of the mechanical power and both electrical loss terms. The block assumes that the speed at which the motor efficiency is defined is in the motoring quadrant and, therefore, positive.

You can operate the block in the reverse direction by changing the sign of the voltage that you apply. The H-Bridge block, for example, reverses motor direction if the voltage at the REV port is greater than the **Reverse threshold voltage** parameter. However, if you are using the block in reverse, specify the torque-speed data for forward operation:

- Positive torques and positive speeds in the motoring quadrant.
- Positive torque and negative speeds in the generating counterclockwise quadrant.
- Negative torque and positive speed in the generating clockwise quadrant.

### Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal port H on the block icon, and adds the **Temperature Dependence** and **Thermal Port** tabs to the block dialog box.

Use the thermal port to simulate the effects of copper resistance losses that convert electrical power to heat. For more information on using thermal ports and on the **Temperature Dependence** and **Thermal Port** tab parameters, see “Simulating Thermal Effects in Rotational and Translational Actuators”.

## Basic Assumptions and Limitations

This block has the following limitations:

- The torque-speed curve data corresponds only to the rated voltage, so the block produces accurate results only when driven by plus or minus the rated voltage.
- In this block requires, you must provide torque-speed data for the full range over which you use the actuator. To use the actuator in the generating and braking regions, provide additional data outside of the normal motoring region.
- Model behavior is sensitive to torque-speed data. For example, no-load speed is correctly defined and finite only when the data crosses the speed axis.
- To drive the block from the H-Bridge block:
  - Do not place any other blocks between the H-Bridge and the Generic Rotary Actuator blocks.

- In the H-Bridge block dialog box, set the **Freewheeling mode** to **Via one semiconductor switch and one freewheeling diode**. Selecting **Via two freewheeling diodes** does not set the bridge output voltage to zero when the PWM input signal is low.
- In the H-Bridge, Generic Rotary Actuator, and Controlled PWM Voltage block dialog boxes, ensure that the **Simulation mode** is the same for all three blocks.

## Parameters

- “Electrical Torque Tab” on page 1-677
- “Mechanical Tab” on page 1-678

### Electrical Torque Tab

#### Speed values

Specify a vector of speeds, including their units, for your torque-speed data. The default value is [ -1.5e+03 -1000 -500 0 500 1000 1.5e+03 2e+03 2.5e+03 3e+03 ] rpm.

#### Torque values

Specify a vector of torques, including their units, for your torque-speed data. The default value is [ 0.04 0.035 0.03 0.025 0.02 0.015 0.01 0.005 0 -0.005 ] Nm.

#### Rated voltage

Indicate the voltage for which the device you are modeling is rated. The default value is 12 V.

#### Motor efficiency (percent)

The efficiency that the block uses to calculate torque-dependent electrical losses. The default value is 80.

#### Speed at which efficiency is measured

The speed that the block uses to calculate torque-dependent electrical losses. The default value is 2e+03 rpm.

#### Torque-independent electrical losses

Fixed electrical loss associated with the actuator when the torque is zero. The default value is 0.1 W.

**Simulation mode**

If you set the **Simulation mode** parameter to PWM, apply a PWM waveform switching between zero and rated volts to the block electrical terminals. The current drawn from the electrical supply is equal to the amount required to deliver the mechanical power and to compensate for electrical losses. If the applied voltage exceeds the rated voltage, the resultant torque scales proportionately. However, applying anything other than the rated voltage can provide unrepresentative results. PWM is the default setting.

If you set the **Simulation mode** parameter to Averaged, the torque generated in response to an applied voltage  $V_{av}$  is

$$\frac{V_{av}}{V_{rated}} \times T(\omega)$$

where  $T(\omega)$  is the torque value at speed  $\omega$ . The current drawn from the supply is such that the product of the current and  $V_{av}$  is equal to the average power that is consumed.

**Mechanical Tab****Rotor inertia**

Rotor resistance to change in motor motion. The default value is  $1e-04$  kg\*m<sup>2</sup>. The value can be zero.

**Rotor damping**

Rotor damping. The default value is  $1e-08$  N\*m/(rad/s). The value can be zero.

**Initial rotor speed**

Speed of the rotor at the start of the simulation. The default value is 0 rpm.

**Ports**

This block has the following ports:

+

Positive electrical conserving port

- Negative electrical conserving port
- C Mechanical rotational conserving port
- R Mechanical rotational conserving port

## **See Also**

Generic Linear Actuator | H-Bridge

## Grounded Neutral

Ground connection for phases of three-phase system



## Library

Simscape / Electrical / Power Systems / Connections

## Description

The Grounded Neutral block connects the phases of a three-phase system to ground.

---

**Note** If you want to connect the neutral point of the three-phase system to other blocks, use the Neutral Port block instead. If you want to create a floating neutral point, use the Floating Neutral block.

---

## Ports

The block has the following ports:

~

Expandable three-phase port

## See Also

### Simscape Blocks

Floating Neutral | Neutral Port | Open Circuit

## **Topics**

“Three-Phase Custom Synchronous Machine”

“Three-Phase Three-Level PWM Generator”

“Three-Phase Bridge Cycloconverter”

**Introduced in R2013b**

# GTO

Gate Turn-Off Thyristor

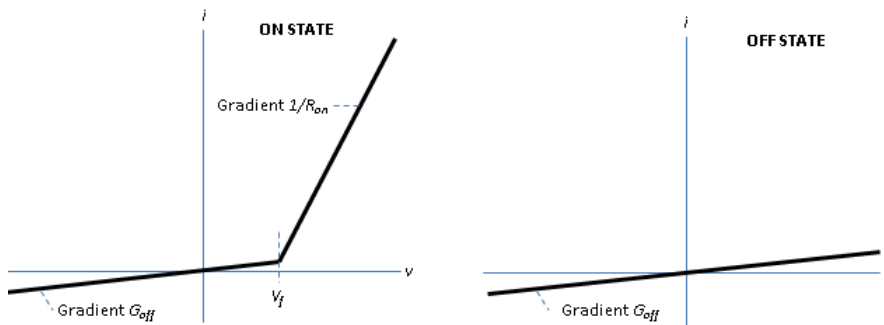


## Library

Simscape / Electrical / Power Systems / Semiconductors / Fundamental Components

## Description

The GTO block models a gate turn-off thyristor (GTO). The I-V characteristic of a GTO is such that if the gate-cathode voltage exceeds the specified gate trigger voltage, the GTO turns on. If the gate-cathode voltage falls below the specified gate turn-off voltage value, or if the load current falls below the specified holding-current value, the device turns off.



In the on state, the anode-cathode path behaves like a linear diode with forward-voltage drop,  $V_f$ , and on-resistance,  $R_{on}$ .

In the off state, the anode-cathode path behaves like a linear resistor with a low off-state conductance value,  $G_{off}$ .



The defining Simscape equations for the block are:

```

if ((v > Vf)&&((G>Vgt)|| (i>Ih)))&&(G>Vgt_off)
    i == (v - Vf*(1-Ron*Goff))/Ron;
else
    i == v*Goff;
end

```

where:

- $v$  is the anode-cathode voltage.
- $V_f$  is the forward voltage.
- $G$  is the gate voltage.
- $V_{gt}$  is the gate trigger voltage.
- $i$  is the anode-cathode current.
- $I_h$  is the holding current.
- $V_{gt\_off}$  is the gate turn-off voltage.
- $R_{on}$  is the on-state resistance.
- $G_{off}$  is the off-state conductance.

Using the Integral Diode tab of the block dialog box, you can include an integral cathode-anode diode. A GTO that includes an integral cathode-anode diode is known as an asymmetrical GTO (A-GTO) or reverse-conducting GTO (RCGTO). An integral diode protects the semiconductor device by providing a conduction path for reverse current. An inductive load can produce a high reverse-voltage spike when the semiconductor device suddenly switches off the voltage supply to the load.

The table shows you how to set the **Integral protection diode** parameter based on your goals.

Goal	Value to Select	Block Behavior
Prioritize simulation speed.	Protection diode with no dynamics	The block includes an integral copy of the Diode block. To parameterise the internal Diode block, use the <b>Protection</b> parameters.

Goal	Value to Select	Block Behavior
Precisely specify reverse-mode charge dynamics.	Protection diode with charge dynamics	The block includes an integral copy of the Commutation Diode block. To parameterise the internal Commutation Diode block, use the <b>Protection</b> parameters.

## Modeling Variants

The block provides four modeling variants. To select the desired variant, right-click the block in your model. From the context menu, select **Simscape > Block choices**, and then one of these variants:

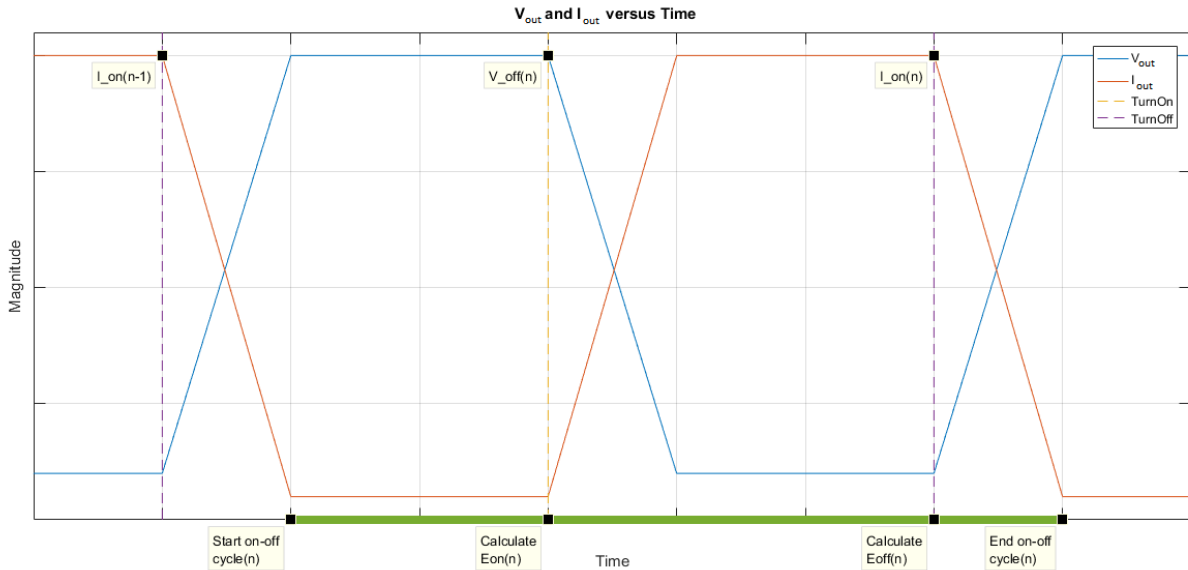
- **PS Control Port** — Contains a physical signal port that is associated with the gate terminal. This variant is the default.
- **Electrical Control Port** — Contains an electrical conserving port that is associated with the gate terminal.
- **PS Control Port | Thermal Port** — Contains a thermal port and a physical signal port that is associated with the gate terminal.
- **Electrical Control Port | Thermal Port** — Contains a thermal port and an electrical conserving port that is associated with the gate terminal.

The variants of this block without the thermal port do not simulate heat generation in the device.

The variants with the thermal port allow you to model the heat that switching events and conduction losses generate. For numerical efficiency, the thermal state does not affect the electrical behavior of the block. The thermal port is hidden by default. To enable the thermal port, select a thermal block variant.

## Thermal Loss Equations

The figure shows an idealized representation of the output voltage,  $V_{out}$ , and the output current,  $I_{out}$ , of the semiconductor device. The interval shown includes the entire  $n$ th switching cycle, during which the block turns off and then on.



When the semiconductor turns on during the  $n$ th switching cycle, the amount of thermal energy that the device dissipates increments by a discrete amount. If you select **Voltage, current, and temperature** for the **Thermal loss dependent on** parameter, the equation for the incremental change is

$$E_{on(n)} = \frac{V_{off(n)}}{V_{off\_data}} fcn(T, I_{on(n-1)}),$$

where:

- $E_{on(n)}$  is the switch-on loss at the  $n$ th switch-on event.
- $V_{off(n)}$  is the off-state output voltage,  $V_{out}$ , just before the device switches on during the  $n$ th switching cycle.
- $V_{off\_data}$  is the **Off-state voltage for losses data** parameter value.
- $T$  is the device temperature.
- $I_{on(n-1)}$  is the on-state output current,  $I_{out}$ , just before the device switches off during the cycle that precedes the  $n$ th switching cycle.

The function  $fcn$  is a 2-D lookup table with linear interpolation and linear extrapolation:

$$E = \text{tablelookup}(T_{j\_data}, I_{out\_data}, E_{on\_data}, T, I_{on(n-1)}),$$

where:

- $T_{j\_data}$  is the **Temperature vector, Tj** parameter value.
- $I_{out\_data}$  is the **Output current vector, Iout** parameter value.
- $E_{on\_data}$  is the **Switch-on loss, Eon=fcn(Tj,Iout)** parameter value.

If you select **Voltage** and **current** for the **Thermal loss dependent on** parameter, when the semiconductor turns on during the  $n$ th switching cycle, the equation that the block uses to calculate the incremental change in the discrete amount of thermal energy that the device dissipates is

$$E_{on(n)} = \left( \frac{V_{off(n)}}{V_{off\_data}} \right) \left( \frac{I_{on(n-1)}}{I_{out\_scalar}} \right) (E_{on\_scalar})$$

where:

- $I_{out\_scalar}$  is the **Output current, Iout** parameter value.
- $E_{on\_scalar}$  is the **Switch-on loss** parameter value.

When the semiconductor turns off during the  $n$ th switching cycle, the amount of thermal energy that the device dissipates increments by a discrete amount. If you select **Voltage**, **current**, and **temperature** for the **Thermal loss dependent on** parameter, the equation for the incremental change is

$$E_{off(n)} = \frac{V_{off(n)}}{V_{off\_data}} \text{fcn}(T, I_{on(n)}),$$

where:

- $E_{off(n)}$  is the switch-off loss at the  $n$ th switch-off event.
- $V_{off(n)}$  is the off-state output voltage,  $V_{out}$ , just before the device switches on during the  $n$ th switching cycle.
- $V_{off\_data}$  is the **Off-state voltage for losses data** parameter value.
- $T$  is the device temperature.

- $I_{on(n)}$  is the on-state output current,  $I_{out}$ , just before the device switches off during the  $n$ th switching cycle.

The function  $fcn$  is a 2-D lookup table with linear interpolation and linear extrapolation:

$$E = \text{tablelookup}(T_{j\_data}, I_{out\_data}, E_{off\_data}, T, I_{on(n)}),$$

where:

- $T_{j\_data}$  is the **Temperature vector**, **Tj** parameter value.
- $I_{out\_data}$  is the **Output current vector**, **Iout** parameter value.
- $E_{off\_data}$  is the **Switch-off loss**, **Eoff=fcn(Tj,Iout)** parameter value.

If you select **Voltage** and **current** for the **Thermal loss dependent on** parameter, when the semiconductor turns off during the  $n$ th switching cycle, the equation that the block uses to calculate the incremental change in the discrete amount of thermal energy that the device dissipates is

$$E_{off(n)} = \left( \frac{V_{off(n)}}{V_{off\_data}} \right) \left( \frac{I_{on(n-1)}}{I_{out\_scalar}} \right) (E_{off\_scalar})$$

where:

- $I_{out\_scalar}$  is the **Output current**, **Iout** parameter value.
- $E_{off\_scalar}$  is the **Switch-off loss** parameter value.

If you select **Voltage**, **current**, and **temperature** for the **Thermal loss dependent on** parameter, then, for both the on state and the off state, the heat loss due to electrical conduction is

$$E_{conduction} = \int fcn(T, I_{out}) dt,$$

where:

- $E_{conduction}$  is the heat loss due to electrical conduction.
- $T$  is the device temperature.
- $I_{out}$  is the device output current.

The function *fcn* is a 2-D lookup table:

$$Q_{conduction} = \text{tablelookup}(T_{j\_data}, I_{out\_data}, I_{out\_data\_repmat} .* V_{on\_data}, T, I_{out}),$$

where:

- $T_{j\_data}$  is the **Temperature vector, Tj** parameter value.
- $I_{out\_data}$  is the **Output current vector, Iout** parameter value.
- $I_{out\_data\_repmat}$  is a matrix that contains length,  $T_{j\_data}$ , copies of  $I_{out\_data}$ .
- $V_{on\_data}$  is the **On-state voltage, Von=fcn(Tj,Iout)** parameter value.

If you select **Voltage** and **current** for the **Thermal loss dependent on** parameter, then, for both the on state and the off state, the heat loss due to electrical conduction is

$$E_{conduction} = \int (I_{out} * V_{on\_scalar}) dt,$$

where  $V_{on\_scalar}$  is the **On-state voltage** parameter value.

The block uses the **Energy dissipation time constant** parameter to filter the amount of heat flow that the block outputs. The filtering allows the block to:

- Avoid discrete increments for the heat flow output
- Handle a variable switching frequency

The filtered heat flow is

$$Q = \frac{1}{\tau} \left( \sum_{i=1}^n E_{on(i)} + \sum_{i=1}^n E_{off(i)} + E_{conduction} - \int Q dt \right),$$

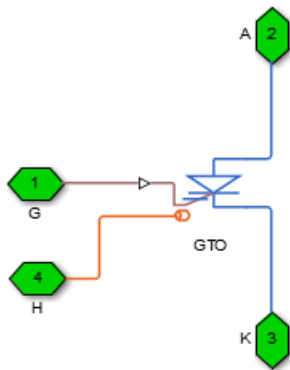
where:

- $Q$  is the heat flow from the component.
- $\tau$  is the **Energy dissipation time constant** parameter value.
- $n$  is the number of switching cycles.
- $E_{on(i)}$  is the switch-on loss at the  $i$ th switch-on event.

- $E_{off(i)}$  is the switch-off loss at the  $i$ th switch-off event.
- $E_{conduction}$  is the heat loss due to electrical conduction.
- $\int Qdt$  is the total heat previously dissipated from the component.

## Ports

The figure shows the block port names.



G

Port associated with the gate terminal. You can set the port to either a physical signal or electrical port.

A

Electrical conserving port associated with the anode terminal.

K

Electrical conserving port associated with the cathode terminal.

H

Thermal conserving port. The thermal port is optional and is hidden by default. To enable this port, select a variant that includes a thermal port.

## Parameters

- “Main Tab” on page 1-690
- “Integral Diode Tab” on page 1-690
- “Thermal Model Tab” on page 1-693

### Main Tab

#### Forward voltage, Vf

Minimum voltage required across the anode and cathode block ports for the gradient of the device I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**. The default value is 0.8 V.

#### On-state resistance

Rate of change of voltage versus current above the forward voltage. The default value is 0.001 Ohm.

#### Off-state conductance

Anode-cathode conductance when the device is off. The value must be less than  $1/R$ , where  $R$  is the value of **On-state resistance**. The default value is  $1e-5$  1/Ohm.

#### Gate trigger voltage, Vgt

Gate-cathode voltage threshold. The device turns on when the gate-cathode voltage is above this value. The default value is 1 V.

#### Gate turn-off voltage, Vgt\_off

Gate-cathode voltage threshold. The device turns off when the gate-cathode voltage is below this value. The default value is -1 V.

#### Holding current

Current threshold. The device stays on when the current is above this value, even when the gate-cathode voltage falls below the gate trigger voltage. The default value is 1 A.

### Integral Diode Tab

#### Integral protection diode

Block integral protection diode. The default value is None.

The diodes you can select are:



- Protection diode with no dynamics
- Protection diode with charge dynamics

When you select Protection diode with no dynamics, additional parameters appear.

### **Additional Parameters for Protection diode with no dynamics**

#### **Forward voltage**

Minimum voltage required across the + and - block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**. The default value is 0.8 V.

#### **On resistance**

Rate of change of voltage versus current above the **Forward voltage**. The default value is 0.001 Ohm.

#### **Off conductance**

Conductance of the reverse-biased diode. The default value is  $1e-5$  1/Ohm.

For more information on these parameters, see Diode.

When you select Protection diode with charge dynamics, additional parameters appear.

### **Additional Parameters for Protection diode with charge dynamics**

#### **Forward voltage**

Minimum voltage required across the + and - block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**. The default value is 0.8 V.

#### **On resistance**

Rate of change of voltage versus current above the **Forward voltage**. The default value is 0.001 Ohm.

#### **Off conductance**

Conductance of the reverse-biased diode. The default value is  $1e-5$  1/Ohm.

#### **Junction capacitance**

Diode junction capacitance. The default value is 50 nF.

**Peak reverse current, *i*RM**

Peak reverse current measured by an external test circuit. This value must be less than zero. The default value is -235 A.

**Initial forward current when measuring *i*RM**

Initial forward current when measuring peak reverse current. This value must be greater than zero. The default value is 300 A.

**Rate of change of current when measuring *i*RM**

Rate of change of current when measuring peak reverse current. This value must be less than zero. The default value is -50 A/ $\mu$ s.

**Reverse recovery time parameterization**

Determines how you specify reverse recovery time in the block. The default value is Specify reverse recovery time directly.

If you select Specify stretch factor or Specify reverse recovery charge, you specify a value that the block uses to derive the reverse recovery time. For more information on these options, see “Alternatives to Specifying trr Directly” on page 1-339.

**Reverse recovery time, trr**

Interval between the time when the current initially goes to zero (when the diode turns off) and the time when the current falls to less than 10% of the peak reverse current. The default value is 15  $\mu$ s.

This parameter is visible only if you set **Reverse recovery time parameterization** to Specify reverse recovery time directly.

The value of the **Reverse recovery time, trr** parameter must be greater than the value of the **Peak reverse current, *i*RM** parameter divided by the value of the **Rate of change of current when measuring *i*RM** parameter.

**Reverse recovery time stretch factor**

Value that the block uses to calculate **Reverse recovery time, trr**. This value must be greater than 1. The default value is 3.

This parameter is visible only if you set **Reverse recovery time parameterization** to Specify stretch factor.

Specifying the stretch factor is an easier way to parameterize the reverse recovery time than specifying the reverse recovery charge. The larger the value of the stretch factor, the longer it takes for the reverse recovery current to dissipate.

### Reverse recovery charge, $Q_{rr}$

Value that the block uses to calculate **Reverse recovery time,  $t_{rr}$** . Use this parameter if the data sheet for your diode device specifies a value for the reverse recovery charge instead of a value for the reverse recovery time.

The reverse recovery charge is the total charge that continues to dissipate when the

diode turns off. The value must be less than  $-\frac{i_{RM}^2}{2a}$ ,

where:

- $i_{RM}$  is the value specified for **Peak reverse current,  $i_{RM}$** .
- $a$  is the value specified for **Rate of change of current when measuring  $i_{RM}$** .

The default value is 1500  $\mu$ As.

The parameter is visible only if you set **Reverse recovery time parameterization** to Specify reverse recovery charge.

For more information on these parameters, see Commutation Diode.

## Thermal Model Tab

The **Thermal Model** tab is enabled only when you select a block variant that includes a thermal port.

### Thermal loss dependent on

Select a parameterization method. The option that you select determines which other parameters are enabled. Options are:

- **Voltage and current** — Use scalar values to specify the output current, switch-on loss, switch-off loss, and on-state voltage data.
- **Voltage, current, and temperature** — Use vectors to specify the output current, switch-on loss, switch-off loss, on-state voltage, and temperature data. This is the default parameterization method.

**Off-state voltage for losses data**

The output voltage of the device during the off state. This is the blocking voltage at which the switch-on loss and switch-off loss data are defined. The default value is 300 V.

**Energy dissipation time constant**

Time constant used to average the switch-on losses, switch-off losses, and conduction losses. This value is equal to the period of the minimum switching frequency. The default value is 1e-4 s.

**Additional Parameters for Parameterizing by Voltage, Current, and Temperature****Temperature vector, Tj**

Temperature values at which the switch-on loss, switch-off loss, and on-state voltage are specified. Specify this parameter using a vector quantity. The default value is [ 298.15 398.15 ] K.

**Output current vector, Iout**

Output currents for which the switch-on loss, switch-off loss and on-state voltage are defined. The first element must be zero. Specify this parameter using a vector quantity. The default value is [ 0 10 50 100 200 400 600 ] A.

**Switch-on loss, Eon=fcn(Tj,Iout)**

Energy dissipated during a single switch on event. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a vector quantity. The default value is [ 0 2.9e-4 0.00143 0.00286 0.00571 0.01314 0.02286; 0 5.7e-4 0.00263 0.00514 0.01029 0.02057 0.03029 ] J.

**Switch-off loss, Eoff=fcn(Tj,Iout)**

Energy dissipated during a single switch-off event. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a vector quantity. The default value is [ 0 2.1e-4 0.00107 0.00214 0.00429 0.009859999999999999 0.01714; 0 4.3e-4 0.00197 0.00386 0.00771 0.01543 0.02271 ] J.

**On-state voltage, Von=fcn(Tj,Iout)**

Voltage drop across the device while it is in a triggered conductive state.. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a vector quantity. The default value is [ 0 1.1 1.3 1.45 1.75 2.25 2.7; 0 1 1.15 1.35 1.7 2.35 3 ] V.

## Additional Parameters for Parameterizing by Voltage and Current

### Output current, $I_{out}$

Output currents for which the switch-on loss, switch-off loss, and on-state voltage are defined. The first element must be zero. Specify this parameter using a scalar quantity. The default value is 600 A.

### Switch-on loss

Energy dissipated during a single switch-on event. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a scalar quantity. The default value is 0.02286 J.

### Switch-off loss

Energy dissipated during a single switch-off event. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a scalar quantity. The default value is 0.01714 J.

### On-state voltage

Voltage drop across the block while it is in a triggered conductive state. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a scalar quantity. The default value is 2.7 V.

## See Also

Commutation Diode | Diode (Piecewise Linear) | IGBT | Ideal Semiconductor Switch | MOSFET | Thyristor (Piecewise Linear)

## Topics

“Quantifying IGBT Thermal Losses”

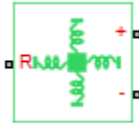
“Simulate Thermal Losses in Semiconductors”

“Switch Between Physical Signal and Electrical Ports”

## Introduced in R2013b

## Gyro

Behavioral model of MEMS gyro



## Library

Sensors

## Description

The Gyro block implements a behavioral model of a MicroElectroMechanical Systems (MEMS) gyro. The gyro provides an output voltage that is proportional to the angular rotation rate presented at the mechanical rotational physical port R. The output voltage is limited according to the values that you provide for maximum and minimum output voltage.

Optionally, you can model sensor dynamics by setting the **Dynamics** parameter to `Model sensor bandwidth`. Including dynamics adds a first-order lag between the angular rate presented at port R and the corresponding voltage applied to the electrical + and - ports.

If running your simulation with a fixed-step solver, or generating code for hardware-in-the-loop testing, MathWorks recommends that you set the **Dynamics** parameter to `No dynamics – Suitable for HIL`, because this avoids the need for a small simulation time step if the sensor bandwidth is high.

## Variables

Use the **Variables** section of the block interface to set the priority and initial target values for the block variables prior to simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

The **Measured angular rate** variable target specifies the initial output for the sensor.

## Parameters

### Sensitivity

The change in output voltage level per unit change in rotation rate when the output is not being limited. The default value is 12.5 mV/(deg/s).

### Output voltage for zero rotation

The output voltage from the sensor when the rotation rate is zero. The default value is 2.5 V.

### Maximum output voltage

The maximum output voltage from the sensor, which determines the sensor maximum measured rotational rate. The default value is 4 V.

### Minimum output voltage

The minimum output voltage from the sensor, which determines the sensor minimum measured rotational rate. The default value is 1 V.

### Dynamics

Select one of the following options for modeling sensor dynamics:

- **No dynamics – Suitable for HIL** — Do not model sensor dynamics. Use this option when running your simulation fixed step or generating code for hardware-in-the-loop testing, because this avoids the need for a small simulation time step if the sensor bandwidth is high. This is the default option.
- **Model sensor bandwidth** — Model sensor dynamics with a first-order lag approximation, based on the **Bandwidth** parameter value. You can control the initial condition for the lag by specifying the **Measured angular rate** variable target.

### Bandwidth

Specifies the 3dB bandwidth for the measured rotational rate assuming a first-order time constant. This parameter is only visible when you select **Model sensor bandwidth** for the **Dynamics** parameter. The default value is 3 kHz.

## **Ports**

The block has the following ports:

R

Mechanical rotational port

+

Positive electrical port

-

Negative electrical port



# H-Bridge

Model H-bridge motor driver



## Library

Drivers

## Description

The H-Bridge block represents an H-bridge motor driver. The block has the following two **Simulation mode** options:

- PWM — The H-Bridge output is a controlled voltage that depends on the input signal at the PWM port. If the input signal has a value greater than the **Enable threshold voltage** parameter value, the H-Bridge output is on and has a value equal to the value of the **Output voltage amplitude** parameter. If it has a value less than the **Enable threshold voltage** parameter value, the block maintains the load circuit using one of the following three **Freewheeling mode** options:
  - Via one semiconductor switch and one freewheeling diode
  - Via two freewheeling diodes
  - Via two semiconductor switches and one freewheeling diode

The first and third options are sometimes referred to as synchronous operation.

The signal at the REV port determines the polarity of the output. If the value of the signal at the REV port is less than the value of the **Reverse threshold voltage** parameter, the output has positive polarity; otherwise, it has negative polarity.

- Averaged — This mode has two **Load current characteristics** options:

- Smoothed
- Unsmoothed or discontinuous

The Smoothed option assumes that the current is practically continuous due to load inductance. In this case, the H-Bridge output is:

$$\frac{V_O V_{PWM}}{A_{PWM}} - I_{OUT} R_{ON}$$

where:

- $V_O$  is the value of the **Output voltage amplitude** parameter.
- $V_{PWM}$  is the value of the voltage at the PWM port.
- $A_{PWM}$  is the value of the **PWM signal amplitude** parameter.
- $I_{OUT}$  is the value of the output current.
- $R_{ON}$  is the **Bridge on resistance** parameter.

The current will be smooth if the PWM frequency is large enough. Synchronous operation where freewheeling is via a bridge arm back to the supply also helps smooth the current. For cases where the current is not smooth, or possibly discontinuous (that is, it goes to zero between PWM cycles), use the **Unsmoothed or discontinuous** option. For this option, you must also provide values for the **Total load series resistance, Total load series inductance** and **PWM frequency**. During simulation, the block uses these values to calculate a more accurate value for H-bridge output voltage that achieves the same average current as would be present if simulating in PWM mode.

Set the **Simulation mode** parameter to **Averaged** to speed up simulations when driving the H-Bridge block with a Controlled PWM Voltage block. You must also set the **Simulation mode** parameter of the Controlled PWM Voltage block to **Averaged** mode. This applies the average of the demanded PWM voltage to the motor. The accuracy of the **Averaged** mode simulation results relies on the validity of your assumption about the load current. If you specify that the current is **Unsmoothed or discontinuous**, then the accuracy also depends on the values you provide for load resistance and inductance being representative. This mode also makes some simplifying assumptions about the underlying equations for the case when current is discontinuous. For typical motor and bridge parameters, accuracy should be within a few percent. To verify **Averaged** mode accuracy, run the simulation using the **PWM** mode and compare the results to those obtained from using the **Averaged** mode.

Braking mode is invoked when the voltage presented at the BRK port is larger than the **Braking threshold voltage**. Regardless of whether in PWM or Averaged mode, when in braking mode the H-bridge is modeled by a series combination of two resistances  $R1$  and  $R2$  where:

- $R1$  is the resistance of a single bridge arm, that is, half the value of the **Total bridge on resistance** parameter.
- $R2$  is the resistance of a single bridge arm in parallel with a diode resistance, that is,  $R1 \cdot Rd / (R1 + Rd)$ , where  $Rd$  is the diode resistance.

## Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal port H on the block icon, and adds the **Temperature Dependence** and **Thermal Port** tabs to the block dialog box. These tabs are described further on this reference page.

When the thermal port is visible:

- The heat generated by the bridge on-resistance and freewheeling diodes is added to the thermal port. The thermal port has an associated thermal mass and initial temperature that you can set from the **Thermal Port** tab.
- The bridge on-resistance and freewheeling diode resistance become functions of temperature. You can define the values for these resistances and the second measurement temperature from the **Temperature Dependence** tab. Resistance is assumed to vary linearly between the two measurement temperatures. Extrapolation is used for temperatures outside of this range, except for when simulating in averaged mode with discontinuous load current characteristics.

## Basic Assumptions and Limitations

The model has the following limitations:

- If you are linearizing your model, set the **Simulation mode** parameter to **Averaged** and ensure that you have specified the operating point correctly. You can only linearize the H-Bridge block for inputs that are greater than zero and less than the PWM signal amplitude.

- No forward voltage is modeled for the freewheeling diodes. They are approximated as ideal resistances when forward biased, with resistance equal to the **Freewheeling diode on** resistance parameter value.
- In Averaged mode, and with the Unsmoothed or discontinuous choice for **Load current characteristics**, you must provide representative values for load inductance and resistance. If driving a DC Motor, then the resistance is the armature resistance, and the inductance is the sum of the armature inductance plus series smoothing inductor (if present). For a Universal Motor, total resistance is the sum of the armature and field windings, and total inductance is the sum of armature and field inductances plus any series smoothing inductance. For a Shunt Motor, MathWorks recommends that you draw a Thévenin equivalent circuit to determine appropriate values.

## Parameters

- “Simulation Mode & Load Assumptions Tab” on page 1-702
- “Input Thresholds Tab” on page 1-704
- “Bridge Parameters Tab” on page 1-705
- “Temperature Dependence Tab” on page 1-705
- “Thermal Port Tab” on page 1-706

## Simulation Mode & Load Assumptions Tab

### Simulation mode

Select one of the following options for the type of output voltage:

- PWM — The output voltage is a pulse-width modulated signal. This is the default option.
- Averaged — The output voltage is a constant whose value is equal to the average value of the PWM signal.

### Freewheeling mode

Select one of the following options for the type of H-Bridge dissipation circuit:

- Via one semiconductor switch and one freewheeling diode — In this mode, the block controls the load by maintaining one high-side bridge arm permanently on and using the PWM signal to modulate the corresponding low-side bridge arm. This means that the block uses only one of the freewheeling diodes in

completing the dissipation circuit when the bridge turns off. This option is the default.

- **Via two freewheeling diodes** — In this mode, all bridge arms are off during the bridge off-state. This means that the block dissipates the load current across the power supply by two freewheeling diodes.
- **Via two semiconductor switches and one freewheeling diode** — In this mode, the block controls the load by maintaining one high-side bridge arm permanently on and using the PWM signal to toggle between enabling the corresponding low-side bridge arm and the opposite high-side bridge arm. This means that the block uses a freewheeling diode in parallel with a bridge arm, plus another series bridge arm, to complete the dissipation circuit when the bridge turns off.

This parameter is only visible when you select PWM for the **Simulation mode** parameter, or when you select Averaged for the **Simulation mode** parameter and Unsmoothed or discontinuous for the **Load current characteristics** parameter.

### Load current characteristics

Select one of the following options for the type of load current:

- **Smoothed** — Assumes that the current is practically continuous due to load inductance. This option is the default.
- **Unsmoothed or discontinuous** — Use this option for cases where the current is not smooth, or possibly discontinuous (that is, it goes to zero between PWM cycles). For this option, you must also provide values for the **Total load series resistance**, **Total load series inductance**, and **PWM frequency** parameters. During simulation, the block uses these values to calculate a more accurate value for H-bridge output voltage that achieves the same average current as would be present if simulating in PWM mode.

This parameter is only visible when you select Averaged for the **Simulation mode** parameter.

### Load total series resistance

The total load series resistance seen by the H-bridge. The default value is 10  $\Omega$ .

This parameter is only visible when you select Averaged for the **Simulation mode** parameter and Unsmoothed or discontinuous for the **Load current characteristics** parameter.

## Load total series inductance

The total load series inductance seen by the H-bridge. As well as motor inductance, you should include any series inductance added external to the motor to smooth current. The default value is  $1e-5$  H.

This parameter is only visible when you select **Averaged** for the **Simulation mode** parameter and **Unsmoothed** or **discontinuous** for the **Load current characteristics** parameter.

## PWM frequency

The PWM frequency at which the H-bridge is driven. For consistency, this should be the same value as the PWM frequency specified by the Controlled PWM Voltage block driving the H-Bridge. The default value is 10 kHz.

This parameter is only visible when you select **Averaged** for the **Simulation mode** parameter and **Unsmoothed** or **discontinuous** for the **Load current characteristics** parameter.

## Input Thresholds Tab

### Enable threshold voltage

Threshold above which the voltage at the PWM port must rise to enable the H-Bridge output. This parameter is used only when the **Simulation mode** parameter on the **Simulation Mode & Load Assumptions** tab is set to PWM. The default value is 2.5 V.

### PWM signal amplitude

The amplitude of the signal at the PWM input. The H-Bridge block uses this parameter only when the **Simulation mode** parameter on the **Simulation Mode & Load Assumptions** tab is set to **Averaged**. The default value is 5 V.

### Reverse threshold voltage

When the voltage at the REV port is greater than this threshold, the output polarity becomes negative. The default value is 2.5 V.

### Braking threshold voltage

When the voltage at the BRK port is greater than this threshold, the H-Bridge output terminals are short-circuited via the following series of devices:

- One bridge arm

- One bridge arm in parallel with a conducting freewheeling diode

The default value is 2.5 V.

## Bridge Parameters Tab

### Output voltage amplitude

The amplitude of the voltage across the H-Bridge output ports when the output is on. The default value is 12 V.

### Total bridge on resistance

The total effective resistance of the two semiconductor switches that connect the load to the two power rails when the voltage at the PWM port is greater than the value of the **Enable threshold voltage** parameter on the **Input Thresholds** tab. The default value is 0.1  $\Omega$ .

### Freewheeling diode on resistance

The total resistance in the freewheeling diodes that dissipate the current that flows through the motor when the voltage at the PWM port is less than the value of the **Enable threshold voltage** parameter on the **Input Thresholds** tab. The default value is 0.05  $\Omega$ .

### Measurement temperature

The temperature for which for which the resistance values on the **Bridge Parameters** tab are specified. This parameter appears only for blocks with exposed thermal port. For more information, see “Thermal Port” on page 1-701. The default value is 298.15 K.

## Temperature Dependence Tab

This tab appears only for blocks with exposed thermal port. For more information, see “Thermal Port” on page 1-701.

### Total bridge on resistance at second measurement temperature

The total effective resistance of the two semiconductor switches that connect the load to the two power rails (as described in the **Total bridge on resistance** parameter definition), quoted at the **Second measurement temperature**. The default value is 0.1  $\Omega$ .

### Freewheeling diode on resistance at second measurement temperature

The total resistance in the freewheeling diodes that dissipate the current that flows through the motor (as described in the **Total bridge on resistance** parameter

definition), quoted at the **Second measurement temperature**. The default value is 0.05  $\Omega$ .

### **Second measurement temperature**

The temperature for which for which the resistance values on the **Temperature Dependence** tab are specified. The default value is 398.15 K.

## **Thermal Port Tab**

This tab appears only for blocks with exposed thermal port. For more information, see “Thermal Port” on page 1-701.

### **Thermal mass**

Thermal mass associated with the thermal port H. It represents the energy required to raise the temperature of the thermal port by one degree. The default value is 100 J/K.

### **Initial temperature**

The temperature of the thermal port at the start of simulation. The default value is 298.15 K.

## **Ports**

The block has the following ports:

+ref

Positive electrical output voltage.

-ref

Negative electrical output voltage.

PWM

Pulse-width modulated signal. The voltage is defined relative to the REF port.

REF

Floating zero volt reference.

REV

Voltage that controls when to reverse the polarity of the H-Bridge output. The voltage is defined relative to the REF port.



**BRK**

Voltage that controls when to short circuit the H-Bridge output. The voltage is defined relative to the REF port.

**H**

Thermal port. For more information, see "Thermal Port" on page 1-701.

## **Examples**

See the PWM-Controlled DC Motor and Linear Electrical Actuator with Control examples.

## Half-Bridge Driver

Abstracted representation of half-bridge driver integrated circuit

**Library:** Simscape / Electrical / Electronics and Mechatronics /  
Actuators & Drivers / Drivers



### Description

The Half-Bridge Driver block provides an abstracted representation of an integrated circuit for driving MOSFET and IGBT half-bridges. The block models input hysteresis, propagation delay, and turn-on/turn-off dynamics. Unless modeling a gate driver circuit explicitly, always use this block or the Gate Driver block to set gate-source voltage on a MOSFET block or gate-emitter voltage on an IGBT block. Do not connect a controlled voltage source directly to a semiconductor gate, because this omits the gate driver output impedance that determines switching dynamics.

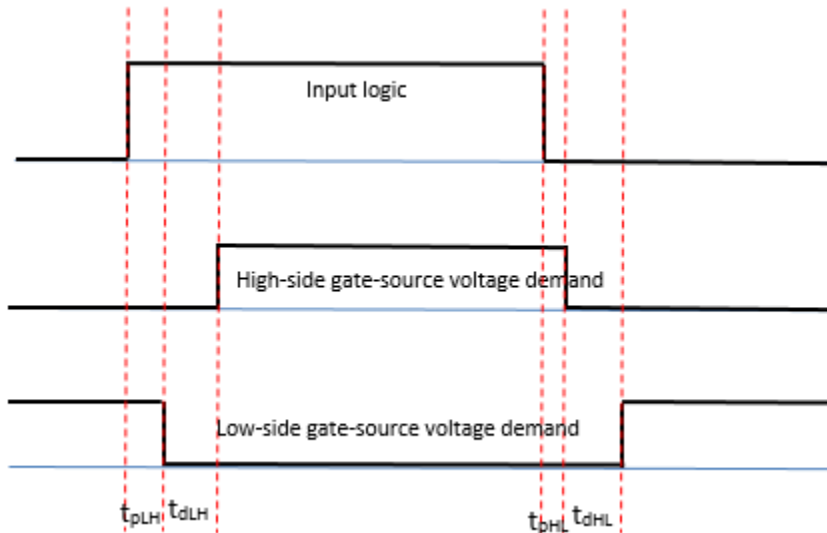
The Half-Bridge Driver block has two modeling variants, accessible by right-clicking the block in your block diagram and then selecting the appropriate option from the context menu, under **Simscape > Block choices**:

- **Physical signal input** — The driver output state is controlled by a physical signal input *u*. Use this variant if all of your controller, including PWM waveform generation, is determined by Simulink blocks. This modeling variant is the default.
- **Electrical input ports** — The driver output state is controlled by two electrical input connections, PWM and REF. Use this variant if your model has upstream analog components, such as the Controlled PWM Voltage source.

The first pair of output electrical ports, HO and HS, behave in the same way as the G and S ports of the Gate Driver block. Connect these ports to the high-side MOSFET or IGBT of the half-bridge. The second pair of ports, LO and LS, connect to the low-side MOSFET or IGBT of the half-bridge. They behave in a similar way, except that their logic is inverted with respect to that of the high side.

The diagram shows the timing properties for the half-bridge driver, where:

- $t_{pLH}$  is low-side propagation delay when the input logic goes from 0 to 1.
- $t_{dLH}$  is high-side dead time when the input logic goes from 0 to 1.
- $t_{pHL}$  is high-side propagation delay when the input logic goes from 1 to 0.
- $t_{dHL}$  is low-side dead time when the input logic goes from 1 to 0.



## Faults

You can insert a fault into one or both of the outputs at a specified simulation time. The fault options are:

- Fail input fixed at logic 0
- Fail input fixed at logic 1
- Fail high side off
- Fail high side on
- Fail low side off
- Fail low side on
- Fail high and low sides off

- Fail high and low sides on

## Ports

### Input

#### **u — Control signal, unitless**

physical signal

Input physical signal that specifies the input control value.

#### **Dependencies**

Enabled for the **Physical signal input** variant of the block.

### Conserving

#### **PWM — Pulse-width modulated signal**

electrical

Electrical conserving port associated with the pulse-width modulated signal.

#### **Dependencies**

Enabled for the **Electrical input ports** variant of the block.

#### **REF — Floating zero volt reference**

electrical

Electrical conserving port associated with the floating zero volt reference.

#### **Dependencies**

Enabled for the **Electrical input ports** variant of the block.

#### **H0 — High-side gate connection**

electrical

Electrical conserving port associated with the gate on the high side of the half bridge. Connect this port to the gate of a MOSFET or IGBT block.

**HS — High-side source or emitter connection**

electrical

Electrical conserving port associated with the source or emitter on the high side of the half bridge. Connect this port to the source of a MOSFET block or the emitter of an IGBT block.

**L0 — Low-side gate connection**

electrical

Electrical conserving port associated with the gate on the low side of the half bridge. Connect this port to the gate of a MOSFET or IGBT block.

**LS — Low-side source or emitter connection**

electrical

Electrical conserving port associated with the source or emitter on the low side of the half bridge. Connect this port to the source of a MOSFET block or the emitter of an IGBT block.

## Parameters

### Input Logic

**Logic 1 input value — Signal value for logic 1**

0.7 (default)

Value of the input signal corresponding to the logic 1 level.

**Dependencies**

Enabled for the **Physical signal input** variant of the block.

**Logic 0 input value — Signal value for logic 0**

0.3 (default)

Value of the input signal corresponding to the logic 0 level.

**Dependencies**

Enabled for the **Physical signal input** variant of the block.

**Logic 1 input voltage — Voltage value for logic 1**

2.0 V (default)

Value of the input voltage corresponding to the logic 1 level.

**Dependencies**

Enabled for the **Electrical input ports** variant of the block.

**Logic 0 input voltage — Voltage value for logic 0**

0.8 V (default)

Value of the input voltage corresponding to the logic 0 level.

**Dependencies**

Enabled for the **Electrical input ports** variant of the block.

## Outputs

**On-state gate-source voltage — Demanded output voltage for on state**

15 V (default)

Demanded output voltage when the driver is in on state.

**Off-state gate-source voltage — Demanded output voltage for off state**

0 V (default)

Demanded output voltage when the driver is in off state.

## Timing

**Low-side propagation delay (logic 0->logic 1) —  $t_{pLH}$  in timing diagram**

50 ns (default)

Low-side propagation delay when the input logic goes from 0 to 1.

**High-side dead time (logic 0->logic 1) —  $t_{dLH}$  in timing diagram**

100 ns (default)

High-side dead time when the input logic goes from 0 to 1.

**High-side propagation delay (logic 1->logic 0) –  $t_{pHL}$  in timing diagram**

50 ns (default)

High-side propagation delay when the input logic goes from 1 to 0.

**Low-side dead time (logic 1->logic 0) –  $t_{dHL}$  in timing diagram**

100 ns (default)

Low-side dead time when the input logic goes from 1 to 0.

## Dynamics

**Parameterization — Select driver parameterization**

Output impedance (default) | Rise and fall times

Select the type of driver parameterization:

- Output impedance — Specify on-state and off-state gate drive resistances.
- Rise and fall times — Specify rise time, fall time, and load capacitance.

**On-state gate drive resistance — Demanded output voltage for on state**2  $\Omega$  (default)

Demanded output voltage when the driver is in on state.

**Dependencies**

Enabled when the **Parameterization** parameter is set to Output impedance.

**Off-state gate drive resistance — Demanded output voltage for off state**2  $\Omega$  (default)

Demanded output voltage when the driver is in off state.

**Dependencies**

Enabled when the **Parameterization** parameter is set to Output impedance.

**Rise time — Driver rise time**

20 ns (default)

Driver rise time from 10% to 90%.

## Dependencies

Enabled when the **Parameterization** parameter is set to Rise and fall times.

## Fall time — Driver fall time

20 ns (default)

Driver fall time from 90% to 10%.

## Dependencies

Enabled when the **Parameterization** parameter is set to Rise and fall times.

## Load capacitance for rise and fall times — Driver load capacitance

10 nF (default)

Driver load capacitance.

## Dependencies

Enabled when the **Parameterization** parameter is set to Rise and fall times.

## Faults

### Enable faults — Select Yes to enable faults modeling

No (default) | Yes

Select Yes to enable faults modeling. The associated parameters in the **Faults** section become visible to let you specify time to fail and the failure mode.

### Simulation time for fault event — Time before entering faulted state

1 s (default)

Set the simulation time at which you want the block to enter the faulted state.

## Dependencies

Enabled when the **Enable faults** parameter is set to Yes.

### Failure mode — State of driver after failure

Fail input fixed at logic 0 (default) | Fail input fixed at logic 1 | Fail high side off | Fail high side on | Fail low side off | Fail low side on | Fail high and low sides off | Fail high and low sides on



Select the driver state after the failure.

**Dependencies**

Enabled when the **Enable faults** parameter is set to Yes.

**See Also**

Controlled PWM Voltage | Gate Driver | N-Channel IGBT | N-Channel MOSFET | P-Channel MOSFET

**Introduced in R2017b**

## Hybrid Excitation Synchronous Machine

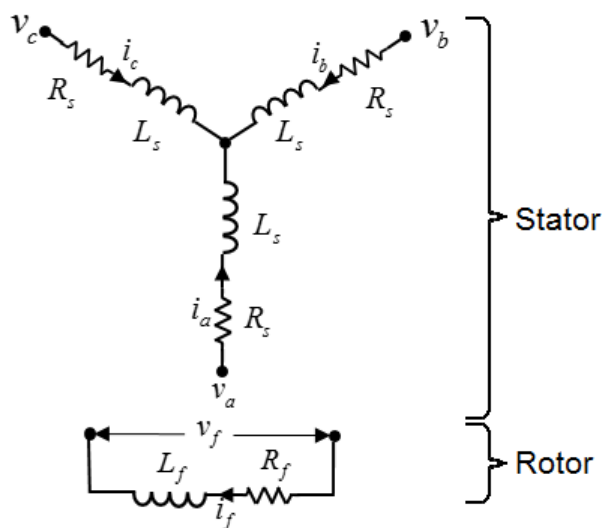
Hybrid excitation synchronous machine with three-phase wye-wound stator

**Library:** Simscape / Electrical / Power Systems / Machines / Permanent Magnet Rotor



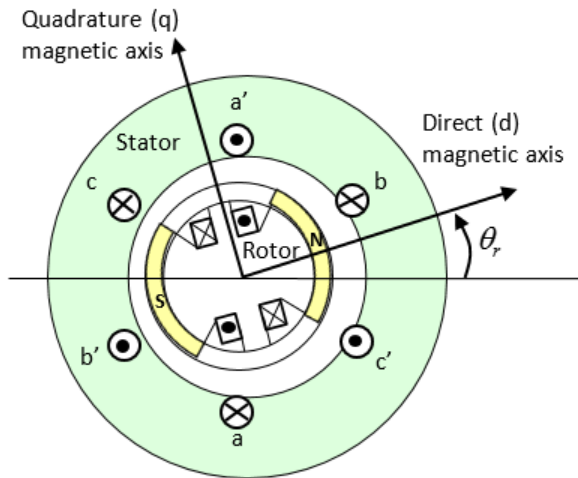
### Description

The Hybrid Excitation Synchronous Machine block represents a hybrid excitation synchronous machine with a three-phase wye-wound stator. Permanent magnets and excitation windings provide the machine excitation. The figure shows the equivalent electrical circuit for the stator and rotor windings.



## Motor Construction

The diagram shows the motor construction with a single pole-pair on the rotor. For the axes convention, when rotor mechanical angle  $\theta_r$  is zero, the  $a$ -phase and permanent magnet fluxes are aligned. The block supports a second rotor axis definition for which rotor mechanical angle is defined as the angle between the  $a$ -phase magnetic axis and the rotor  $q$ -axis.



## Equations

Voltages across the stator windings are defined by

$$\begin{bmatrix} v_a \\ v_b \\ v_c \end{bmatrix} = \begin{bmatrix} R_s & 0 & 0 \\ 0 & R_s & 0 \\ 0 & 0 & R_s \end{bmatrix} \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix} + \begin{bmatrix} \frac{d\psi_a}{dt} \\ \frac{d\psi_b}{dt} \\ \frac{d\psi_c}{dt} \end{bmatrix},$$

where:

- $v_a$ ,  $v_b$ , and  $v_c$  are the individual phase voltages across the stator windings.
- $R_s$  is the equivalent resistance of each stator winding.
- $i_a$ ,  $i_b$ , and  $i_c$  are the currents flowing in the stator windings.
- 
- $\frac{d\psi_a}{dt}$ ,  $\frac{d\psi_b}{dt}$ , and  $\frac{d\psi_c}{dt}$  are the rates of change of magnetic flux in each stator winding.

The voltage across the field winding is expressed as

$$v_f = R_f i_f + \frac{d\psi_f}{dt},$$

where:

- $v_f$  is the individual phase voltage across the field winding.
- $R_f$  is the equivalent resistance of the field winding.
- $i_f$  is the current flowing in the field winding.
- $\frac{d\psi_f}{dt}$  is the rate of change of magnetic flux in the field winding.

The permanent magnet, excitation winding, and the three star-wound stator windings contribute to the flux linking each winding. The total flux is defined by

$$\begin{bmatrix} \psi_a \\ \psi_b \\ \psi_c \end{bmatrix} = \begin{bmatrix} L_{aa} & L_{ab} & L_{ac} \\ L_{ba} & L_{bb} & L_{bc} \\ L_{ca} & L_{cb} & L_{cc} \end{bmatrix} \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix} + \begin{bmatrix} \psi_{am} \\ \psi_{bm} \\ \psi_{cm} \end{bmatrix} + \begin{bmatrix} L_{amf} \\ L_{bmf} \\ L_{cmf} \end{bmatrix} i_f,$$

where:

- $\psi_a$ ,  $\psi_b$ , and  $\psi_c$  are the total fluxes linking each stator winding.
- $L_{aa}$ ,  $L_{bb}$ , and  $L_{cc}$  are the self-inductances of the stator windings.
- $L_{ab}$ ,  $L_{ac}$ ,  $L_{ba}$ ,  $L_{bc}$ ,  $L_{ca}$ , and  $L_{cb}$  are the mutual inductances of the stator windings.
- $\psi_{am}$ ,  $\psi_{bm}$ , and  $\psi_{cm}$  are the magnetization fluxes linking the stator windings.
- $L_{amf}$ ,  $L_{bmf}$ , and  $L_{cmf}$  are the mutual inductances of the field winding.

The inductances in the stator windings are functions of rotor electrical angle and are defined by

$$\theta_e = N \theta_r,$$

$$L_{aa} = L_s + L_m \cos(2\theta_e),$$

$$L_{bb} = L_s + L_m \cos(2(\theta_e - 2\pi/3)),$$

$$L_{cc} = L_s + L_m \cos(2(\theta_e + 2\pi/3)),$$

$$L_{ab} = L_{ba} = -M_s - L_m \cos(2(\theta_e + \pi/6)),$$

$$L_{bc} = L_{cb} = -M_s - L_m \cos(2(\theta_e + \pi/6 - 2\pi/3)),$$

$$L_{ca} = L_{ac} = -M_s - L_m \cos(2(\theta_e + \pi/6 + 2\pi/3)),$$

where:

- $N$  is the number of rotor pole pairs.
- $\theta_r$  is the rotor mechanical angle.
- $\theta_e$  is the rotor electrical angle.
- $L_s$  is the stator self-inductance per phase. This value is the average self-inductance of each of the stator windings.
- $L_m$  is the stator inductance fluctuation. This value is the amplitude of the fluctuation in self-inductance and mutual inductance with changing rotor angle.
- $M_s$  is the stator mutual inductance. This value is the average mutual inductance between the stator windings.

The magnetization flux linking winding,  $a-a'$  is a maximum when  $\theta_r = 0^\circ$  and zero when  $\theta_r = 90^\circ$ . Therefore:

$$\Psi_m = \begin{bmatrix} \Psi_{am} \\ \Psi_{bm} \\ \Psi_{cm} \end{bmatrix} = \begin{bmatrix} \Psi_m \cos \theta_r \\ \Psi_m \cos(\theta_r - 2\pi/3) \\ \Psi_m \cos(\theta_r + 2\pi/3) \end{bmatrix},$$

$$L_{mf} = \begin{bmatrix} L_{amf} \\ L_{bmf} \\ L_{cmf} \end{bmatrix} = \begin{bmatrix} L_{mf} \cos \theta_r \\ L_{mf} \cos(\theta_r - 2\pi/3) \\ L_{mf} \cos(\theta_r + 2\pi/3) \end{bmatrix},$$

and

$$\Psi_f = L_f i_f + L_{mf}^T \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix},$$

where:

- $\Psi_m$  is the linked motor flux.
- $L_{mf}$  is the mutual field armature inductance.
- $\Psi_f$  is the flux linking the field winding.
- $L_f$  is the field winding inductance.
- $[L_{mf}]^T$  is the transform of the  $L_{mf}$  vector, that is,

$$[L_{mf}]^T = \begin{bmatrix} L_{amf} \\ L_{bmf} \\ L_{cmf} \end{bmatrix}^T = [L_{amf} \quad L_{bmf} \quad L_{cmf}].$$

## Simplified Equations

Applying the Park transformation to the block electrical defining equations produces an expression for torque that is independent of rotor angle.

The Park transformation is defined by

$$P = \frac{2}{3} \begin{bmatrix} \cos \theta_e & \cos(\theta_e - 2\pi/3) & \cos(\theta_e + 2\pi/3) \\ -\sin \theta_e & -\sin(\theta_e - 2\pi/3) & -\sin(\theta_e + 2\pi/3) \\ 0.5 & 0.5 & 0.5 \end{bmatrix}.$$

Applying the Park transformation to the first two electrical defining equations produces equations that define the block behavior:

$$v_d = R_s i_d + L_d \frac{di_d}{dt} + L_{mf} \frac{di_f}{dt} - N \omega i_q L_q,$$

$$v_q = R_s i_q + L_q \frac{di_q}{dt} + N \omega (i_d L_d + \psi_m + i_f L_{mf}),$$

$$v_0 = R_s i_0 + L_0 \frac{di_0}{dt},$$

$$v_f = R_f i_f + L_f \frac{di_f}{dt} + \frac{3}{2} L_{mf} \frac{di_d}{dt},$$

$$T = \frac{3}{2} N (i_q (i_d L_d + \psi_m + i_f L_{mf}) - i_d i_q L_q),$$

and

$$J \frac{d\omega}{dt} = T - T_L - B_m \omega.$$

where:

- $v_d$ ,  $v_q$ , and  $v_0$  are the  $d$ -axis,  $q$ -axis, and zero-sequence voltages. These voltages are defined by

$$\begin{bmatrix} v_d \\ v_q \\ v_0 \end{bmatrix} = P \begin{bmatrix} v_a \\ v_b \\ v_c \end{bmatrix}.$$

- $i_d$ ,  $i_q$ , and  $i_0$  are the  $d$ -axis,  $q$ -axis, and zero-sequence currents, defined by

$$\begin{bmatrix} i_d \\ i_q \\ i_0 \end{bmatrix} = P \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix}.$$

- $L_d$  is the stator  $d$ -axis inductance.  $L_d = L_s + M_s + 3/2 L_m$ .
- $\omega$  is the mechanical rotational speed.
- $L_q$  is the stator  $q$ -axis inductance.  $L_q = L_s + M_s - 3/2 L_m$ .
- $L_0$  is the stator zero-sequence inductance.  $L_0 = L_s - 2M_s$ .
- $T$  is the rotor torque. For the Hybrid Excitation Synchronous Machine block, torque flows from the machine case (block conserving port **C**) to the machine rotor (block conserving port **R**).
- $J$  is the rotor inertia.
- $T_L$  is the load torque.
- $B_m$  is the rotor damping.

## Assumptions

The block assumes that the flux distribution is sinusoidal.

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.



## Ports

### Conserving

**R — Machine rotor**

mechanical rotational

Mechanical rotational conserving port associated with the machine rotor.

**C — Machine case**

mechanical rotational

Mechanical rotational conserving port associated with the machine case.

**~ — Three-phase composite**

electrical

Expandable three-phase port associated with the stator windings.

**n — Neutral phase**

electrical

Electrical conserving port associated with the neutral phase.

**fd+ — Field winding positive terminal**

electrical

Electrical conserving port associated with the field winding positive terminal.

**fd- — Field winding negative terminal**

electrical

Electrical conserving port associated with the field winding negative terminal.

## Parameters

### Main

**Number of pole pairs — Rotor pole pairs**

6 (default) | integer

Number of permanent magnet pole pairs on the rotor.

**Permanent magnet flux linkage — Flux linkage**

0.09 Wb (default) | positive integer

Peak permanent magnet flux linkage for any of the stator windings.

**Stator parameterization — Parameterization method**

Specify Ld, Lq and L0 (default) | Specify Ls, Lm, and Ms

Method for parameterizing the stator.

**Dependencies**

Selecting Specify Ld, Lq and L0 enables these parameters:

- **Stator d-axis inductance, Ld**
- **Stator q-axis inductance, Lq**
- **Stator zero-sequence inductance, L0**

Selecting Specify Ls, Lm, and Ms enables these parameters:

- **Stator self-inductance per phase, Ls**
- **Stator inductance fluctuation, Lm**
- **Stator mutual inductance, Ms**

**Stator d-axis inductance, Ld — Inductance**

0.0031 H (default)

Direct-axis inductance of the machine stator.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify Ld, Lq and L0.

**Stator q-axis inductance, Lq — Inductance**

0.0045 H (default)

Quadrature-axis inductance of the machine stator.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify Ld, Lq and L0.

**Stator zero-sequence inductance,  $L_0$  – Inductance**

0.0006 H (default)

Zero-axis inductance for the machine stator.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify  $L_d$ ,  $L_q$  and  $L_0$ .

**Stator self-inductance per phase,  $L_s$  – Inductance**

0.0027 H (default)

Average self-inductance of the three stator windings.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify  $L_s$ ,  $L_m$ , and  $M_s$ .

**Stator inductance fluctuation,  $L_m$  – Inductance**

-0.0005 H (default)

Amplitude of the fluctuation in self-inductance and mutual inductance with the rotor angle.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify  $L_s$ ,  $L_m$ , and  $M_s$ .

**Stator mutual inductance,  $M_s$  – Inductance**

0.0011 H (default)

Average mutual inductance between the stator windings.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify  $L_s$ ,  $L_m$ , and  $M_s$ .

**Field winding inductance,  $L_f$  – Inductance**

0.06 H (default)

Inductance of the field winding.

**Mutual field armature inductance,  $L_{mf}$  – Inductance**

0.0067 H (default)

Armature-field mutual inductance.

**Stator resistance per phase,  $R_s$  — Resistance**

0.7 Ohm (default)

Resistance of each of the stator windings.

**Field winding resistance,  $R_f$  — Resistance**

2.85 Ohm (default)

Resistance of the field winding.

**Zero sequence — Option to neglect zero-sequence terms**

Include (default) | Exclude

Option to neglect zero-sequence terms. Choices are:

- **Include** — Include zero-sequence terms. To prioritize model fidelity, use this default setting. Using this option results in an error for simulations that use the Partitioning solver. For more information, see “Increase Simulation Speed Using the Partitioning Solver” (Simscape).
- **Exclude** — Exclude zero-sequence terms. To prioritize simulation speed for desktop simulation or real-time deployment, select this option.

**Rotor angle definition — Angle**

Angle between the a-phase magnetic axis and the d-axis (default) | Angle between the a-phase magnetic axis and the q-axis

Reference point for the rotor angle measurement. If you select the default value, the rotor and *a*-phase fluxes are aligned for a zero-rotor angle. Otherwise, an *a*-phase current generates the maximum torque value for a zero-rotor angle.

**Mechanical**

**Rotor inertia — Inertia**

0.01 kg\*m<sup>2</sup> (default)

Inertia of the rotor attached to mechanical translational port **R**.

**Rotor Damping — Damping**

0 N\*m/(rad/s) (default)

Rotary damping.

## References

- [1] Kundur, P. *Power System Stability and Control*. New York, NY: McGraw Hill, 1993.
- [2] Mbayed, R. *Analysis of Faulted Power Systems*. Hoboken, NJ: Wiley-IEEE Press, 1995.
- [3] Anderson, P. M. *Contribution to the Control of the Hybrid Excitation Synchronous Machine for Embedded Applications*. Universite de Cergy Pontoise, 2012.
- [4] Luo, X. and T. A. Lipo. "A Synchronous/Permanent Magnet Hybrid AC Machine." *IEEE Transactions of Energy Conversion*. Vol. 15, No 2 (2000), pp. 203-210.

## See Also

Brushless DC Motor | Permanent Magnet Synchronous Motor | Switched Reluctance Machine | Synchronous Machine Field Circuit (SI) | Synchronous Machine Field Circuit (pu) | Synchronous Machine Measurement | Synchronous Reluctance Machine

**Introduced in R2017b**

# Ideal Semiconductor Switch

Ideal Semiconductor Switch



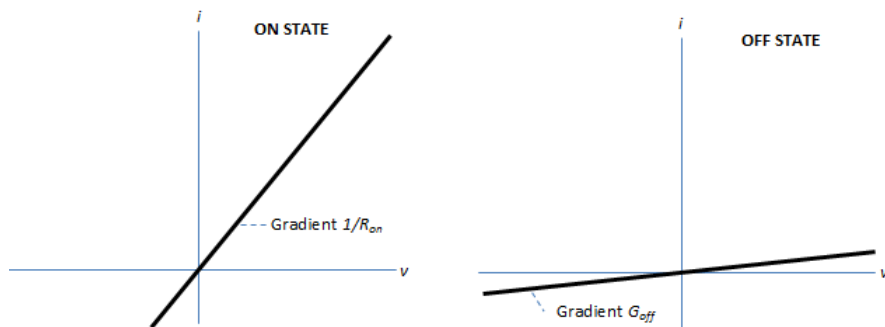
## Library

Simscape / Electrical / Power Systems / Semiconductors / Fundamental Components

## Description

The Ideal Semiconductor Switch block models an ideal semiconductor switching device.

The figure shows a typical i-v characteristic for an ideal semiconductor switch.



If the gate-cathode voltage exceeds the specified threshold voltage, the ideal semiconductor switch is in the on state. Otherwise the device is in the off state.

In the on state, the anode-cathode path behaves like a linear resistor with on-resistance  $R_{on}$ .

In the off state, the anode-cathode path behaves like a linear resistor with a low off-state conductance  $G_{off}$ .

Using the Integral Diode tab of the block dialog box, you can include an integral cathode-anode diode. An integral diode protects the semiconductor device by providing a conduction path for reverse current. An inductive load can produce a high reverse-voltage spike when the semiconductor device suddenly switches off the voltage supply to the load.

The table shows you how to set the **Integral protection diode** parameter based on your goals.

Goal	Value to Select	Block Behavior
Prioritize simulation speed.	Protection diode with no dynamics	The block includes an integral copy of the Diode block. To parameterise the internal Diode block, use the <b>Protection</b> parameters.
Precisely specify reverse-mode charge dynamics.	Protection diode with charge dynamics	The block includes an integral copy of the Commutation Diode block. To parameterise the internal Commutation Diode block, use the <b>Protection</b> parameters.

## Parameters

- “Main Tab” on page 1-729
- “Integral Diode Tab” on page 1-730

### Main Tab

#### On-state resistance

Anode-cathode resistance when the device is on. The default value is 0.001 Ohm.

#### Off-state conductance

Anode-cathode conductance when the device is off. The value must be less than  $1/R$ , where  $R$  is the value of **On-state resistance**. The default value is  $1e-6$  1/Ohm.

### **Threshold voltage, Vth**

Gate-cathode voltage threshold. The device turns on when the gate-cathode voltage is above this value. The default value is 0.5 V.

## **Integral Diode Tab**

### **Integral protection diode**

Specify whether the block includes an integral protection diode. The default value is None.

If you want to include an integral protection diode, there are two options:

- Protection diode with no dynamics
- Protection diode with charge dynamics

When you select **Protection diode with no dynamics**, additional parameters appear.

### **Additional Parameters for Protection diode with no dynamics**

#### **Forward voltage**

Minimum voltage required across the + and - block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**. The default value is 0.8 V.

#### **On resistance**

Rate of change of voltage versus current above the **Forward voltage**. The default value is 0.001 Ohm.

#### **Off conductance**

Conductance of the reverse-biased diode. The default value is  $1e-5$  1/Ohm.

For more information on these parameters, see Diode.

When you select **Protection diode with charge dynamics**, additional parameters appear.



## **Additional Parameters for Protection diode with charge dynamics**

### **Forward voltage**

Minimum voltage required across the + and - block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**. The default value is 0.8 V.

### **On resistance**

Rate of change of voltage versus current above the **Forward voltage**. The default value is 0.001 Ohm.

### **Off conductance**

Conductance of the reverse-biased diode. The default value is  $1e-5$  1/Ohm.

### **Junction capacitance**

Diode junction capacitance. The default value is 50 nF.

### **Peak reverse current, iRM**

Peak reverse current measured by an external test circuit. This value must be less than zero. The default value is -235 A.

### **Initial forward current when measuring iRM**

Initial forward current when measuring peak reverse current. This value must be greater than zero. The default value is 300 A.

### **Rate of change of current when measuring iRM**

Rate of change of current when measuring peak reverse current. This value must be less than zero. The default value is -50 A/ $\mu$ s.

### **Reverse recovery time parameterization**

Determines how you specify reverse recovery time in the block. The default value is Specify reverse recovery time directly.

If you select Specify stretch factor or Specify reverse recovery charge, you specify a value that the block uses to derive the reverse recovery time. For more information on these options, see “Alternatives to Specifying trr Directly” on page 1-339.

### **Reverse recovery time, trr**

Interval between the time when the current initially goes to zero (when the diode turns off) and the time when the current falls to less than 10% of the peak reverse current. The default value is 15  $\mu$ s.

This parameter is visible only if you set **Reverse recovery time parameterization** to **Specify reverse recovery time directly**.

The value of the **Reverse recovery time, trr** parameter must be greater than the value of the **Peak reverse current, iRM** parameter divided by the value of the **Rate of change of current when measuring iRM** parameter.

#### **Reverse recovery time stretch factor**

Value that the block uses to calculate **Reverse recovery time, trr**. This value must be greater than 1. The default value is 3.

This parameter is visible only if you set **Reverse recovery time parameterization** to **Specify stretch factor**.

Specifying the stretch factor is an easier way to parameterize the reverse recovery time than specifying the reverse recovery charge. The larger the value of the stretch factor, the longer it takes for the reverse recovery current to dissipate.

#### **Reverse recovery charge, Qrr**

Value that the block uses to calculate **Reverse recovery time, trr**. Use this parameter if the data sheet for your diode device specifies a value for the reverse recovery charge instead of a value for the reverse recovery time.

The reverse recovery charge is the total charge that continues to dissipate when the

diode turns off. The value must be less than  $-\frac{i_{RM}^2}{2a}$ ,

where:

- $i_{RM}$  is the value specified for **Peak reverse current, iRM**.
- $a$  is the value specified for **Rate of change of current when measuring iRM**.

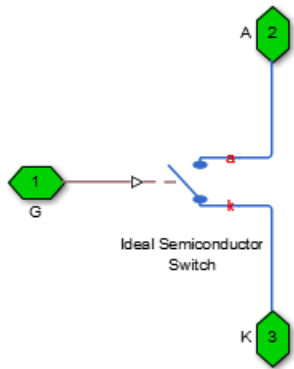
The default value is 1500  $\mu$ As.

The parameter is visible only if you set **Reverse recovery time parameterization** to **Specify reverse recovery charge**.

For more information on these parameters, see Commutation Diode.

## Ports

This figure shows the block port names.



G

Port associated with the gate terminal. You can set the port to either a physical signal or electrical port.

A

Electrical conserving port associated with the anode terminal.

K

Electrical conserving port associated with the cathode terminal.

## See Also

Commutation Diode | Diode (Piecewise Linear) | IGBT | Ideal Semiconductor Switch | MOSFET | Thyristor (Piecewise Linear)

## Topics

“Switch Between Physical Signal and Electrical Ports”

**Introduced in R2013b**

# IGBT

Insulated-Gate Bipolar Transistor

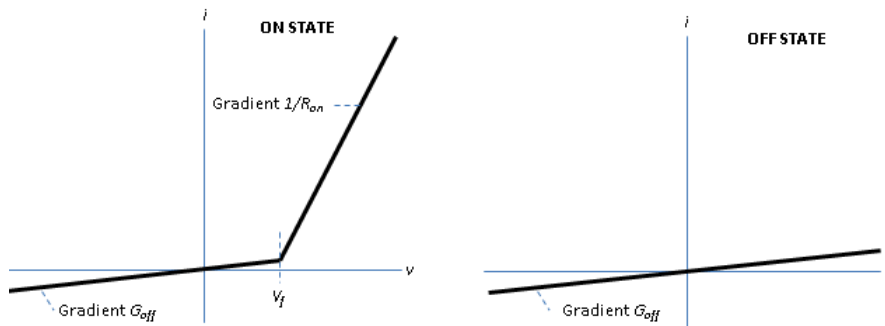


## Library

Simscape / Electrical / Power Systems / Semiconductors / Fundamental Components

## Description

The IGBT block models an insulated-gate bipolar transistor (IGBT). The I-V characteristic of an IGBT is such that if the gate-emitter voltage exceeds the specified threshold voltage,  $V_{th}$ , the IGBT is in the on state. Otherwise, the device is in the off state.



In the on state, the collector-emitter path behaves like a linear diode with forward-voltage drop,  $V_f$ , and on-resistance,  $R_{on}$ .

In the off state, the collector-emitter path behaves like a linear resistor with a low off-state conductance value,  $G_{off}$ .

The defining Simscape equations for the block are:

```

if (v>Vf)&&(G>Vth)
    i == (v - Vf*(1-Ron*Goff))/Ron;
else
    i == v*Goff;
end

```

where:

- $v$  is the collector-emitter voltage.
- $V_f$  is the forward voltage.
- $G$  is the gate-emitter voltage.
- $V_{th}$  is the threshold voltage.
- $i$  is the collector-emitter current.
- $R_{on}$  is the on-state resistance.
- $G_{off}$  is the off-state conductance.

## Integral Protection Diode Option

Using the Integral Diode tab of the block dialog box, you can include an integral emitter-collector diode. An integral diode protects the semiconductor device by providing a conduction path for reverse current. An inductive load can produce a high reverse-voltage spike when the semiconductor device suddenly switches off the voltage supply to the load.

Set the **Integral protection diode** parameter based on your goal.

Goal	Value to Select	Block Behavior
Prioritize simulation speed.	Protection diode with no dynamics	The block includes an integral copy of the Diode block. To parameterise the internal Diode block, use the <b>Protection</b> parameters.

Goal	Value to Select	Block Behavior
Precisely specify reverse-mode charge dynamics.	Protection diode with charge dynamics	The block includes an integral copy of the Commutation Diode block. To parameterise the internal Commutation Diode block, use the <b>Protection</b> parameters.

## Modeling Variants

The block provides four modeling variants. To select the desired variant, right-click the block in your model. From the context menu, select **Simscape > Block choices**, and then one of these variants:

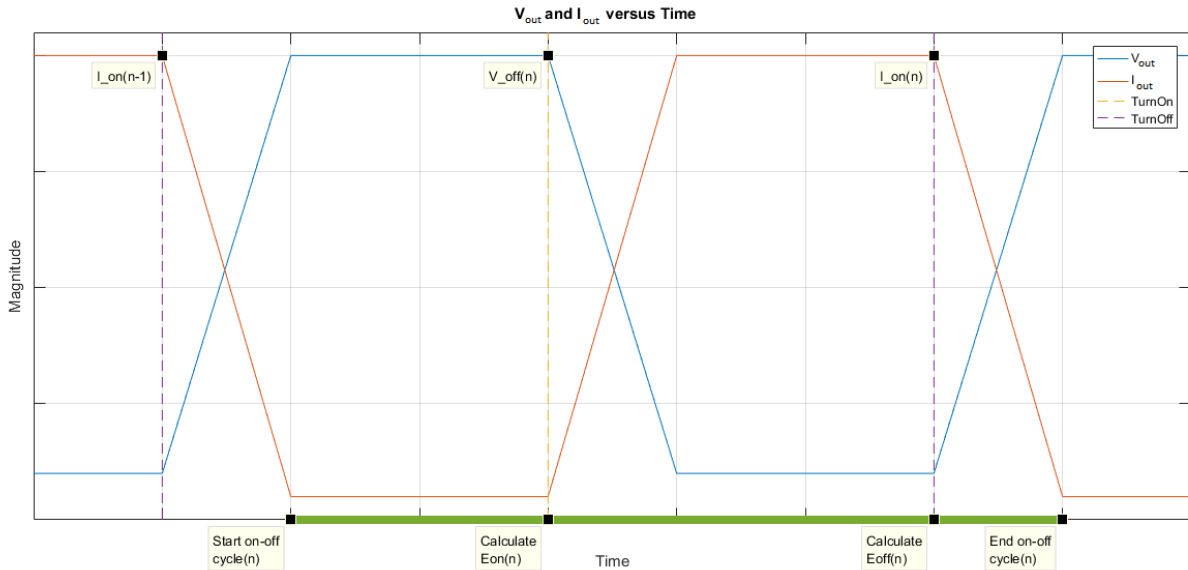
- **PS Control Port** — Contains a physical signal port that is associated with the gate terminal. This variant is the default.
- **Electrical Control Port** — Contains an electrical conserving port that is associated with the gate terminal.
- **PS Control Port | Thermal Port** — Contains a thermal port and a physical signal port that is associated with the gate terminal.
- **Electrical Control Port | Thermal Port** — Contains a thermal port and an electrical conserving port that is associated with the gate terminal.

The variants of this block without the thermal port do not simulate heat generation in the device.

The variants with the thermal port allow you to model the heat that switching events and conduction losses generate. For numerical efficiency, the thermal state does not affect the electrical behavior of the block. The thermal port is hidden by default. To enable the thermal port, select a thermal block variant.

## Thermal Loss Equations

The figure shows an idealized representation of the output voltage,  $V_{out}$ , and the output current,  $I_{out}$ , of the semiconductor device. The interval shown includes the entire  $n$ th switching cycle, during which the block turns off and then on.



When the semiconductor turns on during the  $n$ th switching cycle, the amount of thermal energy that the device dissipates increments by a discrete amount. If you select **Voltage, current, and temperature** for the **Thermal loss dependent on** parameter, the equation for the incremental change is

$$E_{on(n)} = \frac{V_{off(n)}}{V_{off\_data}} fcn(T, I_{on(n-1)}),$$

where:

- $E_{on(n)}$  is the switch-on loss at the  $n$ th switch-on event.
- $V_{off(n)}$  is the off-state output voltage,  $V_{out}$ , just before the device switches on during the  $n$ th switching cycle.
- $V_{off\_data}$  is the **Off-state voltage for losses data** parameter value.
- $T$  is the device temperature.
- $I_{on(n-1)}$  is the on-state output current,  $I_{out}$ , just before the device switches off during the cycle that precedes the  $n$ th switching cycle.

The function  $fcn$  is a 2-D lookup table with linear interpolation and linear extrapolation:

$$E = \text{tablelookup}(T_{j\_data}, I_{out\_data}, E_{on\_data}, T, I_{on(n-1)}),$$

where:

- $T_{j\_data}$  is the **Temperature vector, Tj** parameter value.
- $I_{out\_data}$  is the **Output current vector, Iout** parameter value.
- $E_{on\_data}$  is the **Switch-on loss, Eon=fcn(Tj,Iout)** parameter value.

If you select **Voltage** and **current** for the **Thermal loss dependent on** parameter, when the semiconductor turns on during the  $n$ th switching cycle, the equation that the block uses to calculate the incremental change in the discrete amount of thermal energy that the device dissipates is

$$E_{on(n)} = \left( \frac{V_{off(n)}}{V_{off\_data}} \right) \left( \frac{I_{on(n-1)}}{I_{out\_scalar}} \right) (E_{on\_scalar})$$

where:

- $I_{out\_scalar}$  is the **Output current, Iout** parameter value.
- $E_{on\_scalar}$  is the **Switch-on loss** parameter value.

When the semiconductor turns off during the  $n$ th switching cycle, the amount of thermal energy that the device dissipates increments by a discrete amount. If you select **Voltage**, **current**, and **temperature** for the **Thermal loss dependent on** parameter, the equation for the incremental change is

$$E_{off(n)} = \frac{V_{off(n)}}{V_{off\_data}} \text{fcn}(T, I_{on(n)}),$$

where:

- $E_{off(n)}$  is the switch-off loss at the  $n$ th switch-off event.
- $V_{off(n)}$  is the off-state output voltage,  $V_{out}$ , just before the device switches on during the  $n$ th switching cycle.
- $V_{off\_data}$  is the **Off-state voltage for losses data** parameter value.
- $T$  is the device temperature.



- $I_{on(n)}$  is the on-state output current,  $I_{out}$ , just before the device switches off during the  $n$ th switching cycle.

The function  $fcn$  is a 2-D lookup table with linear interpolation and linear extrapolation:

$$E = \text{tablelookup}(T_{j\_data}, I_{out\_data}, E_{off\_data}, T, I_{on(n)}),$$

where:

- $T_{j\_data}$  is the **Temperature vector**, **Tj** parameter value.
- $I_{out\_data}$  is the **Output current vector**, **Iout** parameter value.
- $E_{off\_data}$  is the **Switch-off loss**, **Eoff=fcn(Tj,Iout)** parameter value.

If you select **Voltage** and **current** for the **Thermal loss dependent on** parameter, when the semiconductor turns off during the  $n$ th switching cycle, the equation that the block uses to calculate the incremental change in the discrete amount of thermal energy that the device dissipates is

$$E_{off(n)} = \left( \frac{V_{off(n)}}{V_{off\_data}} \right) \left( \frac{I_{on(n-1)}}{I_{out\_scalar}} \right) (E_{off\_scalar})$$

where:

- $I_{out\_scalar}$  is the **Output current**, **Iout** parameter value.
- $E_{off\_scalar}$  is the **Switch-off loss** parameter value.

If you select **Voltage**, **current**, and **temperature** for the **Thermal loss dependent on** parameter, then, for both the on state and the off state, the heat loss due to electrical conduction is

$$E_{conduction} = \int fcn(T, I_{out}) dt,$$

where:

- $E_{conduction}$  is the heat loss due to electrical conduction.
- $T$  is the device temperature.
- $I_{out}$  is the device output current.

The function *fcn* is a 2-D lookup table:

$$Q_{conduction} = \text{tablelookup}(T_{j\_data}, I_{out\_data}, I_{out\_data\_repmat} .* V_{on\_data}, T, I_{out}),$$

where:

- $T_{j\_data}$  is the **Temperature vector, Tj** parameter value.
- $I_{out\_data}$  is the **Output current vector, Iout** parameter value.
- $I_{out\_data\_repmat}$  is a matrix that contains length,  $T_{j\_data}$ , copies of  $I_{out\_data}$ .
- $V_{on\_data}$  is the **On-state voltage, Von=fcn(Tj,Iout)** parameter value.

If you select **Voltage** and **current** for the **Thermal loss dependent on** parameter, then, for both the on state and the off state, the heat loss due to electrical conduction is

$$E_{conduction} = \int (I_{out} * V_{on\_scalar}) dt,$$

where  $V_{on\_scalar}$  is the **On-state voltage** parameter value.

The block uses the **Energy dissipation time constant** parameter to filter the amount of heat flow that the block outputs. The filtering allows the block to:

- Avoid discrete increments for the heat flow output
- Handle a variable switching frequency

The filtered heat flow is

$$Q = \frac{1}{\tau} \left( \sum_{i=1}^n E_{on(i)} + \sum_{i=1}^n E_{off(i)} + E_{conduction} - \int Q dt \right),$$

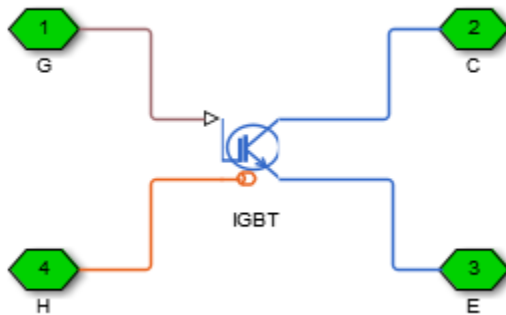
where:

- $Q$  is the heat flow from the component.
- $\tau$  is the **Energy dissipation time constant** parameter value.
- $n$  is the number of switching cycles.
- $E_{on(i)}$  is the switch-on loss at the  $i$ th switch-on event.

- $E_{off(i)}$  is the switch-off loss at the  $i$ th switch-off event.
- $E_{conduction}$  is the heat loss due to electrical conduction.
- $\int Qdt$  is the total heat previously dissipated from the component.

## Ports

The figure shows the block port names.



G

Port associated with the gate terminal. You can set the port to either a physical signal or electrical port

C

Electrical conserving port associated with the collector terminal

E

Electrical conserving port associated with the emitter terminal

H

Thermal conserving port. The thermal port is optional and is hidden by default. To enable this port, select a variant that includes a thermal port.

## Parameters

- “Main Tab” on page 1-742
- “Integral Diode Tab” on page 1-742
- “Thermal Model Tab” on page 1-745

### Main Tab

#### Forward voltage, $V_f$

Minimum voltage required across the collector and emitter block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**. The default value is 0.8 V.

#### On-state resistance

Collector-emitter resistance when the device is on. The default value is 0.001 Ohm.

#### Off-state conductance

Collector-emitter conductance when the device is off. The value must be less than  $1/R$ , where  $R$  is the value of **On-state resistance**. The default value is  $1e-5$  1/Ohm.

#### Threshold voltage, $V_{th}$

Gate-emitter voltage at which the device turns on. The default value is 6 V.

### Integral Diode Tab

#### Integral protection diode

Block integral protection diode. The default value is None.

The diodes you can select are:

- Protection diode with no dynamics
- Protection diode with charge dynamics

When you select Protection diode with no dynamics, additional parameters appear.

## **Additional Parameters for Protection diode with no dynamics**

### **Forward voltage**

Minimum voltage required across the + and - block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**. The default value is 0.8 V.

### **On resistance**

Rate of change of voltage versus current above the **Forward voltage**. The default value is 0.001 Ohm.

### **Off conductance**

Conductance of the reverse-biased diode. The default value is  $1e-5$  1/Ohm.

For more information on these parameters, see Diode.

When you select Protection diode with charge dynamics, additional parameters appear.

## **Additional Parameters for Protection diode with charge dynamics**

### **Forward voltage**

Minimum voltage required across the + and - block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**. The default value is 0.8 V.

### **On resistance**

Rate of change of voltage versus current above the **Forward voltage**. The default value is 0.001 Ohm.

### **Off conductance**

Conductance of the reverse-biased diode. The default value is  $1e-5$  1/Ohm.

### **Junction capacitance**

Diode junction capacitance. The default value is 50 nF.

### **Peak reverse current, iRM**

Peak reverse current measured by an external test circuit. This value must be less than zero. The default value is -235 A.

### **Initial forward current when measuring iRM**

Initial forward current when measuring peak reverse current. This value must be greater than zero. The default value is 300 A.

### **Rate of change of current when measuring $i_{RM}$**

Rate of change of current when measuring peak reverse current. This value must be less than zero. The default value is  $-50 \text{ A}/\mu\text{s}$ .

### **Reverse recovery time parameterization**

Determines how you specify reverse recovery time in the block. The default value is **Specify reverse recovery time directly**.

If you select **Specify stretch factor** or **Specify reverse recovery charge**, you specify a value that the block uses to derive the reverse recovery time. For more information on these options, see “Alternatives to Specifying  $t_{rr}$  Directly” on page 1-339.

### **Reverse recovery time, $t_{rr}$**

Interval between the time when the current initially goes to zero (when the diode turns off) and the time when the current falls to less than 10% of the peak reverse current. The default value is  $15 \mu\text{s}$ .

This parameter is visible only if you set **Reverse recovery time parameterization** to **Specify reverse recovery time directly**.

The value of the **Reverse recovery time,  $t_{rr}$**  parameter must be greater than the value of the **Peak reverse current,  $i_{RM}$**  parameter divided by the value of the **Rate of change of current when measuring  $i_{RM}$**  parameter.

### **Reverse recovery time stretch factor**

Value that the block uses to calculate **Reverse recovery time,  $t_{rr}$** . This value must be greater than 1. The default value is 3.

This parameter is visible only if you set **Reverse recovery time parameterization** to **Specify stretch factor**.

Specifying the stretch factor is an easier way to parameterize the reverse recovery time than specifying the reverse recovery charge. The larger the value of the stretch factor, the longer it takes for the reverse recovery current to dissipate.

### **Reverse recovery charge, $Q_{rr}$**

Value that the block uses to calculate **Reverse recovery time,  $t_{rr}$** . Use this parameter if the data sheet for your diode device specifies a value for the reverse recovery charge instead of a value for the reverse recovery time.

The reverse recovery charge is the total charge that continues to dissipate when the

diode turns off. The value must be less than  $-\frac{i_{RM}^2}{2a}$ ,

where:

- $i_{RM}$  is the value specified for **Peak reverse current, iRM**.
- $a$  is the value specified for **Rate of change of current when measuring iRM**.

The default value is 1500  $\mu$ As.

The parameter is visible only if you set **Reverse recovery time parameterization** to **Specify reverse recovery charge**.

For more information on these parameters, see Commutation Diode.

## Thermal Model Tab

The **Thermal Model** tab is enabled only when you select a block variant that includes a thermal port.

### Thermal loss dependent on

Select a parameterization method. The option that you select determines which other parameters are enabled. Options are:

- **Voltage and current** — Use scalar values to specify the output current, switch-on loss, switch-off loss, and on-state voltage data.
- **Voltage, current, and temperature** — Use vectors to specify the output current, switch-on loss, switch-off loss, on-state voltage, and temperature data. This is the default parameterization method.

### Off-state voltage for losses data

The output voltage of the device during the off state. This is the blocking voltage at which the switch-on loss and switch-off loss data are defined. The default value is 300 V.

### Energy dissipation time constant

Time constant used to average the switch-on losses, switch-off losses, and conduction losses. This value is equal to the period of the minimum switching frequency. The default value is  $1e-4$  s.

**Additional Parameters for Parameterizing by Voltage, Current, and Temperature****Temperature vector, Tj**

Temperature values at which the switch-on loss, switch-off loss, and on-state voltage are specified. Specify this parameter using a vector quantity. The default value is [ 298.15 398.15 ] K.

**Output current vector, Iout**

Output currents for which the switch-on loss, switch-off loss and on-state voltage are defined. The first element must be zero. Specify this parameter using a vector quantity. The default value is [ 0 10 50 100 200 400 600 ] A.

**Switch-on loss, Eon=fcn(Tj,Iout)**

Energy dissipated during a single switch on event. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a vector quantity. The default value is [ 0 2.9e-4 0.00143 0.00286 0.00571 0.01314 0.02286; 0 5.7e-4 0.00263 0.00514 0.01029 0.02057 0.03029 ] J.

**Switch-off loss, Eoff=fcn(Tj,Iout)**

Energy dissipated during a single switch-off event. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a vector quantity. The default value is [ 0 2.1e-4 0.00107 0.00214 0.00429 0.009859999999999999 0.01714; 0 4.3e-4 0.00197 0.00386 0.00771 0.01543 0.02271 ] J.

**On-state voltage, Von=fcn(Tj,Iout)**

Voltage drop across the device while it is in a triggered conductive state.. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a vector quantity. The default value is [ 0 1.1 1.3 1.45 1.75 2.25 2.7; 0 1 1.15 1.35 1.7 2.35 3 ] V.

**Additional Parameters for Parameterizing by Voltage and Current****Output current, Iout**

Output currents for which the switch-on loss, switch-off loss, and on-state voltage are defined. The first element must be zero. Specify this parameter using a scalar quantity. The default value is 600 A.



**Switch-on loss**

Energy dissipated during a single switch-on event. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a scalar quantity. The default value is  $0.02286$  J.

**Switch-off loss**

Energy dissipated during a single switch-off event. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a scalar quantity. The default value is  $0.01714$  J.

**On-state voltage**

Voltage drop across the block while it is in a triggered conductive state. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a scalar quantity. The default value is  $2.7$  V.

**See Also**

Commutation Diode | Diode (Piecewise Linear) | GTO | Ideal Semiconductor Switch | MOSFET | Thyristor (Piecewise Linear)

**Topics**

“Quantifying IGBT Thermal Losses”

“Simulate Thermal Losses in Semiconductors”

“Switch Between Physical Signal and Electrical Ports”

**Introduced in R2013b**

# Incandescent Lamp

Model incandescent lamp, with resistance depending on temperature



## Library

Passive Devices

## Description

The Incandescent Lamp block models an incandescent lamp, the key characteristic of which is that the resistance increases as the filament warms up.

Under the simplifying assumption that the rate of heat loss from the filament is proportional to temperature difference to ambient, the temperature of the filament is governed by

$$kt_c \frac{dT}{dt} = i^2 R - kT$$

and the filament resistance is governed by the following equation

$$R = R_0 (1 + \alpha T)$$

where:

- $R_0$  is the initial resistance at turn-on (when filament is at ambient temperature).
- $T$  is the filament temperature relative to ambient temperature.
- $\alpha$  is the resistance temperature coefficient.

- $t_c$  is the thermal time constant.
- $k$  is the heat transfer coefficient.
- $R$  is the filament resistance.
- $i$  is the filament current.

There are two parameterization options:

- If you select **Specify resistance values directly**, the block uses values that you provide for filament resistance when on and at turn-on to determine the value for the heat transfer coefficient.
- If you select **Specify currents**, the block uses values that you provide for filament current when on and at turn-on to determine the value for the heat transfer coefficient.

Optionally you can specify a simulation time at which the lamp fails by providing a finite value for the **Time at which lamp goes open circuit** parameter on the **Faults** tab. When in the open-circuit state, the lamp resistance is set to be the value of the **Open-circuit resistance** parameter.

## Parameters

- “Resistance Tab” on page 1-749
- “Dynamics Tab” on page 1-750
- “Faults Tab” on page 1-751

## Resistance Tab

### Parameterization

Select one of the following methods for block parameterization:

- **Specify resistance values directly** — Provide the values for filament resistance at turn-on and when on in steady state. The block determines the value for the heat transfer coefficient based on these values. This is the default option.
- **Specify currents** — Provide the values for filament current at turn-on and when on in steady state. The block determines the value for the heat transfer coefficient based on these values.

### **Initial resistance at turn-on**

The resistance seen by the external circuit when the lamp is initially turned on. This parameter is only visible when you select **Specify resistance values directly** for the **Parameterization** parameter. The default value is  $0.15 \Omega$ .

### **Steady-state resistance when on**

The resistance seen by the external circuit when the lamp is on and in steady state. This parameter is only visible when you select **Specify resistance values directly** for the **Parameterization** parameter. This resistance should be greater than the **Initial resistance at turn-on**. The default value is  $1 \Omega$ .

### **Inrush current at turn-on**

The current through the lamp when it is initially turned on. This parameter is only visible when you select **Specify currents** for the **Parameterization** parameter. The default value is  $70 \text{ A}$ .

### **Steady-state current when on**

The current through the lamp when it is on and in steady state. This parameter is only visible when you select **Specify currents** for the **Parameterization** parameter. This current should be less than the **Inrush current at turn-on**. The default value is  $10 \text{ A}$ .

### **Rated voltage**

The rated voltage for the lamp, and the voltage value for which the resistance or current values are provided in the on and turn-on states. The default value is  $12 \text{ V}$ .

### **Resistance temperature coefficient**

The fractional increase in resistance per unit increase in temperature. The default value is  $0.004 \text{ 1/K}$ .

## **Dynamics Tab**

### **Thermal time constant**

The first-order thermal time constant for filament temperature when the lamp is turned on or off. The default value is  $25 \text{ ms}$ .

### **Initial lamp state**

Select between **On** and **Off**. The default is **Off**.

## Faults Tab

### Time at which lamp goes open circuit

For simulation times greater than this parameter value the filament resistance becomes equal to the **Open-circuit resistance**. The default value is `inf` seconds. Specifying a finite value for this parameter lets you simulate the fault dynamics when the bulb burns out.

### Open-circuit resistance

The value of the filament resistance used when the lamp goes open-circuit. The default value is  $1e6 \Omega$ .

## Ports

The block has the following ports:

+

Positive electrical port

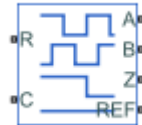
-

Negative electrical port

## Incremental Shaft Encoder

Model device that converts information about angular shaft position into electrical pulses

**Library:** Simscape / Electrical / Electronics and Mechatronics / Sensors  
Simscape / Electrical / Power Systems / Sensors



### Description

The Incremental Shaft Encoder block represents a device that converts information about the angular position of a shaft into electrical pulses. The block produces  $N$  pulses on ports **A** and **B** per shaft revolution, where  $N$  is the value you specify for the **Pulses per revolution** parameter. Pulses *A* and *B* are 90 degrees out of phase. If the shaft rotates in a positive direction, then *A* leads *B*. The block produces a single index pulse on port **Z** once per revolution. The *Z*-pulse positive transition always coincides with an *A*-pulse positive transition, and *Z*-pulse length is equal to the length for the *A* and *B* pulses. The voltages at ports **A**, **B**, and **Z** are defined relative to the **Ref** reference port voltage.

Use this block if you need to model the shaft encoder signals, either to support development of a decoding algorithm or to include the quantization effects. Otherwise, use the Ideal Rotational Motion Sensor block from the Simscape Foundation library.

### Assumptions and Limitations

- The Incremental Shaft Encoder block is not linearizable. For control design studies that require model linearization, use the Ideal Rotational Motion Sensor block from the Simscape Foundation library.

## Ports

### Conserving

#### **R — Rotational velocity**

mechanical rotational

Mechanical rotational conserving port associated with the sensor positive probe.

#### **C — Rotational velocity**

mechanical rotational

Mechanical rotational conserving port associated with the sensor negative (reference) probe.

#### **A — Voltage**

electrical

Encoded electrical output.

#### **B — Voltage**

electrical

Encoded electrical output.

#### **Z — Index or synchronization**

electrical

Index, or synchronization, electrical output.

#### **Ref — Voltage**

electrical

Floating zero-volt reference.

## Parameters

#### **Pulses per revolution — Pulse count**

2 (default)

Number of pulses produced on each of the *A* and *B* phases per revolution of the shaft.

**Output voltage amplitude — Voltage**

5 V (default)

Amplitude of the shaft encoder output voltage when the output is high.

**Index pulse offset relative to shaft initial angle — Position**

0 deg (default)

Offset of the index pulse *Z* relative to the angle of the shaft at the start of the simulation. This parameter lets you set the initial location of the index pulse.

## See Also

**Simscape Blocks**

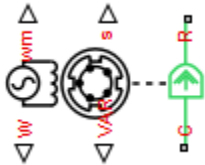
Ideal Rotational Motion Sensor

**Introduced in R2017b**



# Induction Motor

Model induction motor powered by ideal AC supply

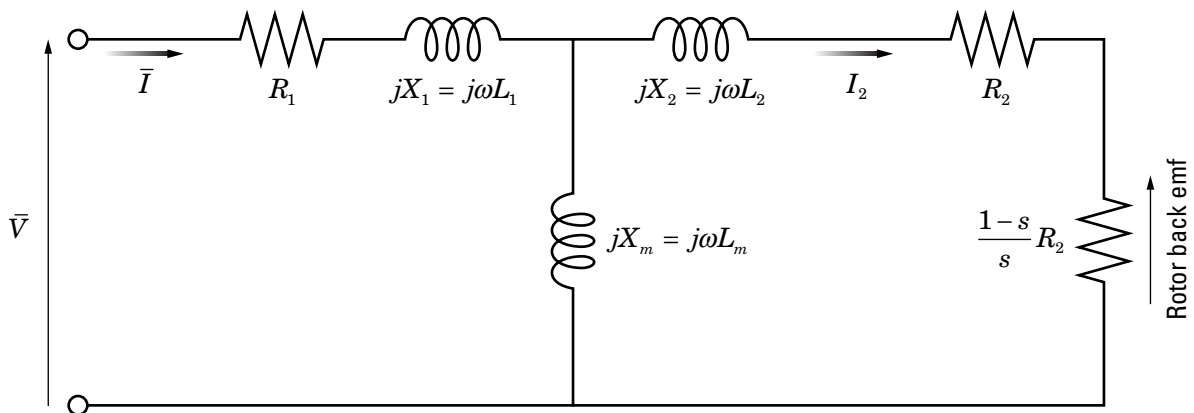


## Library

Rotational Actuators

## Description

The Induction Motor block represents the electrical and torque characteristics of an induction motor powered by an ideal AC supply. The following figure shows the equivalent circuit model of the Induction Motor block.



In the figure:

- $R_1$  is the stator resistance.
- $R_2$  is the rotor resistance with respect to the stator.
- $L_1$  is the stator inductance.
- $L_2$  is the rotor inductance with respect to the stator.
- $L_m$  is magnetizing inductance.
- $s$  is the rotor slip.
- $\bar{V}$  and  $\bar{I}$  are the sinusoidal supply voltage and current phasors.

Rotor slip  $s$  is defined in terms of the mechanical rotational speed  $\omega_m$ , the number of pole pairs  $p$ , and the electrical supply frequency  $\omega$  by

$$s = 1 - \frac{p\omega_m}{\omega}$$

This means that the slip is one when starting, and zero when running synchronously with the supply frequency.

For an  $n$ -phase induction motor the torque-speed relationship is given by:

$$T = \frac{npR_2}{s\omega} \frac{V_{rms}^2}{\left(R_1 + R_2 + \frac{1-s}{s}R_2\right)^2 + (X_1 + X_2)^2}$$

where:

- $V_{rms}$  is the line-neutral supply voltage for a star-configuration induction motor, and the line-to-line voltage for a delta-configuration induction motor.
- $n$  is the number of phases.

You can parameterize this block in terms of the preceding equivalent circuit model parameters or in terms of the motor ratings the block uses to derive these parameters.

This block produces a positive torque acting from the mechanical C to R ports.

## Thermal Ports

The block has two optional thermal ports, one per winding, hidden by default. To expose the thermal ports, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal ports on the block icon, and adds the **Temperature Dependence** and **Thermal Port** tabs to the block dialog box. These tabs are described further on this reference page.

Use the thermal ports to simulate the effects of copper resistance losses that convert electrical power to heat. For more information on using thermal ports in actuator blocks, see “Simulating Thermal Effects in Rotational and Translational Actuators”.

## Basic Assumptions and Limitations

The model is based on the following assumptions:

- The block does not model the starting mechanism for a single-phase induction motor.
- When you parameterize the block by motor ratings, the block derives the equivalent circuit model parameters by assuming that the effect of the magnetizing inductance  $L_m$  is negligible, and the magnetizing inductance is not included in the simulated component.

## Parameters

- “Electrical Torque Tab” on page 1-757
- “Power Supply Tab” on page 1-760
- “Mechanical Tab” on page 1-760
- “Temperature Dependence Tab” on page 1-760
- “Thermal Port Tab” on page 1-761

## Electrical Torque Tab

### Model parameterization

Select one of the following methods for block parameterization:

- **By motor ratings** — Provide electrical torque parameters that the block converts to an equivalent circuit model of the motor assuming that the effect of the magnetizing inductance  $L_m$  is negligible. This is the default method.
- **By equivalent circuit parameters** — Provide electrical parameters for an equivalent circuit model of the motor.

### **Stator resistance R1**

Resistance of the stator winding. The default value is  $1 \Omega$ . This parameter is only visible when you select **By equivalent circuit parameters** for the **Model parameterization** parameter.

### **Rotor resistance R2**

Resistance of the rotor, specified with respect to the stator. The default value is  $1 \Omega$ . This parameter is only visible when you select **By equivalent circuit parameters** for the **Model parameterization** parameter.

### **Stator inductance L1**

Inductance of the stator winding. The default value is  $0.02 \text{ H}$ . This parameter is only visible when you select **By equivalent circuit parameters** for the **Model parameterization** parameter.

### **Rotor inductance L2**

Inductance of the rotor, specified with respect to the stator. The default value is  $0.02 \text{ H}$ . This parameter is only visible when you select **By equivalent circuit parameters** for the **Model parameterization** parameter.

### **Magnetizing inductance Lm**

Magnetizing inductance of the stator. This parameter is only visible when you select **By equivalent circuit parameters** for the **Model parameterization** parameter. Its value is hard to estimate from motor parameters, but the effect is usually small. If you do not know its value, use a typical value of 25 times the **Stator inductance L1** value. The default value is  $0.5 \text{ H}$ .

### **Rated mechanical power**

Mechanical power the motor delivers when running at the rated speed. The default value is  $825 \text{ W}$ . This parameter is only visible when you select **By motor ratings** for the **Model parameterization** parameter.

### **Rated speed**

Speed at which the motor delivers the specified **Rated mechanical power** value. The default value is  $3.5e+03 \text{ rpm}$ . This parameter is only visible when you select **By motor ratings** for the **Model parameterization** parameter.

**Rated RMS line-to-line voltage**

Line-to-line voltage at which the motor ratings are specified. The default value is 200 V. This parameter is only visible when you select **By motor ratings** for the **Model parameterization** parameter.

**Rated supply frequency**

Frequency of the AC supply voltage at which the motor ratings are specified. The default value is 60 hertz. This parameter is only visible when you select **By motor ratings** for the **Model parameterization** parameter.

**Rated RMS line current**

Line current at which the motor delivers the specified **Rated mechanical power** value. The default value is 2.7 A. This parameter is only visible when you select **By motor ratings** for the **Model parameterization** parameter.

**R1 parameterization**

Select one of the following parameterizations for the equivalent circuit resistance,  $R_1$ , of the motor:

- **From motor efficiency** — Calculate  $R_1$  from the motor efficiency. This is the default method.
- **From power factor** — Calculate  $R_1$  from the motor power factor.
- **Use measured stator resistance R1** — Measure  $R_1$  directly.

This parameter is only visible when you select **By motor ratings** for the **Model parameterization** parameter.

**Motor efficiency (percent)**

the percentage of input power to the motor that gets delivered to the mechanical load when running at the **Rated speed** value. The default value is 95. This parameter is only visible when you select **By motor ratings** for the **Model parameterization** parameter and **From motor efficiency** for the **R1 parameterization** parameter.

**Motor power factor**

The cosine of the angle by which the supply current lags the supply voltage when running at the **Rated mechanical power** value. The default value is 0.93. This parameter is only visible when you select **By motor ratings** for the **Model parameterization** parameter and **From power factor** for the **R1 parameterization** parameter.

**Measured stator resistance R1**

the measured stator resistance. The default value is 1  $\Omega$ . This parameter is only visible when you select **By motor ratings** for the **Model parameterization**

parameter and Use measured stator resistance R1 for the **R1 parameterization** parameter.

### **Number of pole pairs**

Total number of pole pairs for the motor. The default value is 1.

### **Number of phases**

Number of supply phases. The default value is 3.

### **Stator connections**

Select one of the following motor configurations:

- **Delta configuration** — Connect the motor stator windings in delta configuration. This is the default method.
- **Star configuration** — Connect the motor stator windings in star configuration.

## **Power Supply Tab**

### **Supply RMS line-to-line voltage**

The line-to-line voltage that supplies the motor. The default value is 200 V.

### **Supply frequency**

Frequency of the AC supply voltage. The default value is 60 hertz.

## **Mechanical Tab**

### **Rotor inertia**

Rotor inertia. The default value is 0.1 kg\*m<sup>2</sup>. The value can be zero.

### **Rotor damping**

Rotor damping. The default value is 2e-06 N\*m/(rad/s). The value can be zero.

### **Initial rotor speed**

Speed of the rotor at the start of the simulation. The default value is 0 rpm.

## **Temperature Dependence Tab**

This tab appears only for blocks with exposed thermal ports. For more information, see “Thermal Ports” on page 1-757.

**Resistance temperature coefficients, [alpha\_1 alpha\_2]**

A 1 by 2 row vector defining the coefficient  $\alpha$  in the equation relating resistance to temperature, as described in “Thermal Model for Actuator Blocks”. The first element corresponds to the stator, and the second to rotor. The default value is for copper, and is [ 0.00393 0.00393 ] 1/K.

**Measurement temperature**

The temperature for which motor parameters are defined. The default value is 25 °C.

**Thermal Port Tab**

This tab appears only for blocks with exposed thermal ports. For more information, see “Thermal Ports” on page 1-757.

**Thermal masses, [M\_1 M\_2]**

A 1 by 2 row vector defining the thermal mass for the stator and rotor windings. The thermal mass is the energy required to raise the temperature by one degree. The default value is [ 100 100 ] J/K.

**Initial temperatures, [T\_A T\_B]**

A 1 by 2 row vector defining the temperature of the stator and rotor thermal ports at the start of simulation. The default value is [ 25 25 ] °C.

**Ports**

The block has the following ports:

W

Real power.

wm

Mechanical speed.

VAR

Imaginary power.

s

Motor slip.

C

Mechanical rotational conserving port.

R

Mechanical rotational conserving port.

H1

Stator thermal port. For more information, see “Thermal Ports” on page 1-757.

H2

Rotor thermal port. For more information, see “Thermal Ports” on page 1-757.

## References

[1] S.E. Lyshevski. *Electromechanical Systems, Electric Machines, and Applied Mechatronics*, CRC, 1999.

## See Also

DC Motor | Servomotor | Shunt Motor | Universal Motor



# Inductor

Inductor model including tolerance, operational limits, and fault behavior

**Library:** Simscape / Electrical / Electronics and Mechatronics /  
Passive Devices



## Description

The Inductor block lets you model linear inductors, including the following effects:

- “Tolerances” on page 1-764
- “Operating Limits” on page 1-764
- “Faults” on page 1-765

You can turn these modeling options on and off independently of each other. When all the additional options are turned off, the component behavior is identical to the Simscape Foundation library Inductor block.

In its simplest form, the Inductor block models a linear inductor, described with the following equation:

$$V = L \frac{dI}{dt}$$

where:

- $V$  is voltage.
- $L$  is inductance.
- $I$  is current.
- $t$  is time.

To model a nonlinear inductor, use the Nonlinear Inductor block.

## Tolerances

You can apply tolerances to the nominal value you provide for the **Inductance** parameter. Datasheets typically provide a tolerance percentage for a given inductor type. The table shows how the block applies tolerances and calculates inductance based on the selected **Tolerance application** option.

Option	Inductance Value
None – use nominal value	$L$
Random tolerance	Uniform distribution: $L \cdot (1 - tol + 2 \cdot tol \cdot rand)$ Gaussian distribution: $L \cdot (1 + tol \cdot randn / nSigma)$
Apply maximum tolerance value	$L \cdot (1 + tol)$
Apply minimum tolerance value	$L \cdot (1 - tol)$

In the table,

- $L$  is the **Inductance** parameter value, nominal inductance.
- $tol$  is fractional tolerance, **Inductance tolerance (%)** /100.
- $nSigma$  is the value you provide for the **Number of standard deviations for quoted tolerance** parameter.
- $rand$  and  $randn$  are standard MATLAB functions for generating uniform and normal distribution random numbers.

## Operating Limits

Inductors are typically rated with a particular saturation current, and possibly with a maximum allowable power dissipation. You can specify operating limits in terms of these values, to generate warnings or errors if the inductor is driven outside its specification.

When an operating limit is exceeded, the block can either generate a warning or stop the simulation with an error. For more information, see the “Operating Limits” on page 1-768 parameters section.

## Faults

Instantaneous changes in inductor parameters are unphysical. Therefore, when the Inductor block enters the faulted state, short-circuit and open-circuit voltages transition to their faulted values over a period of time based on this formula:

$$\text{CurrentValue} = \text{FaultedValue} - (\text{FaultedValue} - \text{UnfaultedValue}) \cdot \text{sech}(\Delta t / \tau)$$

where:

- $\Delta t$  is time since the onset of the fault condition.
- $\tau$  is user-defined time constant associated with the fault transition.

For short-circuit faults, the conductance of the short-circuit path also changes according to the  $\text{sech}(\Delta t / \tau)$  function from a small value (representing an open-circuit path) to a large value.

The block can trigger the start of fault transition:

- At a specific time
- After voltage exceeds the maximum permissible value a certain number of times
- When current exceeds the maximum permissible value for longer than a specific time interval

You can enable or disable these trigger mechanisms separately, or use them together if more than one trigger mechanism is required in a simulation. When more than one mechanism is enabled, the first mechanism to trigger the fault transition takes precedence. In other words, a component fails no more than once per simulation.

You can also choose whether to issue an assertion when a fault occurs, by using the **Reporting when a fault occurs** parameter. The assertion can take the form of a warning or an error. By default, the block does not issue an assertion.

Faultable inductors often require that you use the fixed-step local solver rather than the variable-step solver. In particular, if you model transitions to a faulted state that include short circuits, MathWorks recommends that you use the fixed-step local solver. For more information, see “Making Optimal Solver Choices for Physical Simulation” (Simscape).

## Variables

Use the **Variables** section of the block interface to set the priority and initial target values for the block variables prior to simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

The **Inductor current** variable lets you specify a high-priority target for the initial inductor current at the start of simulation.

## Ports

### Conserving

#### + — Positive terminal

electrical

Electrical conserving port associated with the inductor positive terminal.

#### - — Negative terminal

electrical

Electrical conserving port associated with the inductor negative terminal.

## Parameters

### Main

#### Inductance — Nominal inductance value

1  $\mu$ H (default)

The nominal inductance value. Inductance value must be greater than zero.

#### Tolerance (%) — Inductor tolerance, in percent

20 (default)

The inductor tolerance as defined on the manufacturer datasheet.

**Tolerance application – Select how to apply tolerance during simulation**

None – use nominal value (default) | Random tolerance | Apply maximum tolerance value | Apply minimum tolerance value

Select how to apply tolerance during simulation:

- **None – use nominal value** – The block does not apply tolerance, it uses the nominal inductance value.
- **Random tolerance** – The block applies random offset to the inductance value, within the tolerance value limit. You can choose Uniform or Gaussian distribution for calculating the random number by using the **Tolerance distribution** parameter.
- **Apply maximum tolerance value** – The inductance is increased by the specified tolerance percent value.
- **Apply minimum tolerance value** – The inductance is decreased by the specified tolerance percent value.

**Tolerance distribution – Select the distribution type**

Uniform (default) | Gaussian

Select the distribution type for random tolerance:

- **Uniform** – Uniform distribution
- **Gaussian** – Gaussian distribution

**Dependencies**

Enabled when the **Tolerance application** parameter is set to **Random tolerance**.

**Number of standard deviations for quoted tolerance – Used for calculating the Gaussian random number**

4 (default)

Number of standard deviations for calculating the Gaussian random number.

**Dependencies**

Enabled when the **Tolerance distribution** parameter is set to **Gaussian**.

**Series resistance – Equivalent series resistance of the inductor**

0  $\Omega$  (default)

Equivalent series resistance (ESR) of the inductor, as sometimes specified on manufacturer datasheets. The default value is consistent with the Simscape Foundation library Inductor block. If you model faults, specify a positive value for this parameter.

### **Parallel conductance — Parallel leakage path associated with the inductor**

1e-9 1/Ω (default)

Parallel leakage path associated with the inductor. Simulation of some circuits may require the presence of a small parallel conductance. You can also use this parameter to model the inductor core losses.

## **Operating Limits**

### **Enable operating limits — Select Yes to enable reporting when the operational limits are exceeded**

No (default) | Yes

Select Yes to enable reporting when the operational limits are exceeded. The associated parameters in the **Operating Limits** section become visible to let you select the reporting method and specify the operating limits in terms of power and current.

### **Reporting when operating limits exceeded — Select the reporting method**

Warn (default) | Error

Select what happens when an operating limit is exceeded:

- Warn — The block issues a warning.
- Error — Simulation stops with an error.

### **Dependencies**

Enabled when the **Enable operating limits** parameter is set to Yes.

### **Saturation current — Inductor saturation current**

1 A (default)

Inductor saturation current, as defined in the manufacturer datasheets. If the current exceeds this value, the core material enters saturation.

### **Dependencies**

Enabled when the **Enable operating limits** parameter is set to Yes.

**Power rating — Maximum power dissipation in the inductor**

1 W (default)

Maximum instantaneous power dissipation in the resistance and conductance elements associated with the inductor.

**Dependencies**

Enabled when the **Enable operating limits** parameter is set to Yes.

**Faults****Enable faults — Select Yes to enable faults modeling**

No (default) | Yes

Select Yes to enable faults modeling. The associated parameters in the **Faults** section become visible to let you select the reporting method and specify the trigger mechanism (temporal or behavioral). You can enable these trigger mechanisms separately or use them together.

**Reporting when a fault occurs — Choose whether to issue an assertion when a fault occurs**

None (default) | Warn | Error

Choose whether to issue an assertion when a fault occurs:

- None — The block does not issue an assertion.
- Warn — The block issues a warning.
- Error — Simulation stops with an error.

**Dependencies**

Enabled when the **Enable faults** parameter is set to Yes.

**Location of fault node (% of total turns from - terminal) — Percentage of turns in the subinductor that is in contact with the - port of the block**

50 (default)

In practice, faults are enabled by segmenting the inductor into two coupled subinductors, connected in a series. The inductance is proportional to the square of the number of turns

in the respective segment, and the series resistance of each subinductor is proportional to the number of turns in each segment. The parallel conductance spans both segments.

This parameter indicates the percentage of turns that are assigned to the subinductor that is in contact with the - port of the block. The remaining turns are assigned to the other subinductor. The default value is 50, which means that the overall inductance is divided into two equal, coupled subinductors.

### Dependencies

Enabled when the **Enable faults** parameter is set to Yes.

### Faulted coupling factor — Mutual coupling between the two subinductors

0.9999 (default)

The faulted value for the mutual coupling between the two subinductors. The differential equations governing such a construction break down in the limit of perfect coupling, so the coupling should be less than unity. A value of 0 corresponds to no coupling at all between the subinductors. Physically, this corresponds to a fault that affects the flux within the inductor core. This could be a crack in the core material, or windings coming away from the core.

The default value of this parameter is also the internal value the block uses when computing a faultable inductor in the unfaulted state. For an unfaultable inductor, there is only a single equation being solved, and this corresponds to the ideal case of perfect mutual coupling.

### Dependencies

Enabled when the **Enable faults** parameter is set to Yes.

### Short-circuit turns — Select whether fault results in one of the segments being short-circuited

No (default) | To negative terminal | To positive terminal

Select whether the fault results in one of the subinductor segments being short-circuited:

- No — The fault does not produce a short circuit.
- To negative terminal — The fault short-circuits the subinductor that is in contact with the - port of the block.
- To positive terminal — The fault short-circuits the subinductor that is in contact with the + port of the block.



### Dependencies

Enabled when the **Enable faults** parameter is set to Yes.

### Open-circuit at fault node — Select whether to apply an open-circuit fault between the segments

No (default) | Yes

Select whether to apply an open-circuit fault between the two subinductor segments. The default is No. Even with an open-circuit fault, the characteristics of the subinductors may still be related, depending on the value of the **Faulted coupling factor** parameter:

- If the coupling factor is not zero, the subinductors are galvanically isolated from each other, but they are still magnetically coupled. Physically, this corresponds to a break in the winding.
- With zero coupling factor, the subinductors are galvanically and magnetically isolated.

### Dependencies

Enabled when the **Enable faults** parameter is set to Yes.

### Ground fault — Select whether fault results in one of the segments being short-circuited

No (default) | Negative terminal side of fault node | Positive terminal side of fault node

Select whether, in case of fault, there is a path for current to flow towards the ground node:

- No — The fault does not result in a connection to ground.
- Negative terminal side of fault node — The side that is in contact with the - port of the block is connected to ground.
- Positive terminal side of fault node — The side that is in contact with the + port of the block is connected to ground.

If the **Open-circuit at fault node** parameter is set to Yes, you need to specify which side (negative or positive) is connected to ground. If there is no open circuit, the two options behave similarly. Physically, this corresponds to a breakdown in the insulation between the windings and the grounded core or chassis.

**Dependencies**

Enabled when the **Enable faults** parameter is set to Yes.

**Conductance of faulted ground path — Mutual coupling between the two subinductors**

0 1/Ω (default)

If there is a ground fault, this parameter represents the conductance of the current path to ground. For example, if the path to ground is through the core material, then specify a small conductance value depending on the core material being used. For highly conductive core material or for chassis-shorts, specify a higher conductance value.

**Dependencies**

Enabled when the **Ground fault** parameter is set to Negative terminal side of fault node or Positive terminal side of fault node.

**Fault transition time constant — Time constant for the transition to faulted state**

1 ms (default)

Time constant associated with the transition to the faulted state, as described in “Faults” on page 1-769.

**Dependencies**

Enabled when the **Enable faults** parameter is set to Yes.

**Enable temporal fault trigger — Select Yes to enable time-based fault triggering**

No (default) | Yes

Select Yes to enable time-based fault triggering. You can enable the temporal and behavioral trigger mechanisms separately or use them together.

**Dependencies**

Enabled when the **Enable faults** parameter is set to Yes.

**Simulation time for a fault event — Time before entering faulted state**

1 s (default)

Set the simulation time at which you want the block to start entering the fault state.

**Dependencies**

Enabled when the **Enable temporal fault trigger** parameter is set to Yes.

**Enable behavioral fault trigger – Select Yes to enable behavioral fault triggering**

No (default) | Yes

Select Yes to enable behavioral fault triggering. You can enable the temporal and behavioral trigger mechanisms separately or use them together.

**Dependencies**

Enabled when the **Enable faults** parameter is set to Yes.

**Maximum permissible voltage – Voltage threshold to fault transition**

100 V (default)

Define the voltage threshold to a fault transition. If the voltage value exceeds this threshold a certain number of times, specified by the **Number of events to fail when exceeding voltage** parameter value, then the block starts entering the fault state.

**Dependencies**

Enabled when the **Enable behavioral fault trigger** parameter is set to Yes.

**Number of events to fail when exceeding voltage – Maximum number of times the voltage exceeds the threshold**

1 (default)

Since the physical mechanism underlying voltage-based failures depends on one or more partial discharge events occurring, this parameter allows you to set the number of voltage overshoots that the inductor can withstand before the fault transition begins. Note that the block does not check the time spent in the overvoltage condition, only the number of transitions.

**Dependencies**

Enabled when the **Enable behavioral fault trigger** parameter is set to Yes.

**Maximum permissible current – Current threshold to fault transition**

1 A (default)

Define the current threshold to a fault transition. If the current value exceeds this threshold for longer than the **Time to fail when exceeding current** parameter value, then the block starts entering the fault state.

**Dependencies**

Enabled when the **Enable behavioral fault trigger** parameter is set to Yes.

**Time to fail when exceeding current — Maximum length of time the current exceeds the threshold**

1 s (default)

Set the maximum length of time that the current can exceed the maximum permissible value without triggering the fault.

**Dependencies**

Enabled when the **Enable behavioral fault trigger** parameter is set to Yes.

**See Also**

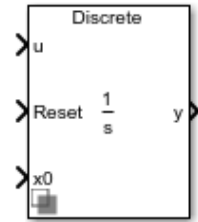
Capacitor | Fault | Mutual Inductor | Nonlinear Inductor | Resistor

**Introduced in R2016a**

# Integrator (Discrete or Continuous)

Discrete-time or continuous-time integrator

**Library:** Simscape / Electrical / Power Systems / Control /  
General Control



## Description

The Integrator (Discrete or Continuous) block implements a simple integrator in conformance with IEEE 421.5-2016<sup>[1]</sup>.

You can switch between continuous and discrete implementations of the integrator using the **Sample time** parameter.

## Equations

To configure the integrator for continuous time, set the **Sample time** property to 0. This representation is equivalent to the continuous transfer function:

$$G(s) = \frac{1}{s}.$$

From the preceding transfer function, the integrator defining equations are:

$$\begin{cases} \dot{x}(t) = u(t) \\ y(t) = x(t) \end{cases} \quad x(0) = x_0,$$

where:

- $u$  is the integrator input.
- $x$  is the integrator state.
- $y$  is the integrator output.
- $t$  is the simulation time.
- $x_0$  is the initial state of the integrator.

To configure the integrator for discrete time, set the **Sample time** property to a positive, nonzero value, or to -1 to inherit the sample time from an upstream block. The discrete representation is equivalent to the transfer function:

$$G(z) = \frac{T_s}{z-1},$$

where  $T_s$  is the sample time. From the discrete transfer function, the integrator equations are defined using the forward Euler method:

$$\begin{cases} x(n+1) = x(n) + T_s u(n) \\ y(n) = x(n) \end{cases} \quad x(0) = x_0,$$

where:

- $u$  is the integrator input.
- $x$  is the integrator state.
- $y$  is the integrator output.
- $n$  is the simulation time step.
- $x_0$  is the initial state of the integrator.

## Defining Initial Conditions

You can define the state initial conditions using the input port **x0**. The integrator state reverts to the initial condition any time it is reset.

## Limiting the Integral

You can limit the integral output using one of two methods:

- Set **Limit type** to **Anti-windup** to use the anti-windup saturation method.

The anti-windup method limits the integrator state  $x$  between the lower saturation limit  $A$  and upper saturation limit  $B$ :

$$A \leq x \leq B.$$

Because the state is limited, the output can respond immediately to a reversal of the input sign when the integral is saturated.

- Set **Limit type** to **Windup** to use the windup saturation method.

The windup method limits the integrator output  $y$  between the lower saturation limit  $A$  and upper saturation limit  $B$ :

$$A \leq y \leq B.$$

Because the output is limited, the state can continue to grow when the integrator is saturated. As a result, the output cannot respond to a reversal of the input sign until the state has reached the limiting saturation point.

## Resetting the State

You can reset the state of the integrator by passing a nonzero signal to the **Reset** port of the block.

## Ports

### Input

#### **u** — Integrator input

vector

Integrator input.

Data Types: `single` | `double`

#### **Reset** — Revert to initial state

scalar

Integrator reset. To reset the integrator state to the value of the **x0** port, pass a nonzero value to this port. Alternatively, attach a zero-valued Constant block to this port to override the external reset.

Data Types: `single` | `double`

### **x0 — Initial state**

vector

Integrator initial state. To specify the value of the state after a reset, pass a signal to this port.

Data Types: `single` | `double`

## **Output**

### **y — Integrator output**

vector

Integrator output.

Data Types: `single` | `double`

## **Parameters**

### **External reset — Reset strategy**

`level` (default) | `rising` | `falling` | `either`

Select the external reset strategy for the integrator:

- Select `rising` to reset the state when the reset signal rises from a negative or zero value to a positive value.
- Select `falling` to reset the state when the reset signal falls from a positive value to a zero or negative value.
- Select `either` to reset the state when the reset signal changes from zero to a nonzero value, from a nonzero value to zero, or changes sign.
- Select `level` to reset the state when the reset signal is nonzero at the current time step or changes from nonzero at the previous time step to zero at the current time step.



**Limit type – Saturation strategy**

Anti-windup (default) | Windup

Select the limit type of the integrator:

- Select `Anti-windup` to limit the state of the integrator, preventing windup.
- Select `windup` to limit the output of the integrator, allowing windup of the integrator state.

**Upper saturation limit – State upper limit**`inf` (default) | real number

Integrator upper saturation limit. Set this to `inf` for an unsaturated upper limit, or to a finite value to saturate the integrator using the strategy set by **Limit type**.

**Lower saturation limit – State lower limit**`-inf` (default) | real number

Integrator lower saturation limit. Set this to `-inf` for an unsaturated lower limit, or to a finite value to saturate the integrator using the strategy set by **Limit type**.

**Sample time (-1 for inherited) – Block sample time**`-1` (default) | 0 | positive scalar

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For inherited discrete-time operation, specify `-1`. For discrete-time operation, specify a positive integer. For continuous-time operation, specify `0`.

If this block is in a masked subsystem, or other variant subsystem that allows you to switch between continuous operation and discrete operation, promote the sample time parameter. Promoting the sample time parameter ensures correct switching between the continuous and discrete implementations of the block. For more information, see “Promote Parameter to Mask” (Simulink).

**References**

- [1] *IEEE Recommended Practice for Excitation System Models for Power System Stability Studies*. IEEE Std 421.5-2016. Piscataway, NJ: IEEE-SA, 2016.

## See Also

### Blocks

Filtered Derivative (Discrete or Continuous) | Integrator with Wrapped State (Discrete or Continuous) | Lead-Lag (Discrete or Continuous) | Low-Pass Filter (Discrete or Continuous) | Washout (Discrete or Continuous)

**Introduced in R2017b**

# Integrator with Wrapped State (Discrete or Continuous)

Discrete-time or continuous-time integrator with wrapped state

**Library:** Simscape / Electrical / Power Systems / Control / General Control



## Description

The Integrator with Wrapped State (Discrete or Continuous) block implements a wrapped state integrator in conformance with IEEE 421.5-2016<sup>[1]</sup>.

Use this block to generate periodic signals such as angles or to represent a voltage-controlled oscillator. You can switch between continuous and discrete implementations of the integrator using the **Sample time** parameter.

## Equations

To configure the integrator for continuous time, set the **Sample time** property to 0. This representation is equivalent to the continuous transfer function:

$$G(s) = \frac{1}{s}.$$

From the preceding transfer function, the integrator defining equations are:

$$\begin{cases} \dot{x}(t) = u(t) \\ y(t) = x(t) \end{cases} \quad x(0) = x_0,$$

where:

- $u$  is the integrator input.
- $x$  is the integrator state.
- $y$  is the integrator output.
- $t$  is the simulation time.
- $x_0$  is the initial state of the integrator.

To configure the integrator for discrete time, set the **Sample time** property to a positive, nonzero value, or to -1 to inherit the sample time from an upstream block. The discrete representation is equivalent to the transfer function:

$$G(z) = \frac{T_s}{z-1},$$

where  $T_s$  is the sample time. From the discrete transfer function, the integrator equations are defined using the forward Euler method:

$$\begin{cases} x(n+1) = x(n) + T_s u(n) \\ y(n) = x(n) \end{cases} \quad x(0) = x_0,$$

where:

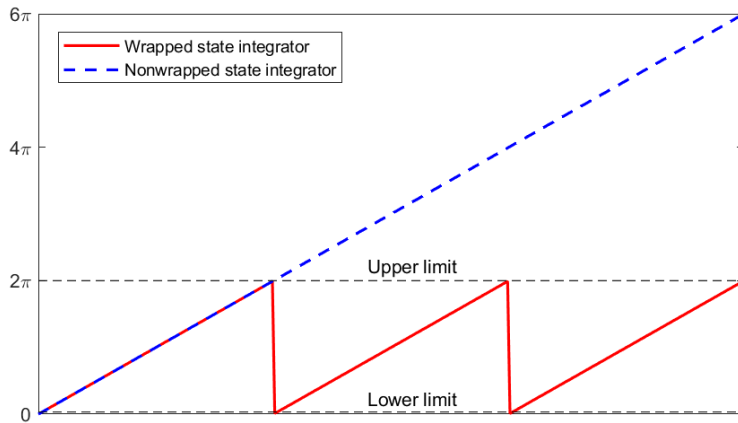
- $u$  is the integrator input.
- $x$  is the integrator state.
- $y$  is the integrator output.
- $n$  is the simulation time step.
- $x_0$  is the initial state of the integrator.

## Defining Initial Conditions

You can define the state initial conditions using **Initial condition** parameter.

## Wrapping Cyclic States

The integrator wraps its state between the specified lower and upper values. This diagram shows the outputs of a wrapped and nonwrapped state integrator for a constant input.



In the diagram, the lower and upper limits are  $0$  and  $2\pi$ , respectively.

## Ports

### Input

#### **u** — Integrator input

vector

Integrator input.

Data Types: `single` | `double`

### Output

#### **y** — Integrator output

vector

Integrator output.

Data Types: `single` | `double`

## Parameters

### **Wrapped state upper limit — State upper limit**

2\*pi (default) | real number

Integrator upper limit.

### **Wrapped state lower limit — State lower limit**

0 (default) | real number

Integrator lower limit.

### **Initial condition — State initial value**

0 (default) | real number

Integrator initial state.

### **Sample time (-1 for inherited) — Block sample time**

-1 (default) | 0 | positive scalar

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For inherited discrete-time operation, specify -1. For discrete-time operation, specify a positive integer. For continuous-time operation, specify 0.

If this block is in a masked subsystem, or other variant subsystem that allows you to switch between continuous operation and discrete operation, promote the sample time parameter. Promoting the sample time parameter ensures correct switching between the continuous and discrete implementations of the block. For more information, see “Promote Parameter to Mask” (Simulink).

## References

- [1] *IEEE Recommended Practice for Excitation System Models for Power System Stability Studies*. IEEE Std 421.5-2016. Piscataway, NJ: IEEE-SA, 2016.

## See Also

### Blocks

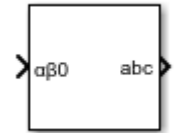
Filtered Derivative (Discrete or Continuous) | Integrator (Discrete or Continuous) | Lead-Lag (Discrete or Continuous) | Low-Pass Filter (Discrete or Continuous) | Washout (Discrete or Continuous)

**Introduced in R2017b**

## Inverse Clarke Transform

Implement  $\alpha\beta 0$  to  $abc$  transform

**Library:** Simscape / Electrical / Power Systems / Control / Mathematical Transforms

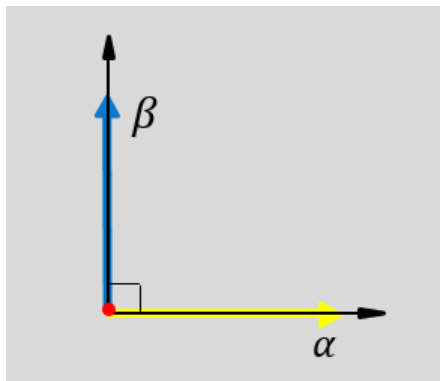


### Description

The Inverse Clarke Transform block converts the time-domain alpha, beta, and zero components in a stationary reference frame to three-phase components in an  $abc$  reference frame. The block can preserve the active and reactive powers with the powers of the system in the stationary reference frame by implementing an invariant power version of the inverse Clarke transform. If the zero component is zero, the components in the three-phase system are balanced.

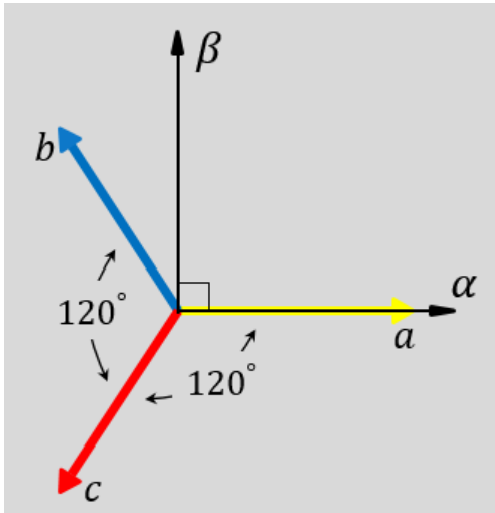
The figures show:

- Balanced  $\alpha$ ,  $\beta$ , and zero components in a stationary reference frame

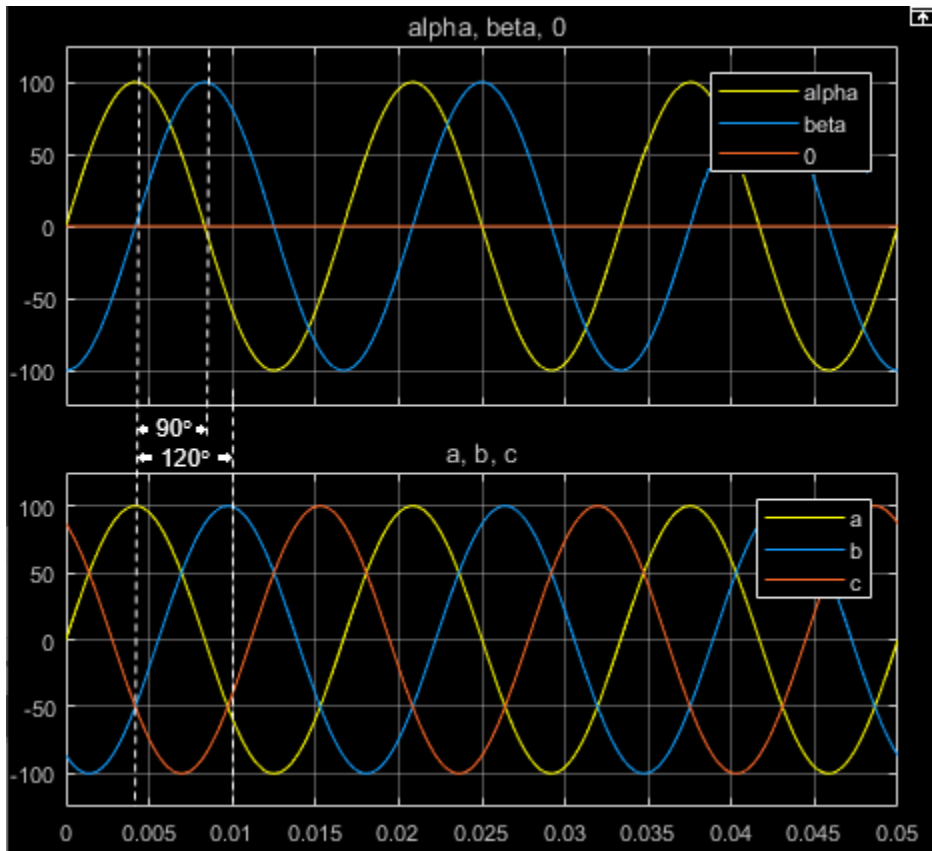


- The direction of the magnetic axes of the stator windings in the stationary  $\alpha\beta 0$  reference frame and the  $abc$  reference frame





- The time-response of the individual components of equivalent balanced  $\alpha\beta 0$  and  $abc$  systems



## Equations

The block implements the inverse Clarke transform as

$$\begin{bmatrix} a \\ b \\ c \end{bmatrix} = \begin{bmatrix} 1 & 0 & 1 \\ -\frac{1}{2} & \frac{\sqrt{3}}{2} & 1 \\ -\frac{1}{2} & -\frac{\sqrt{3}}{2} & 1 \end{bmatrix} \begin{bmatrix} \alpha \\ \beta \\ 0 \end{bmatrix}$$

where:

- $\alpha$  and  $\beta$  are the components in the stationary reference frame.
- $0$  is the zero component in the stationary reference frame.
- $a$ ,  $b$ , and  $c$  are the components of the three-phase system in the  $abc$  reference frame.

The block implements this power invariant version of the inverse Clarke transform as

$$\begin{bmatrix} a \\ b \\ c \end{bmatrix} = \sqrt{\frac{2}{3}} \begin{bmatrix} 1 & 0 & \frac{1}{\sqrt{2}} \\ -\frac{1}{2} & \frac{\sqrt{3}}{2} & \frac{1}{\sqrt{2}} \\ -\frac{1}{2} & -\frac{\sqrt{3}}{2} & \frac{1}{\sqrt{2}} \end{bmatrix} \begin{bmatrix} \alpha \\ \beta \\ 0 \end{bmatrix}$$

## Ports

### Input

#### $\alpha\beta 0$ — $\alpha$ - $\beta$ axis and zero components

vector

Alpha-axis component,  $\alpha$ , beta-axis component  $\beta$ , and zero component in the stationary reference frame.

Data Types: single | double

### Output

#### $abc$ — $a$ -, $b$ -, and $c$ -phase components

vector

Components of the three-phase system in the  $abc$  reference frame.

Data Types: single | double

## Parameters

### **Power Invariant — Power invariant transform**

off (default) | on

Preserve the active and reactive power of the system in the rotating reference frame.

## References

[1] Krause, P., O. Wasynczuk, S. D. Sudhoff, and S. Pekarek. *Analysis of Electric Machinery and Drive Systems*. Piscataway, NJ: Wiley-IEEE Press, 2013.

## See Also

### **Blocks**

Clarke Transform | Clarke to Park Angle Transform | Inverse Park Transform | Park Transform | Park to Clarke Angle Transform

**Introduced in R2017b**

# Inverse Park Transform

Implement  $dq0$  to  $abc$  transform

**Library:** Simscape / Electrical / Power Systems / Control /  
Mathematical Transforms

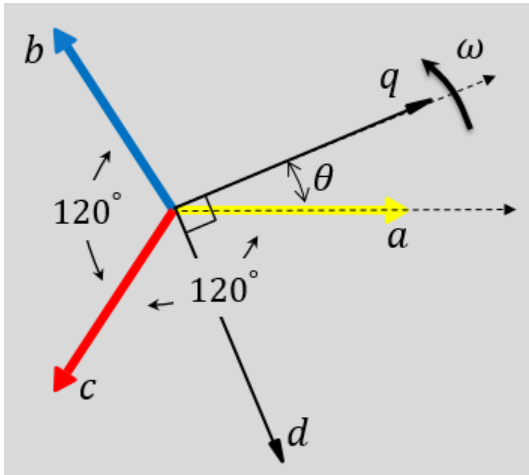


## Description

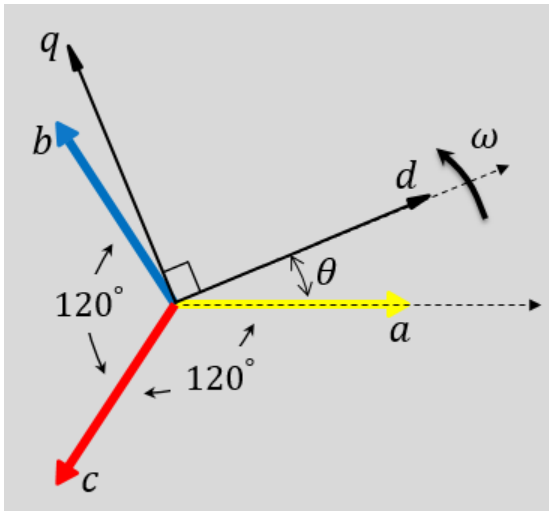
The Inverse Park Transform block converts the time-domain direct, quadrature, and zero components in a rotating reference frame to the components of a three-phase system in an  $abc$  reference frame. The block can preserve the active and reactive powers with the powers of the system in the rotating reference frame by implementing an invariant version of the Park transform. For a balanced system, the zero component is equal to zero.

You can configure the block to align the  $a$ -axis of the three-phase system to either the  $d$ - or  $q$ -axis of the rotating reference frame at time,  $t = 0$ . The figures show the direction of the magnetic axes of the stator windings in an  $abc$  reference frame and a rotating  $d$ - $q$  reference frame where:

- The  $a$ -axis and the  $q$ -axis are initially aligned.



- The  $a$ -axis and the  $d$ -axis are initially aligned.



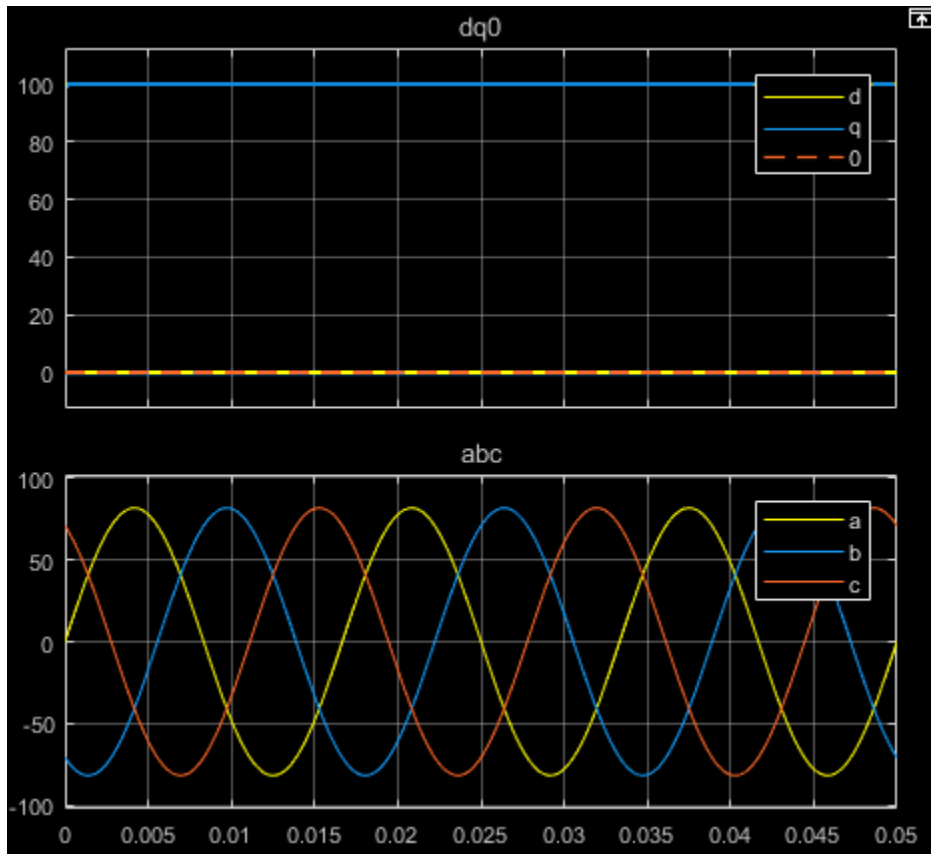
In both cases, the angle  $\theta = \omega t$ , where

- $\theta$  is the angle between the  $a$  and  $q$  axes for the  $q$ -axis alignment or the angle between the  $a$  and  $d$  axes for the  $d$ -axis alignment.
- $\omega$  is the rotational speed of the  $d$ - $q$  reference frame.

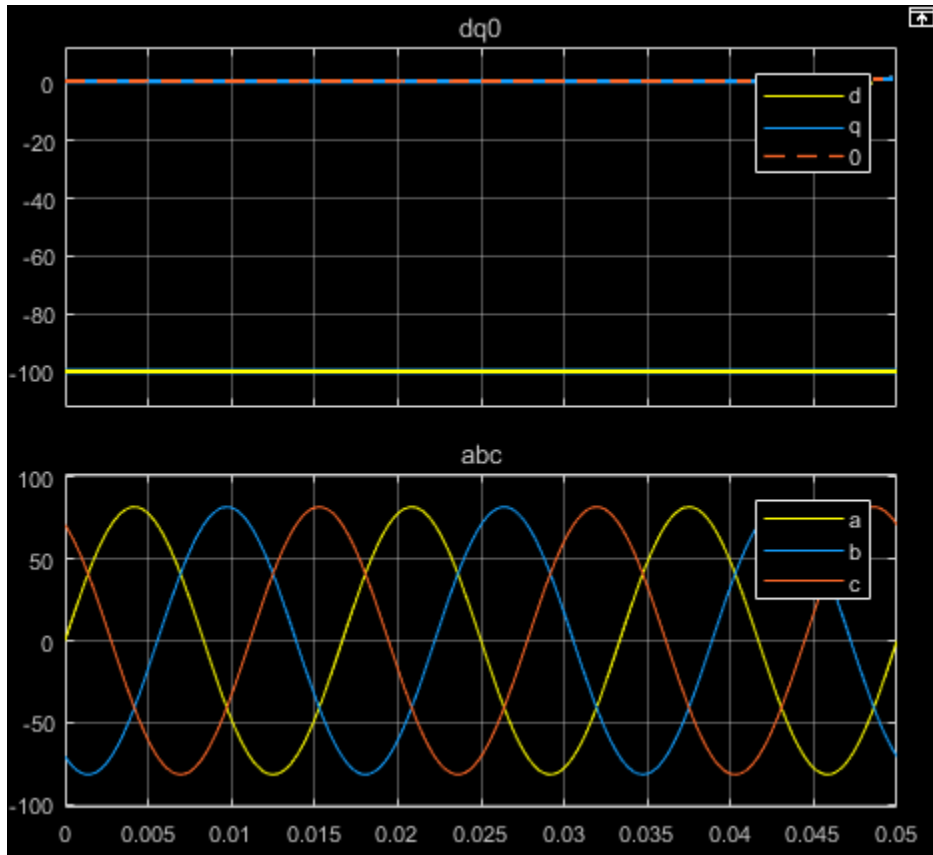
- $t$  is the time, in s, from the initial alignment.

The figures show the time-response of the individual components of equivalent balanced  $dq0$  and  $abc$  for an:

- Alignment of the  $a$ -phase vector to the  $q$ -axis



- Alignment of the  $a$ -phase vector to the  $d$ -axis



## Defining Equations

The Inverse Park Transform block implements the transform for an  $a$ -phase to  $q$ -axis alignment as

$$\begin{bmatrix} a \\ b \\ c \end{bmatrix} = \begin{bmatrix} \sin(\theta) & \cos(\theta) & 1 \\ \sin(\theta - \frac{2\pi}{3}) & \cos(\theta - \frac{2\pi}{3}) & 1 \\ \sin(\theta + \frac{2\pi}{3}) & \cos(\theta + \frac{2\pi}{3}) & 1 \end{bmatrix} \begin{bmatrix} d \\ q \\ 0 \end{bmatrix},$$



where:

- $d$  and  $q$  are the components of the two-axis system in the rotating reference frame.
- $a$ ,  $b$ , and  $c$  are the components of the three-phase system in the  $abc$  reference frame.
- $0$  is the zero component of the two-axis system in the stationary reference frame.

For a power invariant  $a$ -phase to  $q$ -axis alignment, the block implements the transform using this equation:

$$\begin{bmatrix} a \\ b \\ c \end{bmatrix} = \sqrt{\frac{2}{3}} \begin{bmatrix} \sin(\theta) & \cos(\theta) & \sqrt{\frac{1}{2}} \\ \sin(\theta - \frac{2\pi}{3}) & \cos(\theta - \frac{2\pi}{3}) & \sqrt{\frac{1}{2}} \\ \sin(\theta + \frac{2\pi}{3}) & \cos(\theta + \frac{2\pi}{3}) & \sqrt{\frac{1}{2}} \end{bmatrix} \begin{bmatrix} d \\ q \\ 0 \end{bmatrix}.$$

For an  $a$ -phase to  $d$ -axis alignment, the block implements the transform using this equation:

$$\begin{bmatrix} a \\ b \\ c \end{bmatrix} = \begin{bmatrix} \cos(\theta) & -\sin(\theta) & 1 \\ \cos(\theta - \frac{2\pi}{3}) & -\sin(\theta - \frac{2\pi}{3}) & 1 \\ \cos(\theta + \frac{2\pi}{3}) & -\sin(\theta + \frac{2\pi}{3}) & 1 \end{bmatrix} \begin{bmatrix} d \\ q \\ 0 \end{bmatrix}.$$

The block implements a power invariant  $a$ -phase to  $d$ -axis alignment as

$$\begin{bmatrix} a \\ b \\ c \end{bmatrix} = \sqrt{\frac{2}{3}} \begin{bmatrix} \cos(\theta) & -\sin(\theta) & \sqrt{\frac{1}{2}} \\ \cos(\theta - \frac{2\pi}{3}) & -\sin(\theta - \frac{2\pi}{3}) & \sqrt{\frac{1}{2}} \\ \cos(\theta + \frac{2\pi}{3}) & -\sin(\theta + \frac{2\pi}{3}) & \sqrt{\frac{1}{2}} \end{bmatrix} \begin{bmatrix} d \\ q \\ 0 \end{bmatrix}.$$

## Ports

### Input

#### **$dq0$ — $d$ - $q$ axis and zero components**

vector

Direct-axis and quadrature-axis components and the zero component of the system in the rotating reference frame.

Data Types: `single` | `double`

#### **$\theta_{abc}$ — Rotational angle**

scalar | in radians

Angular position of the rotating reference frame. The value of this parameter is equal to the polar distance from the vector of the  $a$ -phase in the  $abc$  reference frame to the initially aligned axis of the  $dq0$  reference frame.

Data Types: `single` | `double`

### Output

#### **$abc$ — $a$ -, $b$ -, and $c$ -phase components**

vector

Components of the three-phase system in the  $abc$  reference frame.

Data Types: `single` | `double`

## Parameters

#### **Power Invariant — Power invariant transform**

`off` (default) | `on`

Option to preserve the active and reactive power of the  $abc$  reference frame.

#### **Phase-a axis alignment — $dq0$ reference frame alignment**

`Q-axis` (default) | `D-axis`

Align the  $a$ -phase vector of the  $abc$  reference frame to the  $d$ - or  $q$ -axis of the rotating reference frame.

## References

- [1] Krause, P., O. Wasynczuk, S. D. Sudhoff, and S. Pekarek. *Analysis of Electric Machinery and Drive Systems*. Piscataway, NJ: Wiley-IEEE Press, 2013.

## See Also

### Blocks

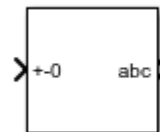
Clarke Transform | Clarke to Park Angle Transform | Inverse Clarke Transform | Park Transform | Park to Clarke Angle Transform

**Introduced in R2017b**

# Inverse Symmetrical-Components Transform

Implement  $+0$  to  $abc$  transform

**Library:** Simscape / Electrical / Power Systems / Control /  
Mathematical Transforms



## Description

The Inverse Symmetrical-Components Transform block implements an inverse symmetrical transform of a positive, negative, and zero phasor. The transform splits a symmetrical set of three phasors into the equivalent unbalanced set of  $a$ ,  $b$ , and  $c$  phasors.

Use this transform to regenerate a three-phase signal from a system that was decoupled using the Symmetrical-Components Transform block.

Use the Power invariant property to choose between the Fortescue transform, and the alternative, power-invariant version.

## Equations

The inverse symmetrical-components transform regenerates an unbalanced three-phase signal  $[V_a, V_b, V_c]$  from the  $a$  components of a balanced set of phasors  $[V_{a+}, V_{a-}, V_{a0}]$ , given in the  $+0$  domain:

$$\begin{bmatrix} V_a \\ V_b \\ V_c \end{bmatrix} = \frac{1}{K} \begin{bmatrix} 1 & 1 & 1 \\ a^2 & a & 1 \\ a & a^2 & 1 \end{bmatrix} \begin{bmatrix} V_{a+} \\ V_{a-} \\ V_{a0} \end{bmatrix}.$$

where,  $a$  is the complex rotation operator

$$a = e^{2\pi i/3},$$

and  $K$  is the constant that determines the type of transform:

$$\begin{cases} K = 1 & \text{Fortescue transform} \\ K = \sqrt{3} & \text{Power-invariant transform} \end{cases}$$

If the transform was performed using the power-invariant option, enable the **Power invariant** property to select the power-invariant inverse transform and regenerate the correct  $abc$  signal.

## Symmetrical-Components Transform

The symmetrical-components transform separates an unbalanced three-phase signal given in phasor quantities into three balanced sets of phasors:

$$\begin{bmatrix} v_a \\ v_b \\ v_c \end{bmatrix} = \begin{bmatrix} v_{a+} \\ v_{b+} \\ v_{c+} \end{bmatrix} + \begin{bmatrix} v_{a-} \\ v_{b-} \\ v_{c-} \end{bmatrix} + \begin{bmatrix} v_{a0} \\ v_{b0} \\ v_{c0} \end{bmatrix},$$

where:

- $v_a$ ,  $v_b$ , and  $v_c$  make up the original, unbalanced set of phasors.
- $v_{a+}$ ,  $v_{b+}$ , and  $v_{c+}$  make up the balanced, positive set of phasors.
- $v_{a-}$ ,  $v_{b-}$ , and  $v_{c-}$  make up the balanced, negative set of phasors.
- $v_{a0}$ ,  $v_{b0}$ , and  $v_{c0}$  make up the balanced, zero set of phasors.

The symmetrical-components transform calculates the symmetric  $a$ -phase as:

$$\begin{bmatrix} V_{a+} \\ V_{a-} \\ V_{a0} \end{bmatrix} = \frac{K}{3} \begin{bmatrix} 1 & a & a^2 \\ 1 & a^2 & a \\ 1 & 1 & 1 \end{bmatrix} \begin{bmatrix} V_a \\ V_b \\ V_c \end{bmatrix}.$$

Because the remaining two sets of symmetrical phasors are not often used in calculation, the transformation only generates the first set. However, you can calculate the  $b$ - and  $c$ -sets in terms of simple rotations of the first:

$$\begin{bmatrix} V_{b+} \\ V_{b-} \\ V_{b0} \end{bmatrix} = \begin{bmatrix} a^2 & 0 & 0 \\ 0 & a & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} V_{a+} \\ V_{a-} \\ V_{a0} \end{bmatrix},$$

and

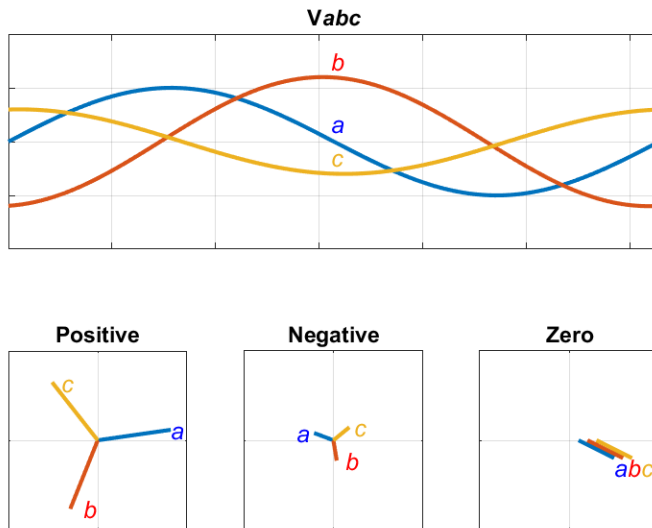
$$\begin{bmatrix} V_{c+} \\ V_{c-} \\ V_{c0} \end{bmatrix} = \begin{bmatrix} a & 0 & 0 \\ 0 & a^2 & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} V_{a+} \\ V_{a-} \\ V_{a0} \end{bmatrix}.$$

## Operating Principle

The three sets of balanced phasors generated by the symmetrical-components transform have the following properties:

- The positive set has the same order as the unbalanced set of phasors  $a$ - $b$ - $c$ .
- The negative set has the opposite order as the unbalanced set of phasors  $a$ - $c$ - $b$ .
- The zero set has no order because all three phasor angles are equal.

This diagram visualizes the separation performed by the transform.



In the diagram, the top axis shows an unbalanced three-phase signal with components  $a$ ,  $b$ , and  $c$ . The bottom set of axes separates the three-phase signal into symmetrical positive, negative, and zero phasors.

Observe that in each case, the  $a$ ,  $b$ , and  $c$  components are symmetrical and are separated by:

- +120 degrees for the positive set.
- -120 degrees for the negative set.
- 0 degrees for the zero set.

## Ports

### Input

**+ - 0 — Balanced a phasor components**  
vector

Positive, negative, and zero  $a$  phasors given as a complex signal. Use the rotations given in the Symmetrical-Components Transform section to compute the  $b$  and  $c$  phasor sets.

Data Types: `single` | `double`

### Output

#### **abc — $a$ , $b$ , and $c$ phasors**

vector

Regenerated three-phase set of unbalanced phasors, output as a complex signal.

Data Types: `single` | `double`

### Parameters

#### **Power invariant — Transform type**

`off` (default) | `on`

Power invariant toggle. Select this parameter to use the power-invariant alternative of the original Fortescue transform.

### References

[1] Anderson, P. M. *Analysis of Faulted Power Systems*. Hoboken, NJ: Wiley-IEEE Press, 1995.

### See Also

#### **Blocks**

[Clarke Transform](#) | [Clarke to Park Angle Transform](#) | [Inverse Clarke Transform](#) | [Inverse Park Transform](#) | [Park to Clarke Angle Transform](#) | [Symmetrical-Components Transform](#)

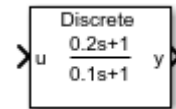
**Introduced in R2017b**



## Lead-Lag (Discrete or Continuous)

Discrete-time or continuous-time lead-lag compensator

**Library:** Simscape / Electrical / Power Systems / Control / General Control



### Description

The Lead-Lag (Discrete or Continuous) block implements a lead-lag compensator in conformance with IEEE 421.5-2016<sup>[1]</sup>.

You can switch between continuous and discrete implementations of the block using the **Sample time** parameter.

### Equations

To configure the compensator for continuous time, set the **Sample time** property to 0. This representation is equivalent to the continuous transfer function:

$$G(s) = \frac{T_1s + 1}{T_2s + 1},$$

where:

- $T_1$  is the lead time constant.
- $T_2$  is the lag time constant.

From the preceding transfer function, the compensator defining equations are:

$$\begin{cases} \dot{x}(t) = \frac{1}{T_2}(u(t) - x(t)) \\ y(t) = \frac{T_1}{T_2}u(t) + \left(1 - \frac{T_1}{T_2}\right)x(t) \end{cases} \quad y(0) = x(0) = u_0,$$

where:

- $u$  is the block input.
- $x$  is the block state.
- $y$  is the block output.
- $t$  is the simulation time.
- $u_0$  is the initial input to the block.

To configure the compensator for discrete time, set the **Sample time** property to a positive, nonzero value, or to -1 to inherit the sample time from an upstream block. The discrete representation is equivalent to the transfer function:

$$\frac{T_1 z + (T_s - T_1)}{T_2 z + (T_s - T_2)},$$

where:

- $T_1$  is the lead time constant.
- $T_2$  is the lag time constant.
- $T_s$  is the compensator sample time.

From the discrete transfer function, the compensator equations are defined using the forward Euler method:

$$\begin{cases} x(n+1) = \left(1 - \frac{T_s}{T_2}\right)x(n) + \left(\frac{T_s}{T_2}\right)u(n) \\ y(n) = \left(1 - \frac{T_1}{T_2}\right)x(n) + \left(\frac{T_1}{T_2}\right)u(n) \end{cases} \quad y(0) = x(0) = u_0,$$

where:

- $u$  is the block input.
- $x$  is the state.
- $y$  is the block output.
- $n$  is the simulation time step.
- $u_0$  is the initial input to the block.

## Initial Conditions

The block sets the state and output initial conditions to the initial input.

## Limiting the Integral

Set the **Upper saturation limit** and **Lower saturation limit** parameters to use the anti-windup saturation method.

The anti-windup method limits the compensator state between the lower saturation limit  $A$  and upper saturation limit  $B$ :

$$A \leq x \leq B.$$

Because the state is limited, the output can respond immediately to a reversal of the input sign when the integral is saturated.

This block does not provide a windup saturation method. To use the windup saturation method, set the **Upper saturation limit** parameter to `inf`, the **Lower saturation limit** parameter to `-inf`, and attach a Saturation block to the output.

## Bypass Compensator Dynamics

Set the lag time constant to zero or to a value equal to that of the lead time constant to ignore the dynamics of the compensator. When bypassed, the block feeds the input directly to the output:

$$\left. \begin{array}{l} T_1 = 0 \\ T_2 = 0 \\ T_1 = T_2 \end{array} \right\} y = u.$$

In the continuous case, both the sample time and at least one time constant must be zero.

## Ports

### Input

#### **u — Compensator input**

vector

Lead-lag compensator input signal. The block uses the input initial value to determine the state initial value.

Data Types: `single` | `double`

### Output

#### **y — Compensator output**

vector

Lead-lag compensator output.

Data Types: `single` | `double`

## Parameters

#### **Lead time constant, T1 — Lead time constant**

0.2 (default) | positive number

Compensator lead time constant. To bypass the dynamics of the compensator, set this value to 0 or to the value of the **Lag time constant, T2** parameter.

#### **Lag time constant, T2 — Lag time constant**

0.1 (default) | positive number

Compensator lag time constant. To bypass the dynamics of the compensator, set this value to 0 or to the value of the **Lead time constant, T1** parameter.

#### **Upper saturation limit — State upper limit**

inf (default) | real number

Compensator upper state limit. Set this to `inf` for an unsaturated upper limit, or to a finite value to prevent upper windup of the system's integrator.

### **Lower saturation limit – State lower limit**

`-inf` (default) | real number

Compensator lower state limit. Set this to `-inf` for an unsaturated lower limit, or to a finite value to prevent lower windup of the system's integrator.

### **Sample time (-1 for inherited) – Block sample time**

`-1` (default) | 0 | positive scalar

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For inherited discrete-time operation, specify `-1`. For discrete-time operation, specify a positive integer. For continuous-time operation, specify `0`.

If this block is in a masked subsystem, or other variant subsystem that allows you to switch between continuous operation and discrete operation, promote the sample time parameter. Promoting the sample time parameter ensures correct switching between the continuous and discrete implementations of the block. For more information, see “Promote Parameter to Mask” (Simulink).

## **References**

[1] *IEEE Recommended Practice for Excitation System Models for Power System Stability Studies*. IEEE Std 421.5-2016. Piscataway, NJ: IEEE-SA, 2016.

## **See Also**

### **Blocks**

Filtered Derivative (Discrete or Continuous) | Integrator (Discrete or Continuous) | Integrator with Wrapped State (Discrete or Continuous) | Low-Pass Filter (Discrete or Continuous) | Washout (Discrete or Continuous)

**Introduced in R2017b**

## Light-Emitting Diode

Model light-emitting diode as exponential diode and current sensor in series



### Library

Sensors

### Description

The Light-Emitting Diode block represents a light-emitting diode as an exponential diode in series with a current sensor. The optical power presented at the signal port W is equal to the product of the current flowing through the diode and the **Optical power per unit current** parameter value.

The exponential diode model provides the following relationship between the diode current  $I$  and the diode voltage  $V$ :

$$I = IS \cdot \left( e^{\frac{qV}{NkT_{m1}}} - 1 \right)$$

where:

- $q$  is the elementary charge on an electron (1.602176e-19 Coulombs).
- $k$  is the Boltzmann constant (1.3806503e-23 J/K).
- $N$  is the emission coefficient.
- $IS$  is the saturation current.
- $T_{m1}$  is the temperature at which the diode parameters are specified, as defined by the **Measurement temperature** parameter value.

When  $(qV / NkT_{m1}) > 80$ , the block replaces  $e^{\frac{qV}{NkT_{m1}}}$  with  $(qV / NkT_{m1} - 79)e^{80}$ , which matches the gradient of the diode current at  $(qV / NkT_{m1}) = 80$  and extrapolates linearly.

When  $(qV / NkT_{m1}) < -79$ , the block replaces  $e^{\frac{qV}{NkT_{m1}}}$  with  $(qV / NkT_{m1} + 80)e^{-79}$ , which also matches the gradient and extrapolates linearly. Typical electrical circuits do not reach these extreme values. The block provides this linear extrapolation to help convergence when solving for the constraints during simulation.

When you select Use parameters **IS** and **N** for the **Parameterization** parameter, you specify the diode in terms of the **Saturation current IS** and **Emission coefficient N** parameters. When you select Use **I-V curve data points** for the **Parameterization** parameter, you specify two voltage and current measurement points on the diode I-V curve and the block derives the **IS** and **N** values. When you specify current and voltage measurements, the block calculates **IS** and **N** as follows:

- $$N = ((V_1 - V_2) / V_t) / (\log(I_1) - \log(I_2))$$
- $$IS = (I_1 / (\exp(V_1 / (NV_t)) - 1) + I_2 / (\exp(V_2 / (NV_t)) - 1)) / 2$$

where:

- $V_t = kT_{m1} / q$ .
- $V_1$  and  $V_2$  are the values in the **Voltages [V1 V2]** vector.
- $I_1$  and  $I_2$  are the values in the **Currents [I1 I2]** vector.

The exponential diode model provides the option to include a junction capacitance:

- When you select **Fixed** or **zero junction capacitance** for the **Junction capacitance** parameter, the capacitance is fixed.
- When you select Use parameters **CJO**, **VJ**, **M** & **FC** for the **Junction capacitance** parameter, the block uses the coefficients **CJO**, **VJ**, **M**, and **FC** to calculate a junction capacitance that depends on the junction voltage.
- When you select Use **C-V curve data points** for the **Junction capacitance** parameter, the block uses three capacitance values on the C-V capacitance curve to estimate **CJO**, **VJ**, and **M** and uses these values with the specified value of **FC** to calculate a junction capacitance that depends on the junction voltage. The block calculates **CJO**, **VJ**, and **M** as follows:

- $$CJ0 = C_1((V_{R2} - V_{R1}) / (V_{R2} - V_{R1}(C_2 / C_1)^{-1/M}))^M$$
- $$VJ = -(-V_{R2}(C_1 / C_2)^{-1/M} + V_{R1}) / (1 - (C_1 / C_2)^{-1/M})$$
- $$M = \log(C_3 / C_2) / \log(V_{R2} / V_{R3})$$

where:

- $V_{R1}$ ,  $V_{R2}$ , and  $V_{R3}$  are the values in the **Reverse bias voltages [VR1 VR2 VR3]** vector.
- $C_1$ ,  $C_2$ , and  $C_3$  are the values in the **Corresponding capacitances [C1 C2 C3]** vector.

It is not possible to estimate  $FC$  reliably from tabulated data, so you must specify its value using the **Capacitance coefficient FC** parameter. In the absence of suitable data for this parameter, use a typical value of 0.5.

The reverse bias voltages (defined as positive values) should satisfy  $V_{R3} > V_{R2} > V_{R1}$ . This means that the capacitances should satisfy  $C_1 > C_2 > C_3$  as reverse bias widens the depletion region and hence reduces capacitance. Violating these inequalities results in an error. Voltages  $V_{R2}$  and  $V_{R3}$  should be well away from the Junction potential  $VJ$ . Voltage  $V_{R1}$  should be less than the Junction potential  $VJ$ , with a typical value for  $V_{R1}$  being 0.1 V.

The voltage-dependent junction is defined in terms of the capacitor charge storage  $Q_j$  as:

- For  $V < FC \cdot VJ$ :

$$Q_j = CJ0 \cdot (VJ / (M - 1)) \cdot ((1 - V / VJ)^{1-M} - 1)$$

- For  $V \geq FC \cdot VJ$ :

$$Q_j = CJ0 \cdot F_1 + (CJ0 / F_2) \cdot (F_3 \cdot (V - FC \cdot VJ) + 0.5(M / VJ) \cdot (V^2 - (FC \cdot VJ)^2))$$

where:

- $$F_1 = (VJ / (1 - M)) \cdot (1 - (1 - FC)^{1-M})$$
- $$F_2 = (1 - FC)^{1+M}$$



- $$F_3 = 1 - FC \cdot (1 + M)$$

These equations are the same as used in [2], except that the temperature dependence of  $V_J$  and  $FC$  is not modeled. This model does not include the diffusion capacitance term that affects performance for high frequency switching applications.

The Light-Emitting Diode block contains several options for modeling the dependence of the diode current-voltage relationship on the temperature during simulation. Temperature dependence of the junction capacitance is not modeled, this being a much smaller effect. For details, see the Diode reference page.

## Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal port H on the block icon, and adds the **Thermal Port** tab to the block dialog box.

Use the thermal port to simulate the effects of generated heat and device temperature. For more information on using thermal ports and on the **Thermal Port** tab parameters, see “Simulating Thermal Effects in Semiconductors”.

## Variables

Use the **Variables** section of the block interface to set the priority and initial target values for the block variables prior to simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

## Basic Assumptions and Limitations

The Light-Emitting Diode block has the following limitations:

- When you select Use I-V curve data points for the **Parameterization** parameter, choose a pair of voltages near the diode turn-on voltage. Typically this is in the range from 0.05 to 1 Volt. Using values outside of this region may lead to numerical issues and poor estimates for  $I_S$  and  $N$ .

- You may need to use nonzero ohmic resistance and junction capacitance values to prevent numerical simulation issues, but the simulation may run faster with these values set to zero.

## Parameters

- “Main” on page 1-812
- “Junction Capacitance” on page 1-813
- “Temperature Dependence” on page 1-814

## Main

### Optical power per unit current

The amount of optical power the light-emitting diode generates per unit of current flowing through the diode. The default value is 0.005 W/A.

### Parameterization

Select one of the following methods for model parameterization:

- Use I-V curve data points — Specify measured data at two points on the diode I-V curve. This is the default method.
- Use parameters  $I_S$  and  $N$  — Specify saturation current and emission coefficient.

### Currents [I1 I2]

A vector of the current values at the two points on the diode I-V curve that the block uses to calculate  $I_S$  and  $N$ . This parameter is only visible when you select Use I-V curve data points for the **Parameterization** parameter. The default value is [ 0.0017 0.003 ] A.

### Voltages [V1 V2]

A vector of the voltage values at the two points on the diode I-V curve that the block uses to calculate  $I_S$  and  $N$ . This parameter is only visible when you select Use I-V curve data points for the **Parameterization** parameter. The default value is [ 0.9 1.05 ] V.

### Saturation current, $I_S$

The magnitude of the current that the ideal diode equation approaches asymptotically for very large reverse bias levels. This parameter is only visible when you select Use

parameters **IS** and **N** for the **Parameterization** parameter. The default value is  $5e-5$  A.

### **Emission coefficient, N**

The diode emission coefficient or ideality factor. This parameter is only visible when you select Use parameters **IS** and **N** for the **Parameterization** parameter. The default value is 10.

### **Ohmic resistance, RS**

The series diode connection resistance. The default value is  $0.1 \Omega$ .

### **Measurement temperature**

The temperature at which **IS** or the I-V curve was measured. The default value is  $25^\circ\text{C}$ .

## **Junction Capacitance**

### **Junction capacitance**

Select one of the following options for modeling the junction capacitance:

- **Fixed or zero junction capacitance** — Model the junction capacitance as a fixed value.
- **Use C-V curve data points** — Specify measured data at three points on the diode C-V curve.
- **Use parameters CJ0, VJ, M & FC** — Specify zero-bias junction capacitance, junction potential, grading coefficient, and forward-bias depletion capacitance coefficient.

### **Zero-bias junction capacitance, CJ0**

The value of the capacitance placed in parallel with the exponential diode term. This parameter is only visible when you select **Fixed or zero junction capacitance** or Use parameters **CJ0**, **VJ**, **M** & **FC** for the **Junction capacitance** parameter. The default value is  $20 \text{ pF}$ .

### **Reverse bias voltages [VR1 VR2 VR3]**

A vector of the reverse bias voltage values at the three points on the diode C-V curve that the block uses to calculate **CJ0**, **VJ**, and **M**. This parameter is only visible when you select **Use C-V curve data points** for the **Junction capacitance** parameter. The default value is  $[ 0.1 \ 10 \ 100 ] \text{ V}$ .

### **Corresponding capacitances [C1 C2 C3]**

A vector of the capacitance values at the three points on the diode C-V curve that the block uses to calculate  $CJ0$ ,  $VJ$ , and  $M$ . This parameter is only visible when you select Use C-V curve data points for the **Junction capacitance** parameter. The default value is [ 15 10 2 ] pF.

### **Junction potential, VJ**

The junction potential. This parameter is only visible when you select Use parameters  $CJ0$ ,  $VJ$ ,  $M$  &  $FC$  for the **Junction capacitance** parameter. The default value is 1 V.

### **Grading coefficient, M**

The grading coefficient. This parameter is only visible when you select Use parameters  $CJ0$ ,  $VJ$ ,  $M$  &  $FC$  for the **Junction capacitance** parameter. The default value is 0.5.

### **Capacitance coefficient, FC**

Fitting coefficient that quantifies the decrease of the depletion capacitance with applied voltage. This parameter is only visible when you select Use C-V curve data points or Use parameters  $CJ0$ ,  $VJ$ ,  $M$  &  $FC$  for the **Junction capacitance** parameter. The default value is 0.5.

## **Temperature Dependence**

### **Parameterization**

Select one of the following methods for temperature dependence parameterization:

- **None – Simulate at parameter measurement temperature –** Temperature dependence is not modeled, or the model is simulated at the measurement temperature  $T_{m1}$  (as specified by the **Measurement temperature** parameter on the **Main** tab). This is the default method.
- **Use an I-V data point at second measurement temperature T2 –** If you select this option, you specify a second measurement temperature  $T_{m2}$ , and the current and voltage values at this temperature. The model uses these values, along with the parameter values at the first measurement temperature  $T_{m1}$ , to calculate the energy gap value.
- **Specify saturation current at second measurement temperature T2 –** If you select this option, you specify a second measurement temperature  $T_{m2}$ , and saturation current value at this temperature. The model uses these values,

along with the parameter values at the first measurement temperature  $T_{m1}$ , to calculate the energy gap value.

- Specify the energy gap EG — Specify the energy gap value directly.

### **Current $I1$ at second measurement temperature**

Specify the diode current  $I1$  value when the voltage is  $V1$  at the second measurement temperature. This parameter is only visible when you select Use an I-V data point at second measurement temperature T2 for the **Parameterization** parameter. The default value is 0.0034 A.

### **Voltage $V1$ at second measurement temperature**

Specify the diode voltage  $V1$  value when the current is  $I1$  at the second measurement temperature. This parameter is only visible when you select Use an I-V data point at second measurement temperature T2 for the **Parameterization** parameter. The default value is 1.05 V.

### **Saturation current, $IS$ , at second measurement temperature**

Specify the saturation current  $IS$  value at the second measurement temperature. This parameter is only visible when you select Specify saturation current at second measurement temperature T2 for the **Parameterization** parameter. The default value is  $1.8e-4$  A.

### **Second measurement temperature**

Specify the value for the second measurement temperature. This parameter is only visible when you select either Use an I-V data point at second measurement temperature T2 or Specify saturation current at second measurement temperature T2 for the **Parameterization** parameter. The default value is 125 °C.

### **Energy gap parameterization**

This parameter is only visible when you select Specify the energy gap EG for the **Parameterization** parameter. It lets you select a value for the energy gap from a list of predetermined options, or specify a custom value:

- Use nominal value for silicon (EG=1.11eV) — This is the default.
- Use nominal value for 4H-SiC silicon carbide (EG=3.23eV)
- Use nominal value for 6H-SiC silicon carbide (EG=3.00eV)
- Use nominal value for germanium (EG=0.67eV)
- Use nominal value for gallium arsenide (EG=1.43eV)

- Use nominal value for selenium ( $EG=1.74\text{eV}$ )
- Use nominal value for Schottky barrier diodes ( $EG=0.69\text{eV}$ )
- Specify a custom value — If you select this option, the **Energy gap, EG** parameter appears in the dialog box, to let you specify a custom value for  $EG$ .

## Energy gap, EG

Specify a custom value for the energy gap,  $EG$ . This parameter is only visible when you select Specify a custom value for the **Energy gap parameterization** parameter. The default value is 1.11 eV.

## Saturation current temperature exponent parameterization

Select one of the following options to specify the saturation current temperature exponent value:

- Use nominal value for pn-junction diode ( $XTI=3$ ) — This is the default.
- Use nominal value for Schottky barrier diode ( $XTI=2$ )
- Specify a custom value — If you select this option, the **Saturation current temperature exponent, XTI** parameter appears in the dialog box, to let you specify a custom value for  $XTI$ .

## Saturation current temperature exponent, XTI

Specify a custom value for the saturation current temperature exponent,  $XTI$ . This parameter is only visible when you select Specify a custom value for the **Saturation current temperature exponent parameterization** parameter. The default value is 3.

## Device simulation temperature

Specify the value for the temperature  $T_s$ , at which the device is to be simulated. The default value is 25 °C.

# Ports

The block has the following ports:

W

Optical output power

+

Electrical conserving port associated with the diode positive terminal

-

Electrical conserving port associated with the diode negative terminal

## References

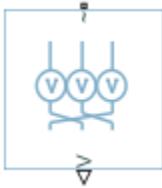
- [1] H. Ahmed and P.J. Spreadbury. *Analogue and digital electronics for engineers*. 2nd Edition, Cambridge University Press, 1984.
- [2] G. Massobrio and P. Antognetti. *Semiconductor Device Modeling with SPICE*. 2nd Edition, McGraw-Hill, 1993.

## See Also

Diode | Optocoupler | Photodiode

## Line Voltage Sensor

Measure line voltages in three-phase system



### Library

Simscape / Electrical / Power Systems / Sensors

### Description

The Line Voltage Sensor block represents an ideal three-phase line voltage sensor. The block measures the line-line voltages of a three-phase system and outputs a three-element physical signal vector. Each element of the physical signal output vector is proportional to the voltage between the phases as follows:

- Element 1:  $V_{ab} = V_a - V_b$
- Element 2:  $V_{bc} = V_b - V_c$
- Element 3:  $V_{ca} = V_c - V_a$

where  $V_a, V_b$  and  $V_c$  are the absolute phase voltages.

### Ports

The block has the following ports:

~1

Expandable three-phase port



V

Three-element physical signal vector output port associated with the voltages between the phases

## **See Also**

Phase Voltage Sensor

## **Topics**

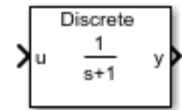
“Expand and Collapse Three-Phase Ports on a Block”

**Introduced in R2013b**

## Low-Pass Filter (Discrete or Continuous)

Discrete-time or continuous-time low-pass filter

**Library:** Simscape / Electrical / Power Systems / Control / General Control



### Description

The Low-Pass Filter (Discrete or Continuous) block implements a low-pass filter in conformance with IEEE 421.5-2016<sup>[1]</sup>. In the standard, the filter is referred to as a Simple Time Constant.

You can switch between continuous and discrete implementations of the integrator using the **Sample time** parameter.

### Equations

To configure the filter for continuous time, set the **Sample time** property to 0. This representation is equivalent to the continuous transfer function:

$$G(s) = \frac{K}{Ts + 1},$$

where:

- $K$  is the filter gain.
- $T$  is the filter time constant.

From the preceding transfer function, the filter defining equations are:

$$\begin{cases} \dot{x}(t) = \frac{1}{T}(Ku(t) - x(t)) & y(0) = x(0) = Ku_0, \\ y(t) = x(t) \end{cases}$$

where:

- $u$  is filter input.
- $x$  is filter state.
- $y$  is filter output.
- $t$  is simulation time.
- $u_0$  is the initial input to the block.

To configure the filter for discrete time, set the **Sample time** property to a positive, nonzero value, or to -1 to inherit the sample time from an upstream block. The discrete representation is equivalent to the transfer function:

$$G(z) = K \frac{(T_s / T)z^{-1}}{1 + (T_s / T - 1)z^{-1}},$$

where:

- $K$  is the filter gain.
- $T$  is the filter time constant.
- $T_s$  is the filter sample time.

From the discrete transfer function, the filter equations are defined using the forward Euler method:

$$\begin{cases} x(n+1) = \left(1 - \frac{T_s}{T}\right)x(n) + K\left(\frac{T_s}{T}\right)u(n) \\ y(n) = x(n) \end{cases} \quad y(0) = x(0) = Ku_0,$$

where:

- $u$  is the filter input.
- $x$  is the filter state.
- $y$  is the filter output.
- $n$  is the simulation time step.
- $u_0$  is the initial input to the block.

## Initial Conditions

The block sets the state and output initial conditions proportionally to the initial input.

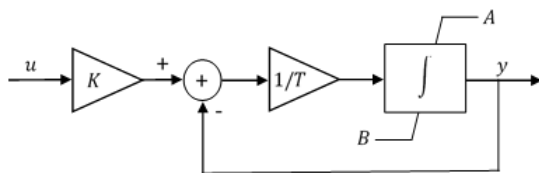
## Limiting the Integral

Set the **Upper saturation limit** and **Lower saturation limit** parameters to use the anti-windup saturation method.

The anti-windup method limits the integrator state between the lower saturation limit  $A$  and upper saturation limit  $B$ :

$$A \leq x \leq B.$$

Because the state is limited, the output can respond immediately to a reversal of the input sign when the integral is saturated. This block diagram depicts the implementation of the anti-windup saturation method in the filter.



This block does not provide a windup saturation method. To use the windup saturation method, set the **Upper saturation limit** parameter to `inf`, the **Lower saturation limit** parameter to `-inf`, and attach a saturation block to the output.

## Bypass Filter Dynamics

Set the time constant to a value smaller than or equal to the sample time to ignore the dynamics of the filter. When bypassed, the block feeds the gain-scaled input directly to the output:

$$T \leq T_s \rightarrow y = Ku$$

In the continuous case, the sample time and time constant must both be zero.

## Ports

### Input

**u — Filter input**

vector

Low-pass filter input signal. The block uses the input initial value to determine the state initial value.

Data Types: `single` | `double`

### Output

**y — Filter output**

vector

Low-pass filter output.

Data Types: `single` | `double`

## Parameters

**Gain — Filter gain**

1 (default) | positive number

Low-pass filter gain.

**Time constant — Filter time constant**

1 (default) | positive number

Low-pass filter time constant. In the discrete implementation, set this value to less than the **Sample time** to bypass the dynamics of the filter.

**Upper saturation limit — State upper limit**`inf` (default) | real number

Low-pass filter upper state limit. Set this to `inf` for an unsaturated upper limit, or to a finite value to prevent upper windup of the filter's integrator.

## Lower saturation limit — State lower limit

-inf (default) | real number

Low-pass filter lower state limit. Set this to -inf for an unsaturated lower limit, or to a finite value to prevent lower windup of the filter's integrator.

## Sample time (-1 for inherited) — Block sample time

-1 (default) | 0 | positive scalar

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For inherited discrete-time operation, specify -1. For discrete-time operation, specify a positive integer. For continuous-time operation, specify 0.

If this block is in a masked subsystem, or other variant subsystem that allows you to switch between continuous operation and discrete operation, promote the sample time parameter. Promoting the sample time parameter ensures correct switching between the continuous and discrete implementations of the block. For more information, see “Promote Parameter to Mask” (Simulink).

## References

[1] *IEEE Recommended Practice for Excitation System Models for Power System Stability Studies*. IEEE Std 421.5-2016. Piscataway, NJ: IEEE-SA, 2016.

## See Also

### Blocks

Filtered Derivative (Discrete or Continuous) | Integrator (Discrete or Continuous) | Integrator with Wrapped State (Discrete or Continuous) | Lead-Lag (Discrete or Continuous) | Washout (Discrete or Continuous)

### Introduced in R2017b

# Luenberger Observer

Discrete-time Luenberger observer

**Library:** Simscape / Electrical / Power Systems / Control / Observers



## Description

The Luenberger Observer block implements a discrete time Luenberger Observer. Use this block to estimate the states of an observable system using:

- The discrete inputs and outputs of the system.
- A discrete state-space representation of the system.

The Luenberger Observer is also sometimes referred to as a state observer or simply an observer.

You can control multi-input, multi-output systems by passing the output state vector of this block to a State Feedback Controller block.

## Defining Equations

The block implements a discrete time Luenberger Observer using the backward Euler method due to its simplicity and stability.

The estimator is given by this difference equation:

$$\check{x}(k+1) = A_d \check{x}(k) + B_d u(k) + L_d (y(k) - \check{y}(k)),$$

where:

- $\check{x}(k)$  is the  $k^{\text{th}}$  estimated state vector.
- $\check{y}(k)$  is the  $k^{\text{th}}$  estimated output vector.

- $u(k)$  is the  $k^{\text{th}}$  input vector.
- $y(k)$  is the  $k^{\text{th}}$  measured output vector.
- $A_d$  is the discretized state matrix.
- $B_d$  is the discretized input matrix.
- $L_d$  is the discretized observer gain matrix.

The dynamics of the estimation error are described by:

$$e(k+1) = (A_d - L_d C_d) e(k),$$

where:

- $e(k)$  is the  $k^{\text{th}}$  error vector.
- $C_d$  is the output matrix.

The estimation error converges to zero when  $A_d - L_d C_d$  has its eigenvalues inside the unit circle. Therefore, the value of  $L_d$  should be such that this goal is achieved. The block computes the observer gain by solving

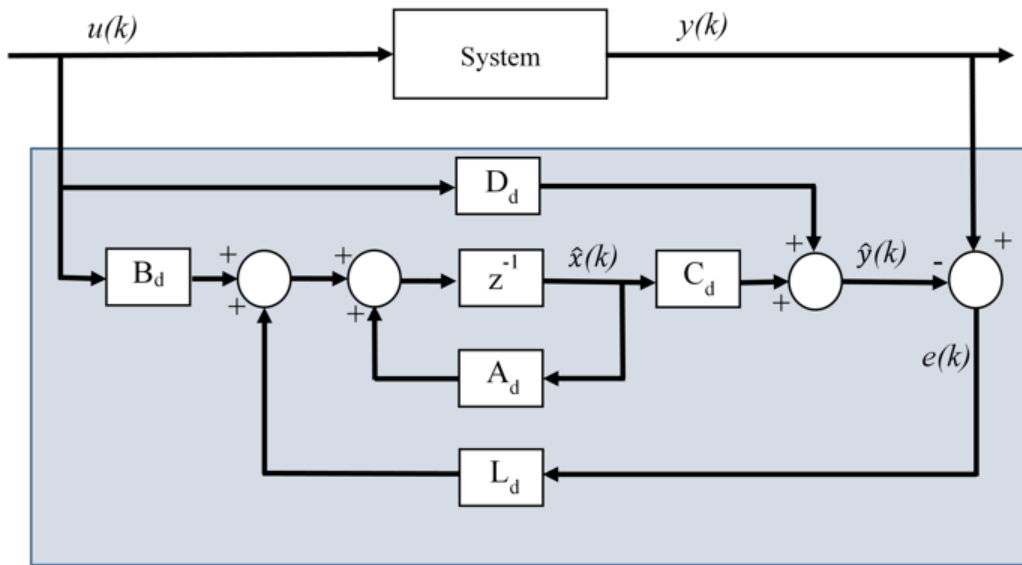
$$L_d^T = G X^{-1},$$

where  $G$  is an arbitrary matrix and  $X$  is obtained by solving the Sylvester equation:

$$A_d^T X - X \Lambda = C_d^T G.$$

Here,  $\Lambda$  is a matrix with the desired eigenvalues, which are not the same as the eigenvalues of  $A_d$ . This diagram shows the basic structure of a discrete time Luenberger Observer.





## Assumptions

The system is observable, which is true if the state of the system can be determined from the input and output in a finite time. Mathematically, this means that the system observability matrix has full rank.

## Limitations

The desired eigenvalues are not the same as the eigenvalues of the open-loop model.

## Ports

### Input

**u** — Control input  
vector

Input signal to the system whose state we want to estimate, specified as a vector.

Data Types: `single` | `double`

**y — System output**

vector

Measured output of the system whose state we want to estimate, specified as a vector.

Data Types: `single` | `double`

## Output

**xhat — State estimate**

vector

Estimate of the state of the system, specified as a vector.

Data Types: `single` | `double`

## Parameters

**State-space parameterization — State-space parameterization**

`Discrete-time` (default) | `Continuous-time`

Select the strategy for parameterizing the state-space matrices and desired poles for the observer. The block implementation is discrete regardless of this parameterization.

**Discrete A matrix — A matrix in discrete time**

1 (default) | real scalar or matrix

State matrix of the discrete-time state-space model. The A matrix must be square, with the number of rows and columns equal to the order of the system.

**Dependencies**

To enable this parameter, set **State-space parameterization** to `Discrete-time`.

**Discrete B matrix — B matrix in discrete time**

1 (default) | real scalar or matrix

Input matrix of the discrete-time state-space model. The B matrix must have the number of rows equal to the order of the system, and the number of columns equal to the number of system inputs.

**Dependencies**

To enable this parameter, set **State-space parameterization** to Discrete-time.

**Discrete C matrix — C matrix in discrete time**

1 (default) | real scalar or matrix

Output matrix of the discrete-time state-space model. The C matrix must have the number of rows equal the number of outputs of the system, and the number of columns equal to the order of the system.

**Dependencies**

To enable this parameter, set **State-space parameterization** to Discrete-time.

**Discrete D matrix — D matrix in discrete time**

1 (default) | real scalar or matrix

Feedthrough matrix of the discrete-time state-space model. The D matrix must have the number of rows equal to the number of system outputs, and the number of columns equal to the number of system inputs.

**Dependencies**

To enable this parameter, set **State-space parameterization** to Discrete-time.

**Continuous A matrix — A matrix in continuous time**

1 (default) | real scalar or matrix

State matrix of the continuous-time state-space model. The A matrix must be square, with the number of rows and columns equal to the order of the system.

**Dependencies**

To enable this parameter, set **State-space parameterization** to Continuous-time.

**Continuous B matrix — B matrix in continuous time**

1 (default) | real scalar or matrix

Input matrix of the continuous-time state-space model. The B matrix must have the number of rows equal to the order of the system, and the number of columns equal to the number of system inputs.

### **Dependencies**

To enable this parameter, set **State-space parameterization** to Continuous-time.

### **Continuous C matrix — C matrix in continuous time**

1 (default) | real scalar or matrix

Output matrix of the continuous-time state-space model. The C matrix must have the number of rows equal the number of outputs of the system, and the number of columns equal to the order of the system.

### **Dependencies**

To enable this parameter, set **State-space parameterization** to Continuous-time.

### **Continuous D matrix — D matrix in continuous time**

1 (default) | real scalar or matrix

Feedthrough matrix of the continuous-time state-space model. The D matrix must have the number of rows equal to the number of system outputs, and the number of columns equal to the number of system inputs.

### **Dependencies**

To enable this parameter, set **State-space parameterization** to Continuous-time.

### **Observer design — State-space parameterization**

Desired eigenvalues (default) | Observer gain

Select the strategy for parameterizing observer gain.

### **Dependencies**

To enable this parameter, set **State-space parameterization** to Discrete-time.

### **Observer gain — Observer gain**

1 (default) | real scalar or matrix

Specify the observer gain that puts all eigenvalues of the matrix  $A_d - L_d C_d$  inside the unit circle. The gain matrix must have the number of rows equal to number of system inputs and the number of columns equal to the order of the system.

### **Dependencies**

To enable this parameter, set:

- **State-space parameterization** to Discrete-time.
- **Observer design** to Observer gain.

### Desired eigenvalues — Observer eigenvalues

0 (default) | real vector

Specify the location of the eigenvalues:

- To have negative real part if **State-space parameterization** is set to Continuous-time. In this case, the eigenvalues of the continuous-time system are approximated to the discrete ones based on the **Discretization sample time**.
- To lie within the unit circle if **State-space parameterization** is set to Discrete-time.

The Observer gain is then calculated based on these eigenvalues. The size of the vector should be the same as the system order.

### Initial conditions — Initial conditions

0 (default) | real vector with length equal to system order

Select the initial condition of each state.

### Discretization sample time — Discretization sample time

0.1 (default) | positive real number

Value used to discretize the state space matrices and also approximate the discrete-time eigenvalues.

### Dependencies

To enable this parameter, set **State-space parameterization** to Continuous-time.

### Sample time — Sample time

0.1 (default) | -1 or positive real number

Value used to simulate the dynamics of the model. Choose the same value as **Discretization sample time**, unless the block is placed within a triggered subsystem, in which case you must set it to -1.

## References

- [1] Luenberger, D. G. "An Introduction to Observers." *IEEE Transactions on Automatic Control*. Vol. 16, Number 6, 1971, pp. 596-602.
- [2] Alessandri, A., and P. Coletta. "Design of Luenberger observers for a class of hybrid linear systems." *In International Workshop on Hybrid Systems: Computation and Control*, Berlin, March 2001.
- [3] Varga, A. "Robust pole assignment via Sylvester equation based state feedback parametrization." *In Computer-Aided Control System Design*, pp. 13-18., Anchorage, Alaska, 2000.

## See Also

### Blocks

ASM Flux Observer | State Feedback Controller

**Introduced in R2017b**

# Machine Inertia

Machine inertia

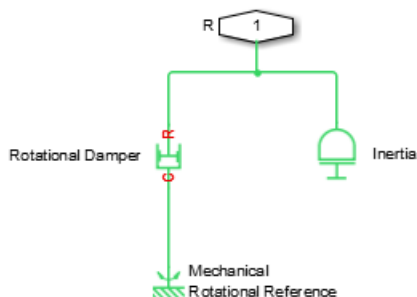


## Library

Simscape / Electrical / Power Systems / Machines

## Description

The Machine Inertia block models inertia and damping that you connect to the mechanical rotational R port of a three-phase machine. The block has an internal connection to a mechanical rotational reference. The figure shows an equivalent configuration to the Machine Inertia block using Simscape mechanical rotational components.



Based on the value you select for the `Specify inertia` parameterization by parameter, you specify inertia  $J$  directly or using the machine inertia constant  $H$ .

If you specify the inertia constant, the block calculates inertia by

$$J = \frac{2HS_{rated}}{(2\pi F_{rated} / N)^2},$$

where:

- $J$  is inertia in  $\text{kg}\cdot\text{m}^2$ .
- $H$  is the inertia constant in  $\text{sW/VA}$ .
- $S_{rated}$  is the machine rated apparent power in VA.
- $F_{rated}$  is the machine rated electrical frequency in Hz.
- $N$  is the number of machine pole pairs.

You specify damping that represents viscous friction between the machine rotor and mechanical rotational reference. Based on the value you select for the **Specify damper parameterization** by parameter, you specify a damping coefficient in SI units or in per-unit. If you specify the damping coefficient in per-unit, the block calculates the damping coefficient in SI units by

$$\omega_{base} = \frac{2\pi F_{rated}}{N},$$

$$T_{base} = \frac{S_{rated}}{\omega_{base}},$$

$$D_{base} = \frac{T_{base}}{\omega_{base}},$$

and

$$D = D_{pu} D_{base},$$

where:

- $\omega_{base}$  is the base mechanical speed in rad/s.
- $T_{base}$  is the base damping torque in Nm.



- $D_{base}$  is the base damping coefficient in Nm/(rad/s).
- $D_{pu}$  is the damping coefficient in per-unit.
- $D$  is the damping coefficient in SI units of Nm/(rad/s).

## Display Option

You can display machine parameters using the **Electrical** menu on the block context menu.

Right-click the block and, from the **Electrical** menu, select **Display Parameters** to display the machine per-unit base values and inertia parameters in the MATLAB Command Window.

## Parameters

- “Main Tab” on page 1-835
- “Inertia Tab” on page 1-835
- “Initial Conditions Tab” on page 1-836

### Main Tab

#### Rated apparent power

Machine rated apparent power. The default value is 555e6 V\*A.

#### Rated electrical frequency

Nominal electrical frequency corresponding to the machine rated apparent power. The default value is 60 Hz.

#### Number of pole pairs

Number of machine pole pairs. The default value is 1.

### Inertia Tab

#### Specify inertia parameterization by

Inertia specification. The default value is Inertia constant, H.

### **Inertia constant, H**

Inertia constant. This parameter is visible only if you set **Specify inertia parameterization by** to `Inertia constant, H`. The default value is 3.525 sW/VA.

### **Actual inertia, J**

Inertia. This parameter is visible only if you set **Specify inertia parameterization by** to `Actual inertia, J`. The default value is 27548 kg·m<sup>2</sup>.

### **Specify damper parameterization by**

Damping specification. The default value is `Per-unit damping coefficient, pu_D`.

### **Per-unit damping coefficient**

Damping coefficient in per-unit. This parameter is visible only if you set **Specify damper parameterization by** to `Per-unit damping coefficient, pu_D`. The default value is 0.01.

### **SI damping coefficient**

Damping coefficient in SI units. This parameter is visible only if you set **Specify damper parameterization by** to `SI damping coefficient, D`. The default value is 39.0509 Nm/(rad/s).

## **Initial Conditions Tab**

### **Specify initialization by**

Frequency initialization. The default value is `Initial electrical frequency`.

### **Initial electrical frequency**

Initial electrical frequency. This parameter is visible only if you set **Specify initialization by** to `Initial electrical frequency`. The default value is 60 Hz.

### **Initial mechanical frequency**

Initial mechanical frequency. This parameter is visible only if you set **Specify initialization by** to `Initial mechanical frequency`. The default value is 60 Hz.

## **Ports**

The block has the following ports:

R

Mechanical rotational conserving port associated with the machine rotor

## **References**

[1] Kundur, P. *Power System Stability and Control*. New York, NY: McGraw Hill, 1993.

**Introduced in R2013b**

## Machine Mechanical Power (pu)

Machine mechanical power defined in the per-unit system



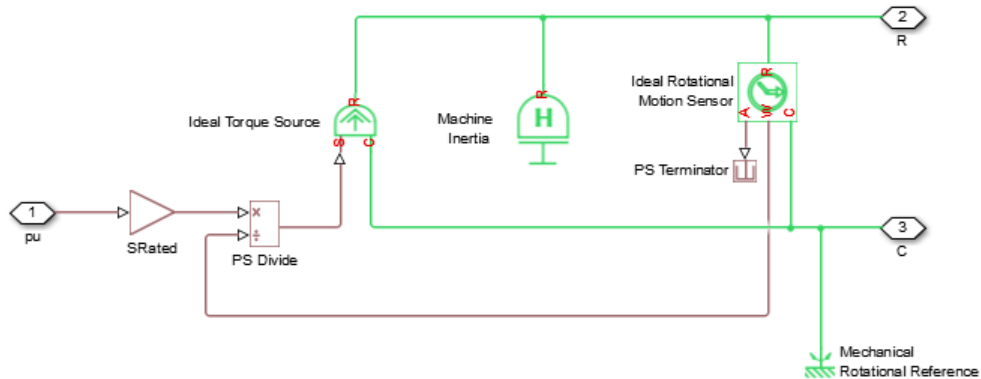
### Library

Simscape / Electrical / Power Systems / Machines

### Description

The Machine Mechanical Power (pu) block supplies specified power to, or draws specified power from, the machine that it connects to. It includes a representation of machine inertia and a mechanical rotational reference. In generator mode, the physical signal input pu defines the per-unit mechanical power that is input to the machine. In motor mode, it defines the mechanical power output from the machine. The pu input must always be positive.

The figure shows an equivalent configuration to the Machine Mechanical Power (pu) block using Simscape mechanical rotational components.



## Electrical Defining Equations

To calculate the torque that it applies to the inertia, the block divides the power demand by the present speed. To set the peak torque limit, specify a value for the **Peak torque to rated torque ratio** parameter. Use the **Specify inertia parameterization by** parameter to specify inertia,  $J$ , directly or indirectly, with the inertia constant for the machine,  $H$ .

If you specify the inertia constant for the machine, the block calculates inertia as

$$J = \frac{2HS_{rated}}{(2\pi F_{rated} / N)^2},$$

where:

- $J$  is inertia in  $\text{kg}\cdot\text{m}^2$ .
- $H$  is the inertia constant in  $\text{sW/VA}$ .
- $S_{rated}$  is the rated apparent power of the connected machine in VA.
- $F_{rated}$  is the rated electrical frequency of the connected machine in Hz.
- $N$  is the number of machine pole pairs.

Damping represents viscous friction between the machine rotor and mechanical rotational reference. Based on the value you select for the **Specify damper parameterization by** parameter, you specify a damping coefficient in per-unit or in SI

units. If you specify the damping coefficient in per-unit, the block calculates the damping coefficient in SI units using these equations:

$$\omega_{base} = \frac{2\pi F_{rated}}{N},$$

$$T_{base} = \frac{S_{rated}}{\omega_{base}},$$

$$D_{base} = \frac{T_{base}}{\omega_{base}},$$

and

$$D = D_{pu} D_{base},$$

where:

- $\omega_{base}$  is the base mechanical speed in rad/s.
- $T_{base}$  is the base damping torque in Nm.
- $D_{base}$  is the base damping coefficient in Nm/(rad/s).
- $D_{pu}$  is the damping coefficient in per-unit.
- $D$  is the damping coefficient in SI units of Nm/(rad/s).

## Parameters

- “Main Tab” on page 1-840
- “Inertia Tab” on page 1-841
- “Initial Conditions Tab” on page 1-842

### Main Tab

#### Input power sign convention

Machine type specification. The choices are `Generator` and `Motor`. The default value is `Generator`.

**Rated apparent power**

Rated apparent power of the connected machine. The default value is 555e6 V\*A.

**Rated electrical frequency**

Nominal electrical frequency corresponding to the rated apparent power of the connected machine. The default value is 60 Hz.

**Number of pole pairs**

Number of pole pairs of the connected machine. The default value is 1.

**Peak torque to rated torque ratio**

Ratio that the block multiplies by the base torque to provide the upper limit for the torque that accelerates the inertia. The default value is 2.

**Inertia Tab****Specify inertia parameterization by**

Inertia specification. The choices are between Actual Inertia, J and Inertia constant, H. The default value is Inertia constant, H.

**Inertia constant, H**

Inertia constant. This parameter is visible only if you set **Specify inertia parameterization by** to Inertia constant, H. The default value is 3.525 s\*W/VA.

**Actual inertia, J**

Inertia. This parameter is visible only if you set **Specify inertia parameterization by** to Actual Inertia, J. The default value is 27548 kg·m<sup>2</sup>.

**Specify damper parameterization by**

Damping specification. The choices are Per-unit damping coefficient, pu\_D and SI damping coefficient, D. The default value is Per-unit damping coefficient, pu\_D.

**Per-unit damping coefficient**

Damping coefficient in per-unit. This parameter is visible only if you set **Specify damper parameterization by** to Per-unit damping coefficient, pu\_D. The default value is 0.01.

### **SI damping coefficient**

Damping coefficient in SI units. This parameter is visible only if you set **Specify damper parameterization by** to SI damping coefficient, D. The default value is 39.0509 Nm/(rad/s).

## **Initial Conditions Tab**

### **Specify initialization by**

Frequency initialization. The choices are Initial electrical frequency and Initial mechanical frequency. The default value is Initial electrical frequency.

### **Initial electrical frequency**

Initial electrical frequency. This parameter is visible only if you set **Specify initialization by** to Initial electrical frequency. The default value is 60 Hz.

### **Initial mechanical frequency**

Initial mechanical frequency. This parameter is visible only if you set **Specify initialization by** to Initial mechanical frequency. The default value is 60 Hz.

## **Ports**

The block has the following ports:

pu

Physical signal input port associated with mechanical power, per-unit

R

Mechanical rotational conserving port associated with the machine rotor

C

Mechanical rotational conserving port associated with the machine case

## **See Also**

Machine Inertia | Machine Mechanical Power (SI)



**Introduced in R2014b**

## Machine Mechanical Power (SI)

Machine mechanical power defined in SI units



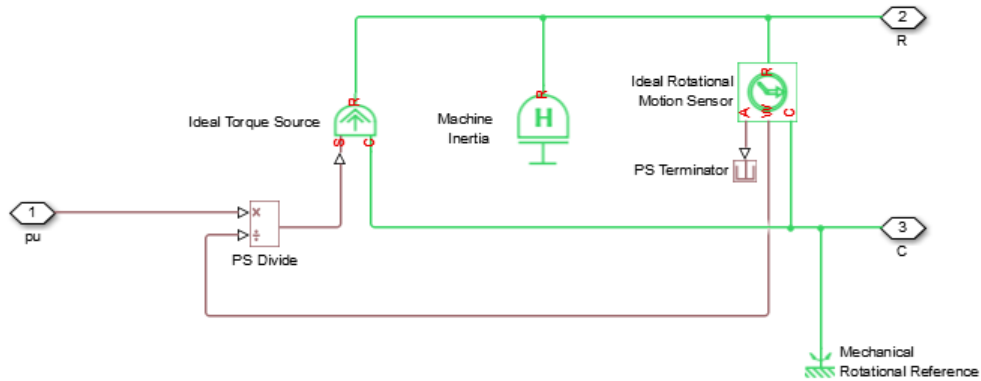
### Library

Simscape / Electrical / Power Systems / Machines

### Description

The Machine Mechanical Power (SI) block supplies specified power to, or draws specified power from, the machine that it connects to. It includes a representation of machine inertia and a mechanical rotational reference. In generator mode, the physical signal input  $P_m$  defines the mechanical power in SI units that is input to the machine. In motor mode, it defines the mechanical power output from the machine. The  $P_m$  input must always be positive.

The figure shows an equivalent configuration to the Machine Mechanical Power (SI) block using Simscape mechanical rotational components.



## Electrical Defining Equations

To calculate the torque that it applies to the inertia, the block divides the power demand by the present speed. To set the peak torque limit, specify a value for the **Peak torque to rated torque ratio** parameter. Use the **Specify inertia parameterization by** parameter to specify inertia,  $J$ , directly or indirectly, with the inertia constant for the machine,  $H$ .

If you specify the inertia constant for the machine, the block calculates inertia as

$$J = \frac{2HS_{rated}}{(2\pi F_{rated} / N)^2},$$

where:

- $J$  is inertia in  $\text{kg}\cdot\text{m}^2$ .
- $H$  is the inertia constant in  $\text{sW/VA}$ .
- $S_{rated}$  is the rated apparent power of the connected machine in VA.
- $F_{rated}$  is the rated electrical frequency of the connected machine in Hz.
- $N$  is the number of pole pairs in the connected machine.

Damping represents viscous friction between the machine rotor and mechanical rotational reference. Based on the value you select for the **Specify damper parameterization by** parameter, you specify a damping coefficient in per-unit or in SI

units. If you specify the damping coefficient in per-unit, the block calculates the damping coefficient in SI units using these equations:

$$\omega_{base} = \frac{2\pi F_{rated}}{N},$$

$$T_{base} = \frac{S_{rated}}{\omega_{base}},$$

$$D_{base} = \frac{T_{base}}{\omega_{base}},$$

and

$$D = D_{pu} D_{base},$$

where:

- $\omega_{base}$  is the base mechanical speed in rad/s.
- $T_{base}$  is the base damping torque in Nm.
- $D_{base}$  is the base damping coefficient in Nm/(rad/s).
- $D_{pu}$  is the damping coefficient in per-unit.
- $D$  is the damping coefficient in SI units of Nm/(rad/s).

## Parameters

- “Main Tab” on page 1-846
- “Inertia Tab” on page 1-847
- “Initial Conditions Tab” on page 1-848

### Main Tab

#### Input power sign convention

Machine type specification. The choices are `Generator` and `Motor`. The default value is `Generator`.

**Rated apparent power**

Rated apparent power of the connected machine. The default value is 555e6 V\*A.

**Rated electrical frequency**

Nominal electrical frequency corresponding to the rated apparent power of the connected machine. The default value is 60 Hz.

**Number of pole pairs**

Number of pole pairs of the connected machine. The default value is 1.

**Peak torque to rated torque ratio**

Ratio that the block multiplies by the base torque to provide the upper limit for the torque that accelerates the inertia. The default value is 2.

**Inertia Tab****Specify inertia parameterization by**

Inertia specification. The choices are Actual Inertia, J and Inertia constant, H. The default value is Inertia constant, H.

**Inertia constant, H**

Inertia constant. This parameter is visible only if you set **Specify inertia parameterization by** to Inertia constant, H. The default value is 3.525 s\*W/VA.

**Actual inertia, J**

Inertia. This parameter is visible only if you set **Specify inertia parameterization by** to Actual Inertia, J. The default value is 27548 kg·m<sup>2</sup>.

**Specify damper parameterization by**

Damping specification. The choices are Per-unit damping coefficient, pu\_D and SI damping coefficient, D. The default value is Per-unit damping coefficient, pu\_D.

**Per-unit damping coefficient**

Damping coefficient in per-unit. This parameter is visible only if you set **Specify damper parameterization by** to Per-unit damping coefficient, pu\_D. The default value is 0.01.

### **SI damping coefficient**

Damping coefficient in SI units. This parameter is visible only if you set **Specify damper parameterization by** to SI damping coefficient, D. The default value is 39.0509 Nm/(rad/s).

## **Initial Conditions Tab**

### **Specify initialization by**

Frequency initialization. The choices are Initial electrical frequency and Initial mechanical frequency. The default value is Initial electrical frequency.

### **Initial electrical frequency**

Initial electrical frequency. This parameter is visible only if you set **Specify initialization by** to Initial electrical frequency. The default value is 60 Hz.

### **Initial mechanical frequency**

Initial mechanical frequency. This parameter is visible only if you set **Specify initialization by** to Initial mechanical frequency. The default value is 60 Hz.

## **Ports**

The block has the following ports:

Pm

Physical signal input port associated with mechanical power, W

R

Mechanical rotational conserving port associated with the machine rotor

C

Mechanical rotational conserving port associated with the machine case

## **See Also**

Machine Inertia | Machine Mechanical Power (pu)

## **Topics**

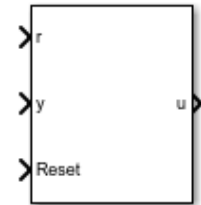
Three-Phase Asynchronous Machine Starting

**Introduced in R2014b**

# Model Reference Adaptive Controller

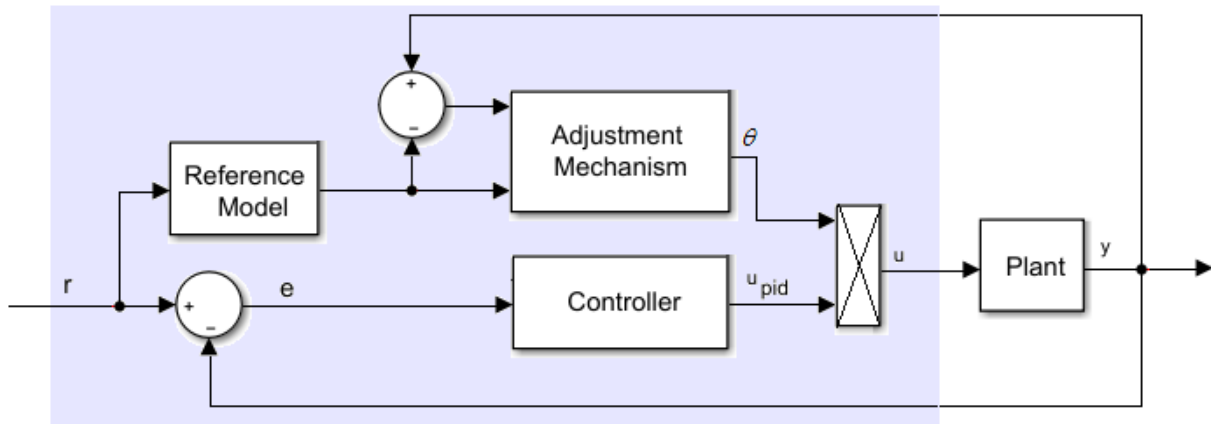
Discrete-time PID-based model reference adaptive control

**Library:** Simscape / Electrical / Power Systems / Control / General Control



## Description

The Model Reference Adaptive Controller block implements discrete-time proportional-integral-derivative (PID) model reference adaptive control (MRAC). The three main components of an MRAC system are the reference model, the adjustment mechanism, and the controller.





## Equations

The control equation is

$$u_{pid}(k) = \left[ K_p + K_i \frac{T_s z}{z-1} + K_d \frac{z-1}{T_s z} \right] e(k),$$

where:

- $u_{pid}$  is the controller output.
- $K_p$  is the proportional gain.
- $K_i$  is the integral gain.
- $K_d$  is the differential gain.
- $T_s$  is the sample time.
- $e$  is the error.

The reference model is the transfer function for the closed-loop system. This model captures the desired behavior of the closed-loop system. It is implemented as the discrete-time transfer function

$$G_m(z) = \frac{B(z)}{A(z)}.$$

The adaptation mechanism adjusts the control action based on the error between the plant output and the reference model output as

$$\theta = (y - y_m) y_m \frac{-\gamma T_s z}{z-1},$$

where:

- $\theta$  is the adaptation parameter.
- $y$  is the plant output.
- $y_m$  is the reference model output.
- $\gamma$  is the learning rate.

Increasing the value of  $\gamma$  results in faster adaptation to plant changes.

The adjusted control signal,  $u$ , is

$$u(k) = u_{pid}(k)\theta(k).$$

## Ports

### Input

**r — Plant reference,  $r$**

scalar

Plant system reference signal.

Data Types: `single` | `double`

**y — Plant output,  $y$**

scalar

Plant system output signal.

Data Types: `single` | `double`

**Reset — Integrator reset**

scalar

External reset signal (rising edge) for the integrator.

Data Types: `Boolean`

### Output

**u — Controller adjusted output,  $u$**

scalar

Adjusted control signal.

Data Types: `single` | `double`

# Parameters

## Controller Parameters

### Proportional gain — Controller proportional gain, $K_p$

1 (default) | positive scalar

Proportional gain,  $K_p$ , of the controller.

### Integral gain — Controller integral gain, $K_i$

1 (default) | positive scalar

Integral gain,  $K_i$ , of the controller.

### Derivative gain — Controller derivative gain, $K_d$

0 (default)

Derivative gain,  $K_d$ , of the controller.

### Anti-windup gain — Controller anti-windup gain

1 (default) | positive scalar

Anti-windup gain of the controller.

### Use filtered derivative — Filter option

on (default) | off

Choose whether to use a filter coefficient for the reference signal.

#### Dependencies

The **Filter coefficient** parameter is only visible when the **Use filtered derivative** check box is selected.

### Filter coefficient — Reference signal filter coefficient

100 (default)

Filter coefficient for the reference signal.

#### Dependencies

This parameter is only visible when the **Use filtered derivative** check box is selected.

**Control action upper limit — Control signal upper limit,  $u_{pid\_max}$**   
5 (default)

Upper bound for the control signal.

**Control action lower limit — Control signal lower limit,  $u_{pid\_min}$**   
-5 (default)

Lower bound for the control signal.

**Sample time (-1 for inherited) — Block sample time**  
-1 (default) | positive scalar

Time, in s, between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

If this block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly using a positive scalar.

#### **Dependencies**

If you set **Sample time (-1 for inherited)** to -1 and, in the **Reference Model** settings, set **Model parameterization** to Continuous-time, the **Discretization sample time** parameter becomes visible in the **Reference Model** settings.

## **Reference Model**

**Model parameterization — Discrete or continuous model option**  
Discrete-time (default) | Continuous-time

Mathematical model for the controller.

#### **Dependencies**

Choosing:

- Discrete-time makes the discrete-time parameters visible.
- Continuous-time makes the continuous-time parameters visible. Also, in the **Controller Parameter** settings, if **Sample time (-1 for inherited)** is set to -1, choosing this option makes the **Discretization sample time** parameter visible in the **Reference Model** settings.

**Discrete-time numerator — Discrete-time transfer function numerator**

[0.01 -0.0099] (default)

Numerator for the discrete-time transfer function.

**Dependencies**

Choosing Discrete-time for **Model parameterization** makes this parameter visible.

**Discrete-time denominator — Discrete-time transfer function denominator**

[1 -1.9801 0.9802] (default) | vector

Denominator for the discrete-time transfer function.

**Dependencies**

Choosing Discrete-time for **Model parameterization** makes this parameter visible.

**Discretization sample time — Sample time for discretization**

0.01 (default) | -1 or positive number

Time, in seconds, between consecutive discretizations. If block sample time is inherited, specify the discretization sample time explicitly.

**Dependencies**

This parameter is only visible when both of these conditions are met:

- In the **Control Parameters** settings, **Sample time (-1 for inherited)** is set to -1.
- In the **Reference Model** settings, **Model parameterization** is set to Continuous-time.

**Continuous-time numerator — Continuous-time transfer function numerator**

[1 1] (default) | vector

Numerator for the continuous-time transfer function.

**Dependencies**

Choosing Continuous-time for **Model parameterization** makes this parameter visible.

**Continuous-time denominator — Continuous-time transfer function denominator**

[1 2 1] (default) | vector

Denominator for the continuous-time transfer function.

**Dependencies**

Choosing Continuous-time for **Model parameterization** makes this parameter visible.

**Adjustment Mechanism**

**Learning rate —  $\gamma$**

0.5 (default)

Rate of adjustment to plant changes.

**References**

[1] Butler, H. *Model-Reference Adaptive Control-From Theory to Practice*. Upper Saddle River, NJ: Prentice Hall, 1992.

**See Also**

RST Controller

**Introduced in R2018a**

# Monostable Flip-Flop

Monostable flip-flop

**Library:** Simscape / Electrical / Power Systems / Control / General Control



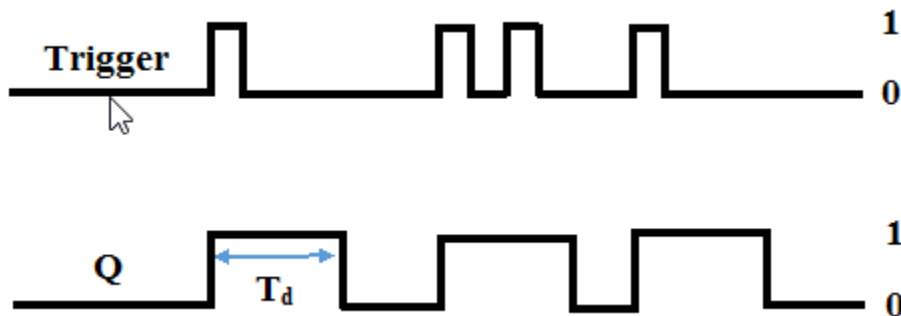
## Description

The Monostable Flip-Flop (or monostable multivibrator) block generates a single output pulse of a specified duration when it is triggered externally. The external trigger is a Boolean signal. Pulse generation is triggered when a change is detected in the external trigger signal. The change detection can be:

- Rising edge
- Falling edge
- Either edge

When the output is true, a change in the external trigger signal has no effect.

The operation of a monostable flip-flop is represented in the following figure:



## Ports

### Input

#### **Trigger — Flip-flop trigger**

0 | 1

Input trigger signal to the flip-flop.

Data Types: Boolean

### Output

#### **Q — Flip-flop output**

0 | 1

Flip-flop output signal.

Data Types: Boolean

## Parameters

#### **Change detection — Change in trigger signal**

Rising edge (default) | Falling edge | Either edge

Change detection strategy for output pulse generation.

#### **Initial condition — Flip-flop initial condition**

0 (default) | scalar

The initial condition of the flip-flop.

#### **Pulse duration (s) — Output pulse duration**

0.01 (default) | positive scalar

Duration of the output pulse, in s.

#### **Sample time — Block sample time**

0.001 (default) | 0 | positive scalar



Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For discrete-time operation, set the sample time to a positive value. For continuous-time operation, set the sample time to 0.

If this block is in a masked subsystem, or other variant subsystem that allows either continuous and discrete operation, promote the sample time parameter. Promoting the sample time parameter ensures correct switching between the continuous and discrete implementations of the block. For more information, see “Promote Parameter to Mask” (Simulink).

## See Also

### Blocks

Change Detector | Set-Reset Flip-Flop | Signal Sample and Hold

**Introduced in R2018b**

# MOSFET

N-channel MOSFET



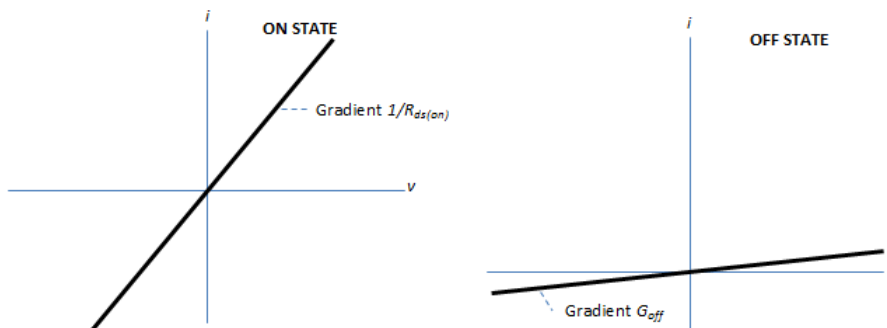
## Library

Simscape / Electrical / Power Systems / Semiconductors / Fundamental Components

## Description

The MOSFET block models an n-channel metal-oxide-semiconductor field-effect transistor (MOSFET).

The I-V characteristic of an n-channel MOSFET is such that if the gate-source voltage exceeds the specified threshold voltage, the MOSFET is in the on state. Otherwise, the device is in the off state.



In the on state, the drain-source path behaves like a linear resistor with resistance,  $R_{ds\_on}$ .

In the off state, the drain-source path behaves like a linear resistor with low off-state conductance,  $G_{off}$ .

The defining Simscape equations for the block are:

```

if G > Vth
    v == i*Rds_on;
else
    v == i/Goff;
end

```

where:

- $G$  is the gate-source voltage.
- $Vth$  is the threshold voltage.
- $v$  is the drain-source voltage.
- $i$  is the drain-source current.
- $Rds\_on$  is the on-state resistance.
- $Goff$  is the off-state conductance.

Using the Integral Diode tab of the block dialog box, you can include an integral source-drain diode. An integral diode protects the semiconductor device by providing a conduction path for reverse current. An inductive load can produce a high reverse-voltage spike when the semiconductor device suddenly switches off the voltage supply to the load.

Set the **Integral protection diode** parameter based on your goal.

Goal	Value to Select	Block Behavior
Prioritize simulation speed.	Protection diode with no dynamics	The block includes an integral copy of the Diode block. To parameterise the internal Diode block, use the <b>Protection</b> parameters.

Goal	Value to Select	Block Behavior
Precisely specify reverse-mode charge dynamics.	Protection diode with charge dynamics	The block includes an integral copy of the Commutation Diode block. To parameterise the internal Commutation Diode block, use the <b>Protection</b> parameters.

## Modeling Variants

The block provides four modeling variants. To select the desired variant, right-click the block in your model. From the context menu, select **Simscape > Block choices**, and then one of these variants:

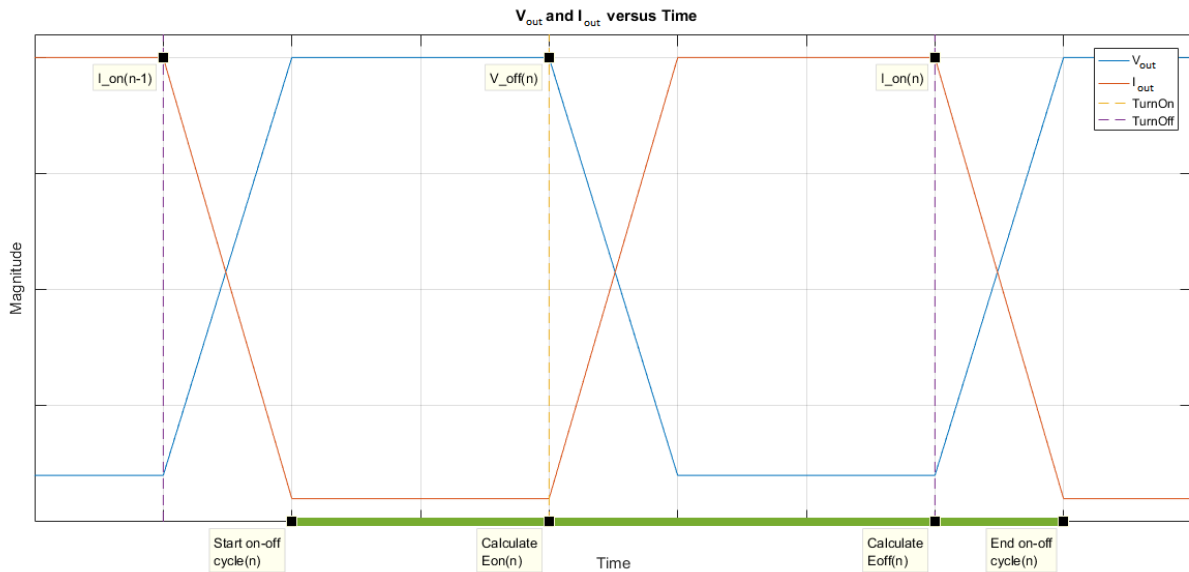
- **PS Control Port** — Contains a physical signal port that is associated with the gate terminal. This variant is the default.
- **Electrical Control Port** — Contains an electrical conserving port that is associated with the gate terminal.
- **PS Control Port | Thermal Port** — Contains a thermal port and a physical signal port that is associated with the gate terminal.
- **Electrical Control Port | Thermal Port** — Contains a thermal port and an electrical conserving port that is associated with the gate terminal.

The variants of this block without the thermal port do not simulate heat generation in the device.

The variants with the thermal port allow you to model the heat that switching events and conduction losses generate. For numerical efficiency, the thermal state does not affect the electrical behavior of the block. The thermal port is hidden by default. To enable the thermal port, select a thermal block variant.

## Thermal Loss Equations

The figure shows an idealized representation of the output voltage,  $V_{out}$ , and the output current,  $I_{out}$ , of the semiconductor device. The interval shown includes the entire  $n$ th switching cycle, during which the block turns off and then on.



When the semiconductor turns on during the  $n$ th switching cycle, the amount of thermal energy that the device dissipates increments by a discrete amount. If you select **Voltage, current, and temperature** for the **Thermal loss dependent on** parameter, the equation for the incremental change is

$$E_{on(n)} = \frac{V_{off(n)}}{V_{off\_data}} fcn(T, I_{on(n-1)}),$$

where:

- $E_{on(n)}$  is the switch-on loss at the  $n$ th switch-on event.
- $V_{off(n)}$  is the off-state output voltage,  $V_{out}$ , just before the device switches on during the  $n$ th switching cycle.
- $V_{off\_data}$  is the **Off-state voltage for losses data** parameter value.
- $T$  is the device temperature.
- $I_{on(n-1)}$  is the on-state output current,  $I_{out}$ , just before the device switches off during the cycle that precedes the  $n$ th switching cycle.

The function  $fcn$  is a 2-D lookup table with linear interpolation and linear extrapolation:

$$E = \text{tablelookup}(T_{j\_data}, I_{out\_data}, E_{on\_data}, T, I_{on(n-1)}),$$

where:

- $T_{j\_data}$  is the **Temperature vector, Tj** parameter value.
- $I_{out\_data}$  is the **Output current vector, Iout** parameter value.
- $E_{on\_data}$  is the **Switch-on loss, Eon=fcn(Tj,Iout)** parameter value.

If you select **Voltage** and **current** for the **Thermal loss dependent on** parameter, when the semiconductor turns on during the  $n$ th switching cycle, the equation that the block uses to calculate the incremental change in the discrete amount of thermal energy that the device dissipates is

$$E_{on(n)} = \left( \frac{V_{off(n)}}{V_{off\_data}} \right) \left( \frac{I_{on(n-1)}}{I_{out\_scalar}} \right) (E_{on\_scalar})$$

where:

- $I_{out\_scalar}$  is the **Output current, Iout** parameter value.
- $E_{on\_scalar}$  is the **Switch-on loss** parameter value.

When the semiconductor turns off during the  $n$ th switching cycle, the amount of thermal energy that the device dissipates increments by a discrete amount. If you select **Voltage, current, and temperature** for the **Thermal loss dependent on** parameter, the equation for the incremental change is

$$E_{off(n)} = \frac{V_{off(n)}}{V_{off\_data}} \text{fcn}(T, I_{on(n)}),$$

where:

- $E_{off(n)}$  is the switch-off loss at the  $n$ th switch-off event.
- $V_{off(n)}$  is the off-state output voltage,  $V_{out}$ , just before the device switches on during the  $n$ th switching cycle.
- $V_{off\_data}$  is the **Off-state voltage for losses data** parameter value.
- $T$  is the device temperature.

- $I_{on(n)}$  is the on-state output current,  $I_{out}$ , just before the device switches off during the  $n$ th switching cycle.

The function  $fcn$  is a 2-D lookup table with linear interpolation and linear extrapolation:

$$E = \text{tablelookup}(T_{j\_data}, I_{out\_data}, E_{off\_data}, T, I_{on(n)}),$$

where:

- $T_{j\_data}$  is the **Temperature vector**, **Tj** parameter value.
- $I_{out\_data}$  is the **Output current vector**, **Iout** parameter value.
- $E_{off\_data}$  is the **Switch-off loss**, **Eoff=fcn(Tj,Iout)** parameter value.

If you select **Voltage** and **current** for the **Thermal loss dependent on** parameter, when the semiconductor turns off during the  $n$ th switching cycle, the equation that the block uses to calculate the incremental change in the discrete amount of thermal energy that the device dissipates is

$$E_{off(n)} = \left( \frac{V_{off(n)}}{V_{off\_data}} \right) \left( \frac{I_{on(n-1)}}{I_{out\_scalar}} \right) (E_{off\_scalar})$$

where:

- $I_{out\_scalar}$  is the **Output current**, **Iout** parameter value.
- $E_{off\_scalar}$  is the **Switch-off loss** parameter value.

If you select **Voltage**, **current**, and **temperature** for the **Thermal loss dependent on** parameter, then, for both the on state and the off state, the heat loss due to electrical conduction is

$$E_{conduction} = \int fcn(T, I_{out}) dt,$$

where:

- $E_{conduction}$  is the heat loss due to electrical conduction.
- $T$  is the device temperature.
- $I_{out}$  is the device output current.

The function *fcn* is a 2-D lookup table:

$$Q_{conduction} = \text{tablelookup}(T_{j\_data}, I_{out\_data}, I_{out\_data\_repmat} .* V_{on\_data}, T, I_{out}),$$

where:

- $T_{j\_data}$  is the **Temperature vector, Tj** parameter value.
- $I_{out\_data}$  is the **Output current vector, Iout** parameter value.
- $I_{out\_data\_repmat}$  is a matrix that contains length,  $T_{j\_data}$ , copies of  $I_{out\_data}$ .
- $V_{on\_data}$  is the **On-state voltage, Von=fcn(Tj,Iout)** parameter value.

If you select **Voltage** and **current** for the **Thermal loss dependent on** parameter, then, for both the on state and the off state, the heat loss due to electrical conduction is

$$E_{conduction} = \int (I_{out} * V_{on\_scalar}) dt,$$

where  $V_{on\_scalar}$  is the **On-state voltage** parameter value.

The block uses the **Energy dissipation time constant** parameter to filter the amount of heat flow that the block outputs. The filtering allows the block to:

- Avoid discrete increments for the heat flow output
- Handle a variable switching frequency

The filtered heat flow is

$$Q = \frac{1}{\tau} \left( \sum_{i=1}^n E_{on(i)} + \sum_{i=1}^n E_{off(i)} + E_{conduction} - \int Q dt \right),$$

where:

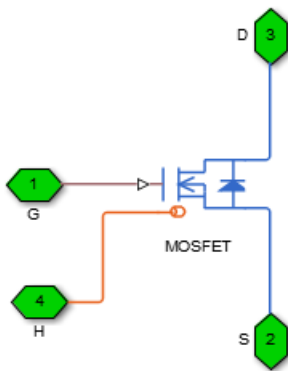
- $Q$  is the heat flow from the component.
- $\tau$  is the **Energy dissipation time constant** parameter value.
- $n$  is the number of switching cycles.
- $E_{on(i)}$  is the switch-on loss at the  $i$ th switch-on event.



- $E_{off(i)}$  is the switch-off loss at the  $i$ th switch-off event.
- $E_{conduction}$  is the heat loss due to electrical conduction.
- $\int Qdt$  is the total heat previously dissipated from the component.

## Ports

The figure shows the block port names.



G

Port associated with the gate terminal. You can set the port to either a physical signal or electrical port.

S

Electrical conserving port associated with the source terminal.

D

Electrical conserving port associated with the drain terminal.

H

Thermal conserving port. The thermal port is optional and is hidden by default. To enable this port, select a variant that includes a thermal port.

## Parameters

- “Main Tab” on page 1-868
- “Integral Diode Tab” on page 1-868
- “Thermal Model Tab” on page 1-871

### Main Tab

#### On-state resistance, $R_{DS(on)}$

Drain-source resistance when the device is on. The default value is 0.01 Ohm.

#### Off-state conductance

Drain-source conductance when the device is off. The value must be less than  $1/R$ , where  $R$  is the value of **On-state resistance**. The default value is  $1e-6$  1/Ohm.

#### Threshold voltage, $V_{th}$

Gate-source voltage threshold. The device turns on when the gate-source voltage is above this value. The default value is 2 V.

### Integral Diode Tab

#### Integral protection diode

Block integral protection diode. The default value is Protection diode with no dynamics.

The diodes you can select are:

- Protection diode with no dynamics
- Protection diode with charge dynamics

When you select Protection diode with no dynamics, additional parameters appear.

## **Additional Parameters for Protection diode with no dynamics**

### **Forward voltage**

Minimum voltage required across the + and - block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**. The default value is 0.8 V.

### **On resistance**

Rate of change of voltage versus current above the **Forward voltage**. The default value is 0.001 Ohm.

### **Off conductance**

Conductance of the reverse-biased diode. The default value is  $1e-5$  1/Ohm.

For more information on these parameters, see Diode.

When you select Protection diode with charge dynamics, additional parameters appear.

## **Additional Parameters for Protection diode with charge dynamics**

### **Forward voltage**

Minimum voltage required across the + and - block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**. The default value is 0.8 V.

### **On resistance**

Rate of change of voltage versus current above the **Forward voltage**. The default value is 0.001 Ohm.

### **Off conductance**

Conductance of the reverse-biased diode. The default value is  $1e-5$  1/Ohm.

### **Junction capacitance**

Diode junction capacitance. The default value is 50 nF.

### **Peak reverse current, iRM**

Peak reverse current measured by an external test circuit. This value must be less than zero. The default value is -235 A.

### **Initial forward current when measuring iRM**

Initial forward current when measuring peak reverse current. This value must be greater than zero. The default value is 300 A.

### **Rate of change of current when measuring $i_{RM}$**

Rate of change of current when measuring peak reverse current. This value must be less than zero. The default value is  $-50 \text{ A}/\mu\text{s}$ .

### **Reverse recovery time parameterization**

Determines how you specify reverse recovery time in the block. The default value is **Specify reverse recovery time directly**.

If you select **Specify stretch factor** or **Specify reverse recovery charge**, you specify a value that the block uses to derive the reverse recovery time. For more information on these options, see “Alternatives to Specifying  $t_{rr}$  Directly” on page 1-339.

### **Reverse recovery time, $t_{rr}$**

Interval between the time when the current initially goes to zero (when the diode turns off) and the time when the current falls to less than 10% of the peak reverse current. The default value is  $15 \mu\text{s}$ .

This parameter is visible only if you set **Reverse recovery time parameterization** to **Specify reverse recovery time directly**.

The value of the **Reverse recovery time,  $t_{rr}$**  parameter must be greater than the value of the **Peak reverse current,  $i_{RM}$**  parameter divided by the value of the **Rate of change of current when measuring  $i_{RM}$**  parameter.

### **Reverse recovery time stretch factor**

Value that the block uses to calculate **Reverse recovery time,  $t_{rr}$** . This value must be greater than 1. The default value is 3.

This parameter is visible only if you set **Reverse recovery time parameterization** to **Specify stretch factor**.

Specifying the stretch factor is an easier way to parameterize the reverse recovery time than specifying the reverse recovery charge. The larger the value of the stretch factor, the longer it takes for the reverse recovery current to dissipate.

### **Reverse recovery charge, $Q_{rr}$**

Value that the block uses to calculate **Reverse recovery time,  $t_{rr}$** . Use this parameter if the data sheet for your diode device specifies a value for the reverse recovery charge instead of a value for the reverse recovery time.

The reverse recovery charge is the total charge that continues to dissipate when the

diode turns off. The value must be less than  $-\frac{i_{RM}^2}{2a}$ ,

where:

- $i_{RM}$  is the value specified for **Peak reverse current, iRM**.
- $a$  is the value specified for **Rate of change of current when measuring iRM**.

The default value is 1500  $\mu$ As.

The parameter is visible only if you set **Reverse recovery time parameterization** to **Specify reverse recovery charge**.

For more information on these parameters, see Commutation Diode.

## Thermal Model Tab

The **Thermal Model** tab is enabled only when you select a block variant that includes a thermal port.

### Thermal loss dependent on

Select a parameterization method. The option that you select determines which other parameters are enabled. Options are:

- **Voltage and current** — Use scalar values to specify the output current, switch-on loss, switch-off loss, and on-state voltage data.
- **Voltage, current, and temperature** — Use vectors to specify the output current, switch-on loss, switch-off loss, on-state voltage, and temperature data. This is the default parameterization method.

### Off-state voltage for losses data

The output voltage of the device during the off state. This is the blocking voltage at which the switch-on loss and switch-off loss data are defined. The default value is 300 V.

### Energy dissipation time constant

Time constant used to average the switch-on losses, switch-off losses, and conduction losses. This value is equal to the period of the minimum switching frequency. The default value is  $1e-4$  s.

**Additional Parameters for Parameterizing by Voltage, Current, and Temperature****Temperature vector, Tj**

Temperature values at which the switch-on loss, switch-off loss, and on-state voltage are specified. Specify this parameter using a vector quantity. The default value is [ 298.15 398.15 ] K.

**Output current vector, Iout**

Output currents for which the switch-on loss, switch-off loss and on-state voltage are defined. The first element must be zero. Specify this parameter using a vector quantity. The default value is [ 0 10 50 100 200 400 600 ] A.

**Switch-on loss, Eon=fcn(Tj,Iout)**

Energy dissipated during a single switch on event. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a vector quantity. The default value is [ 0 2.9e-4 0.00143 0.00286 0.00571 0.01314 0.02286; 0 5.7e-4 0.00263 0.00514 0.01029 0.02057 0.03029 ] J.

**Switch-off loss, Eoff=fcn(Tj,Iout)**

Energy dissipated during a single switch-off event. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a vector quantity. The default value is [ 0 2.1e-4 0.00107 0.00214 0.00429 0.009859999999999999 0.01714; 0 4.3e-4 0.00197 0.00386 0.00771 0.01543 0.02271 ] J.

**On-state voltage, Von=fcn(Tj,Iout)**

Voltage drop across the device while it is in a triggered conductive state.. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a vector quantity. The default value is [ 0 1.1 1.3 1.45 1.75 2.25 2.7; 0 1 1.15 1.35 1.7 2.35 3 ] V.

**Additional Parameters for Parameterizing by Voltage and Current****Output current, Iout**

Output currents for which the switch-on loss, switch-off loss, and on-state voltage are defined. The first element must be zero. Specify this parameter using a scalar quantity. The default value is 600 A.

**Switch-on loss**

Energy dissipated during a single switch-on event. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a scalar quantity. The default value is  $0.02286$  J.

**Switch-off loss**

Energy dissipated during a single switch-off event. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a scalar quantity. The default value is  $0.01714$  J.

**On-state voltage**

Voltage drop across the block while it is in a triggered conductive state. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a scalar quantity. The default value is  $2.7$  V.

**See Also**

Commutation Diode | Diode (Piecewise Linear) | GTO | IGBT | Ideal Semiconductor Switch | Thyristor (Piecewise Linear)

**Topics**

“Quantifying IGBT Thermal Losses”

“Simulate Thermal Losses in Semiconductors”

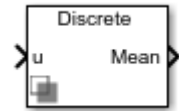
“Switch Between Physical Signal and Electrical Ports”

**Introduced in R2013b**

# Moving Average

Moving average-value computation

**Library:** Simscape / Electrical / Power Systems / Control /  
General Control



## Description

This block computes the moving average value of the input signal. Use this block to filter higher frequency signal components and to smoothen noisy signals.

## Equations

The moving average is computed based on a moving time window. The moving average for continuous-time is calculated as

$$\bar{u} = \frac{1}{T_0} \int_{t_0}^{t_0+T_0} u(t) dt$$

where:

- $u(t)$  is the input signal,
- $T_0 = \frac{1}{f}$
- $f$  is the fundamental frequency of the signal.

The moving average for discrete-time is calculated as:

$$\bar{u}(k) = \frac{1}{T_0} \sum_{i=0}^{n-1} u(k-i)$$



## Assumptions

- The output is initialized with the initial condition in the time interval  $[0, T_0]$

## Ports

### Input

**u — Moving average input**

scalar | vector

Input signal.

Data Types: single | double

### Output

**Mean — Moving average output**

scalar | vector

Moving average of the input signal.

Data Types: single | double

## Parameters

**Fundamental frequency (Hz) — Signal fundamental frequency**

60 (default) | positive scalar | vector with positive values

Fundamental frequency of the signal, in Hz. If you specify the fundamental frequency using a vector, it must match the input vector dimension.

**Initial value — Initial input signal value**

0 (default) | scalar

The initial value of the input signal.

**Sample time (-1 for inherited) — Block sample time**

-1 (default) | 0 | positive scalar

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For inherited discrete-time operation, specify  $-1$ . For discrete-time operation, specify a positive integer. For continuous-time operation, specify  $0$ .

If this block is in a masked subsystem, or other variant subsystem that allows you to switch between continuous operation and discrete operation, promote the sample time parameter. Promoting the sample time parameter ensures correct switching between the continuous and discrete implementations of the block. For more information, see “Promote Parameter to Mask” (Simulink).

## See Also

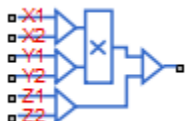
### Blocks

Fourier Analysis | Signal Sample and Hold

**Introduced in R2018b**

# Multiplier

Model integrated circuit multiplier



## Library

Integrated Circuits

## Description

The Multiplier block models an integrated circuit multiplier. The block implements the following equation, which defines the voltage applied to the output port:

$$V_{out} = A \left( \frac{(X_1 - X_2)(Y_1 - Y_2)}{K} - (Z_1 - Z_2) \right)$$

where  $X_1$ ,  $X_2$ ,  $Y_1$ ,  $Y_2$ ,  $Z_1$ ,  $Z_2$  are the voltages presented at the input ports,  $A$  is the gain, and  $K$  is the scale factor.

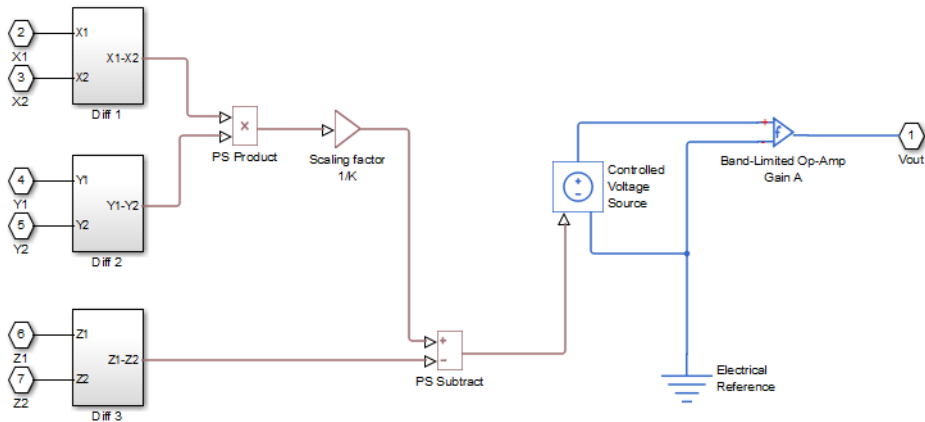
In a typical multiplication circuit, the output is fed back into input  $Z_1$ , which results in the following gain (assuming that  $A$  is large):

$$V_{out} = \left( \frac{(X_1 - X_2)(Y_1 - Y_2)}{K} + Z_2 \right)$$

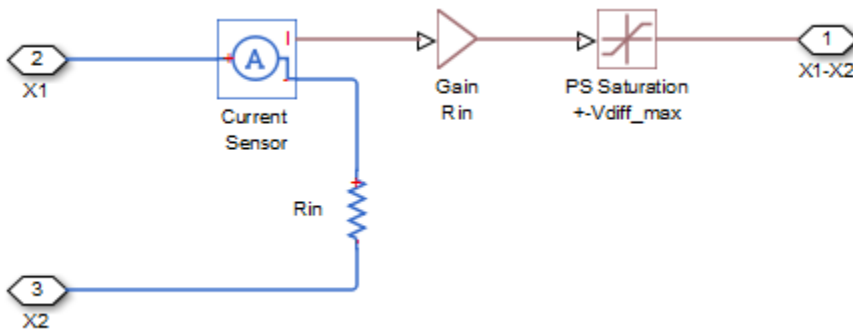
The value of the scale factor  $K$  is usually altered by an external resistor bias network. The Multiplier block implements  $K$  as an internal gain, and the external bias network is not necessary for system simulation. A typical value for  $K$  is 10, with a typical adjustment down to 3.

You can use the Multiplier block to implement a number of other functions, as well as multiplication. Examples include division, squares, and square roots. For example circuits, consult manufacturer datasheets.

The following figure shows the internal model structure of the Multiplier block. It includes the Band-Limited Op-Amp block to model finite bandwidth and slew-rate limiting.



The next figure shows one of the differential subsystem blocks. All three differential subsystem blocks are identical in structure.



## Basic Assumptions and Limitations

The Multiplier block has the following limitations:

- Only differential limiting of the inputs is implemented. You must ensure that the absolute values of the inputs you use keep the actual device operating in its linear region.
- Output current is such that the integrated circuit is operating in the linear I-V region, which can be approximated by a voltage source plus a series output resistance.
- Input offset voltage is not modeled, and the input voltage-current relationship is treated as linear within the differential signal voltage range.

## Parameters

- “Main Tab” on page 1-879
- “Inputs Tab” on page 1-879
- “Outputs Tab” on page 1-880

### Main Tab

#### Scaling factor, $K$

The scaling factor  $K$  in the equation that defines output voltage. Datasheets sometimes refer to it as the scale factor, or SF. The default value is 10 V.

#### Gain, $A$

The gain of the internal operational amplifier, corresponding to the gain  $A$  in the equation that defines output voltage. The default value is  $3e3$ .

### Inputs Tab

#### Differential resistance, $R_{in}$

Each of the differential inputs is approximated as a linear resistor with value  $R_{in}$ . Set this value to the datasheet value for differential resistance. The default value is  $1e7 \Omega$ .

### **Differential signal voltage range**

This value,  $V_{diff\_max}$ , is used to limit the magnitude of each of the three differential input voltages. Set this value to the datasheet value for differential signal voltage range. The default value is 10 V.

## **Outputs Tab**

### **Output resistance, $R_{out}$**

The multiplier output stage is modeled as a voltage source plus series resistor inside the Band-Limited Op-Amp block. This parameter specifies the value of this series resistor. The default value is 0.1  $\Omega$ .

### **Minimum output, $V_{min}$**

The lower limit of the output voltage. The default value is -11 V.

### **Maximum output, $V_{max}$**

The upper limit of the output voltage. The default value is 11 V.

### **Maximum slew rate, $V_{dot}$**

The maximum positive or negative rate of change of output voltage magnitude. The default value is 20 V/ $\mu$ s.

### **Bandwidth, $f$**

The bandwidth of the Band-Limited Op-Amp block. The default value is 1 MHz.

### **Initial output voltage, $V_0$**

The value of the initial Multiplier block output if the **Start simulation from steady-state** option is not selected in the Solver block. The default value is 0 V.

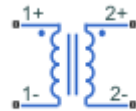
## **Ports**

The block has six electrical conserving ports that serve as signal input ports and one electrical conserving port that outputs the multiplied signal.

# Mutual Inductor

Mutual inductor model with nominal inductance tolerances for each winding, operating limits, and faults

**Library:** Simscape / Electrical / Electronics and Mechatronics / Passive Devices  
Simscape / Foundation Library / Electrical / Electrical Elements



## Description

The Mutual Inductor block lets you model a mutual inductor (two-winding transformer) with nominal inductance tolerances for each winding. The model includes the following effects:

- “Tolerances” on page 1-882
- “Operating Limits” on page 1-883
- “Faults” on page 1-883

You can turn these modeling options on and off independently of each other.

In the unfaulted state, the following equations describe the Mutual Inductor block behavior:

$$v_1 = L_1 \frac{di_{L1}}{dt} + M \frac{di_{L2}}{dt} + i_{L1} R_1$$

$$v_2 = L_2 \frac{di_{L2}}{dt} + M \frac{di_{L1}}{dt} + i_{L2} R_2$$

$$M = k \sqrt{L_1 L_2}$$

where:

- $v_1$  and  $v_2$  are voltages across the primary and secondary winding, respectively.
- $L_1$  and  $L_2$  are inductances of the primary and secondary winding.
- $R_1$  and  $R_2$  are series resistances of the primary and secondary winding.
- $M$  is mutual inductance.
- $k$  is coefficient of coupling. To reverse one of the winding directions, use a negative value.
- $t$  is time.

A parallel conductance is placed across the + and - terminals of the primary and secondary windings, so that  $i_{L1} = i_1 - G_1 v_1$ , where  $G_1$  is the parallel conductance of the primary winding, and  $i_1$  is the terminal current into the primary. Similar definitions and equation apply to  $i_{L2}$ .

## Tolerances

You can apply tolerances separately for each winding. Datasheets typically provide a tolerance percentage for a given inductor type. Therefore, this value is the same for both windings. The table shows how the block applies tolerances to the nominal inductance value and calculates inductance based on the selected tolerance application option for the winding, **L1 tolerance application** or **L2 tolerance application**.

Option	Inductance Value
None – use nominal value	$L$
Random tolerance	Uniform distribution: $L \cdot (1 - tol + 2 \cdot tol \cdot rand)$ Gaussian distribution: $L \cdot (1 + tol \cdot randn / nSigma)$
Apply maximum tolerance value	$L \cdot (1 + tol)$
Apply minimum tolerance value	$L \cdot (1 - tol)$

In the table:

- $L$  is nominal inductance for the primary or secondary winding, **Inductance L1** or **Inductance L2** parameter value.
- $tol$  is fractional tolerance, **Tolerance (%)** /100.



- $nSigma$  is the value you provide for the **Number of standard deviations for quoted tolerance** parameter.
- `rand` and `randn` are standard MATLAB functions for generating uniform and normal distribution random numbers.

## Operating Limits

Inductors are typically rated with a particular saturation current, and possibly with a maximum allowable power dissipation. You can specify operating limits in terms of these values, to generate warnings or errors if the inductor is driven outside its specification.

When an operating limit is exceeded, the block can either generate a warning or stop the simulation with an error. For more information, see the “Operating Limits” on page 1-888 parameters section.

## Faults

Instantaneous changes in inductor parameters are unphysical. Therefore, when the Mutual Inductor block enters the faulted state, short-circuit and open-circuit voltages transition to their faulted values over a period of time based on this formula:

$$CurrentValue = FaultedValue - (FaultedValue - UnfaultedValue) \cdot \operatorname{sech}(\Delta t / \tau)$$

where:

- $\Delta t$  is time since the onset of the fault condition.
- $\tau$  is user-defined time constant associated with the fault transition.

For short-circuit faults, the conductance of the short-circuit path also changes according to the  $\operatorname{sech}(\Delta t / \tau)$  function from a small value (representing an open-circuit path) to a large value.

The Mutual Inductor block lets you select whether the faults occur in the primary or secondary winding. The block models the faulted winding as a faulted inductor. The unfaulted winding is coupled to the faulted winding. As a result, the actual equations involve a total of three coupled windings: two for the faulted winding and one for the unfaulted winding. The coupling between the primary and secondary windings is defined by the **Coefficient of coupling** parameter.

The block can trigger the start of fault transition:

- At a specific time
- After voltage exceeds the maximum permissible value a certain number of times
- When current exceeds the maximum permissible value for longer than a specific time interval

You can enable or disable these trigger mechanisms separately, or use them together if more than one trigger mechanism is required in a simulation. When more than one mechanism is enabled, the first mechanism to trigger the fault transition takes precedence. In other words, a component fails no more than once per simulation.

You can also choose whether to issue an assertion when a fault occurs by using the **Reporting when a fault occurs** parameter. The assertion can take the form of a warning or an error. By default, the block does not issue an assertion.

Faultable inductors often require that you use the fixed-step local solver, rather than the variable-step solver. In particular, if you model transitions to a faulted state that include short circuits, MathWorks recommends that you use the fixed-step local solver. For more information, see “Making Optimal Solver Choices for Physical Simulation” (Simscape).

## Variables

Use the **Variables** section of the block interface to set the priority and initial target values for the block variables prior to simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

The **Primary current** and **Secondary current** variables let you specify a high-priority target for the initial inductor current in the respective winding at the start of simulation.

## Ports

### Conserving

#### **1+ — Positive terminal of the primary winding**

electrical

Electrical conserving port associated with the primary winding positive terminal.

#### **1- — Negative terminal of the primary winding**

electrical

Electrical conserving port associated with the primary winding negative terminal.

**2+ — Positive terminal of the secondary winding**

electrical

Electrical conserving port associated with the secondary winding positive terminal.

**2- — Negative terminal of the secondary winding**

electrical

Electrical conserving port associated with the secondary winding negative terminal.

## Parameters

### Main

**Inductance L1 — Nominal inductance value in the primary winding**

10 H (default)

The nominal inductance value in the primary winding. Inductance value must be greater than zero.

**Inductance L2 — Nominal inductance value in the secondary winding**

0.1 H (default)

The nominal inductance value in the secondary winding. Inductance value must be greater than zero.

**Coefficient of coupling — Mutual inductance coupling between windings**

0.9 (default)

The coupling between the primary and secondary windings. This coefficient defines the mutual inductance. To reverse one of the winding directions, use a negative value.

**Tolerance (%) — Inductor tolerance, in percent**

20 (default)

The inductor tolerance as defined on the manufacturer datasheet. Datasheets typically provide a tolerance percentage for a given inductor type. Therefore, this value is the same for both windings.

**L1 tolerance application — Select how to apply tolerance to primary winding**

None — use nominal value (default) | Random tolerance | Apply maximum tolerance value | Apply minimum tolerance value

Select how to apply tolerance during simulation to the primary winding:

- **None — use nominal value** — The block does not apply tolerance, it uses the nominal inductance value.
- **Random tolerance** — The block applies random offset to the inductance value, within the tolerance value limit. You can choose Uniform or Gaussian distribution for calculating the random number by using the **Tolerance distribution** parameter.
- **Apply maximum tolerance value** — The inductance is increased by the specified tolerance percent value.
- **Apply minimum tolerance value** — The inductance is decreased by the specified tolerance percent value.

**L1 tolerance distribution — Select the distribution type for primary winding**

Uniform (default) | Gaussian

Select the distribution type for random tolerance:

- **Uniform** — Uniform distribution
- **Gaussian** — Gaussian distribution

**Dependencies**

Enabled when the **L1 tolerance application** parameter is set to Random tolerance.

**L1 number of standard deviations for quoted tolerance — Used for calculating the Gaussian random number for primary winding**

4 (default)

Number of standard deviations for calculating the Gaussian random number.

**Dependencies**

Enabled when the **L1 tolerance distribution** parameter is set to Gaussian.

**L2 tolerance application — Select how to apply tolerance to secondary winding**

None — use nominal value (default) | Random tolerance | Apply maximum tolerance value | Apply minimum tolerance value

Select how to apply tolerance during simulation to the secondary winding:

- **None – use nominal value** — The block does not apply tolerance, it uses the nominal inductance value.
- **Random tolerance** — The block applies random offset to the inductance value, within the tolerance value limit. You can choose Uniform or Gaussian distribution for calculating the random number by using the **Tolerance distribution** parameter.
- **Apply maximum tolerance value** — The inductance is increased by the specified tolerance percent value.
- **Apply minimum tolerance value** — The inductance is decreased by the specified tolerance percent value.

### **L2 tolerance distribution — Select the distribution type for secondary winding**

Uniform (default) | Gaussian

Select the distribution type for random tolerance:

- **Uniform** — Uniform distribution
- **Gaussian** — Gaussian distribution

#### **Dependencies**

Enabled when the **L2 tolerance application** parameter is set to Random tolerance.

### **L2 number of standard deviations for quoted tolerance — Used for calculating the Gaussian random number for secondary winding**

4 (default)

Number of standard deviations for calculating the Gaussian random number.

#### **Dependencies**

Enabled when the **L2 tolerance distribution** parameter is set to Gaussian.

## **Resistance**

### **Series resistance, [R\_primary R\_secondary] — Equivalent series resistance of the primary and secondary winding**

[0.001, 0.001]  $\Omega$  (default)

Equivalent series resistance of the primary and secondary winding, specified as a two-element vector. The first number corresponds to the primary winding, the second number to the secondary winding. For a faulted winding, the block allocates the resistance to each segment in proportion to the number of turns in that segment.

### **Parallel conductance, [G\_primary G\_secondary] — Parallel leakage path associated with the primary and secondary winding**

[1e-9,1e-9] 1/Ω (default)

Parallel leakage path associated with the primary and secondary winding, specified as a two-element vector. The first number corresponds to the primary winding, the second number to the secondary winding. The parallel conductances are placed directly across the + and - terminals of the primary and secondary winding, respectively.

## **Operating Limits**

### **Enable operating limits — Select Yes to enable reporting when the operational limits are exceeded**

No (default) | Yes

Select Yes to enable reporting when the operational limits are exceeded. The associated parameters in the **Operating Limits** section become visible to let you select the reporting method and specify the operating limits in terms of power and current.

### **Reporting when operating limits exceeded — Select the reporting method**

Warn (default) | Error

Select what happens when an operating limit is exceeded:

- Warn — The block issues a warning.
- Error — Simulation stops with an error.

### **Dependencies**

Enabled when the **Enable operating limits** parameter is set to Yes.

### **Saturation current — Inductor saturation current**

1 A (default)

Inductor saturation current, as defined in the manufacturer datasheets. If the net current into the primary and secondary windings exceeds this value, the core material enters

saturation, and the block reports an operating limits violation. That is, the block compares the limit against  $|i_1 + i_2|$ , where currents are defined as being positive when they are into the + nodes.

### Dependencies

Enabled when the **Enable operating limits** parameter is set to Yes.

### Power rating — Maximum power dissipation in the inductor

1 W (default)

Maximum instantaneous (total) power dissipation in the resistance and conductance elements associated with the mutual inductor. If the total power (including both primary and secondary winding) exceeds this number, the block reports an operating limits violation.

### Dependencies

Enabled when the **Enable operating limits** parameter is set to Yes.

## Faults

### Enable faults — Select Yes to enable faults modeling

No (default) | Yes

Select Yes to enable faults modeling. The associated parameters in the **Faults** section become visible to let you select the reporting method and specify the trigger mechanism (temporal or behavioral). You can enable these trigger mechanisms separately or use them together.

### Reporting when a fault occurs — Choose whether to issue an assertion when a fault occurs

None (default) | Warn | Error

Choose whether to issue an assertion when a fault occurs:

- None — The block does not issue an assertion.
- Warn — The block issues a warning.
- Error — Simulation stops with an error.

### Dependencies

Enabled when the **Enable faults** parameter is set to Yes.

**Faulted winding — Select winding to use for fault modeling**

Primary (default) | Secondary

Select whether the faults occur in the primary or secondary winding.

**Dependencies**

Enabled when the **Enable faults** parameter is set to Yes.

**Location of fault node (% of total turns from - terminal) — Percentage of turns in the subinductor that is in contact with the - port of the faulted winding**

50 (default)

In practice, faults are enabled by segmenting the faulted winding into two coupled subinductors, connected in a series. The inductance is proportional to the square of the number of turns in the respective segment, and the series resistance of each subinductor is proportional to the number of turns in each segment. The parallel conductance spans both segments.

This parameter indicates the percentage of turns that are assigned to the subinductor that is in contact with the - port of the faulted winding. The remaining turns are assigned to the other subinductor. The default value is 50, which means that the overall inductance of the faulted winding is divided into two equal, coupled subinductors.

**Dependencies**

Enabled when the **Enable faults** parameter is set to Yes.

**Short-circuit turns — Select whether fault results in one of the segments being short-circuited**

No (default) | To negative terminal | To positive terminal

Select whether the fault results in one of the subinductor segments being short-circuited:

- **No** — The fault does not produce a short circuit.
- **To negative terminal** — The fault short-circuits the subinductor that is in contact with the - port of the block.
- **To positive terminal** — The fault short-circuits the subinductor that is in contact with the + port of the block.



**Dependencies**

Enabled when the **Enable faults** parameter is set to Yes.

**Open-circuit at fault node — Select whether to apply an open-circuit fault between the segments**

No (default) | Yes

Select whether to apply an open-circuit fault between the two subinductor segments. The default is No. Even with an open-circuit fault, the characteristics of the subinductors are still related because they are magnetically coupled even in the faulted state.

**Dependencies**

Enabled when the **Enable faults** parameter is set to Yes.

**Ground fault — Select whether fault results in one of the segments being short-circuited**

No (default) | Negative terminal side of fault node | Positive terminal side of fault node

Select whether, in case of fault, there is a path for current to flow towards the ground node:

- No — The fault does not result in a connection to ground.
- Negative terminal side of fault node — The side that is in contact with the - port of the block is connected to ground.
- Positive terminal side of fault node — The side that is in contact with the + port of the block is connected to ground.

If the **Open-circuit at fault node** parameter is set to Yes, you need to specify which side (negative or positive) is connected to ground. If there is no open circuit, the two options behave similarly. Physically, this corresponds to a breakdown in the insulation between the windings and the grounded core or chassis.

**Dependencies**

Enabled when the **Enable faults** parameter is set to Yes.

**Conductance of faulted ground path — Mutual coupling between the two subinductors**

0 1/Ω (default)

If there is a ground fault, this parameter represents conductance of the current path to ground. For example, if the path to ground is through the core material, then specify a small conductance value depending on the core material being used. For highly conductive core material or for chassis-shorts, specify a higher conductance value.

**Dependencies**

Enabled when the **Ground fault** parameter is set to Negative terminal side of fault node or Positive terminal side of fault node.

**Fault transition time constant — Time constant for the transition to faulted state**

1 ms (default)

Time constant associated with the transition to the faulted state, as described in “Faults” on page 1-883.

**Dependencies**

Enabled when the **Enable faults** parameter is set to Yes.

**Enable temporal fault trigger — Select Yes to enable time-based fault triggering**

No (default) | Yes

Select Yes to enable time-based fault triggering. You can enable the temporal and behavioral trigger mechanisms separately or use them together.

**Dependencies**

Enabled when the **Enable faults** parameter is set to Yes.

**Simulation time for a fault event — Time before entering faulted state**

1 s (default)

Set the simulation time at which you want the block to start entering the fault state.

**Dependencies**

Enabled when the **Enable temporal fault trigger** parameter is set to Yes.

**Enable behavioral fault trigger — Select Yes to enable behavioral fault triggering**

No (default) | Yes

Select **Yes** to enable behavioral fault triggering. You can enable the temporal and behavioral trigger mechanisms separately or use them together.

**Dependencies**

Enabled when the **Enable faults** parameter is set to **Yes**.

**Maximum permissible voltage – Voltage threshold to fault transition**

100 V (default)

Define the voltage threshold to a fault transition. If the voltage value exceeds this threshold a certain number of times, specified by the **Number of events to fail when exceeding voltage** parameter value, then the block starts entering the fault state.

**Dependencies**

Enabled when the **Enable behavioral fault trigger** parameter is set to **Yes**.

**Number of events to fail when exceeding voltage – Maximum number of times the voltage exceeds the threshold**

1 (default)

Because the physical mechanism underlying voltage-based failures depends on one or more partial discharge events occurring, this parameter allows you to set the number of voltage overshoots that the inductor can withstand before the fault transition begins. Note that the block does not check the time spent in the overvoltage condition, only the number of transitions.

**Dependencies**

Enabled when the **Enable behavioral fault trigger** parameter is set to **Yes**.

**Maximum permissible current – Current threshold to fault transition**

1 A (default)

Define the current threshold to a fault transition. If the current value exceeds this threshold for longer than the **Time to fail when exceeding current** parameter value, then the block starts entering the fault state.

**Dependencies**

Enabled when the **Enable behavioral fault trigger** parameter is set to **Yes**.

**Time to fail when exceeding current — Maximum length of time the current exceeds the threshold**

1 s (default)

Set the maximum length of time that the current can exceed the maximum permissible value without triggering the fault.

**Dependencies**

Enabled when the **Enable behavioral fault trigger** parameter is set to Yes.

## Block Characteristics

<b>Data Types</b>	
<b>Direct Feedthrough</b>	No
<b>Zero-Crossing Detection</b>	No

## See Also

Fault | Inductor

**Introduced in R2017a**

# N-Channel IGBT

Model N-Channel IGBT



## Library

Semiconductor Devices

## Description

The N-Channel IGBT block models an Insulated Gate Bipolar Transistor (IGBT). The block provides two main modeling variants, accessible by right-clicking the block in your block diagram and then selecting the appropriate option from the context menu, under **Simscape > Block choices**:

- **Full I-V and capacitance characteristics** — This variant is a detailed component model suitable for simulating detailed switching characteristics and predicting component losses. This variant, in turn, provides two ways of modeling an IGBT:
  - As an equivalent circuit based on a PNP bipolar transistor and N-channel MOSFET. For more information on using this model, see “Representation by Equivalent Circuit” on page 1-896, “Fine-Tuning the Current-Voltage Characteristics” on page 1-904, and “Modeling Temperature Dependence” on page 1-904.
  - By a 2-D lookup table approximation to the I-V (current-voltage) curve. For details, see “Representation by 2-D Lookup Table” on page 1-899.
  - By a 3-D lookup table approximation to the I-V (current-voltage) curve that includes temperature data. For details, see “Representation by 3-D Lookup Table” on page 1-899.

The gate junction capacitance in the detailed model is represented as a fixed gate-emitter capacitance  $C_{GE}$  and either a fixed or a nonlinear gate-collector capacitance  $C_{GC}$ . For details, see “Charge Model” on page 1-900.

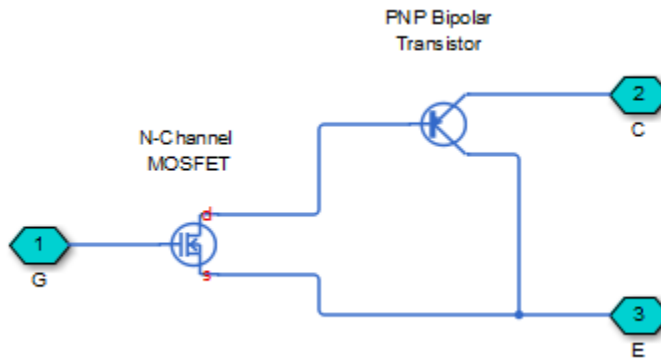
- **Simplified I-V characteristics and event-based timing** — This variant models the IGBT more simply by using just the on-state I-V data corresponding to the gate voltage used in your circuit. Switching between states is achieved by linearly ramping the collector-emitter voltage. This simplified model is suitable when approximate dynamic characteristics are sufficient, and simulation speed is of paramount importance. For details, see “Event-Based IGBT Variant” on page 1-905.

Together with the thermal port variants (see “Thermal Port” on page 1-906), the block therefore provides you with four choices. To select the desired variant, right-click the block in your model. From the context menu, select **Simscape > Block choices**, and then one of the following options:

- **Full I-V and capacitance characteristics | No thermal port** — Detailed model that does not simulate the effects of generated heat and device temperature. This is the default.
- **Full I-V and capacitance characteristics | Show thermal port** — Detailed model with exposed thermal port.
- **Simplified I-V characteristics and event-based timing | No thermal port** — Simplified event-based model, which also does not simulate the effects of generated heat and device temperature.
- **Simplified I-V characteristics and event-based timing | Show thermal port** — Simplified event-based model with exposed thermal port.

## Representation by Equivalent Circuit

The equivalent circuit of the detailed block variant consists of a PNP Bipolar Transistor block driven by an N-Channel MOSFET block, as shown in the following figure:



The MOSFET source is connected to the bipolar transistor collector, and the MOSFET drain is connected to the bipolar transistor base. The MOSFET uses the equations shown in the N-Channel MOSFET block reference page. The bipolar transistor uses the equations shown in the PNP Bipolar Transistor block reference page, but with the addition of an emission coefficient parameter  $N$  that scales  $kT/q$ .

The N-Channel IGBT block uses the on and off characteristics you specify in the block dialog box to estimate the parameter values for the underlying N-Channel MOSFET and PNP bipolar transistor.

The block uses the off characteristics to calculate the base-emitter voltage,  $V_{be}$ , and the saturation current,  $I_s$ .

When the transistor is off, the gate-emitter voltage is zero and the IGBT base-collector voltage is large, so the PNP base and collector current equations simplify to:

$$I_b = 0 = -I_s \left[ \frac{1}{\beta_F} (e^{-qV_{bc}/(NkT)} - 1) - \frac{1}{\beta_R} \right]$$

$$I_c = -I_s \left[ e^{-qV_{bc}/(NkT)} \left( 1 + \frac{V_{bc}}{V_{AF}} \right) + \frac{1}{\beta_R} \right]$$

where  $N$  is the **Emission coefficient**,  $N$  parameter value,  $V_{AF}$  is the forward Early voltage, and  $I_c$  and  $I_b$  are defined as positive flowing into the collector and base, respectively. See the PNP Bipolar Transistor reference page for definitions of the remaining variables. The first equation can be solved for  $V_{be}$ .

The base current is zero in the off-condition, and hence  $I_c = -I_{ces}$ , where  $I_{ces}$  is the Zero gate voltage collector current. The base-collector voltage,  $V_{bc}$ , is given by  $V_{bc} = V_{ces} + V_{ces}$ , where  $V_{ces}$  is the voltage at which  $I_{ces}$  is measured. Hence we can rewrite the second equation as follows:

$$I_{ces} = I_s \left[ e^{-qV_{be}/(NKT)} \left( 1 + \frac{V_{ces} + V_{be}}{V_{AF}} \right) + \frac{1}{\beta_R} \right]$$

The block sets  $\beta_R$  and  $\beta_F$  to typical values of 1 and 50, so these two equations can be used to solve for  $V_{be}$  and  $I_s$ :

$$V_{be} = \frac{-NkT}{q} \log \left( 1 + \frac{\beta_F}{\beta_R} \right)$$

$$I_s = \frac{I_c}{e^{-qV_{be}/(NKT)} + \frac{1}{\beta_R}}$$

---

**Note** The block does not require an exact value for  $\beta_F$  because it can adjust the MOSFET gain  $K$  to ensure the overall device gain is correct.

---

The block parameters **Collector-emitter saturation voltage, Vce(sat)** and **Collector current at which Vce(sat) is defined** are used to determine  $V_{be(sat)}$  by solving the following equation:

$$I_{ce(sat)} = I_s \left[ e^{-qV_{be(sat)}/(NKT)} \left( 1 + \frac{V_{ce(sat)} + V_{be(sat)}}{V_{AF}} \right) + \frac{1}{\beta_R} \right]$$

Given this value, the block calculates the MOSFET gain,  $K$ , using the following equation:

$$I_{ds} = I_b = K \left[ (V_{GE(sat)} - V_{th})V_{ds} - \frac{V_{ds}^2}{2} \right]$$



where  $V_{th}$  is the **Gate-emitter threshold voltage, Vge(th)** parameter value and  $V_{GE(sat)}$  is the **Gate-emitter voltage at which Vce(sat) is defined** parameter value.

$V_{ds}$  is related to the transistor voltages as  $V_{ds} = V_{ce} - V_{be}$ . The block substitutes this relationship for  $V_{ds}$ , sets the base-emitter voltage and base current to their saturated values, and rearranges the MOSFET equation to give

$$K = \frac{I_{b(sat)}}{\left[ (V_{GE(sat)} - V_{th})(V_{be(sat)} + V_{ce(sat)}) - \frac{(V_{be(sat)} + V_{ce(sat)})^2}{2} \right]}$$

where  $V_{ce(sat)}$  is the **Collector-emitter saturation voltage, Vce(sat)** parameter value.

These calculations ensure the zero gate voltage collector current and collector-emitter saturation voltage are exactly met at these two specified conditions. However, the current-voltage plots are very sensitive to the emission coefficient  $N$  and the precise value of  $V_{th}$ . If the manufacturer datasheet gives current-voltage plots for different  $V_{GE}$  values, then the  $N$  and  $V_{th}$  can be tuned by hand to improve the match.

## Representation by 2-D Lookup Table

For the lookup table representation of the detailed block variant, you provide tabulated values for collector current as a function of gate-emitter voltage and collector-emitter voltage. The main advantage of using this option is simulation speed. It also lets you parameterize the device from either measured data or from data obtained from another simulation environment. To generate your own data from the equivalent circuit representation, you can use a test harness, such as shown in the IGBT Characteristics example.

The lookup table representation combines all of the equivalent circuit components (PNP transistor, N-channel MOSFET, collector resistor and emitter resistor) into one equivalent lookup table. Therefore, if you use this option, the **Advanced** tab has no parameters.

## Representation by 3-D Lookup Table

For the temperature-dependent lookup table representation of the detailed block variant, you provide tabulated values for collector current as a function of gate-emitter voltage, collector-emitter voltage, and temperature.

The lookup table representation combines all of the equivalent circuit components (PNP transistor, N-channel MOSFET, collector resistor and emitter resistor) into one equivalent lookup table. Therefore, if you use this option, the **Advanced** tab has no parameters.

If the block thermal port is not exposed, then the **Device simulation temperature** parameter on the **Temperature Dependence** tab lets you specify the simulation temperature. If the block has an exposed thermal port, the **Temperature Dependence** tab is empty because the temperature variable T on the thermal port sets the device simulation temperature.

## Charge Model

The detailed variant of the block models junction capacitances either by fixed capacitance values, or by tabulated values as a function of the collector-emitter voltage. In either case, you can either directly specify the gate-emitter and gate-collector junction capacitance values, or let the block derive them from the input and reverse transfer capacitance values. Therefore, the **Parameterization** options for charge model on the **Junction Capacitance** tab are:

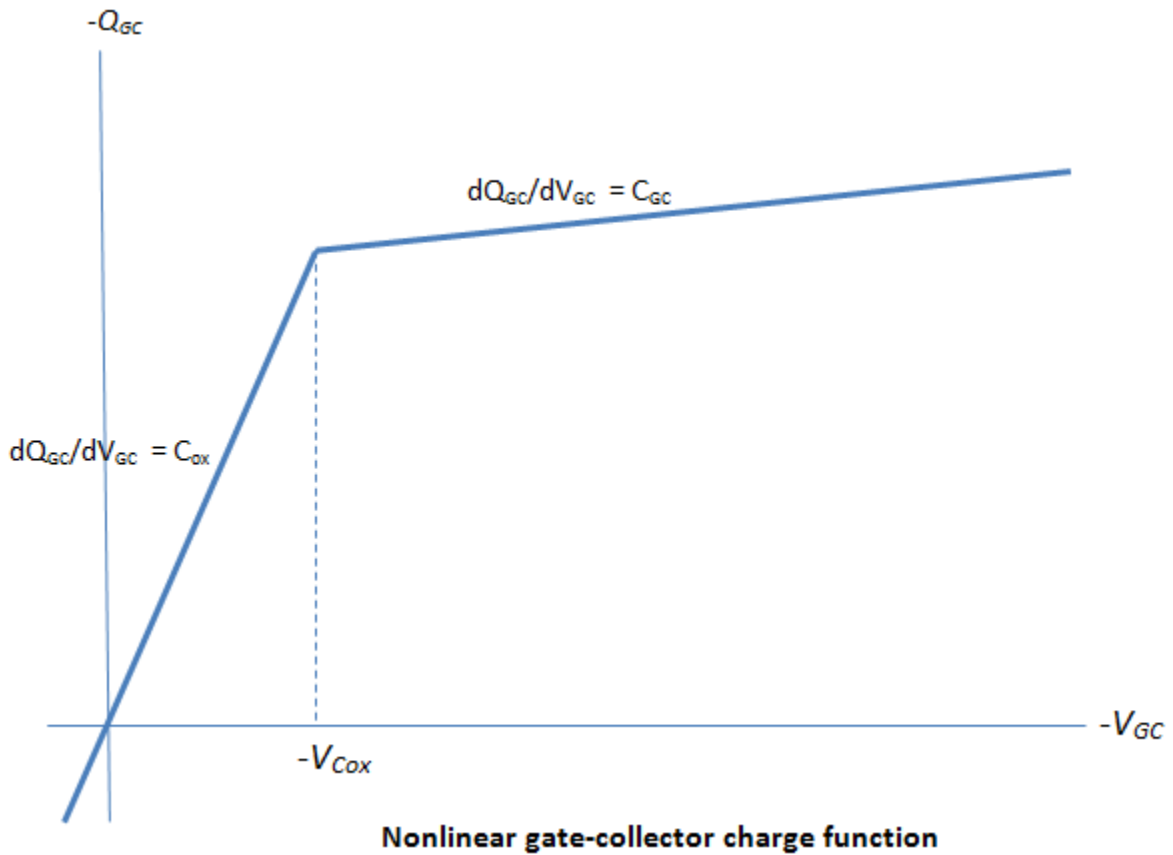
- **Specify fixed input, reverse transfer and output capacitance** — Provide fixed parameter values from datasheet and let the block convert the input and reverse transfer capacitance values to junction capacitance values, as described below. This is the default method.
- **Specify fixed gate-emitter, gate-collector and collector-emitter capacitance** — Provide fixed values for junction capacitance parameters directly.
- **Specify tabulated input, reverse transfer and output capacitance** — Provide tabulated capacitance and collector-emitter voltage values based on datasheet plots. The block converts the input and reverse transfer capacitance values to junction capacitance values, as described below.
- **Specify tabulated gate-emitter, gate-collector and collector-emitter capacitance** — Provide tabulated values for junction capacitances and collector-emitter voltage.

Use one of the tabulated capacitance options (**Specify tabulated input, reverse transfer and output capacitance** or **Specify tabulated gate-emitter, gate-collector and collector-emitter capacitance**) when the datasheet provides a plot of junction capacitances as a function of collector-emitter voltage. Using tabulated capacitance values will give more accurate dynamic characteristics, and avoids the need to iteratively tune parameters to fit the dynamics.

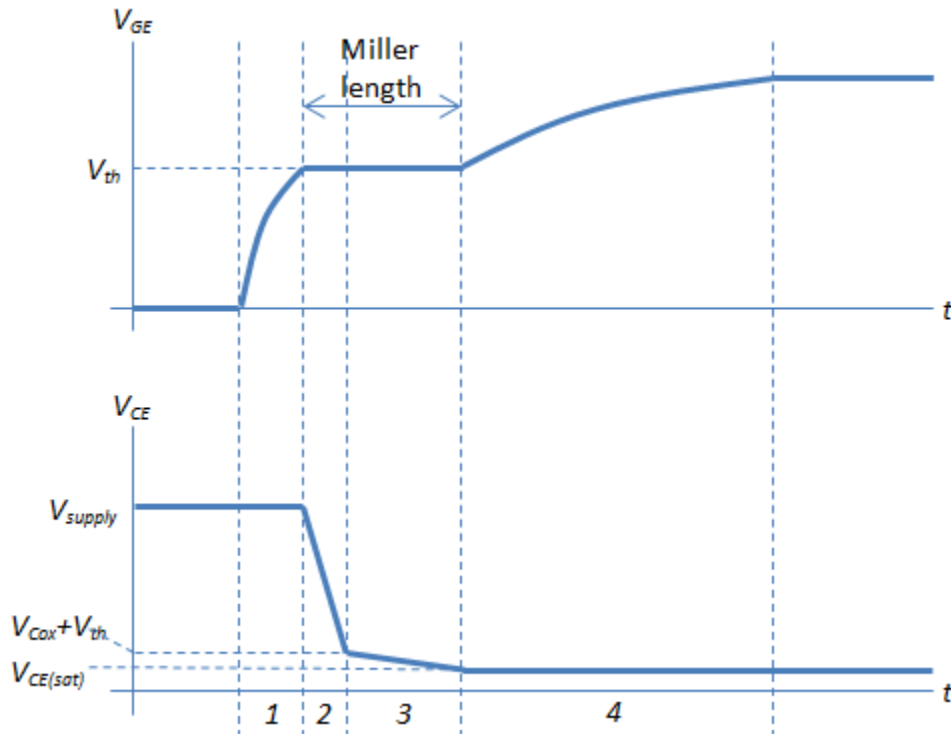
If you use the Specify fixed gate-emitter, gate-collector and collector-emitter capacitance or Specify tabulated gate-emitter, gate-collector and collector-emitter capacitance option, the **Junction Capacitance** tab lets you specify the **Gate-emitter junction capacitance**, **Gate-collector junction capacitance**, and **Collector-emitter junction capacitance** parameter values (fixed or tabulated) directly. Otherwise, the block derives them from the **Input capacitance**, **Cies**, **Reverse transfer capacitance**, **Cres**, and **Output capacitance**, **Coes** parameter values. These two parameterization methods are related as follows:

- $C_{GE} = C_{res}$
- $C_{GC} = C_{ies} - C_{res}$
- $C_{CE} = C_{oes} - C_{res}$

The two fixed capacitance options (Specify fixed input, reverse transfer and output capacitance or Specify fixed gate-emitter, gate-collector and collector-emitter capacitance) let you model gate junction capacitance as a fixed gate-emitter capacitance  $C_{GE}$  and either a fixed or a nonlinear gate-collector capacitance  $C_{GC}$ . If you select the Gate-collector charge function is nonlinear option for the **Charge-voltage linearity** parameter, then the gate-collector charge relationship is defined by the piecewise-linear function shown in the following figure.



With this nonlinear capacitance, the gate-emitter and collector-emitter voltage profiles take the form shown in the next figure, where the collector-emitter voltage fall has two regions (labeled 2 and 3) and the gate-emitter voltage has two time-constants (before and after the threshold voltage  $V_{th}$ ):



You can determine the capacitor values for  $C_{ies}$ ,  $C_{res}$ , and  $C_{ox}$  as follows, assuming that the IGBT gate is driven through an external resistance  $R_G$ :

- 1 Set  $C_{ies}$  to get correct time-constant for  $V_{GE}$  in Region 1. The time constant is defined by the product of  $C_{ies}$  and  $R_G$ . Alternatively, you can use a datasheet value for  $C_{ies}$ .
- 2 Set  $C_{res}$  so as to achieve the correct  $V_{CE}$  gradient in Region 2. The gradient is given by  $(V_{GE} - V_{th}) / (C_{res} \cdot R_G)$ .
- 3 Set  $V_{Cox}$  to the voltage at which the  $V_{CE}$  gradient changes minus the threshold voltage  $V_{th}$ .
- 4 Set  $C_{ox}$  to get correct Miller length and time constant in Region 4.

Because the underlying model is a simplification of an actual charge distribution, some iteration of these four steps may be required to get a best overall fit to measured data.

The collector current tail when the IGBT is turned off is determined by the **Total forward transit time** parameter.

---

**Note** Because this block implementation includes a charge model, you must model the impedance of the circuit driving the gate to obtain representative turn-on and turn-off dynamics. Therefore, if you are simplifying the gate drive circuit by representing it as a controlled voltage source, you must include a suitable series resistor between the voltage source and the gate.

---

## Fine-Tuning the Current-Voltage Characteristics

For the equivalent circuit representation of the detailed model, use the parameters on the **Advanced** tab to fine-tune the current-voltage characteristics of the modeled device. To use these additional parameters effectively, you will need a manufacturer datasheet that provides plots of the collector current versus collector-emitter voltage for different values of gate-emitter voltage. The parameters on the **Advanced** tab have the following effects:

- The **Emission coefficient, N** parameter controls the shape of the current-voltage curves around the origin.
- The **Collector resistance, RC** and **Emitter resistance, RE** parameters affect the slope of the current-voltage curve at higher currents, and when fully turned on by a high gate-emitter voltage.
- The **Forward Early voltage, VAF** parameter affects the shape of the current-voltage curves for gate-emitter voltages around the **Gate-emitter threshold voltage, Vge(th)**.

## Modeling Temperature Dependence

For the 2-D lookup table representation, the electrical equations do not depend on temperature. However, you can model temperature dependence by either using the 3-D lookup table representation, or by using the equivalent circuit representation of the detailed model.

For the equivalent circuit representation, temperature dependence is modeled by the temperature dependence of the constituent components. See the N-Channel MOSFET and PNP Bipolar Transistor block reference pages for further information on the defining equations.

Some datasheets do not provide information on the zero gate voltage collector current,  $I_{ces}$ , at a higher measurement temperature. In this case, you can alternatively specify the energy gap,  $EG$ , for the device, using a typical value for the semiconductor type. For silicon, the energy gap is usually 1.11 eV.

## Event-Based IGBT Variant

This implementation has much simpler equations than that with full I-V and capacitance characteristics. Use the event-based variant when the focus of the analysis is to understand overall circuit behavior rather than to verify the precise IGBT timing or losses characteristics.

The device is always in one of the following four states:

- Off
- Turning on
- On
- Turning off

In the off state, the relationship between collector current ( $i_c$ ) and collector-emitter voltage ( $v_{ce}$ ) is

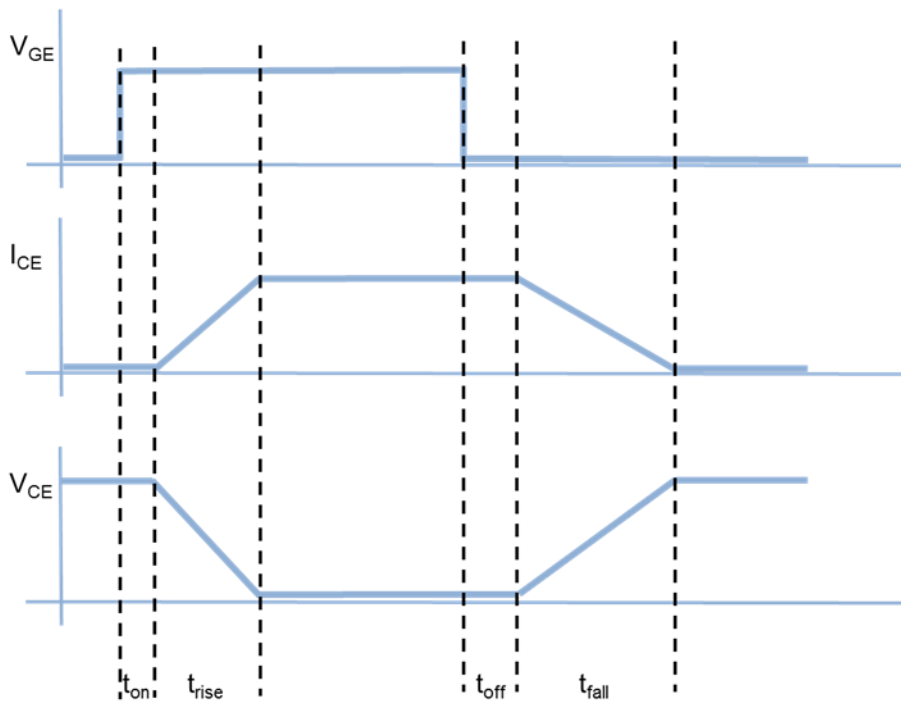
$$i_c = G_{\text{off}} v_{ce}$$

In the on state, the relationship between collector current ( $i_c$ ) and collector-emitter voltage ( $v_{ce}$ ) is

$$v_{ce} = \text{tablelookup}(i_c)$$

When turning on, the collector-emitter voltage is ramped down to zero over the rise time, the device moving into the on state when the voltage falls below the tabulated on-state value. Similarly when turning off, the collector-emitter voltage is ramped up over the (current) fall time to the specified blocking voltage value.

The following figure shows the resulting voltage and current profiles when driving a resistive load.



## Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and select the appropriate block variant:

- For a detailed model, select **Simscape > Block choices > Full I-V and capacitance characteristics | Show thermal port**. This action displays the thermal port H on the block icon, and adds the **Thermal Port** tab to the block dialog box.
- For a simplified event-based model, select **Simscape > Block choices > Simplified I-V characteristics and event-based timing | Show thermal port**. This action displays the thermal port H on the block icon, adds the **Thermal Port** tab to the block dialog box, and displays additional parameters on the **Main** tab. To simulate thermal effects, you must provide additional tabulated data for turn-on and turn-off losses and define the collector-emitter on-state voltage as a function of both current and temperature.



Use the thermal port to simulate the effects of generated heat and device temperature. For more information on using thermal ports and on the **Thermal Port** tab parameters, see “Simulating Thermal Effects in Semiconductors”.

## Basic Assumptions and Limitations

The detailed model is based on the following assumptions:

- This block does not allow you to specify initial conditions on the junction capacitances. If you select the **Start simulation from steady state** option in the Solver Configuration block, the block solves the initial voltages to be consistent with the calculated steady state. Otherwise, voltages are zero at the start of the simulation.
- You may need to use nonzero junction capacitance values to prevent numerical simulation issues, but the simulation may run faster with these values set to zero.
- The block does not account for temperature-dependent effects on the junction capacitances.

The simplified, event-based model is based on the following assumptions:

- When you use a pair of IGBTs in a bridge arm, normally the gate drive circuitry will prevent a device turning on until the corresponding device has turned off, thereby implementing a minimum dead band. If you need to simulate the case where there is no minimum dead band and both devices are momentarily partially on, use the detailed IGBT model variant (**Full I-V and capacitance characteristics**). The assumption used by the event-based variant that the collector-emitter voltages can be ramped between on and off states is not valid for such cases.
- A minimum pulse width is applied when turning on or off; at the point where the gate-collector voltage rises above the threshold, any subsequent gate voltage changes are ignored for a time equal to the sum of the turn-on delay and current rise time. Similarly at the point where the gate collector voltage falls below the threshold, any subsequent gate voltage changes are ignored for a time equal to the sum of the turn-off delay and current fall time. This feature is normally implemented in the gate drive circuitry.
- This model does not account for charge. Hence there is no current tail when turning off an inductive load.
- Representative modeling of the current spike during turn-on of an inductive load with preexisting freewheeling current requires tuning of the **Miller resistance** parameter.

- The tabulated turn-on switching loss uses the previous on-state current, not the current value (which is not known until the device reaches the final on state).
- Due to high model stiffness that can arise from the simplified equations, you may get minimum step size violation warnings when using this block. Open the Solver pane of the Configuration Parameters dialog box and increase the **Number of consecutive min steps** parameter value as necessary to remove these warnings.

## Parameters

- “Main Tab (Default Block Variant)” on page 1-908
- “Junction Capacitance Tab (Default Block Variant)” on page 1-911
- “Advanced Tab (Default Block Variant)” on page 1-914
- “Temperature Dependence Tab (Default Block Variant)” on page 1-914
- “Main Tab (Event-Based Block Variant)” on page 1-916
- “Dynamics Tab (Event-Based Block Variant)” on page 1-917

## Main Tab (Default Block Variant)

This configuration of the **Main** tab corresponds to the detailed block variant, which is the default. If you are using the simplified, event-based variant of the block, see “Main Tab (Event-Based Block Variant)” on page 1-916.

### I-V characteristics defined by

Select the IGBT representation:

- **Fundamental nonlinear equations** — Use an equivalent circuit based on a PNP bipolar transistor and N-channel MOSFET. This is the default.
- **Lookup table (2-D, temperature independent)** — Use 2-D table lookup for collector current as a function of gate-emitter voltage and collector-emitter voltage.
- **Lookup table (3-D, temperature dependent)** — Use 3-D table lookup for collector current as a function of gate-emitter voltage, collector-emitter voltage, and temperature.

### Zero gate voltage collector current, *I*<sub>ces</sub>

The collector current that flows when the gate-emitter voltage is set to zero, and a large collector-emitter voltage is applied, that is, the device is in the off-state. The

value of the large collector-emitter voltage is defined by the parameter **Voltage at which  $I_{ces}$  is defined**. The default value is 2 mA. This parameter is visible only when you select Fundamental nonlinear equations for the **I-V characteristics defined by** parameter.

#### **Voltage at which $I_{ces}$ is defined**

The voltage used when measuring the **Zero gate voltage collector current,  $I_{ces}$** . The default value is 600 V. This parameter is visible only when you select Fundamental nonlinear equations for the **I-V characteristics defined by** parameter.

#### **Gate-emitter threshold voltage, $V_{ge(th)}$**

The threshold voltage used in the MOSFET equations. The default value is 6 V. This parameter is visible only when you select Fundamental nonlinear equations for the **I-V characteristics defined by** parameter.

#### **Collector-emitter saturation voltage, $V_{ce(sat)}$**

The collector-emitter voltage for a typical on-state as specified by the manufacturer. The default value is 2.8 V. This parameter is visible only when you select Fundamental nonlinear equations for the **I-V characteristics defined by** parameter.

#### **Collector current at which $V_{ce(sat)}$ is defined**

The collector-emitter current when the gate-emitter voltage is  $V_{ge(sat)}$  and collector-emitter voltage is  $V_{ce(sat)}$ . The default value is 400 A. This parameter is visible only when you select Fundamental nonlinear equations for the **I-V characteristics defined by** parameter.

#### **Gate-emitter voltage at which $V_{ce(sat)}$ is defined**

The gate voltage used when measuring  $V_{ce(sat)}$  and  $I_{ce(sat)}$ . The default value is 10 V. This parameter is visible only when you select Fundamental nonlinear equations for the **I-V characteristics defined by** parameter.

#### **Measurement temperature**

The temperature for which the parameters are quoted ( $T_m$ ). The default value is 25 °C. This parameter is visible only when you select Fundamental nonlinear equations for the **I-V characteristics defined by** parameter.

#### **Vector of gate-emitter voltages, $V_{ge}$**

The vector of gate-emitter voltages, to be used for table lookup. The vector values must be strictly increasing. The values can be nonuniformly spaced. The default values, in V, are [-2 6 7 8 10 12 15 20]. This parameter is visible only when you select Lookup table (2-D, temperature independent) or Lookup table

(3-D, temperature dependent) for the **I-V characteristics defined by** parameter.

### Vector of collector-emitter voltages, Vce

The vector of collector-emitter voltages, to be used for table lookup. The vector values must be strictly increasing. The values can be nonuniformly spaced. The default values, in V, are [-1 0 0.5 1 1.5 2 2.5 3 3.5 4]. This parameter is visible only when you select **Lookup table** (2-D, temperature independent) or **Lookup table** (3-D, temperature dependent) for the **I-V characteristics defined by** parameter.

### Vector of temperatures, T

The vector of temperatures, to be used for table lookup. The vector values must be strictly increasing. The values can be nonuniformly spaced. The default values, in degC, are [25 125]. This parameter is visible only when you select **Lookup table** (3-D, temperature dependent) for the **I-V characteristics defined by** parameter.

### Tabulated collector currents, Ic=fcn(Vge,Vce)

Tabulated values for collector current as a function of gate-emitter voltage and collector-emitter voltage, to be used for 2-D table lookup. Each value in the matrix specifies the collector current for a specific combination of gate-emitter voltage and collector-emitter voltage. The matrix size must match the dimensions defined by the gate-emitter voltage and collector-emitter voltage vectors. The default values, in A, are:

```
[-1.015e-5 1.35e-8 4.7135e-4 5.092e-4 5.105e-4 5.1175e-4 5.1299e-4 5.1423e-4 5.1548e-4 5.1672e-4;
-9.9869e-6 1.35e-8 4.7135e-4 5.092e-4 5.105e-4 5.1175e-4 5.1299e-4 5.1423e-4 5.1548e-4 5.1672e-4;
-9.955e-6 1.35e-8 0.0065225 3.3324 48.154 93.661 105.52 105.72 105.93 106.14;
-9.955e-6 1.35e-8 0.0065235 3.5783 70.264 166.33 252.4 317.67 353.38 357.39;
-9.955e-6 1.35e-8 0.006524 3.7206 89.171 228.09 371.63 511.02 642.69 764.04;
-9.9549e-6 1.35e-8 0.0065242 3.7716 97.793 256.21 424.27 592.92 759.2 921.52;
-9.9549e-6 1.35e-8 0.0065243 3.8067 104.52 278.11 464.6 654.37 844.57 1.0339e+3;
-9.9549e-6 1.35e-8 0.0065244 3.8324 109.92 295.67 496.54 702.28 909.96 1.1183e+3]
```

This parameter is visible only when you select **Lookup table** (2-D, temperature independent) for the **I-V characteristics defined by** parameter.

### Tabulated collector currents, Ic=fcn(Vge,Vce,T)

Tabulated values for collector current as a function of gate-emitter voltage, collector-emitter voltage, and temperature, to be used for 3-D table lookup. Each value in the matrix specifies the collector current for a specific combination of gate-emitter voltage and collector-emitter voltage at a specific temperature. The matrix size must match the dimensions defined by the gate-emitter voltage, collector-emitter voltage, and temperature vectors. The default value, in A, is zeros(8, 10, 2).

This parameter is visible only when you select Lookup table (3-D, temperature dependent) for the **I-V characteristics defined by** parameter.

## Junction Capacitance Tab (Default Block Variant)

### Parameterization

Select one of the following methods for block parameterization:

- Specify fixed input, reverse transfer and output capacitance — Provide fixed parameter values from datasheet and let the block convert the input, output, and reverse transfer capacitance values to junction capacitance values, as described in “Charge Model” on page 1-900. This is the default method.
- Specify fixed gate-emitter, gate-collector and collector-emitter capacitance — Provide fixed values for junction capacitance parameters directly.
- Specify tabulated input, reverse transfer and output capacitance — Provide tabulated capacitance and collector-emitter voltage values based on datasheet plots. The block converts the input, output, and reverse transfer capacitance values to junction capacitance values, as described in “Charge Model” on page 1-900.
- Specify tabulated gate-emitter, gate-collector and collector-emitter capacitance — Provide tabulated values for junction capacitances and collector-emitter voltage.

### Input capacitance, Cies

The gate-emitter capacitance with the collector shorted to the source. This parameter is visible only for the following two values for the **Parameterization** parameter:

- If you select Specify fixed input, reverse transfer and output capacitance, the default value is 26.4 nF.
- If you select Specify tabulated input, reverse transfer and output capacitance, the default value is [80 40 32 28 27.5 27 26.5 26.5 26.5] nF.

### Reverse transfer capacitance, Cres

The collector-gate capacitance with the emitter connected to ground. This parameter is visible only for the following two values for the **Parameterization** parameter:

- If you select Specify fixed input, reverse transfer and output capacitance, the default value is 2.7 nF.
- If you select Specify tabulated input, reverse transfer and output capacitance, the default value is [55 9 5.5 3.1 2.5 2.1 1.9 1.8 1.7] nF.

### Output capacitance, Coes

The collector-gate capacitance with the gate and emitter shorted. This parameter is visible only for the following two values for the **Parameterization** parameter:

- If you select Specify fixed input, reverse transfer and output capacitance, the default value is 0 nF.
- If you select Specify tabulated input, reverse transfer and output capacitance, the default value is [60 20 12 8 6 4.8 4 3.5 3.1] nF.

### Gate-emitter junction capacitance

The value of the capacitance placed between the gate and the emitter. This parameter is visible only for the following two values for the **Parameterization** parameter:

- If you select Specify fixed gate-emitter, gate-collector and collector-emitter capacitance, the default value is 23.7 nF.
- If you select Specify tabulated gate-emitter, gate-collector and collector-emitter capacitance, the default value is [25 31 26.5 24.9 25 24.9 24.6 24.7 24.8] nF.

### Gate-collector junction capacitance

The value of the capacitance placed between the gate and the collector. This parameter is visible only for the following two values for the **Parameterization** parameter:

- If you select Specify fixed gate-emitter, gate-collector and collector-emitter capacitance, the default value is 2.7 nF.
- If you select Specify tabulated gate-emitter, gate-collector and collector-emitter capacitance, the default value is [55 9 5.5 3.1 2.5 2.1 1.9 1.8 1.7] nF.

### Collector-emitter junction capacitance

The value of the capacitance placed between the collector and the emitter. This parameter is visible only for the following two values for the **Parameterization** parameter:

- If you select Specify fixed gate-emitter, gate-collector and collector-emitter capacitance, the default value is 0 nF.
- If you select Specify tabulated gate-emitter, gate-collector and collector-emitter capacitance, the default value is [5 11 6.5 4.9 3.5 2.7 2.1 1.7 1.4] nF.

### Corresponding collector-emitter voltages

The collector-emitter voltages corresponding to the tabulated capacitance values. This parameter is visible only for tabulated capacitance models (Specify tabulated input, reverse transfer and output capacitance or Specify tabulated gate-emitter, gate-collector and output capacitance). The default value is [0 1 2 5 10 15 20 25 30] V.

### Charge-voltage linearity

Select whether gate-drain capacitance is fixed or nonlinear:

- Gate-collector capacitance is constant — The capacitance value is constant and defined according to the selected parameterization option, either directly or derived from a datasheet. This is the default method.
- Gate-collector charge function is nonlinear — The gate-collector charge relationship is defined according to the piecewise-nonlinear function described in “Charge Model” on page 1-900. Two additional parameters appear to let you define the gate-collector charge function.

This parameter is visible only for fixed capacitance models (Specify fixed input, reverse transfer and output capacitance or Specify fixed gate-emitter, gate-collector and output capacitance).

### Gate-collector oxide capacitance

The gate-collector capacitance when the device is on and the collector-gate voltage is small. This parameter is visible only when you select Gate-collector charge function is nonlinear for the **Charge-voltage linearity** parameter. The default value is 20 nF.

### Collector-gate voltage below which oxide capacitance becomes active

The collector-gate voltage at which the collector-gate capacitance switches between off-state ( $C_{GC}$ ) and on-state ( $C_{ox}$ ) capacitance values. This parameter is visible only when you select Gate-collector charge function is nonlinear for the **Charge-voltage linearity** parameter. The default value is -5 V.

### **Total forward transit time**

The forward transit time for the PNP transistor used as part of the underlying IGBT model. It affects how quickly charge is removed from the channel when the IGBT is turned off. The default value is 0  $\mu$ s.

## **Advanced Tab (Default Block Variant)**

The lookup table representation combines all the equivalent circuit components into one lookup table, and therefore this tab is empty. If you use the equivalent circuit representation, this tab has the following parameters.

### **Emission coefficient, N**

The emission coefficient or ideality factor of the bipolar transistor. The default value is 1.

### **Forward Early voltage, VAF**

The forward Early voltage for the PNP transistor used in the IGBT model. See the PNP Bipolar Transistor block reference page for more information. The default value is 200 V.

### **Collector resistance, RC**

Resistance at the collector. The default value is 0.001 Ohm.

### **Emitter resistance, RE**

Resistance at the emitter. The default value is 0.001 Ohm.

### **Internal gate resistance, RG**

The value of the internal gate resistor at the measurement temperature. Note that this is not the value of the external circuit series gate resistance, which you should model externally to the IGBT. The default value is 0.001 Ohm.

### **Forward current transfer ratio, BF**

Ideal maximum forward current gain for the PNP transistor used in the IGBT model. See the PNP Bipolar Transistor block reference page for more information. The default value is 50.

## **Temperature Dependence Tab (Default Block Variant)**

For the 2-D lookup table representation, the electrical equations do not depend on temperature and therefore this tab is empty. For the 3-D lookup table representation with exposed thermal port, this tab is also empty because the 3-D matrix on the **Main** tab



captures the temperature dependence. If the block thermal port is not exposed for the 3-D lookup table representation, then this tab contains only the **Device simulation temperature** parameter. If you use the equivalent circuit representation, this tab has the following parameters.

### Parameterization

Select one of the following methods for temperature dependence parameterization:

- **None – Simulate at parameter measurement temperature –** Temperature dependence is not modeled, and none of the other parameters on this tab are visible. This is the default method.
- **Specify  $I_{ces}$  and  $V_{ce(sat)}$  at second measurement temperature –** Model temperature-dependent effects by providing values for the zero gate voltage collector current,  $I_{ces}$ , and collector-emitter voltage,  $V_{ce(sat)}$ , at the second measurement temperature.
- **Specify  $V_{ce(sat)}$  at second measurement temperature plus the energy gap, EG –** Use this option when the datasheet does not provide information on the zero gate voltage collector current,  $I_{ces}$ , at a higher measurement temperature.

### Energy gap, EG

Energy gap value. This parameter is visible only when you select **Specify  $V_{ce(sat)}$  at second measurement temperature plus the energy gap, EG** for the **Parameterization** parameter. The default value is 1.11 eV.

### Zero gate voltage collector current, $I_{ces}$ , at second measurement temperature

The zero gate collector current value at the second measurement temperature. This parameter is visible only when you select **Specify  $I_{ces}$  and  $V_{ce(sat)}$  at second measurement temperature** for the **Parameterization** parameter. The default value is 100 mA.

### Collector-emitter saturation voltage, $V_{ce(sat)}$ , at second measurement temperature

The collector-emitter saturation voltage value at the second measurement temperature, and when the collector current and gate-emitter voltage are as defined by the corresponding parameters on the **Main** tab. The default value is 3 V.

### Second measurement temperature

Second temperature  $T_{m2}$  at which **Zero gate voltage collector current,  $I_{ces}$ , at second measurement temperature** and **Collector-emitter saturation voltage,**

**Vce(sat), at second measurement temperature** are measured. The default value is 125 °C.

#### **Saturation current temperature exponent, XTI**

The saturation current exponent value for your device type. If you have graphical data for the value of  $I_{ces}$  as a function of temperature, you can use it to fine-tune the value of  $XTI$ . The default value is 3.

#### **Mobility temperature exponent, BEX**

Mobility temperature coefficient value. You can use the default value for most devices. If you have graphical data for  $V_{ce(sat)}$  at different temperatures, you can use it to fine-tune the value of  $BEX$ . The default value is -1.5.

#### **Internal gate resistance temperature coefficient**

Represents the fractional rate of change ( $\alpha$ ) of internal gate resistance (RG) with temperature. Thus the gate resistance is  $R = R_{meas}(1 + \alpha(T_s - T_{m1}))$ , where  $R_{meas}$  is the **Internal gate resistance, RG** parameter value. The default value is 0 1/K.

#### **Device simulation temperature**

Temperature  $T_s$  at which the device is simulated. The default value is 25 °C.

### **Main Tab (Event-Based Block Variant)**

This configuration of the **Main** tab corresponds to the simplified, event-based block variant. If you are using the detailed variant of the block, see “Main Tab (Default Block Variant)” on page 1-908.

#### **Vector of temperatures, Tj**

Temperature values at which the collector-emitter and turn-on/turn-off losses are quoted. The default values, in K, are [298.15, 398.15]. This parameter is visible only if your block has an exposed thermal port.

#### **Vector of collector currents, Ic**

Collector currents for which the on-state collector-emitter voltages are defined. The first element must be zero. The default values, in A, are [0, 10, 50, 100, 200, 400, 600].

#### **Corresponding on-state collector-emitter voltages**

Collector-emitter voltages corresponding to the vector of collector currents. The first element must be zero. The default values, in V, are [0, 1.1, 1.3, 1.45, 1.75, 2.25, 2.7]. If your block has an exposed thermal port, this parameter is replaced

with the **Collector-emitter on-state voltages,  $V_{ce}=f_{cn}(T_j, I_c)$**  parameter, which defines the voltages in terms of both temperature and current.

### **Collector-emitter on-state voltages, $V_{ce}=f_{cn}(T_j, I_c)$**

Collector-emitter voltages when in the on state, defined as a function of both temperature and current. The default values, in V, are [0, 1.1, 1.3, 1.45, 1.75, 2.25, 2.7; 0, 1.0, 1.15, 1.35, 1.7, 2.35, 3.0]. This parameter is visible only if your block has an exposed thermal port.

### **Turn-on switching losses, $E_{on}=f_{cn}(T_j, I_c)$**

Energy loss when turning the device on, defined as a function of temperature and final on-state current. The default values, in J, are [0, 0.2, 1, 2, 4, 8, 15; 0, 0.3, 1.3, 2.5, 5, 11, 18]\*1e-3. This parameter is visible only if your block has an exposed thermal port.

### **Turn-off switching losses, $E_{off}=f_{cn}(T_j, I_c)$**

Energy loss when turning the device off, defined as a function of temperature and final on-state current. The default values, in J, are [0, 0.3, 1.5, 3, 6, 15, 25; 0, 0.7, 3.3, 6.5, 13, 25, 35]\*1e-3. This parameter is visible only if your block has an exposed thermal port.

### **Miller resistance**

When the device turns on, it has a constant-value Miller resistance in series with the demanded voltage ramp. This resistance represents the partial conductance path through the device during turn on, and can be used to match the voltage spike observed when reconnecting a current-carrying inductor and corresponding freewheeling diode. A typical value is 10 to 50 times the effective on-state resistance. The default value is 0.1 Ohm.

### **Off-state conductance**

Conductance when the device is in the off state. The default value is 1e-5 1/Ohm.

### **Threshold voltage, $V_{th}$**

The gate-emitter voltage must be greater than this value for the device to turn on. The default value is 6 V.

## **Dynamics Tab (Event-Based Block Variant)**

### **Turn-on delay**

Time before which the device starts to ramp on. The default value is 0.07 us.

**Current rise time**

Time taken for the current to ramp up when driving a resistive load. The default value is 0.7  $\mu$ s.

**Turn-off delay**

Time before which the device starts to ramp off. The default value is 0.2  $\mu$ s.

**Current fall time**

Time taken for the current to ramp down when driving a resistive load. The default value is 0.5  $\mu$ s.

**Off-state voltage for rise and fall times**

Off-state collector-emitter voltage used when specifying the rise and fall times. The default value is 300 V. If your block has an exposed thermal port, this parameter is replaced with the **Off-state voltage for timing and losses data** parameter, which defines the voltage used when specifying the rise and fall times and also the losses data, also with the default value of 300 V.

## Ports

The block has the following ports:

C

Electrical conserving port associated with the PNP emitter terminal

G

Electrical conserving port associated with the IGBT gate terminal

E

Electrical conserving port associated with the PNP collector terminal

# N-Channel JFET

Model N-Channel JFET

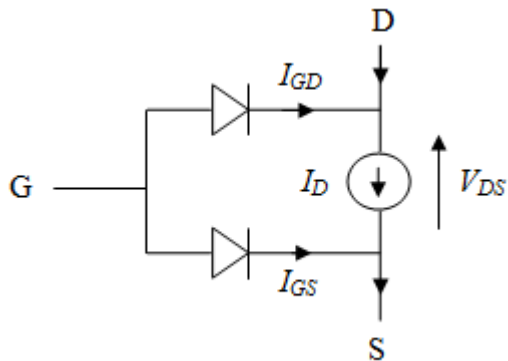


## Library

Semiconductor Devices

## Description

The N-Channel JFET block uses the Shichman and Hodges equations to represent an N-Channel JFET using a model with the following structure:



G is the transistor gate, D is the transistor drain, and S is the transistor source. The drain current,  $I_D$ , depends on the region of operation and whether the transistor is operating in normal or inverse mode.

- In normal mode ( $V_{DS} \geq 0$ ), the block provides the following relationship between the drain current  $I_D$  and the drain-source voltage  $V_{DS}$ .

Region	Applicable Range of $V_{GS}$ and $V_{DS}$ Values	Corresponding $I_D$ Equation
Off	$V_{GS} - V_{t0} \leq 0$	$I_D = 0$
Linear	$0 < V_{DS} < V_{GS} - V_{t0}$	$I_D = \beta V_{DS}(2(V_{GS} - V_{t0}) - V_{DS})(1 + \lambda V_{DS})$
Saturated	$0 < V_{GS} - V_{t0} \leq V_{DS}$	$I_D = \beta (V_{GS} - V_{t0})^2 (1 + \lambda V_{DS})$

- In inverse mode ( $V_{DS} < 0$ ), the block provides the following relationship between the drain current  $I_D$  and the drain-source voltage  $V_{DS}$ .

Region	Applicable Range of $V_{GS}$ and $V_{DS}$ Values	Corresponding $I_D$ Equation
Off	$V_{GD} - V_{t0} \leq 0$	$I_D = 0$
Linear	$0 < -V_{DS} < V_{GD} - V_{t0}$	$I_D = \beta V_{DS}(2(V_{GD} - V_{t0}) + V_{DS})(1 - \lambda V_{DS})$
Saturated	$0 < V_{GD} - V_{t0} \leq -V_{DS}$	$I_D = \beta (V_{GD} - V_{t0})^2 (1 - \lambda V_{DS})$

In the preceding equations:

- $V_{GS}$  is the gate-source voltage.
- $V_{GD}$  is the gate-drain voltage.
- $V_{t0}$  is the threshold voltage. If you select **Specify using equation parameters directly** for the **Parameterization** parameter,  $V_{t0}$  is the **Threshold voltage** parameter value. Otherwise, the block calculates  $V_{t0}$  from the datasheet parameters you specify.
- $\beta$  is the transconductance parameter. If you select **Specify using equation parameters directly** for the **Parameterization** parameter,  $\beta$  is the **Transconductance parameter** parameter value. Otherwise, the block calculates  $\beta$  from the datasheet parameters you specify.
- $\lambda$  is the channel-length modulation parameter. If you select **Specify using equation parameters directly** for the **Parameterization** parameter,  $\lambda$  is the **Channel-length modulation** parameter value. Otherwise, the block calculates  $\lambda$  from the datasheet parameters you specify.

The currents in each of the diodes satisfy the exponential diode equation

$$I_{GD} = IS \cdot \left( e^{\frac{qV_{GD}}{kT_{m1}}} - 1 \right)$$

$$I_{GS} = IS \cdot \left( e^{\frac{qV_{GS}}{kT_{m1}}} - 1 \right)$$

where:

- $IS$  is the saturation current. If you select **Specify using equation parameters directly** for the **Parameterization** parameter,  $IS$  is the **Saturation current** parameter value. Otherwise, the block calculates  $IS$  from the datasheet parameters you specify.
- $q$  is the elementary charge on an electron (1.602176e-19 Coulombs).
- $k$  is the Boltzmann constant (1.3806503e-23 J/K).
- $T_{m1}$  is the measurement temperature. The value comes from the **Measurement temperature** parameter.

The block models gate junction capacitance as a fixed gate-drain capacitance  $C_{GD}$  and a fixed gate-source capacitance  $C_{GS}$ . If you select **Specify using equation parameters directly** for the **Parameterization** parameter, you specify these values directly using the **Gate-drain junction capacitance** and **Gate-source junction capacitance** parameters. Otherwise, the block derives them from the **Input capacitance Ciss** and **Reverse transfer capacitance Crss** parameter values. The two parameterizations are related as follows:

- $C_{GD} = Crss$
- $C_{GS} = Ciss - Crss$

## Modeling Temperature Dependence

The default behavior is that dependence on temperature is not modeled, and the device is simulated at the temperature for which you provide block parameters. You can optionally include modeling the dependence of the transistor static behavior on temperature during simulation. Temperature dependence of the junction capacitances is not modeled, this being a much smaller effect.

When including temperature dependence, the transistor defining equations remain the same. The measurement temperature value,  $T_{m1}$ , is replaced with the simulation temperature,  $T_s$ . The transconductance,  $\beta$ , and the threshold voltage,  $V_{t0}$ , become a function of temperature according to the following equations:

$$\beta_{T_s} = \beta_{T_{m1}} \left( \frac{T_s}{T_{m1}} \right)^{BEX}$$

$$V_{t0s} = V_{t01} + \alpha (T_s - T_{m1})$$

where:

- $T_{m1}$  is the temperature at which the transistor parameters are specified, as defined by the **Measurement temperature** parameter value.
- $T_s$  is the simulation temperature.
- $\beta_{T_{m1}}$  is JFET transconductance at the measurement temperature.
- $\beta_{T_s}$  is JFET transconductance at the simulation temperature. This is the transconductance value used in the JFET equations when temperature dependence is modeled.
- $V_{t01}$  is the threshold voltage at measurement temperature.
- $V_{t0s}$  is the threshold voltage at simulation temperature. This is the threshold voltage value used in the JFET equations when temperature dependence is modeled.
- $BEX$  is the mobility temperature exponent. A typical value of  $BEX$  is -1.5.
- $\alpha$  is the gate threshold voltage temperature coefficient,  $dV_{th}/dT$ .

For most JFETS, you can use the default value of -1.5 for  $BEX$ . Some datasheets quote the value for  $\alpha$ , but most typically they provide the temperature dependence for the saturated drain current,  $I_{dss}$ . Depending on the block parameterization method, you have two ways of specifying  $\alpha$ :

- If you parameterize the block from a datasheet, you have to provide  $I_{dss}$  at a second measurement temperature. The block then calculates the value for  $\alpha$  based on this data.
- If you parameterize by specifying equation parameters, you have to provide the value for  $\alpha$  directly.



If you have more data comprising drain current as a function of gate-source voltage for fixed drain-source voltage plotted at more than one temperature, then you can also use Simulink Design Optimization software to help tune the values for  $\alpha$  and  $BEX$ .

In addition, the saturation current term,  $IS$ , in the gate-drain and gate-source current equations depends on temperature

$$IS_{T_s} = IS_{T_{m1}} \cdot (T_s / T_{m1})^{XTI} \cdot \exp\left(-\frac{EG}{kT_s}(1 - T_s / T_{m1})\right)$$

where:

- $IS_{T_{m1}}$  is the saturation current at the measurement temperature.
- $IS_{T_s}$  is the saturation current at the simulation temperature. This is the saturation current value used in the bipolar transistor equations when temperature dependence is modeled.
- $EG$  is the energy gap.
- $k$  is the Boltzmann constant (1.3806503e-23 J/K).
- $XTI$  is the saturation current temperature exponent.

Similar to  $\alpha$ , you have two ways of specifying  $EG$  and  $XTI$ :

- If you parameterize the block from a datasheet, you have to specify the gate reverse current,  $I_{gss}$ , at a second measurement temperature. The block then calculates the value for  $\overline{EG}$  based on this data and assuming a p-n junction nominal value of 3 for  $XTI$ .
- If you parameterize by specifying equation parameters, you have to provide the values for  $EG$  and  $XTI$  directly. This option gives you most flexibility to match device behavior, for example, if you have a graph of  $I_{gss}$  as a function of temperature. With this data you can use Simulink Design Optimization software to help tune the values for  $EG$  and  $XTI$ .

## Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal port H on the block icon, and adds the **Thermal Port** tab to the block dialog box.

Use the thermal port to simulate the effects of generated heat and device temperature. For more information on using thermal ports and on the **Thermal Port** tab parameters, see “Simulating Thermal Effects in Semiconductors”.

## Basic Assumptions and Limitations

The model is based on the following assumptions:

- This block does not allow you to specify initial conditions on the junction capacitances. If you select the **Start simulation from steady state** option in the Solver Configuration block, the block solves the initial voltages to be consistent with the calculated steady state. Otherwise, voltages are zero at the start of the simulation.
- You may need to use nonzero ohmic resistance and junction capacitance values to prevent numerical simulation issues, but the simulation may run faster with these values set to zero.
- The block does not account for temperature-dependent effects on the junction capacitances.
- When you specify  $I_{dss}$  at a second measurement temperature, it must be quoted for the same working point (that is, the same drain current and gate-source voltage) as for the  $I_{dss}$  value on the **Main** tab. Inconsistent values for  $I_{dss}$  at the higher temperature will result in unphysical values for  $\alpha$  and unrepresentative simulation results.
- You may need to tune the value of  $BEX$  to replicate the  $I_D$ - $V_{GS}$  relationship (if available) for a given device. The value of  $BEX$  affects whether the  $I_D$ - $V_{GS}$  curves for different temperatures cross each other, or not, for the ranges of  $I_D$  and  $V_{GS}$  considered.

## Parameters

- “Main Tab” on page 1-925
- “Ohmic Resistance Tab” on page 1-926
- “Junction Capacitance Tab” on page 1-926
- “Temperature Dependence Tab” on page 1-927

## Main Tab

### Parameterization

Select one of the following methods for block parameterization:

- **Specify from a datasheet** — Provide parameters that the block converts to equations that describe the transistor. This is the default method.
- **Specify using equation parameters directly** — Provide equation parameters  $\beta$ ,  $IS$ ,  $V_{t0}$ , and  $\lambda$ .

### Gate reverse current, $I_{gss}$

The reverse current that flows in the diode when the drain and source are short-circuited and a large negative gate-source voltage is applied. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is -1 nA.

### Saturated drain current, $I_{dss}$

The current that flows when a large positive drain-source voltage is applied for a specified gate-source voltage. For a depletion-mode device, this gate-source voltage may be zero, in which case  $I_{dss}$  may be referred to as the zero-gate voltage drain current. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is 3 mA.

### $I_{dss}$ measurement point, [V<sub>gs</sub> V<sub>ds</sub>]

A vector of the values of  $V_{GS}$  and  $V_{DS}$  at which  $I_{dss}$  is measured. Normally  $V_{GS}$  is zero.  $V_{DS}$  should be greater than zero. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is [ 0 15 ] V.

### Small-signal parameters, [g<sub>fs</sub> g<sub>os</sub>]

A vector of the values of  $g_{fs}$  and  $g_{os}$ .  $g_{fs}$  is the forward transfer conductance, that is, the conductance for a fixed drain-source voltage.  $g_{os}$  is the output conductance, that is, the conductance for a fixed gate-source voltage. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is [ 3e+03 10 ] uS.

### Small-signal measurement point, [V<sub>gs</sub> V<sub>ds</sub>]

A vector of the values of  $V_{GS}$  and  $V_{DS}$  at which  $g_{fs}$  and  $g_{os}$  are measured.  $V_{DS}$  should be greater than zero. For depletion-mode devices,  $V_{GS}$  is typically zero. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is [ 0 15 ] V.

### **Transconductance parameter**

The derivative of drain current with respect to gate voltage. This parameter is only visible when you select **Specify using equation parameters directly** for the **Parameterization** parameter. The default value is  $1e-04 \text{ A/V}^2$ .

### **Saturation current**

The magnitude of the current that the ideal diode equation approaches asymptotically for very large reverse bias levels. This parameter is only visible when you select **Specify using equation parameters directly** for the **Parameterization** parameter. The default value is  $1e-14 \text{ A}$ .

### **Threshold voltage**

The gate-source voltage above which the transistor produces a nonzero drain current. For an enhancement device,  $V_{t0}$  should be positive. For a depletion mode device,  $V_{t0}$  should be negative. This parameter is only visible when you select **Specify using equation parameters directly** for the **Parameterization** parameter. The default value is  $-2 \text{ V}$ .

### **Channel-length modulation**

The channel-length modulation. This parameter is only visible when you select **Specify using equation parameters directly** for the **Parameterization** parameter. The default value is  $0 \text{ 1/V}$ .

### **Measurement temperature**

The temperature for which the datasheet parameters are quoted. The default value is  $25 \text{ }^\circ\text{C}$ .

## **Ohmic Resistance Tab**

### **Source ohmic resistance**

The transistor source resistance. The default value is  $1e-4 \text{ } \Omega$ . The value must be greater than or equal to  $0$ .

### **Drain ohmic resistance**

The transistor drain resistance. The default value is  $0.01 \text{ } \Omega$ . The value must be greater than or equal to  $0$ .

## **Junction Capacitance Tab**

### **Parameterization**

Select one of the following methods for block parameterization:

- **Specify from a datasheet** — Provide parameters that the block converts to junction capacitance values. This is the default method.
- **Specify using equation parameters directly** — Provide junction capacitance parameters directly.

### **Input capacitance, Ciss**

The gate-source capacitance with the drain shorted to the source. This parameter is only visible when you select **Specify from a datasheet** for the **Model junction capacitance** parameter. The default value is 4.5 pF.

### **Reverse transfer capacitance, Crss**

The drain-gate capacitance with the source connected to ground. This parameter is only visible when you select **Specify from a datasheet** for the **Model junction capacitance** parameter. The default value is 1.5 pF.

### **Gate-source junction capacitance**

The value of the capacitance placed between the gate and the source. This parameter is only visible when you select **Specify using equation parameters directly** for the **Model junction capacitance** parameter. The default value is 3 pF.

### **Gate-drain junction capacitance**

The value of the capacitance placed between the gate and the drain. This parameter is only visible when you select **Specify using equation parameters directly** for the **Model junction capacitance** parameter. The default value is 1.5 pF.

## **Temperature Dependence Tab**

### **Parameterization**

Select one of the following methods for temperature dependence parameterization:

- **None** — Simulate at parameter measurement temperature — Temperature dependence is not modeled. This is the default method.
- **Model temperature dependence** — Model temperature-dependent effects. You also have to provide a set of additional parameters depending on the block parameterization method. If you parameterize the block from a datasheet, you have to provide values for  $I_{gss}$  and  $I_{dss}$  at second measurement temperature. If you parameterize by directly specifying equation parameters, you have to provide the values for  $EG$ ,  $XTI$ , and the gate threshold voltage temperature coefficient,  $dV_{t0}/dT$ . Regardless of the block parameterization method, you also have to provide values for  $BEX$  and for the simulation temperature,  $T_s$ .

**Gate reverse current,  $I_{gss}$ , at second measurement temperature**

The value of the gate reverse current,  $I_{gss}$ , at the second measurement temperature. This parameter is only visible when you select **Specify** from a datasheet for the **Parameterization** parameter on the **Main** tab. It must be quoted for the same working point (drain current and gate-source voltage) as the **Drain-source on resistance,  $R_{DS(on)}$**  parameter on the **Main** tab. The default value is -200 nA.

**Saturated drain current,  $I_{dss}$ , at second measurement temperature**

The value of the saturated drain current,  $I_{dss}$ , at the second measurement temperature, and when the  $I_{dss}$  measurement point is the same as defined by the  **$I_{dss}$  measurement point,  $[V_{gs} V_{ds}]$**  parameter on the **Main** tab. This parameter is only visible when you select **Specify** from a datasheet for the **Parameterization** parameter on the **Main** tab. The default value is 2.5 mA.

**Second measurement temperature**

Second temperature  $T_{m2}$  at which **Gate reverse current,  $I_{gss}$ , at second measurement temperature** and **Saturated drain current,  $I_{dss}$ , at second measurement temperature** are measured. This parameter is only visible when you select **Specify** from a datasheet for the **Parameterization** parameter on the **Main** tab. The default value is 125 °C.

**Energy gap, EG**

Energy gap value. This parameter is only visible when you select **Specify** using equation parameters directly for the **Parameterization** parameter on the **Main** tab. The default value is 1.11 eV.

**Saturation current temperature exponent, XTI**

Saturation current temperature coefficient value. This parameter is only visible when you select **Specify** using equation parameters directly for the **Parameterization** parameter on the **Main** tab. The default value is 3.

**Gate threshold voltage temperature coefficient,  $dV_{t0}/dT$** 

The rate of change of gate threshold voltage with temperature. This parameter is only visible when you select **Specify** using equation parameters directly for the **Parameterization** parameter on the **Main** tab. The default value is -6 mV/K.

**Mobility temperature exponent, BEX**

Mobility temperature coefficient value. You can use the default value for most JFETs. See the “Basic Assumptions and Limitations” on page 1-924 section for additional considerations. The default value is -1.5.

**Device simulation temperature**

Temperature  $T_s$  at which the device is simulated. The default value is 25 °C.

## Ports

The block has the following ports:

G

Electrical conserving port associated with the transistor gate terminal

D

Electrical conserving port associated with the transistor drain terminal

S

Electrical conserving port associated with the transistor source terminal

## References

- [1] H. Shichman and D. A. Hodges, *Modeling and simulation of insulated-gate field-effect transistor switching circuits*. IEEE J. Solid State Circuits, SC-3, 1968.
- [2] G. Massobrio and P. Antognetti. *Semiconductor Device Modeling with SPICE*. 2nd Edition, McGraw-Hill, 1993. Chapter 2.

## See Also

P-Channel JFET

## N-Channel LDMOS FET

Model N-Channel LDMOS or VDMOS transistors suitable for high voltage



### Library

Semiconductor Devices

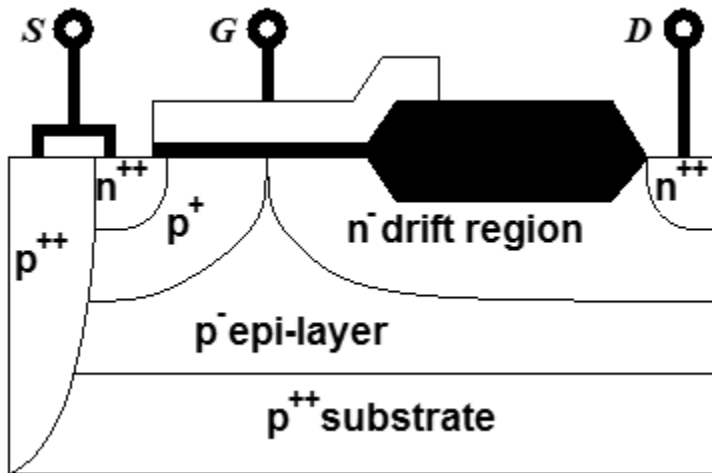
### Description

The N-Channel LDMOS FET block lets you model LDMOS (or VDMOS) transistors suitable for high voltage. The model is based on surface potential and includes effects due to an extended drain (drift) region:

- Nonlinear capacitive effects associated with the drift region
- Surface scattering and velocity saturation in the drift region
- Velocity saturation and channel-length modulation in the channel region
- Charge conservation inside the model, so you can use the model for charge sensitive simulations
- The intrinsic body diode
- Reverse recovery in the body diode model
- Temperature scaling of physical parameters
- For the thermal variant (see “Thermal Port” on page 1-936), dynamic self-heating

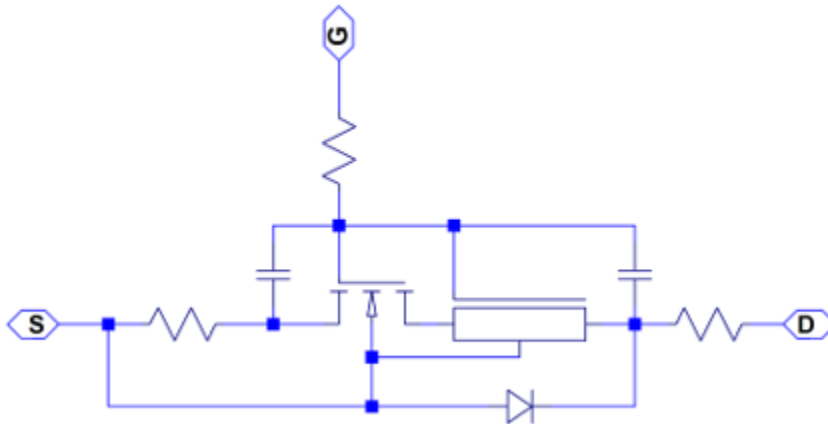
The physical structure of the model is shown in the following figure.





The channel region is in the  $p^+$  region, from the heavily  $n$ -doped source well to the end of the  $p^+$  region. The drift region is a lightly doped drain extension. Further down, there is a  $p$ -type epi-layer, and then the entire structure is on a heavily  $p$ -doped substrate. The gate oxide is thin over the entire channel region and over part of the drift region. Further into the drift region, the gate oxide has a greater thickness in the local-oxidation-of-silicon (LOCOS) region.

The next figure shows the equivalent circuit of the model.



The modeling approach is similar to [1]. The overlaps of the gate contact with the source and drain n-wells are modeled as lumped linear capacitances. The channel (p+) region is modeled using the surface-potential-based MOSFET model. The pn-junction between the source/bulk and drain is modeled using an ideal diode, including both junction and diffusion capacitances. The drift region underneath the thin gate oxide is modeled according to a surface-potential formulation, which includes:

- The current due to the accumulation layer at the semiconductor-oxide interface
- The current due to the electrons flowing towards the drain deeper inside the drift region

The space-charge region between the epi-layer and the drift region is represented using a pinching effect on the current flowing through the bulk of the drift region. The LOCOS part of the drift region is modeled as a lumped, series resistor, and there are also series resistances added to the source and gate contacts.

For detailed description of the channel model, see the surface-potential-based model of the N-Channel MOSFET block. The drift region model is similarly derived from the surface potential using the Poisson equation. For an n-type semiconductor under the gradual-channel approximation, the defining equations are:

$$\frac{\partial^2 \psi}{\partial y^2} \approx \frac{qN_D}{\epsilon_{Si}} \left[ -1 - \exp\left(\frac{-\psi - 2\phi_B}{\phi_T}\right) + \exp\left(\frac{\psi - V_{CB}}{\phi_T}\right) \right]$$

$$\phi_T = \frac{k_B T}{q}$$

where:

- $\psi$  is the electrostatic potential.
- $q$  is the magnitude of the electronic charge.
- $N_D$  is the doping density of the drift region.
- $\epsilon_{Si}$  is the dielectric permittivity of the semiconductor material (for example, silicon).
- $\phi_B$  is the difference between the intrinsic Fermi level and the Fermi level deep in the drift region.
- $V_{CB}$  is the quasi-Fermi potential of the drift region referenced to the bulk.
- $\phi_T$  is the thermal voltage.

- $k_B$  is Boltzmann's constant.
- $T$  is temperature.

If we neglect inversion for the DC current model, we obtain the following current expression:

$$I_D = \frac{1}{1 + \theta_{sat} V_{DK}} \left[ \frac{f_{lin}}{R_D} V_{DK} + \frac{\beta}{2} \cdot \frac{V_{GK}^2 - V_{GD}^2}{1 + \frac{\theta_{surf}}{2} (V_{GK} + V_{GD})} \right]$$

where:

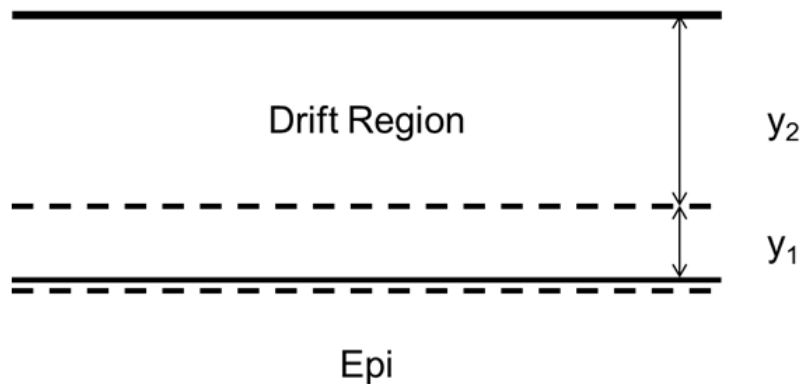
- $I_D$  is the drain current.
- $\theta_{sat}$  is the velocity saturation.
- $V_{ij}$  is the voltage difference between nodes  $i$  and  $j$ , where subscripts D and K refer to the drain and to the junction of the channel and drift regions, respectively, and subscript G refers to the gate with a correction due to the flatband voltage being applied.
- $f_{lin}/R_D$  represents the conductance of the bulk of the drift region, including the effect of pinching due to depletion from the epi-drift interface.
- $\beta$  is the gain of the accumulation layer at the interface between the drift region and the thin gate oxide.
- $\theta_{surf}$  is the parameter that accounts for scattering in the accumulation layer due to the vertical electric field.

The pinching off of the bulk part of the drift region is described by

$$f_{lin} = 1 - \lambda_D \frac{\sqrt{V_{bi} + V_{SB}} - \sqrt{V_{bi}}}{\sqrt{V_{bi}}}$$

where:

- $\lambda_D$  is the parameter representing the n-side vertical depth of the space-charge region along the epi-drift interface at zero bias divided by the vertical depth of the undepleted part of the drift region at zero bias.



In the figure, the top solid line is the semiconductor surface. The lower solid line is the junction between the drift region and the epi layer. The dashed lines show the extent of the space-charge region around the drift-epi interface.  $\lambda_D$  is  $y_1/y_2$  at zero bias.

- $V_{bi}$  is the built-in voltage for the epi-drift diode.
- $V_{SB}$  is the source-body voltage, used as an approximation to the bias applied across the epi-drift diode. Using this voltage instead of  $V_{KB}$  is more numerically stable, and is justified because most of the drain-source voltage drops across the drift region in the transistor on-state.

The charge model is similar to that of the surface-potential-based MOSFET model, with additional expressions to account for the charge in the drift region. The block uses the derived equations as described in [1], which include both inversion and accumulation in the drift region.

## Modeling Body Diode

The block models the body diode as an ideal, exponential diode with both junction and diffusion capacitances:

$$I_{dio} = I_s \left[ \exp\left(-\frac{V_{DB}}{n\phi_T}\right) - 1 \right]$$

$$C_j = \frac{C_{j0}}{\sqrt{1 + \frac{V_{DB}}{V_{bi}}}}$$

$$C_{diff} = \frac{\tau I_s}{n\phi_T} \exp\left(-\frac{V_{DB}}{n\phi_T}\right)$$

where:

- $I_{dio}$  is the current through the diode.
- $I_s$  is the reverse saturation current.
- $V_{DB}$  is the drain-body voltage.
- $n$  is the ideality factor.
- $\phi_T$  is the thermal voltage.
- $C_j$  is the junction capacitance of the diode.
- $C_{j0}$  is the zero-bias junction capacitance.
- $V_{bi}$  is the built-in voltage.
- $C_{diff}$  is the diffusion capacitance of the diode.
- $\tau$  is the transit time.

The capacitances are defined through an explicit calculation of charges, which are then differentiated to give the capacitive expressions above. The block computes the capacitive diode currents as time derivatives of the relevant charges, similar to the computation in the surface-potential-based MOSFET model.

## Modeling Temperature Dependence

The default behavior is that dependence on temperature is not modeled, and the device is simulated at the temperature for which you provide block parameters. To model the dependence on temperature during simulation, select **Model temperature dependence** for the **Parameterization** parameter on the **Temperature Dependence** tab.

The model includes temperature effects on the capacitance characteristics, as well as modeling the dependence of the transistor static behavior on temperature during simulation.

The **Measurement temperature** parameter on the **Main** tab specifies temperature  $T_{m1}$  at which the other device parameters have been extracted. The **Temperature Dependence** tab provides the simulation temperature,  $T_s$ , and the temperature-scaling coefficients for the other device parameters. For more information, see “Temperature Dependence Tab” on page 1-940.

## Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal port H on the block icon, and adds the **Thermal Port** tab to the block dialog box.

Use the thermal port to simulate the effects of generated heat and device temperature. For more information on using thermal ports and on the **Thermal Port** tab parameters, see “Simulating Thermal Effects in Semiconductors”.

The thermal variant of the block includes dynamic self-heating, that is, lets you simulate the effect of self-heating on the electrical characteristics of the device.

## Parameters

- “Main Tab” on page 1-936
- “Ohmic Resistance Tab” on page 1-939
- “Capacitances Tab” on page 1-939
- “Body Diode Tab” on page 1-940
- “Temperature Dependence Tab” on page 1-940

## Main Tab

### Gain, [channel drift\_region]

The gain,  $\beta$ , of the MOSFET regions. The parameter value is a two-element vector, with the first element corresponding to the channel, and the second — to the drift region. This parameter primarily defines the linear region of operation on an  $I_D$ - $V_{DS}$  characteristic. The values of both elements must be greater than 0. The default value is [11.6, 0.01] A/V<sup>2</sup>.

**Flatband voltage, [channel drift\_region]**

The flatband voltage,  $V_{FB}$ , defines the gate bias that must be applied in order to achieve the flatband condition at the surface of the silicon. The parameter value is a two-element vector, with the first element corresponding to the channel, and the second — to the drift region. The default value is  $[-1.05, -0.1]$  V. You can also use this parameter to arbitrarily shift the threshold voltage due to material work function differences, and to trapped interface or oxide charges. In practice, however, it is usually recommended to modify the threshold voltage by using the **Body factor** and **Surface potential at strong inversion** parameters first, and only use this parameter for fine-tuning.

The threshold voltage for the channel region, for a short-circuited source-bulk connection, is approximately

$$V_T = V_{FB} + 2\phi_B + 2\phi_T + \gamma\sqrt{2\phi_B + 2\phi_T}$$

where  $2\phi_B$  is the surface potential at strong inversion and  $\gamma$  is the body factor, both at the channel region.

**Body factor, [channel drift\_region]**

Body factor,  $\gamma$ , in the surface-potential equation. The parameter value is a two-element vector, with the first element corresponding to the channel, and the second — to the drift region. The default value is  $[3.4, 2.5]$  V<sup>1/2</sup>.

For the channel region, the body factor is

$$\gamma = \frac{\sqrt{2q\epsilon_{Si}N_A}}{C_{ox}}$$

See the N-Channel MOSFET block reference page for details on this equation. The drift region equation is similar, except that  $N_A$  is replaced by the doping density,  $N_D$ . The channel-region parameter value primarily impacts the threshold voltage. For the drift region, this parameter primarily affects the charge model, and also has a minor effect on the pinch-off behavior of the bulk current through the drift region.

**Surface potential at strong inversion, [channel drift\_region]**

The  $2\phi_B$  term in the surface-potential equation. The parameter value is a two-element vector, with the first element corresponding to the channel, and the second — to the drift region. The default value is  $[0.95, 0.95]$  V.

The channel-region parameter value also primarily impacts the threshold voltage. For the drift region, this parameter affects the charge model only.

**Velocity saturation factor, [channel drift\_region]**

Velocity saturation,  $\theta_{\text{sat}}$ , in the drain-current equation. Use this parameter in cases where a good fit to linear operation leads to a saturation current that is too high. By increasing this parameter value, you reduce the saturation current. The parameter value is a two-element vector, with the first element corresponding to the channel, and the second — to the drift region. The default value is [0.0, 0.1] 1/V, which means that velocity saturation in the channel region is off by default.

**Drift region surface scattering factor**

Surface scattering factor,  $\theta_{\text{surf}}$ , in the drain-current equation. This parameter applies to the drift region only and accounts for scattering in the accumulation layer due to the vertical electric field. The default value is 0 1/V.

**Channel-length modulation factor**

The factor,  $\alpha$ , multiplying the logarithmic term in the  $G_{\Delta L}$  equation. See the N-Channel MOSFET block reference page for details on this equation. This parameter describes the onset of channel-length modulation. For device characteristics that exhibit a positive conductance in saturation, increase the parameter value to fit this behavior. This parameter applies to the channel region only. The default value is 0, which means that channel-length modulation is off by default.

**Channel-length modulation voltage**

The voltage  $V_p$  in the  $G_{\Delta L}$  equation. See the N-Channel MOSFET block reference page for details on this equation. This parameter controls the drain-voltage at which channel-length modulation starts to become active. This parameter applies to the channel region only. The default value is 50 mV.

**Linear-to-saturation transition coefficient**

This parameter controls how smoothly the MOSFET transitions from linear into saturation, particularly when velocity saturation is enabled. This parameter can usually be left at its default value, but you can use it to fine-tune the knee of the  $I_D$ - $V_{DS}$  characteristic. This parameter applies both to the channel and drift regions. The expected range for this parameter value is between 2 and 8. The default value is 8.

**Measurement temperature**

Temperature  $T_{m1}$  at which the block parameters are measured. If the **Device simulation temperature** parameter on the **Temperature Dependence** tab differs from this value, then device parameters will be scaled from their defined values according to the simulation and reference temperatures. For more information, see “Temperature Dependence Tab” on page 1-940. The default value is 25 °C.



## Ohmic Resistance Tab

### Source ohmic resistance

The transistor source resistance, that is, the series resistance associated with the source contact. The default value is  $1e-4 \Omega$ . The value must be greater than or equal to  $\theta$ .

### Drain ohmic resistance

The transistor drain resistance, that is, the series resistance associated with the drain contact and with the LOCOS part of the drift region, which is not heavily impacted by the applied gate voltage. The default value is  $0.07 \Omega$ . The value must be greater than or equal to  $\theta$ .

### Gate ohmic resistance

The transistor gate resistance, that is, the series resistance associated with the gate contact. The default value is  $8.4 \Omega$ . The value must be greater than or equal to  $\theta$ .

### Drift region low-bias resistance for gated region

Resistance  $R_D$  in the drain-current equation. It represents the resistance of the bulk part of the drift region in the absence of depletion from the top and bottom interfaces. The default value is  $0.1 \Omega$ . The value must be greater than or equal to  $\theta$ .

### Drift region depletion layer thickness factor

Parameter  $\lambda_D$  in the drain-current equation. It is the ratio of vertical depths  $y_1$  and  $y_2$  at zero bias, where  $y_1$  represents the space-charge region and  $y_2$  represents the undepleted part of the drift region. The default value is  $0.2$ .

## Capacitances Tab

### Oxide capacitance

The parallel plate gate-channel and gate-drift-region capacitance. The parameter value is a two-element vector, with the first element corresponding to the channel, and the second — to the drift region. The default value is  $[1600.0, 1000.0]$  pF.

### Gate-source overlap capacitance

The fixed, linear capacitance associated with the overlap of the gate electrode with the source well. The default value is 15 pF.

### Gate-drain overlap capacitance

The fixed, linear capacitance associated with the overlap of the gate electrode with the drain well. The default value is 15 pF.

## Body Diode Tab

### Reverse saturation current

The current designated by the  $I_s$  symbol in the body-diode equations. The default value is  $1\text{e-}13$  A.

### Built-in voltage

The built-in voltage of the diode, designated by the  $V_{bi}$  symbol in the body-diode equations. The default value is  $0.6$  V.

### Ideality factor

The factor designated by the  $n$  symbol in the body-diode equations. The default value is  $1$ .

### Zero-bias junction capacitance

The capacitance between the drain and bulk contacts at zero-bias due to the body diode alone. It is designated by the  $C_{j0}$  symbol in the body-diode equations. The default value is  $1800$  pF.

### Transit time

The time designated by the  $\tau$  symbol in the body-diode equations. The default value is  $50$  ns.

## Temperature Dependence Tab

### Parameterization

Select one of the following methods for temperature dependence parameterization:

- **None – Simulate at parameter measurement temperature** — Temperature dependence is not modeled. This is the default method.
- **Model temperature dependence** — Model temperature-dependent effects. Provide a value for the device simulation temperature,  $T_s$ , and the temperature-scaling coefficients for other block parameters.

### Device simulation temperature

Temperature  $T_s$  at which the device is simulated. The default value is  $25$  °C.

### Gain temperature exponent, [channel drift\_region]

The parameter value is a two-element vector, with the first element corresponding to the channel, and the second — to the drift region. Both in the channel and the drift region, the MOSFET gain,  $\beta$ , is assumed to scale exponentially with temperature,  $\beta =$

$\beta_{m1}(T_{m1}/T_s)^{\eta_\beta}$ .  $\beta_{m1}$  is the value of the channel or drift region gain, as specified by the **Gain, [channel drift\_region]** parameter from the **Main** tab.  $\eta_\beta$  is the corresponding element of the **Gain temperature exponent, [channel drift\_region]** parameter. The default value is [1.3, 1.3].

#### **Flatband voltage temperature coefficient, [channel drift\_region]**

The parameter value is a two-element vector, with the first element corresponding to the channel, and the second — to the drift region. The flatband voltage,  $V_{FB}$ , is assumed to scale linearly with temperature,  $V_{FB} = V_{FBm1} + (T_s - T_{m1})S_{T,V_{FB}}$ .  $V_{FBm1}$  is the value of the channel or drift region flatband voltage, as specified by the **Flatband voltage, [channel drift\_region]** parameter from the **Main** tab.  $S_{T,V_{FB}}$  is the corresponding element of the **Flatband voltage temperature coefficient, [channel drift\_region]** parameter. The default value is [0.0005, 0.0005] V/K.

#### **Surface potential at strong inversion temperature coefficient**

The surface potential at strong inversion,  $2\phi_B$ , is assumed to scale linearly with temperature,  $2\phi_B = 2\phi_{Bm1} + (T_s - T_{m1})S_{T,\phi_B}$ .  $2\phi_{Bm1}$  is the value of the **Surface potential at strong inversion** parameter from the **Main** tab and  $S_{T,\phi_B}$  is the **Surface potential at strong inversion temperature coefficient**. The default value is  $-8.5e-4$  V/K.

#### **Velocity saturation temperature exponent, [channel drift\_region]**

The parameter value is a two-element vector, with the first element corresponding to the channel, and the second — to the drift region. The velocity saturation,  $\theta_{sat}$ , is assumed to scale exponentially with temperature,  $\theta_{sat} = \theta_{sat,m1}(T_{m1}/T_s)^{\eta_\theta}$ .  $\theta_{sat,m1}$  is the value of the channel or drift region velocity saturation factor, as specified by the **Velocity saturation factor, [channel drift\_region]** parameter from the **Main** tab.  $\eta_\theta$  is the corresponding element of the **Velocity saturation temperature exponent, [channel drift\_region]** parameter. The default value is [1.04, 1.04].

#### **Ohmic resistance temperature exponent**

The series resistances are assumed to correspond to semiconductor resistances. Therefore, they decrease exponentially with increasing temperature.  $R_i = R_{i,m1}(T_{m1}/T_s)^{\eta_R}$ , where  $i$  is S, D, or G, for the source, drain, or gate series resistance, respectively.  $R_{i,m1}$  is the value of the corresponding series resistance parameter from the **Ohmic Resistance** tab and  $\eta_R$  is the **Ohmic resistance temperature exponent**. The default value is 0.95.

#### **Drift region low-bias resistance temperature exponent for gated portion**

Resistance  $R_D$ , the low-bias resistance of the bulk part of the drift region, scales similarly to the other series resistances. A separate value of the temperature

exponent for this resistance provides an extra degree of freedom. The default value is 0.95.

### **Body diode reverse saturation current temperature exponent**

The reverse saturation current for the body diode is assumed to be proportional to the square of the intrinsic carrier concentration,  $n_i = N_C \exp(-E_G/2k_B T)$ .  $N_C$  is the temperature-dependent effective density of states and  $E_G$  is the temperature-dependent bandgap for the semiconductor material. To avoid introducing another temperature-scaling parameter, the block neglects the temperature dependence of the bandgap and uses the bandgap of silicon at 300K (1.12eV) for all device types. Therefore, the temperature-scaled reverse saturation current is given by

$$I_s = I_{s,m1} \left( \frac{T_s}{T_{m1}} \right)^{\eta_{Is}} \cdot \exp \left( \frac{E_G}{k_B} \cdot \left( \frac{1}{T_{m1}} - \frac{1}{T_s} \right) \right).$$

$I_{s,m1}$  is the value of the **Reverse saturation current** parameter from the **Body Diode** tab,  $k_B$  is Boltzmann's constant (8.617x10-5eV/K), and  $\eta_{Is}$  is the **Body diode reverse saturation current temperature exponent**. The default value is 3, because  $N_C$  for silicon is roughly proportional to  $T^{3/2}$ . You can remedy the effect of neglecting the temperature-dependence of the bandgap by a pragmatic choice of  $\eta_{Is}$ .

## **Ports**

The block has the following ports:

G

Electrical conserving port associated with the transistor gate terminal

D

Electrical conserving port associated with the transistor drain terminal

S

Electrical conserving port associated with the transistor source terminal

## **References**

- [1] Aarts, A., N. D'Halleweyn, and R. Van Langevelde. "A Surface-Potential-Based High-Voltage Compact LDMOS Transistor Model." *IEEE Transactions on Electron Devices*. 52(5):999 - 1007. June 2005.

- [2] Van Langevelde, R., A. J. Scholten, and D. B. M. Klaassen. "Physical Background of MOS Model 11. Level 1101." *Nat.Lab. Unclassified Report 2003/00239*. April 2003.
- [3] Oh, S-Y., D. E. Ward, and R. W. Dutton. "Transient analysis of MOS transistors." *IEEE J. Solid State Circuits*. SC-15, pp. 636-643, 1980.

## **See Also**

N-Channel MOSFET | P-Channel LDMOS FET

## **Topics**

Interactive Generation of LDMOS Characteristics

**Introduced in R2016b**

## N-Channel MOSFET

Model N-Channel MOSFET using either Shichman-Hodges equation or surface-potential-based model



## Library

Semiconductor Devices

## Description

The N-Channel MOSFET block provides two main modeling variants:

- Based on threshold voltage — Uses the Shichman-Hodges equation to represent the device. This modeling approach, based on threshold voltage, has the benefits of simple parameterization and simple current-voltage expressions. However, these models have difficulty in accurately capturing transitions across the threshold voltage and lack some important effects, such as velocity saturation. For details, see “Threshold-Based Model” on page 1-945.
- Based on surface potential — Uses the surface-potential equation to represent the device. This modeling approach provides a greater level of model fidelity than the simple square-law (threshold-voltage-based) models can provide. The trade-off is that there are more parameters that require extraction. For details, see “Surface-Potential-Based Model” on page 1-948.

Together with the thermal port variants (see “Thermal Port” on page 1-954), the block therefore provides you with four choices. To select the desired variant, right-click the block in your model. From the context menu, select **Simscape > Block choices**, and then one of the following options:

- **Threshold-based** — Basic model, which represents the device using the Shichman-Hodges equation (based on threshold voltage) and does not simulate thermal effects. This is the default.
- **Threshold-based with thermal** — Model based on threshold voltage and with exposed thermal port.
- **Surface-potential-based** — Model based on surface potential. This model does not simulate thermal effects.
- **Surface-potential-based with thermal** — Thermal variant of the model based on surface potential.

## Threshold-Based Model

The threshold-based variant of the block uses the Shichman and Hodges equations [1] for an insulated-gate field-effect transistor to represent an N-Channel MOSFET.

The drain-source current,  $I_{DS}$ , depends on the region of operation:

- In the off region ( $V_{GS} < V_{th}$ ), the drain-source current is:

$$I_{DS} = 0$$

- In the linear region ( $0 < V_{DS} < V_{GS} - V_{th}$ ), the drain-source current is:

$$I_{DS} = K \left( (V_{GS} - V_{th})V_{DS} - V_{DS}^2 / 2 \right) (1 + \lambda |V_{DS}|)$$

- In the saturated region ( $0 < V_{GS} - V_{th} < V_{DS}$ ), the drain-source current is:

$$I_{DS} = (K / 2) (V_{GS} - V_{th})^2 (1 + \lambda |V_{DS}|)$$

In the preceding equations:

- $K$  is the transistor gain.
- $V_{DS}$  is the positive drain-source voltage.
- $V_{GS}$  is the gate-source voltage.
- $V_{th}$  is the threshold voltage.
- $\lambda$  is the channel modulation.

## Charge Model for Threshold-Based Variant

The block models junction capacitances either by fixed capacitance values, or by tabulated values as a function of the drain-source voltage. In either case, you can either directly specify the gate-source and gate-drain junction capacitance values, or let the block derive them from the input and reverse transfer capacitance values. Therefore, the **Parameterization** options for charge model on the **Junction Capacitance** tab are:

- **Specify fixed input, reverse transfer and output capacitance** — Provide fixed parameter values from datasheet and let the block convert the input and reverse transfer capacitance values to junction capacitance values, as described below. This is the default method.
- **Specify fixed gate-source, gate-drain and drain-source capacitance** — Provide fixed values for junction capacitance parameters directly.
- **Specify tabulated input, reverse transfer and output capacitance** — Provide tabulated capacitance and drain-source voltage values based on datasheet plots. The block converts the input and reverse transfer capacitance values to junction capacitance values, as described below.
- **Specify tabulated gate-source, gate-drain and drain-source capacitance** — Provide tabulated values for junction capacitances and drain-source voltage.

Use one of the tabulated capacitance options (**Specify tabulated input, reverse transfer and output capacitance** or **Specify tabulated gate-source, gate-drain and drain-source capacitance**) when the datasheet provides a plot of junction capacitances as a function of drain-source voltage. Using tabulated capacitance values gives more accurate dynamic characteristics and avoids the need for interactive tuning of parameters to fit the dynamics.

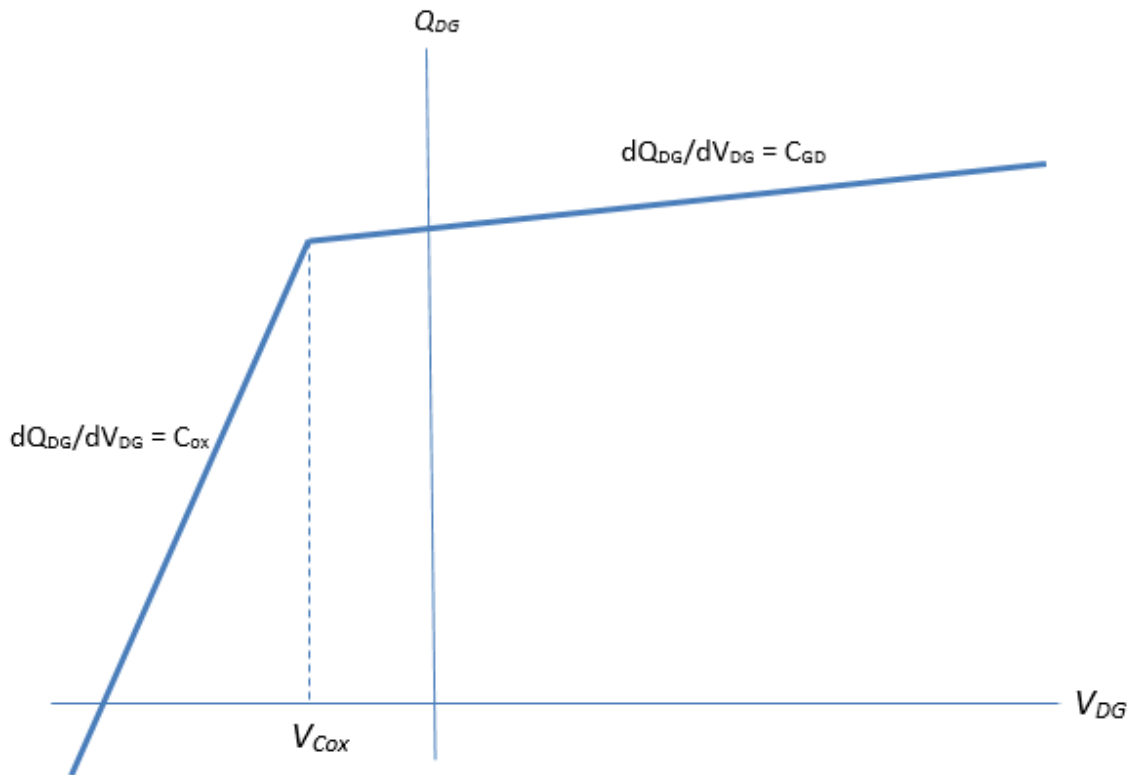
If you use the **Specify fixed gate-source, gate-drain and drain-source capacitance** or **Specify tabulated gate-source, gate-drain and drain-source capacitance** option, the **Junction Capacitance** tab lets you specify the **Gate-drain junction capacitance**, **Gate-source junction capacitance**, and **Drain-source junction capacitance** parameter values (fixed or tabulated) directly. Otherwise, the block derives them from the **Input capacitance**, **Ciss**, **Reverse transfer capacitance**, **Crss**, and **Output capacitance**, **Coss** parameter values. These two parameterization methods are related as follows:

- $C_{GD} = Crss$



- $C_{GS} = C_{iss} - C_{rss}$
- $C_{DS} = C_{oss} - C_{rss}$

The two fixed capacitance options (Specify fixed input, reverse transfer and output capacitance or Specify fixed gate-source, gate-drain and drain-source capacitance) let you model gate junction capacitance as a fixed gate-source capacitance  $C_{GS}$  and either a fixed or a nonlinear gate-drain capacitance  $C_{GD}$ . If you select the Gate-drain charge function is nonlinear option for the **Charge-voltage linearity** parameter, then the gate-drain charge relationship is defined by the piecewise-linear function shown in the following figure.



**Nonlinear gate-drain charge function**

For instructions on how to map a time response to device capacitance values, see the N-Channel IGBT block reference page. However, this mapping is only approximate because the Miller voltage typically varies more from the threshold voltage than in the case for the IGBT.

---

**Note** Because this block implementation includes a charge model, you must model the impedance of the circuit driving the gate to obtain representative turn-on and turn-off dynamics. Therefore, if you are simplifying the gate drive circuit by representing it as a controlled voltage source, you must include a suitable series resistor between the voltage source and the gate.

---

## Surface-Potential-Based Model

The surface-potential-based variant of the block provides a greater level of model fidelity than the simple square-law (threshold-voltage-based) model. The surface-potential-based block variant includes the following effects:

- Fully nonlinear capacitance model (including the nonlinear Miller capacitance)
- Charge conservation inside the model, so you can use the model for charge sensitive simulations
- Velocity saturation and channel-length modulation
- The intrinsic body diode
- Reverse recovery in the body diode model
- Temperature scaling of physical parameters
- For the thermal variant, dynamic self-heating (that is, you can simulate the effect of self-heating on the electrical characteristics of the device)

This model is a minimal version of the world-standard PSP model (see <https://nsti.org/Nanotech2005/WCM2005/WCM2005-GGildenblat.pdf>), including only certain effects from the PSP model in order to strike a balance between model fidelity and complexity. For details of the physical background to the phenomena included in this model, see [2].

The basis of the model is Poisson equation:

$$\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} = \frac{qN_A}{\epsilon_{Si}} \left[ 1 - \exp\left(\frac{-\psi}{\phi_T}\right) + \exp\left(\frac{\psi - 2\phi_B - V_{CB}}{\phi_T}\right) \right]$$

$$\phi_T = \frac{k_B T}{q}$$

where:

- $\psi$  is the electrostatic potential.
- $q$  is the magnitude of the electronic charge.
- $N_A$  is the density of acceptors in the substrate.
- $\epsilon_{Si}$  is the dielectric permittivity of the semiconductor material (for example, silicon).
- $\phi_B$  is the difference between the intrinsic Fermi level and the Fermi level in the bulk silicon.
- $V_{CB}$  is the quasi-Fermi potential of the surface layer referenced to the bulk.
- $\phi_T$  is the thermal voltage.
- $k_B$  is Boltzmann's constant.
- $T$  is temperature.

Poisson equation is used to derive the surface-potential equation:

$$(V_{GB} - V_{FB} - \psi_s)^2 = \gamma^2 \left[ \psi_s + \phi_T \left( \exp\left(\frac{-\psi_s}{\phi_T}\right) - 1 \right) + \phi_T \exp\left(-\frac{2\phi_B + V_{CB}}{\phi_T}\right) \left( \exp\left(\frac{\psi_s}{\phi_T}\right) - 1 \right) \right]$$

where:

- $V_{GB}$  is the applied gate-body voltage.
- $V_{FB}$  is the flatband voltage.
- $\psi_s$  is the surface potential.
- $\gamma$  is the body factor,

$$\gamma = \frac{\sqrt{2q\epsilon_{Si}N_A}}{C_{ox}}$$

- $C_{ox}$  is the capacitance per unit area.

The block uses an explicit approximation to the surface-potential equation, to avoid the need for numerical solution to this implicit equation.

Once the surface potential is known, the drain current  $I_D$  is given by

$$I_D = \frac{W\mu_0}{LG_{\Delta L}G_{mob}\sqrt{1+(\theta_{sat}\Delta\psi)^2}} \left[ -\bar{Q}_{inv}\Delta\psi + \phi_T(Q_{invL} - Q_{inv0}) \right]$$

where:

- $W$  is the device width.
- $L$  is the channel length.
- $\mu_0$  is the low-field mobility.
- $\theta_{sat}$  is the velocity saturation.
- $\Delta\psi$  is the difference in the surface potential between the drain and the source.
- $Q_{inv0}$  and  $Q_{invL}$  are the inversion charge densities at the source and drain, respectively.
- $\bar{Q}_{inv}$  is the average inversion charge density across the channel.
- $G_{mob}$  is the mobility reduction factor. For more information, see the **Surface roughness scattering factor** parameter description in the “Main Tab (Surface-Potential-Based Variant)” on page 1-957 section.
- $G_{\Delta L}$  is the channel-length modulation.

$$G_{\Delta L} = 1 - \frac{\Delta L}{L} = 1 - \alpha \ln \left[ \frac{V_{DB} - V_{DB,eff} + \sqrt{(V_{DB} - V_{DB,eff})^2 + V_p^2}}{V_p} \right]$$

where:

- $\alpha$  is the channel-length modulation factor.
- $V_{DB}$  is the drain-body voltage.
- $V_{DB,eff}$  is the drain-body voltage clipped to a maximum value corresponding to velocity saturation or pinch-off (whichever occurs first).
- $V_p$  is the channel-length modulation voltage.

The block computes the inversion charge densities directly from the surface potential.

The block also computes the nonlinear capacitances from the surface potential. Source and drain charge contributions are assigned via a bias-dependent Ward-Dutton charge-

partitioning scheme, as described in [3]. These charges are computed explicitly, so this model is charge-conserving. The capacitive currents are computed by taking the time derivatives of the relevant charges. In practice, the charges within the simulation are normalized to the oxide capacitance and computed in units of volts.

The MOSFET gain,  $\beta$ , is given by

$$\beta = \frac{W\mu_0 C_{ox}}{L}$$

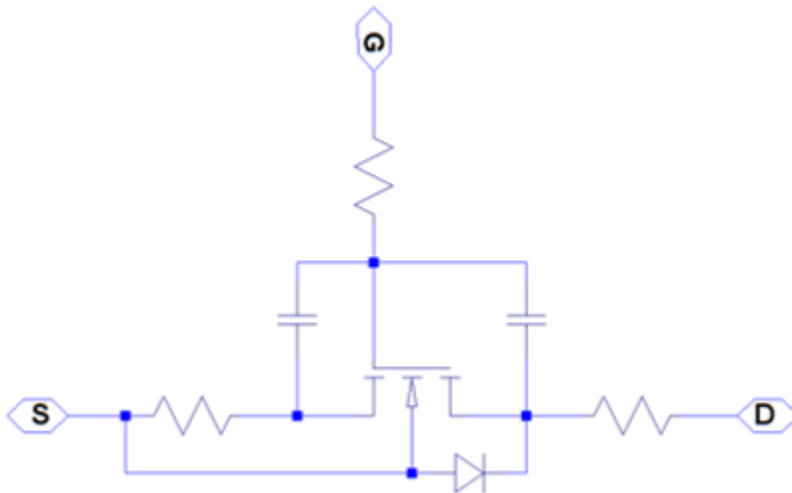
The threshold voltage for a short-circuited source-bulk connection is approximately given by

$$V_T = V_{FB} + 2\phi_B + 2\phi_T + \gamma\sqrt{2\phi_B + 2\phi_T}$$

where:

- $2\phi_B$  is the surface potential at strong inversion.

The overall model consists of an intrinsic MOSFET defined by the surface-potential formulation, a body diode, series resistances, and fixed overlap capacitances, as shown in the schematic.



## Modeling Body Diode

The block models the body diode as an ideal, exponential diode with both junction and diffusion capacitances:

$$I_{dio} = I_s \left[ \exp\left(-\frac{V_{DB}}{n\phi_T}\right) - 1 \right]$$

$$C_j = \frac{C_{j0}}{\sqrt{1 + \frac{V_{DB}}{V_{bi}}}}$$

$$C_{diff} = \frac{\tau I_s}{n\phi_T} \exp\left(-\frac{V_{DB}}{n\phi_T}\right)$$

where:

- $I_{dio}$  is the current through the diode.
- $I_s$  is the reverse saturation current.
- $V_{DB}$  is the drain-body voltage.
- $n$  is the ideality factor.
- $\phi_T$  is the thermal voltage.
- $C_j$  is the junction capacitance of the diode.
- $C_{j0}$  is the zero-bias junction capacitance.
- $V_{bi}$  is the built-in voltage.
- $C_{diff}$  is the diffusion capacitance of the diode.
- $\tau$  is the transit time.

The capacitances are defined through an explicit calculation of charges, which are then differentiated to give the capacitive expressions above. The block computes the capacitive diode currents as time derivatives of the relevant charges, similar to the computation in the surface-potential-based MOSFET model.

## Modeling Temperature Dependence

The default behavior is that dependence on temperature is not modeled, and the device is simulated at the temperature for which you provide block parameters. To model the dependence on temperature during simulation, select **Model temperature dependence** for the **Parameterization** parameter on the **Temperature Dependence** tab.

### Threshold-Based Model

For threshold-based variant, you can include modeling the dependence of the transistor static behavior on temperature during simulation. Temperature dependence of the junction capacitances is not modeled, this being a much smaller effect.

When including temperature dependence, the transistor defining equations remain the same. The gain,  $K$ , and the threshold voltage,  $V_{th}$ , become a function of temperature according to the following equations:

$$K_{T_s} = K_{T_{m1}} \left( \frac{T_s}{T_{m1}} \right)^{BEX}$$

$$V_{th_s} = V_{th1} + \alpha (T_s - T_{m1})$$

where:

- $T_{m1}$  is the temperature at which the transistor parameters are specified, as defined by the **Measurement temperature** parameter value.
- $T_s$  is the simulation temperature.
- $K_{T_{m1}}$  is the transistor gain at the measurement temperature.
- $K_{T_s}$  is the transistor gain at the simulation temperature. This is the transistor gain value used in the MOSFET equations when temperature dependence is modeled.
- $V_{th1}$  is the threshold voltage at the measurement temperature.
- $V_{th_s}$  is the threshold voltage at the simulation temperature. This is the threshold voltage value used in the MOSFET equations when temperature dependence is modeled.
- $BEX$  is the mobility temperature exponent. A typical value of  $BEX$  is -1.5.
- $\alpha$  is the gate threshold voltage temperature coefficient,  $dV_{th}/dT$ .

For most MOSFETS, you can use the default value of -1.5 for *BEX*. Some datasheets quote the value for  $\alpha$ , but most typically they provide the temperature dependence for drain-source on resistance,  $R_{DS(on)}$ . Depending on the block parameterization method, you have two ways of specifying  $\alpha$ :

- If you parameterize the block from a datasheet, you have to provide  $R_{DS(on)}$  at a second measurement temperature. The block then calculates the value for  $\alpha$  based on this data.
- If you parameterize by specifying equation parameters, you have to provide the value for  $\alpha$  directly.

If you have more data comprising drain current as a function of gate-source voltage for more than one temperature, then you can also use Simulink Design Optimization software to help tune the values for  $\alpha$  and *BEX*.

### Surface-Potential-Based Model

The surface-potential-based model includes temperature effects on the capacitance characteristics, as well as modeling the dependence of the transistor static behavior on temperature during simulation.

The **Measurement temperature** parameter on the **Main** tab specifies temperature  $T_{m1}$  at which the other device parameters have been extracted. The **Temperature Dependence** tab provides the simulation temperature,  $T_s$ , and the temperature-scaling coefficients for the other device parameters. For more information, see “Temperature Dependence Tab (Surface-Potential-Based Variant)” on page 1-965.

### Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and select the appropriate block variant:

- For a model based on threshold voltage and with exposed thermal port, select **Simscape > Block choices > Threshold-based with thermal**.
- For a thermal variant of the model based on surface potential, select **Simscape > Block choices > Surface-potential-based with thermal**.

This action displays the thermal port H on the block icon, and adds the **Thermal Port** tab to the block dialog box.



Use the thermal port to simulate the effects of generated heat and device temperature. For more information on using thermal ports and on the **Thermal Port** tab parameters, see “Simulating Thermal Effects in Semiconductors”.

## Basic Assumptions and Limitations

When modeling temperature dependence for threshold-based block variant, consider the following:

- The block does not account for temperature-dependent effects on the junction capacitances.
- When you specify  $R_{DS(on)}$  at a second measurement temperature, it must be quoted for the same working point (that is, the same drain current and gate-source voltage) as for the other  $R_{DS(on)}$  value. Inconsistent values for  $R_{DS(on)}$  at the higher temperature will result in unphysical values for  $\alpha$  and unrepresentative simulation results. Typically  $R_{DS(on)}$  increases by a factor of about 1.5 for a hundred degree increase in temperature.
- You may need to tune the values of  $BEX$  and threshold voltage,  $V_{th}$ , to replicate the  $I_{DS}$ - $V_{GS}$  relationship (if available) for a given device. Increasing  $V_{th}$  moves the  $I_{DS}$ - $V_{GS}$  plots to the right. The value of  $BEX$  affects whether the  $I_{DS}$ - $V_{GS}$  curves for different temperatures cross each other, or not, for the ranges of  $V_{DS}$  and  $V_{GS}$  considered. Therefore, an inappropriate value can result in the different temperature curves appearing to be reordered. Quoting  $R_{DS(on)}$  values for higher currents, preferably close to the current at which it will operate in your circuit, will reduce sensitivity to the precise value of  $BEX$ .

## Parameters

- “Main Tab (Threshold-Based Variant)” on page 1-956
- “Main Tab (Surface-Potential-Based Variant)” on page 1-957
- “Ohmic Resistance Tab” on page 1-959
- “Junction Capacitance Tab” on page 1-959
- “Channel Capacitances Tab” on page 1-962
- “Body Diode Tab” on page 1-963
- “Temperature Dependence Tab (Threshold-Based Variant)” on page 1-963

- “Temperature Dependence Tab (Surface-Potential-Based Variant)” on page 1-965

## Main Tab (Threshold-Based Variant)

This configuration of the **Main** tab corresponds to the threshold-based block variant, which is the default. If you are using the surface-potential-based variant of the block, see “Main Tab (Surface-Potential-Based Variant)” on page 1-957.

### Parameterization

Select one of the following methods for block parameterization:

- **Specify from a datasheet** — Provide the drain-source on resistance and the corresponding drain current and gate-source voltage. The block calculates the transistor gain for the Shichman and Hodges equations from this information. This is the default method.
- **Specify using equation parameters directly** — Provide the transistor gain.

### Drain-source on resistance, $R_{DS(on)}$

The ratio of the drain-source voltage to the drain current for specified values of drain current and gate-source voltage.  $R_{DS(on)}$  should have a positive value. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is  $0.025 \Omega$ .

### Drain current, $I_{DS}$ , for $R_{DS(on)}$

The drain current the block uses to calculate the value of the drain-source resistance.  $I_{DS}$  should have a positive value. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is 6 A.

### Gate-source voltage, $V_{GS}$ , for $R_{DS(on)}$

The gate-source voltage the block uses to calculate the value of the drain-source resistance.  $V_{GS}$  should have a positive value. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is 10 V.

### Gain, $K$

Positive constant gain coefficient for the Shichman and Hodges equations. This parameter is only visible when you select **Specify using equation parameters directly** for the **Parameterization** parameter. The default value is  $5 A/V^2$ .

**Gate-source threshold voltage,  $V_{th}$** 

Gate-source threshold voltage  $V_{th}$  in the Shichman and Hodges equations. For an enhancement device,  $V_{th}$  should be positive. For a depletion mode device,  $V_{th}$  should be negative. The default value is 1.7 V.

**Channel modulation,  $L$** 

The channel-length modulation, usually denoted by the mathematical symbol  $\lambda$ . When in the saturated region, it is the rate of change of drain current with drain-source voltage. The effect on drain current is typically small, and the effect is neglected if calculating transistor gain  $K$  from drain-source on-resistance,  $R_{DS(on)}$ . A typical value is 0.02, but the effect can be ignored in most circuit simulations. However, in some circuits a small nonzero value may help numerical convergence. The default value is 0 1/V.

**Measurement temperature**

Temperature  $T_{m1}$  at which **Drain-source on resistance,  $R_{DS(on)}$**  is measured. The default value is 25 °C.

**Main Tab (Surface-Potential-Based Variant)**

This configuration of the **Main** tab corresponds to the surface-potential-based block variant. If you are using the threshold-based variant of the block, based on the Shichman and Hodges equations, see “Main Tab (Threshold-Based Variant)” on page 1-956.

**Gain**

The MOSFET gain,  $\beta$ . This parameter primarily defines the linear region of operation on an  $I_D$ - $V_{DS}$  characteristic. The value must be greater than 0. The default value is 18 A/V<sup>2</sup>.

**Flatband voltage**

The flatband voltage,  $V_{FB}$ , defines the gate bias that must be applied in order to achieve the flatband condition at the surface of the silicon. The default value is -1.1 V. You can also use this parameter to arbitrarily shift the threshold voltage due to material work function differences, and to trapped interface or oxide charges. In practice, however, it is usually recommended to modify the threshold voltage by using the **Body factor** and **Surface potential at strong inversion** parameters first, and only use this parameter for fine-tuning.

**Body factor**

Body factor,  $\gamma$ , in the surface-potential equation. This parameter primarily impacts the threshold voltage. The default value is 3.5 V<sup>1/2</sup>.

**Surface potential at strong inversion**

The  $2\phi_B$  term in the surface-potential equation. This parameter also primarily impacts the threshold voltage. The default value is 1 V.

**Velocity saturation factor**

Velocity saturation,  $\theta_{\text{sat}}$ , in the drain-current equation. Use this parameter in cases where a good fit to linear operation leads to a saturation current that is too high. By increasing this parameter value, you reduce the saturation current. For high-voltage devices, it is often the case that a good fit to linear operation leads to a saturation current that is too low. In such a case, either increase both the gain and the drain ohmic resistance or use an N-Channel LDMOS FET block instead. The default value is 0.4 1/V.

**Channel-length modulation factor**

The factor,  $\alpha$ , multiplying the logarithmic term in the  $G_{\Delta L}$  equation. This parameter describes the onset of channel-length modulation. For device characteristics that exhibit a positive conductance in saturation, increase the parameter value to fit this behavior. The default value is 0, which means that channel-length modulation is off by default.

**Channel-length modulation voltage**

The voltage  $V_p$  in the  $G_{\Delta L}$  equation. This parameter controls the drain-voltage at which channel-length modulation starts to become active. The default value is 50 mV.

**Surface roughness scattering factor**

Indicates the strength of the mobility reduction. The mobility is  $\mu = \mu_0/G_{\text{mob}}$ , where  $\mu_0$  is the low-field mobility without the effect of surface scattering. The mobility

reduction factor,  $G_{\text{mob}}$ , is given by  $G_{\text{mob}} = \sqrt{1 + (\theta_{\text{sr}} V_{\text{eff}})^4}$ , where  $\theta_{\text{sr}}$  is the surface roughness scattering factor and  $V_{\text{eff}}$  is a voltage that is indicative of the effective vertical electric field strength in the channel,  $E_{\text{eff}}$ . For high vertical electric fields, the mobility is roughly proportional to  $E_{\text{eff}}^2$  for electrons. The default parameter value is 0 1/V.

**Linear-to-saturation transition coefficient**

This parameter controls how smoothly the MOSFET transitions from linear into saturation, particularly when velocity saturation is enabled. This parameter can usually be left at its default value, but you can use it to fine-tune the knee of the  $I_D$ - $V_{\text{DS}}$  characteristic. The expected range for this parameter value is between 2 and 8. The default value is 8.

### Measurement temperature

Temperature  $T_{m1}$  at which the block parameters are measured. If the **Device simulation temperature** parameter on the **Temperature Dependence** tab differs from this value, then device parameters will be scaled from their defined values according to the simulation and reference temperatures. For more information, see “Temperature Dependence Tab (Surface-Potential-Based Variant)” on page 1-965. The default value is 25 °C.

## Ohmic Resistance Tab

### Source ohmic resistance

The transistor source resistance, that is, the series resistance associated with the source contact. The value must be greater than or equal to 0. The default value for threshold-based variants is  $1e-4 \Omega$ . The default value for surface-potential-based variants is  $2e-3 \Omega$ .

### Drain ohmic resistance

The transistor drain resistance, that is, the series resistance associated with the drain contact. The value must be greater than or equal to 0. The default value for threshold-based variants is  $0.01 \Omega$ . The default value for surface-potential-based variants is  $0.17 \Omega$ .

### Gate ohmic resistance

The transistor gate resistance, that is, the series resistance associated with the gate contact. This parameter is visible only for the surface-potential-based block variants. The value must be greater than or equal to 0. The default value is  $8.4 \Omega$ .

## Junction Capacitance Tab

This tab is visible only for the threshold-based variant of the block.

### Parameterization

Select one of the following methods for capacitance parameterization:

- Specify fixed input, reverse transfer and output capacitance — Provide fixed parameter values from datasheet and let the block convert the input, output, and reverse transfer capacitance values to junction capacitance values, as described in “Charge Model for Threshold-Based Variant” on page 1-946. This is the default method.

- Specify fixed gate-source, gate-drain and drain-source capacitance — Provide fixed values for junction capacitance parameters directly.
- Specify tabulated input, reverse transfer and output capacitance — Provide tabulated capacitance and drain-source voltage values based on datasheet plots. The block converts the input, output, and reverse transfer capacitance values to junction capacitance values, as described in “Charge Model for Threshold-Based Variant” on page 1-946.
- Specify tabulated gate-source, gate-drain and drain-source capacitance — Provide tabulated values for junction capacitances and drain-source voltage.

### **Input capacitance, Ciss**

The gate-source capacitance with the drain shorted to the source. This parameter is visible only for the following two values for the **Parameterization** parameter:

- If you select Specify fixed input, reverse transfer and output capacitance, the default value is 350 pF.
- If you select Specify tabulated input, reverse transfer and output capacitance, the default value is [720 700 590 470 390 310] pF.

### **Reverse transfer capacitance, Crss**

The drain-gate capacitance with the source connected to ground, also known as the Miller capacitance. This parameter is visible only for the following two values for the **Parameterization** parameter:

- If you select Specify fixed input, reverse transfer and output capacitance, the default value is 80 pF.
- If you select Specify tabulated input, reverse transfer and output capacitance, the default value is [450 400 300 190 95 55] pF.

### **Output capacitance, Coss**

The drain-source capacitance with the gate and source shorted. This parameter is visible only for the following two values for the **Parameterization** parameter:

- If you select Specify fixed input, reverse transfer and output capacitance, the default value is 0 pF.
- If you select Specify tabulated input, reverse transfer and output capacitance, the default value is [900 810 690 420 270 170] pF.

### Gate-source junction capacitance

The value of the capacitance placed between the gate and the source. This parameter is visible only for the following two values for the **Parameterization** parameter:

- If you select Specify fixed gate-source, gate-drain and drain-source capacitance, the default value is 270 pF.
- If you select Specify tabulated gate-source, gate-drain and drain-source capacitance, the default value is [270 300 290 280 295 255] pF.

### Gate-drain junction capacitance

The value of the capacitance placed between the gate and the drain. This parameter is visible only for the following two values for the **Parameterization** parameter:

- If you select Specify fixed gate-source, gate-drain and drain-source capacitance, the default value is 80 pF.
- If you select Specify tabulated gate-source, gate-drain and drain-source capacitance, the default value is [450 400 300 190 95 55] pF.

### Drain-source junction capacitance

The value of the capacitance placed between the drain and the source. This parameter is visible only for the following two values for the **Parameterization** parameter:

- If you select Specify fixed gate-source, gate-drain and drain-source capacitance, the default value is 0 pF.
- If you select Specify tabulated gate-source, gate-drain and drain-source capacitance, the default value is [450 410 390 230 175 115] pF.

### Corresponding drain-source voltages

The drain-source voltages corresponding to the tabulated capacitance values. This parameter is visible only for tabulated capacitance models (Specify tabulated input, reverse transfer and output capacitance or Specify tabulated gate-source, gate-drain and output capacitance). The default value is [0.1 0.3 1 3 10 30] V.

### Gate-source voltage, $V_{gs}$ , for tabulated capacitances

For tabulated capacitance models, this parameter controls the voltage dependence of the **Reverse transfer capacitance,  $C_{rss}$**  or the **Gate-drain junction capacitance** parameter (depending on the selected parameterization option). These capacitances are a function of the drain-gate voltage. The block calculates drain-gate voltages by

subtracting this gate-source voltage value from the values specified for the **Corresponding drain-source voltages** parameter. The default value is 0 V. This parameter is visible only for tabulated capacitance models (Specify tabulated input, reverse transfer and output capacitance or Specify tabulated gate-source, gate-drain and output capacitance).

## Charge-voltage linearity

The two fixed capacitance options (Specify fixed input, reverse transfer and output capacitance or Specify fixed gate-source, gate-drain and drain-source capacitance) let you model gate junction capacitance as a fixed gate-source capacitance  $C_{GS}$  and either a fixed or a nonlinear gate-drain capacitance  $C_{GD}$ . Select whether the gate-drain capacitance is fixed or nonlinear:

- **Gate-drain capacitance is constant** — The capacitance value is constant and defined according to the selected parameterization option, either directly or derived from a datasheet. This is the default method.
- **Gate-drain charge function is nonlinear** — The gate-drain charge relationship is defined according to the piecewise-nonlinear function described in “Charge Model for Threshold-Based Variant” on page 1-946. Two additional parameters appear to let you define the gate-drain charge function.

## Gate-drain oxide capacitance

The gate-drain capacitance when the drain-gate voltage is less than the **Drain-gate voltage at which oxide capacitance becomes active** parameter value. This parameter is visible only when you select Gate-drain charge function is nonlinear for the **Charge-voltage linearity** parameter. The default value is 200 pF.

## Drain-gate voltage at which oxide capacitance becomes active

The drain-gate voltage at which the drain-gate capacitance switches between off-state ( $C_{GD}$ ) and on-state ( $C_{ox}$ ) capacitance values. This parameter is only visible when you select Gate-drain charge function is nonlinear for the **Charge-voltage linearity** parameter. The default value is -0.5 V.

## Channel Capacitances Tab

This tab is visible only for the surface-potential-based variant of the block.

### Oxide capacitance

The parallel plate gate-channel capacitance. The default value is 1500 pF.



**Gate-source overlap capacitance**

The fixed, linear capacitance associated with the overlap of the gate electrode with the source well. The default value is 100 pF.

**Gate-drain overlap capacitance**

The fixed, linear capacitance associated with the overlap of the gate electrode with the drain well. The default value is 14 pF.

**Body Diode Tab****Reverse saturation current**

The current designated by the  $I_s$  symbol in the body-diode equations. The default value for threshold-based variant is 0 A. The default value for surface-potential-based variant is  $5.2e-13$  A.

**Built-in voltage**

The built-in voltage of the diode, designated by the  $V_{bi}$  symbol in the body-diode equations. Built-in voltage has an impact only on the junction capacitance equation. It does not affect the conduction current. The default value is 0.6 V.

**Ideality factor**

The factor designated by the  $n$  symbol in the body-diode equations. The default value is 1.

**Zero-bias junction capacitance**

The capacitance between the drain and bulk contacts at zero-bias due to the body diode alone. It is designated by the  $C_{j0}$  symbol in the body-diode equations. The default value for threshold-based variant is 0 pF. The default value for surface-potential-based variant is 480 pF.

**Transit time**

The time designated by the  $\tau$  symbol in the body-diode equations. The default value is 50 ns.

**Temperature Dependence Tab (Threshold-Based Variant)**

This configuration of the **Temperature Dependence** tab corresponds to the threshold-based block variant, which is the default. If you are using the surface-potential-based variant of the block, see “Temperature Dependence Tab (Surface-Potential-Based Variant)” on page 1-965

### Parameterization

Select one of the following methods for temperature dependence parameterization:

- **None – Simulate at parameter measurement temperature –** Temperature dependence is not modeled. This is the default method.
- **Model temperature dependence –** Model temperature-dependent effects. Provide a value for simulation temperature,  $T_s$ , a value for  $BEX$ , and a value for the measurement temperature  $T_{m1}$  (using the **Measurement temperature** parameter on the **Main** tab). You also have to provide a value for  $\alpha$  using one of two methods, depending on the value of the **Parameterization** parameter on the **Main** tab. If you parameterize the block from a datasheet, you have to provide  $R_{DS(on)}$  at a second measurement temperature, and the block will calculate  $\alpha$  based on that. If you parameterize by specifying equation parameters, you have to provide the value for  $\alpha$  directly.

### Drain-source on resistance, $R_{DS(on)}$ , at second measurement temperature

The ratio of the drain-source voltage to the drain current for specified values of drain current and gate-source voltage at second measurement temperature. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter on the **Main** tab. It must be quoted for the same working point (drain current and gate-source voltage) as the **Drain-source on resistance,  $R_{DS(on)}$**  parameter on the **Main** tab. The default value is  $0.037 \Omega$ .

### Second measurement temperature

Second temperature  $T_{m2}$  at which **Drain-source on resistance,  $R_{DS(on)}$ , at second measurement temperature** is measured. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter on the **Main** tab. The default value is  $125^\circ\text{C}$ .

### Gate threshold voltage temperature coefficient, $dV_{th}/dT$

The rate of change of gate threshold voltage with temperature. This parameter is only visible when you select **Specify using equation parameters directly** for the **Parameterization** parameter on the **Main** tab. The default value is  $-6 \text{ mV/K}$ .

### Mobility temperature exponent, $BEX$

Mobility temperature coefficient value. You can use the default value for most MOSFETs. See the “Basic Assumptions and Limitations” on page 1-955 section for additional considerations. The default value is  $-1.5$ .

### Body diode reverse saturation current temperature exponent

The reverse saturation current for the body diode is assumed to be proportional to the square of the intrinsic carrier concentration,  $n_i = N_C \exp(-E_G/2k_B T)$ .  $N_C$  is the

temperature-dependent effective density of states and  $E_G$  is the temperature-dependent bandgap for the semiconductor material. To avoid introducing another temperature-scaling parameter, the block neglects the temperature dependence of the bandgap and uses the bandgap of silicon at 300K (1.12eV) for all device types. Therefore, the temperature-scaled reverse saturation current is given by

$$I_s = I_{s,m1} \left( \frac{T_s}{T_{m1}} \right)^{\eta_{Is}} \cdot \exp \left( \frac{E_G}{k_B} \cdot \left( \frac{1}{T_{m1}} - \frac{1}{T_s} \right) \right).$$

$I_{s,m1}$  is the value of the **Reverse saturation current** parameter from the **Body Diode** tab,  $k_B$  is Boltzmann's constant (8.617x10-5eV/K), and  $\eta_{Is}$  is the **Body diode reverse saturation current temperature exponent**. The default value is 3, because  $N_C$  for silicon is roughly proportional to  $T^{3/2}$ . You can remedy the effect of neglecting the temperature-dependence of the bandgap by a pragmatic choice of  $\eta_{Is}$ .

### Device simulation temperature

Temperature  $T_s$  at which the device is simulated. The default value is 25 °C.

## Temperature Dependence Tab (Surface-Potential-Based Variant)

This configuration of the **Temperature Dependence** tab corresponds to the surface-potential-based block variant. If you are using the threshold-based variant of the block, see "Temperature Dependence Tab (Threshold-Based Variant)" on page 1-963

### Parameterization

Select one of the following methods for temperature dependence parameterization:

- **None** – Simulate at parameter measurement temperature – Temperature dependence is not modeled. This is the default method.
- **Model temperature dependence** – Model temperature-dependent effects. Provide a value for the device simulation temperature,  $T_s$ , and the temperature-scaling coefficients for other block parameters.

### Gain temperature exponent

The MOSFET gain,  $\beta$ , is assumed to scale exponentially with temperature,  $\beta = \beta_{m1} (T_{m1}/T_s)^{\eta_{\beta}}$ .  $\beta_{m1}$  is the value of the **Gain** parameter from the **Main** tab and  $\eta_{\beta}$  is the **Gain temperature exponent**. The default value is 1.3.

**Flatband voltage temperature coefficient**

The flatband voltage,  $V_{FB}$ , is assumed to scale linearly with temperature,  $V_{FB} = V_{FBm1} + (T_s - T_{m1})S_{TV_{FB}}$ .  $V_{FBm1}$  is the value of the **Flatband voltage** parameter from the **Main** tab and  $S_{TV_{FB}}$  is the **Flatband voltage temperature coefficient**. The default value is  $5e-4$  V/K.

**Surface potential at strong inversion temperature coefficient**

The surface potential at strong inversion,  $2\phi_B$ , is assumed to scale linearly with temperature,  $2\phi_B = 2\phi_{Bm1} + (T_s - T_{m1})S_{T\phi_B}$ .  $2\phi_{Bm1}$  is the value of the **Surface potential at strong inversion** parameter from the **Main** tab and  $S_{T\phi_B}$  is the **Surface potential at strong inversion temperature coefficient**. The default value is  $-8.5e-4$  V/K.

**Velocity saturation temperature exponent**

The velocity saturation,  $\theta_{sat}$ , is assumed to scale exponentially with temperature,  $\theta_{sat} = \theta_{sat,m1}(T_{m1}/T_s)^{\eta_\theta}$ .  $\theta_{sat,m1}$  is the value of the **Velocity saturation factor** parameter from the **Main** tab and  $\eta_\theta$  is the **Velocity saturation temperature exponent**. The default value is 1.04.

**Surface roughness scattering temperature exponent**

This parameter leads to a temperature-dependent reduction in the MOSFET transconductance at high gate voltage. The surface roughness scattering,  $\theta_{sr}$ , is assumed to scale exponentially with temperature,  $\theta_{sr} = \theta_{sr,m1}(T_{m1}/T_s)^{\eta_{sr}}$ .  $\theta_{sr,m1}$  is the value of the **Surface roughness scattering factor** parameter from the **Main** tab and  $\eta_{sr}$  is the **Surface roughness scattering temperature exponent**. The default value is 0.65.

**Resistance temperature exponent**

The series resistances are assumed to correspond to semiconductor resistances. Therefore, they decrease exponentially with increasing temperature.  $R_i = R_{i,m1}(T_{m1}/T_s)^{\eta_{R_i}}$  where  $i$  is S, D, or G, for the source, drain, or gate series resistance, respectively.  $R_{i,m1}$  is the value of the corresponding series resistance parameter from the **Ohmic Resistance** tab and  $\eta_{R_i}$  is the **Resistance temperature exponent**. The default value is 0.95.

**Body diode reverse saturation current temperature exponent**

The reverse saturation current for the body diode is assumed to be proportional to the square of the intrinsic carrier concentration,  $n_i = N_C \exp(-E_G/2k_B T)$ .  $N_C$  is the temperature-dependent effective density of states and  $E_G$  is the temperature-dependent bandgap for the semiconductor material. To avoid introducing another temperature-scaling parameter, the block neglects the temperature dependence of

the bandgap and uses the bandgap of silicon at 300K (1.12eV) for all device types. Therefore, the temperature-scaled reverse saturation current is given by

$$I_s = I_{s,m1} \left( \frac{T_s}{T_{m1}} \right)^{\eta_{Is}} \cdot \exp \left( \frac{E_G}{k_B} \cdot \left( \frac{1}{T_{m1}} - \frac{1}{T_s} \right) \right).$$

$I_{s,m1}$  is the value of the **Reverse saturation current** parameter from the **Body Diode** tab,  $k_B$  is Boltzmann's constant (8.617x10-5eV/K), and  $\eta_{Is}$  is the **Body diode reverse saturation current temperature exponent**. The default value is 3, because  $N_C$  for silicon is roughly proportional to  $T^{3/2}$ . You can remedy the effect of neglecting the temperature-dependence of the bandgap by a pragmatic choice of  $\eta_{Is}$ .

### Device simulation temperature

Temperature  $T_s$  at which the device is simulated. The default value is 25 °C.

## Ports

The block has the following ports:

G

Electrical conserving port associated with the transistor gate terminal

D

Electrical conserving port associated with the transistor drain terminal

S

Electrical conserving port associated with the transistor source terminal

## References

- [1] Shichman, H. and D. A. Hodges. "Modeling and simulation of insulated-gate field-effect transistor switching circuits." *IEEE J. Solid State Circuits*. SC-3, 1968.
- [2] Van Langevelde, R., A. J. Scholten, and D. B. M. Klaassen. "Physical Background of MOS Model 11. Level 1101." *Nat.Lab. Unclassified Report 2003/00239*. April 2003.
- [3] Oh, S-Y., D. E. Ward, and R. W. Dutton. "Transient analysis of MOS transistors." *IEEE J. Solid State Circuits*. SC-15, pp. 636-643, 1980.

## **See Also**

P-Channel MOSFET

## **Topics**

Interactive Generation of MOSFET Characteristics

# Negative Supply Rail

Model ideal negative supply rail



## Library

Sources

## Description

The Negative Supply Rail block represents an ideal negative supply rail. Use this block instead of the Simscape DC Voltage Source block to define the output voltage relative to the Simscape Electrical Reference block that must appear in each model.

---

**Note** Do not attach more than one Negative Supply Rail block to any connected line.

---

## Parameters

### Constant voltage

The voltage at the output port relative to the Electrical Reference block ground port.  
The default value is -1 V.

## Ports

The block has the following ports:

-

Negative electrical voltage

## **See Also**

DC Voltage Source | Positive Supply Rail



# Neutral Port

Connect phases of three-phase system to electrical conserving port



## Library

Simscape / Electrical / Power Systems / Connections

## Description

The Neutral Port block connects the phases of a three-phase system to an electrical conserving port. You can connect the electrical port to electrical components from the Simscape and Simscape Electrical Electronics and Mechatronics libraries.

---

**Note** If you do not need to connect the neutral port to other blocks, use a Floating Neutral block instead. If you want to ground the neutral port, use a Grounded Neutral block.

---

## Parameters

### Parasitic ground conductance

Parasitic conductance to ground. A nonzero value is required for the simulation of some circuit topologies.

The default value is  $1e-12$  1/Ohm.

## Ports

The block has the following ports:

~

Expandable three-phase port

n

Electrical conserving port associated with the neutral point

## See Also

Floating Neutral | Neutral Port

## Topics

“Expand and Collapse Three-Phase Ports on a Block”

**Introduced in R2013b**

# Nonlinear Inductor

Model inductor with nonideal core



## Library

Simscape / Electrical / Electronics and Mechatronics / Passive Devices

Simscape / Electrical / Power Systems / Passive Devices / Fundamental Components

## Description

The Nonlinear Inductor block represents an inductor with a nonideal core. A core may be nonideal due to its magnetic properties and dimensions. The block provides the following parameterization options:

- Single inductance (linear) on page 1-973
- Single saturation point on page 1-974
- Magnetic flux versus current characteristic on page 1-975
- Magnetic flux density versus magnetic field strength characteristic on page 1-976
- Magnetic flux density versus magnetic field strength characteristic with hysteresis on page 1-977

### Single Inductance (Linear)

The relationships between voltage, current and flux are defined by the following equations:

$$i = i_L + vG_p$$

$$v = N_w \frac{d\Phi}{dt}$$

$$\Phi = \frac{L}{N_w} i_L$$

where:

- $v$  is the terminal voltage.
- $i$  is the terminal current.
- $i_L$  is the current through inductor.
- $G_p$  is the parasitic parallel conductance.
- $N_w$  is the number of winding turns.
- $\Phi$  is the magnetic flux.
- $L$  is the unsaturated inductance.

## Single Saturation Point

The relationships between voltage, current and flux are defined by the following equations:

$$i = i_L + vG_p$$

$$v = N_w \frac{d\Phi}{dt}$$

$$\Phi = \frac{L}{N_w} i_L \text{ (for unsaturated)}$$

$$\Phi = \frac{L_{sat}}{N_w} i_L \pm \Phi_{offset} \text{ (for saturated)}$$

where:

- $v$  is the terminal voltage.

- $i$  is the terminal current.
- $i_L$  is the current through inductor.
- $G_p$  is the parasitic parallel conductance.
- $N_w$  is the number of winding turns.
- $\Phi$  is the magnetic flux.
- $\Phi_{\text{offset}}$  is the magnetic flux saturation offset.
- $L$  is the unsaturated inductance.
- $L_{\text{sat}}$  is the saturated inductance.

## Magnetic Flux Versus Current Characteristic

The relationships between voltage, current and flux are defined by the following equations:

$$i = i_L + vG_p$$

$$v = N_w \frac{d\Phi}{dt}$$

$$\Phi = f(i_L)$$

where:

- $v$  is the terminal voltage.
- $i$  is the terminal current.
- $i_L$  is the current through inductor.
- $G_p$  is the parasitic parallel conductance.
- $N_w$  is the number of winding turns.
- $\Phi$  is the magnetic flux.

Magnetic flux is determined by one-dimensional table lookup, based on the vector of current values and the vector of corresponding magnetic flux values that you provide. You can construct these vectors using either negative and positive data, or positive data only. If using positive data only, the vector must start at 0, and the negative data will be automatically calculated by rotation about (0,0).

## Magnetic Flux Density Versus Magnetic Field Strength Characteristic

The relationships between voltage, current and flux are defined by the following equations:

$$i = i_L + vG_p$$

$$v = N_w \frac{d\Phi}{dt}$$

$$\Phi = B \cdot A_e$$

$$B = f(H)$$

$$H = \frac{N_w}{l_e} i_L$$

where:

- $v$  is the terminal voltage.
- $i$  is the terminal current.
- $i_L$  is the current through inductor.
- $G_p$  is the parasitic parallel conductance.
- $N_w$  is the number of winding turns.
- $\Phi$  is the magnetic flux.
- $B$  is the magnetic flux density.
- $H$  is the magnetic field strength.
- $l_e$  is the effective core length.
- $A_e$  is the effective core cross-sectional area.

Magnetic flux density is determined by one-dimensional table lookup, based on the vector of magnetic field strength values and the vector of corresponding magnetic flux density values that you provide. You can construct these vectors using either negative and

positive data, or positive data only. If using positive data only, the vector must start at 0, and the negative data will be automatically calculated by rotation about (0,0).

## Magnetic Flux Density Versus Magnetic Field Strength Characteristic with Hysteresis

The relationships between voltage, current and flux are defined by the following equations:

$$i = i_L + vG_p$$

$$v = N_w \frac{d\Phi}{dt}$$

$$\Phi = B \cdot A_e$$

$$B = \mu_0 (H + M)$$

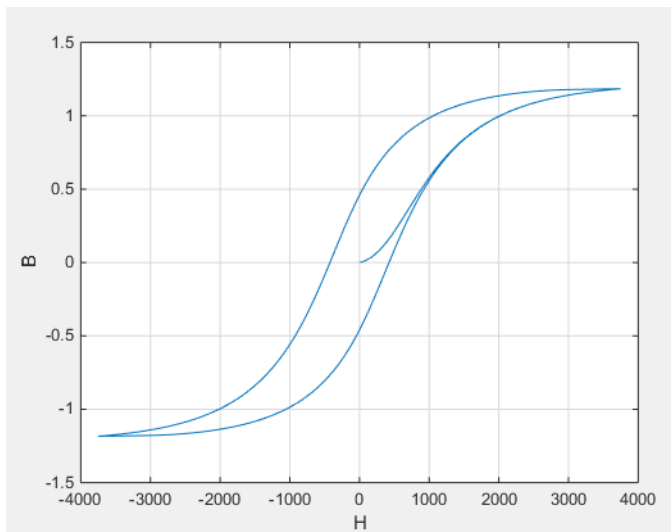
$$H = \frac{N_w}{l_e} i_L$$

where:

- $v$  is the terminal voltage.
- $i$  is the terminal current.
- $i_L$  is the current through inductor.
- $G_p$  is the parasitic parallel conductance.
- $N_w$  is the number of winding turns.
- $\Phi$  is the magnetic flux.
- $B$  is the magnetic flux density.
- $\mu_0$  is the magnetic constant, permeability of free space.
- $H$  is the magnetic field strength.
- $M$  is the magnetization of the inductor core.
- $l_e$  is the effective core length.

- $A_e$  is the effective core cross-sectional area.

The magnetization acts to increase the magnetic flux density, and its value depends on both the current value and the history of the field strength  $H$ . The Jiles-Atherton [1 on page 1-986, 2 on page 1-986] equations are used to determine  $M$  at any given time. The figure below shows a typical plot of the resulting relationship between  $B$  and  $H$ .



In this case the magnetization starts as zero, and hence the plot starts at  $B = H = 0$ . As the field strength increases, the plot tends to the positive-going hysteresis curve; then on reversal the rate of change of  $H$ , it follows the negative-going hysteresis curve. The difference between positive-going and negative-going curves is due to the dependence of  $M$  on the trajectory history. Physically the behavior corresponds to magnetic dipoles in the core aligning as the field strength increases, but not then fully recovering to their original position as field strength decreases.

The starting point for the Jiles-Atherton equation is to split the magnetization effect into two parts, one that is purely a function of effective field strength ( $H_{\text{eff}}$ ) and the other an irreversible part that depends on past history:

$$M = cM_{an} + (1 - c)M_{irr}$$



The  $M_{an}$  term is called the anhysteretic magnetization because it exhibits no hysteresis. It is described by the following function of the current value of the effective field strength,  $H_{eff}$ :

$$M_{an} = M_s \left( \coth \left( \frac{H_{eff}}{\alpha} \right) - \frac{\alpha}{H_{eff}} \right)$$

This function defines a saturation curve with limiting values  $\pm M_s$  and point of saturation determined by the value of  $\alpha$ , the anhysteretic shape factor. It can be approximately thought of as describing the average of the two hysteretic curves. In the Nonlinear

Inductor block, you provide values for  $dM_{an} / dH_{eff}$  when  $H_{eff} = 0$  and a point  $[H_1, B_1]$  on the anhysteretic B-H curve, and these are used to determine values for  $\alpha$  and  $M_s$ .

The parameter  $c$  is the coefficient for reversible magnetization, and dictates how much of the behavior is defined by  $M_{an}$  and how much by the irreversible term  $M_{irr}$ . The Jiles-Atherton model defines the irreversible term by a partial derivative with respect to field strength:

$$\frac{dM_{irr}}{dH} = \frac{M_{an} - M_{irr}}{K\delta - \alpha(M_{an} - M_{irr})}$$

$$\delta = \begin{cases} 1 & \text{if } H \geq 0 \\ -1 & \text{if } H < 0 \end{cases}$$

Comparison of this equation with a standard first order differential equation reveals that as increments in field strength,  $H$ , are made, the irreversible term  $M_{irr}$  attempts to track

the reversible term  $M_{an}$ , but with a variable tracking gain of  $1 / (K\delta - \alpha(M_{an} - M_{irr}))$ . The tracking error acts to create the hysteresis at the points where  $\delta$  changes sign. The main parameter that shapes the irreversible characteristic is  $K$ , which is called the bulk coupling coefficient. The parameter  $\alpha$  is called the inter-domain coupling factor, and is also used to define the effective field strength used when defining the anhysteretic curve:

$$H_{eff} = H + \alpha M$$

The value of  $\alpha$  affects the shape of the hysteresis curve, larger values acting to increase the B-axis intercepts. However, notice that for stability the term  $K\delta - \alpha(M_{an} - M_{irr})$

must be positive for  $\delta > 0$  and negative for  $\delta < 0$ . Therefore not all values of  $\alpha$  are permissible, a typical maximum value being of the order  $1e-3$ .

## Procedure for Finding Approximate Values for Jiles-Atherton Equation Coefficients

You can determine representative parameters for the equation coefficients by using the following procedure:

- 1 Provide a value for the **Anhyseretic B-H gradient when H is zero** parameter ( $dM_{an} / dH_{eff}$  when  $H_{eff} = 0$ ) plus a data point  $[H_1, B_1]$  on the anhyseretic B-H curve. From these values, the block initialization determines values for  $\alpha$  and  $M_s$ .
- 2 Set the **Coefficient for reversible magnetization, c** parameter to achieve correct initial B-H gradient when starting a simulation from  $[H B] = [0 0]$ . The value of  $c$  is approximately the ratio of this initial gradient to the **Anhyseretic B-H gradient when H is zero**. The value of  $c$  must be greater than 0 and less than 1.
- 3 Set the **Bulk coupling coefficient, K** parameter to the approximate magnitude of  $H$  when  $B = 0$  on the positive-going hysteresis curve.
- 4 Start with  $\alpha$  very small, and gradually increase to tune the value of  $B$  when crossing  $H = 0$  line. A typical value is in the range of  $1e-4$  to  $1e-3$ . Values that are too large will cause the gradient of the B-H curve to tend to infinity, which is nonphysical and generates a run-time assertion error.

Sometimes you need to iterate on these four steps to get a good match against a predefined B-H curve.

## Parameters

- “Main Tab” on page 1-980
- “Initial Conditions Tab” on page 1-984

## Main Tab

### Parameterized by

Select one of the following methods for block parameterization:

- **Single inductance (linear)** — Provide the values for number of turns, unsaturated inductance, and parasitic parallel conductance.
- **Single saturation point** — Provide the values for number of turns, unsaturated and saturated inductances, saturation magnetic flux, and parasitic parallel conductance. This is the default option.
- **Magnetic flux versus current characteristic** — In addition to the number of turns and the parasitic parallel conductance value, provide the current vector and the magnetic flux vector, to populate the magnetic flux versus current lookup table.
- **Magnetic flux density versus magnetic field strength characteristic** — In addition to the number of turns and the parasitic parallel conductance value, provide the values for effective core length and cross-sectional area, as well as the magnetic field strength vector and the magnetic flux density vector, to populate the magnetic flux density versus magnetic field strength lookup table.
- **Magnetic flux density versus magnetic field strength characteristic with hysteresis** — In addition to the number of turns and the effective core length and cross-sectional area, provide the values for the initial anhysteretic B-H curve gradient, the magnetic flux density and field strength at a certain point on the B-H curve, as well as the coefficient for the reversible magnetization, bulk coupling coefficient, and inter-domain coupling factor, to define magnetic flux density as a function of both the current value and the history of the field strength.

### **Number of turns**

The total number of turns of wire wound around the inductor core. The default value is 10.

### **Unsaturated inductance**

The value of inductance used when the inductor is operating in its linear region. This parameter is visible only when you select **Single inductance (linear)** or **Single saturation point** for the **Parameterized by** parameter. The default value is  $2e-4$  H.

### **Saturated inductance**

The value of inductance used when the inductor is operating beyond its saturation point. This parameter is visible only when you select **Single saturation point** for the **Parameterized by** parameter. The default value is  $1e-4$  H.

### **Saturation magnetic flux**

The value of magnetic flux at which the inductor saturates. This parameter is visible only when you select **Single saturation point** for the **Parameterized by** parameter. The default value is 1.3e-5 Wb.

### **Current, i**

The current data used to populate the magnetic flux versus current lookup table. This parameter is visible only when you select **Magnetic flux versus current characteristic** for the **Parameterized by** parameter. The default value is [ 0 0.64 1.28 1.92 2.56 3.20 ] A.

### **Magnetic flux vector, phi**

The magnetic flux data used to populate the magnetic flux versus current lookup table. This parameter is visible only when you select **Magnetic flux versus current characteristic** for the **Parameterized by** parameter. The default value is [0 1.29 2.00 2.27 2.36 2.39 ].\*1e-5 Wb.

### **Magnetic field strength vector, H**

The magnetic field strength data used to populate the magnetic flux density versus magnetic field strength lookup table. This parameter is visible only when you select **Magnetic flux density versus magnetic field strength characteristic** for the **Parameterized by** parameter. The default value is [ 0 200 400 600 800 1000 ] A/m.

### **Magnetic flux density vector, B**

The magnetic flux density data used to populate the magnetic flux density versus magnetic field strength lookup table. This parameter is visible only when you select **Magnetic flux density versus magnetic field strength characteristic** for the **Parameterized by** parameter. The default value is [ 0 0.81 1.25 1.42 1.48 1.49 ] T.

### **Effective length**

The effective core length, that is, the average distance of the magnetic path. This parameter is visible only when you select **Magnetic flux density versus magnetic field strength characteristic** or **Magnetic flux density versus magnetic field strength characteristic with hysteresis** for the **Parameterized by** parameter. The default value is 0.032 m.

### **Effective cross-sectional area**

The effective core cross-sectional area, that is, the average area of the magnetic path. This parameter is visible only when you select **Magnetic flux density versus magnetic field strength characteristic** or **Magnetic flux density**

versus magnetic field strength characteristic with hysteresis for the **Parameterized by** parameter. The default value is  $1.6e-5 \text{ m}^2$ .

#### **Anhyysteretic B-H gradient when H is zero**

The gradient of the anhyysteretic (no hysteresis) B-H curve around zero field strength. Set it to the average gradient of the positive-going and negative-going hysteresis curves. This parameter is visible only when you select **Magnetic flux density versus magnetic field strength characteristic with hysteresis** for the **Parameterized by** parameter. The default value is  $0.005 \text{ m}^2/\text{A}$ .

#### **Flux density point on anhyysteretic B-H curve**

Specify a point on the anhyysteretic curve by providing its flux density value. Picking a point at high field strength where the positive-going and negative-going hysteresis curves align is the most accurate option. This parameter is visible only when you select **Magnetic flux density versus magnetic field strength characteristic with hysteresis** for the **Parameterized by** parameter. The default value is  $1.49 \text{ T}$ .

#### **Corresponding field strength**

The corresponding field strength for the point that you define by the **Flux density point on anhyysteretic B-H curve** parameter. This parameter is visible only when you select **Magnetic flux density versus magnetic field strength characteristic with hysteresis** for the **Parameterized by** parameter. The default value is  $1000 \text{ A/m}$ .

#### **Coefficient for reversible magnetization, c**

The proportion of the magnetization that is reversible. The value should be greater than zero and less than one. This parameter is visible only when you select **Magnetic flux density versus magnetic field strength characteristic with hysteresis** for the **Parameterized by** parameter. The default value is  $0.1$ .

#### **Bulk coupling coefficient, K**

The Jiles-Atherton parameter that primarily controls the field strength magnitude at which the B-H curve crosses the zero flux density line. This parameter is visible only when you select **Magnetic flux density versus magnetic field strength characteristic with hysteresis** for the **Parameterized by** parameter. The default value is  $200 \text{ A/m}$ .

#### **Inter-domain coupling factor, alpha**

The Jiles-Atherton parameter that primarily affects the points at which the B-H curves intersect the zero field strength line. Typical values are in the range of  $1e-4$  to  $1e-3$ . This parameter is visible only when you select **Magnetic flux density versus**

magnetic field strength characteristic with hysteresis for the **Parameterized by** parameter. The default value is  $1e-4$ .

### **Averaging period for power logging**

Averaging period for the hysteresis losses calculation. These losses are proportional to the area enclosed by the B-H trajectory. If the block is excited at a known, fixed frequency, you can set this value to the corresponding excitation period to calculate the hysteresis loss. In this case, the block logs the hysteresis loss once per AC cycle to the variable `power_dissipated`. If you are using a fixed-step solver, this value must be an integer multiple of the simulation step size.

If the block is not excited at a known, fixed frequency, set this parameter to  $0$ . In this case, the block sets `power_dissipated` to zero, and you can calculate the actual hysteresis loss by post-processing the logged variable `power_instantaneous`.

The default value is  $0$  s. This parameter is visible only when you select **Magnetic flux density versus magnetic field strength characteristic with hysteresis** for the **Parameterized by** parameter.

### **Parasitic parallel conductance**

Use this parameter to represent small parasitic effects. A small parallel conductance may be required for the simulation of some circuit topologies. The default value is  $1e-9$   $1/\Omega$ .

### **Interpolation option**

The lookup table interpolation option. This parameter is visible only when you select **Magnetic flux versus current characteristic** or **Magnetic flux density versus magnetic field strength characteristic** for the **Parameterized by** parameter. Select one of the following interpolation methods:

- **Linear** — Select this option to get the best performance.
- **Smooth** — Select this option to produce a continuous curve with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (1D) block reference page.

## **Initial Conditions Tab**

### **Specify initial state by**

Select the appropriate initial state specification option:

- **Current** — Specify the initial state of the inductor by the initial current through the inductor ( $i_L$ ). This is the default option.
- **Magnetic flux** — Specify the initial state of the inductor by the magnetic flux.

This parameter is not visible when you select **Magnetic flux density versus magnetic field strength characteristic with hysteresis** for the **Parameterized by** parameter on the **Main** tab.

### **Initial current**

The initial current value used to calculate the value of magnetic flux at time zero. This is the current passing through the inductor. Component current consists of current passing through the inductor and current passing through the parasitic parallel conductance. This parameter is visible only when you select **Current** for the **Specify initial state by** parameter. The default value is 0 A.

### **Initial magnetic flux**

The value of magnetic flux at time zero. This parameter is visible only when you select **Magnetic flux** for the **Specify initial state by** parameter. The default is 0 Wb.

### **Initial magnetic flux density**

The value of magnetic flux density at time zero. This parameter is visible only when you select **Magnetic flux density versus magnetic field strength characteristic with hysteresis** for the **Parameterized by** parameter on the **Main** tab. The default is 0 T.

### **Initial field strength**

The value of magnetic field strength at time zero. This parameter is visible only when you select **Magnetic flux density versus magnetic field strength characteristic with hysteresis** for the **Parameterized by** parameter on the **Main** tab. The default is 0 A/m.

## **Ports**

The block has the following ports:

+

Positive electrical port

-

Negative electrical port

## Examples

For comparison of nonlinear inductor behavior with different parameterization options, see the Nonlinear Inductor Characteristics example.

The Inductor With Hysteresis example shows how modifying the equation coefficients of the Jiles-Atherton magnetic hysteresis equations affects the resulting B-H curve.

## References

- [1] Jiles, D. C. and D. L. Atherton. "Theory of ferromagnetic hysteresis." *Journal of Magnetism and Magnetic Materials*. Vol. 61, 1986, pp. 48-60.
- [2] Jiles, D. C. and D. L. Atherton. "Ferromagnetic hysteresis." *IEEE Transactions on Magnetics*. Vol. 19, No. 5, 1983, pp. 2183-2184.

## See Also

Inductor



# Nonlinear Transformer

Model transformer with nonideal core



## Library

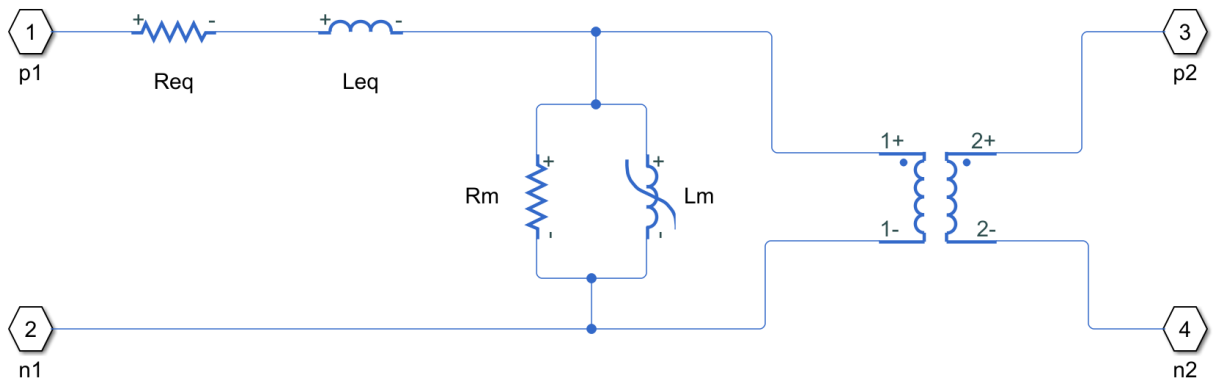
Simscape / Electrical / Electronics and Mechatronics / Passive Devices

Simscape / Electrical / Power Systems / Passive Devices / Fundamental Components

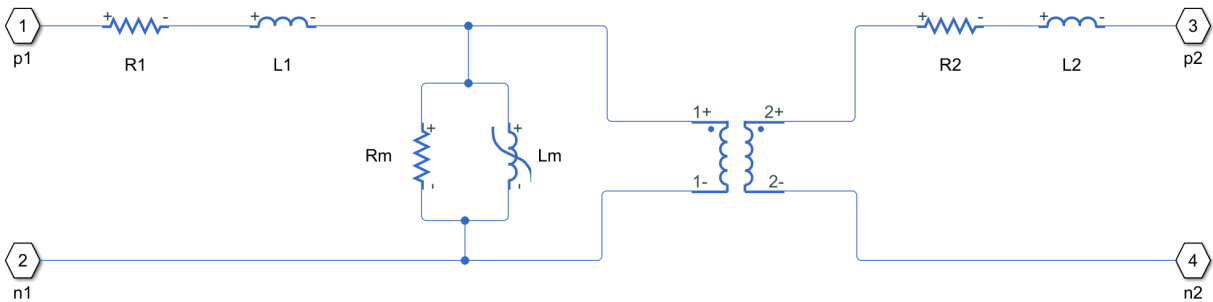
## Description

The Nonlinear Transformer block represents a transformer with a nonideal core. A core may be nonideal due to its magnetic properties and dimensions. The equivalent circuit topology depends upon which of the two winding leakage parameterization options you select:

- Combined primary and secondary values



- Separate primary and secondary values



where:

- $R_{eq}$  is the combined leakage resistance.
- $L_{eq}$  is the combined leakage inductance.
- $R1$  is the primary leakage resistance.
- $L1$  is the primary leakage inductance.
- $R2$  is the secondary leakage resistance.
- $L2$  is the secondary leakage inductance.
- $R_m$  is the magnetization resistance.
- $L_m$  is the magnetization inductance.

The block provides the following parameterization options for the nonlinear magnetization inductance:

- Single inductance (linear)
- Single saturation point
- Magnetic flux versus current characteristic
- Magnetic flux density versus magnetic field strength characteristic
- Magnetic flux density versus magnetic field strength characteristic with hysteresis

For more information, see the Nonlinear Inductor block reference page.

## Parameters

- “Main Tab” on page 1-989
- “Magnetization Tab” on page 1-990
- “Initial Conditions Tab” on page 1-994
- “Parasitics Tab” on page 1-995

### Main Tab

#### Primary number of turns

The number of turns of wire on the primary winding of the transformer. The default value is 100.

#### Secondary number of turns

The number of turns of wire on the secondary winding of the transformer. The default value is 200.

#### Winding parameterized by

Select one of the following methods for the winding leakage parameterization:

- Combined primary and secondary values — Use the lumped resistance and inductance values representing the combined leakage in the primary and secondary windings. This is the default option.

- Separate primary and secondary values — Use separate resistances and inductances to represent leakages in the primary and secondary windings.

### **Combined leakage resistance**

The lumped equivalent resistance  $Req$ , which represents the combined power loss of the primary and secondary windings. This parameter is visible only when you select Combined primary and secondary values for the **Winding parameterized by** parameter. The default value is  $0.01 \Omega$ .

### **Combined leakage inductance**

The lumped equivalent inductance  $Leq$ , which represents the combined magnetic flux loss of the primary and secondary windings. This parameter is visible only when you select Combined primary and secondary values for the **Winding parameterized by** parameter. The default value is  $1e-4$  H.

### **Primary leakage resistance**

The resistance  $R1$ , which represents the power loss of the primary winding. This parameter is visible only when you select Separate primary and secondary values for the **Winding parameterized by** parameter. The default value is  $0.01 \Omega$ .

### **Primary leakage inductance**

The inductance  $L1$ , which represents the magnetic flux loss of the primary winding. This parameter is visible only when you select Separate primary and secondary values for the **Winding parameterized by** parameter. The default value is  $1e-4$  H.

### **Secondary leakage resistance**

The resistance  $R2$ , which represents the power loss of the secondary winding. This parameter is visible only when you select Separate primary and secondary values for the **Winding parameterized by** parameter. The default value is  $0.01 \Omega$ .

### **Secondary leakage inductance**

The inductance  $L2$ , which represents the magnetic flux loss of the secondary winding. This parameter is visible only when you select Separate primary and secondary values for the **Winding parameterized by** parameter. The default value is  $1e-4$  H.

## **Magnetization Tab**

### **Magnetization resistance**

The resistance  $Rm$ , which represents the magnetic losses in the transformer core. The default value is  $100 \Omega$ .

### Magnetization inductance parameterized by

Select one of the following methods for the nonlinear magnetization inductance parameterization:

- **Single inductance (linear)** — Provide the unsaturated inductance value.
- **Single saturation point** — Provide the values for the unsaturated and saturated inductances, as well as saturation magnetic flux. This is the default option.
- **Magnetic flux versus current characteristic** — Provide the current vector and the magnetic flux vector, to populate the magnetic flux versus current lookup table.
- **Magnetic flux density versus magnetic field strength characteristic** — Provide the values for effective core length and cross-sectional area, as well as the magnetic field strength vector and the magnetic flux density vector, to populate the magnetic flux density versus magnetic field strength lookup table.
- **Magnetic flux density versus magnetic field strength characteristic with hysteresis** — In addition to the number of turns and the effective core length and cross-sectional area, provide the values for the initial anhysteretic B-H curve gradient, the magnetic flux density and field strength at a certain point on the B-H curve, as well as the coefficient for the reversible magnetization, bulk coupling coefficient, and inter-domain coupling factor, to define magnetic flux density as a function of both the current value and the history of the field strength.

### Unsaturated inductance

The value of inductance used when the magnetization inductance  $L_m$  is operating in its linear region. This parameter is visible only when you select **Single inductance (linear)** or **Single saturation point** for the **Magnetization inductance parameterized by** parameter. The default value is 0.04 H.

### Saturated inductance

The value of inductance used when the magnetization inductance  $L_m$  is operating beyond its saturation point. This parameter is visible only when you select **Single saturation point** for the **Magnetization inductance parameterized by** parameter. The default value is 0.01 H.

### Saturation magnetic flux

The value of magnetic flux at which the magnetization inductance  $L_m$  saturates. This parameter is visible only when you select **Single saturation point** for the

**Magnetization inductance parameterized by** parameter. The default value is  $1.6e-4$  Wb.

**Current, i**

The current data used to populate the magnetic flux versus current lookup table. This parameter is visible only when you select Magnetic flux versus current characteristic for the **Magnetization inductance parameterized by** parameter. The default value is [ 0 0.4 0.8 1.2 1.6 2.0 ] A.

**Magnetic flux vector, phi**

The magnetic flux data used to populate the magnetic flux versus current lookup table. This parameter is visible only when you select Magnetic flux versus current characteristic for the **Magnetization inductance parameterized by** parameter. The default value is [0 0 0.161 0.25 0.284 0.295 0.299 ] \* $1e-3$  Wb.

**Magnetic field strength vector, H**

The magnetic field strength data used to populate the magnetic flux density versus magnetic field strength lookup table. This parameter is visible only when you select Magnetic flux density versus magnetic field strength characteristic for the **Magnetization inductance parameterized by** parameter. The default value is [ 0 200 400 600 800 1000 ] A/m.

**Magnetic flux density vector, B**

The magnetic flux density data used to populate the magnetic flux density versus magnetic field strength lookup table. This parameter is visible only when you select Magnetic flux density versus magnetic field strength characteristic for the **Magnetization inductance parameterized by** parameter. The default value is [ 0 0.81 1.25 1.42 1.48 1.49 ] T.

**Effective length**

The effective core length, that is, the average distance of the magnetic path around the transformer core. This parameter is visible only when you select Magnetic flux density versus magnetic field strength characteristic for the **Magnetization inductance parameterized by** parameter. The default value is 0.2 m.

**Effective cross-sectional area**

The effective core cross-sectional area, that is, the average area of the magnetic path around the transformer core. This parameter is visible only when you select Magnetic flux density versus magnetic field strength characteristic for the **Magnetization inductance parameterized by** parameter. The default value is  $2e-4$  m<sup>2</sup>.

**Anhysteretic B-H gradient when H is zero**

The gradient of the anhysteretic (no hysteresis) B-H curve around zero field strength. Set it to the average gradient of the positive-going and negative-going hysteresis curves. This parameter is visible only when you select **Magnetic flux density versus magnetic field strength characteristic with hysteresis** for the **Magnetization inductance parameterized by** parameter. The default value is  $0.005 \text{ m}^2/\text{T}\cdot\text{A}$ .

**Flux density point on anhysteretic B-H curve**

Specify a point on the anhysteretic curve by providing its flux density value. Picking a point at high field strength where the positive-going and negative-going hysteresis curves align is the most accurate option. This parameter is visible only when you select **Magnetic flux density versus magnetic field strength characteristic with hysteresis** for the **Magnetization inductance parameterized by** parameter. The default value is 1.49 T.

**Corresponding field strength**

The corresponding field strength for the point that you define by the **Flux density point on anhysteretic B-H curve** parameter. This parameter is visible only when you select **Magnetic flux density versus magnetic field strength characteristic with hysteresis** for the **Magnetization inductance parameterized by** parameter. The default value is 1000 A/m.

**Coefficient for reversible magnetization, c**

The proportion of the magnetization that is reversible. The value should be greater than zero and less than one. This parameter is visible only when you select **Magnetic flux density versus magnetic field strength characteristic with hysteresis** for the **Magnetization inductance parameterized by** parameter. The default value is 0.1.

**Bulk coupling coefficient, K**

The Jiles-Atherton parameter that primarily controls the field strength magnitude at which the B-H curve crosses the zero flux density line. This parameter is visible only when you select **Magnetic flux density versus magnetic field strength characteristic with hysteresis** for the **Magnetization inductance parameterized by** parameter. The default value is 200 A/m.

**Inter-domain coupling factor, alpha**

The Jiles-Atherton parameter that primarily affects the points at which the B-H curves intersect the zero field strength line. Typical values are in the range of  $1\text{e-}4$  to  $1\text{e-}3$ . This parameter is visible only when you select **Magnetic flux density versus magnetic field strength characteristic with hysteresis** for the

**Magnetization inductance parameterized by** parameter. The default value is  $1e-4$ .

### **Interpolation option**

The lookup table interpolation option. This parameter is visible only when you select Magnetic flux versus current characteristic or Magnetic flux density versus magnetic field strength characteristic for the **Magnetization inductance parameterized by** parameter. Select one of the following interpolation methods:

- **Linear** — Select this option to get the best performance.
- **Smooth** — Select this option to produce a continuous curve with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (1D) block reference page.

## **Initial Conditions Tab**

### **Combined leakage inductance initial current**

The value of current through the combined leakage inductance  $Leq$  at time zero. This parameter is visible only when you select Combined primary and secondary values for the **Winding parameterized by** parameter on the **Main** tab. The default value is 0 A.

### **Primary leakage inductance initial current**

The value of current through the primary leakage inductance  $L1$  at time zero. This parameter is visible only when you select Separate primary and secondary values for the **Winding parameterized by** parameter on the **Main** tab. The default value is 0 A.

### **Secondary leakage inductance initial current**

The value of current through the secondary leakage inductance  $L2$  at time zero. This parameter is visible only when you select Separate primary and secondary values for the **Winding parameterized by** parameter on the **Main** tab. The default value is 0 A.

### **Specify magnetization inductance initial state by**

Select the appropriate initial state specification option:

- **Current** — Specify the initial state of the magnetization inductance  $Lm$  by the initial current. This is the default option.



- **Magnetic flux** — Specify the initial state of the magnetization inductance  $L_m$  by the magnetic flux.

This parameter is not visible when you select **Magnetic flux density versus magnetic field strength characteristic with hysteresis** for the **Magnetization inductance parameterized by** parameter on the **Magnetization** tab.

#### **Magnetization inductance initial current**

The initial current value used to calculate the value of magnetic flux within the magnetization inductance  $L_m$  at time zero. This is the current passing through the magnetization inductance  $L_m$ . Total magnetization current consists of current passing through the magnetization resistance  $R_m$  and current passing through the magnetization inductance  $L_m$ . This parameter is visible only when you select **Current** for the **Specify magnetization inductance initial state by** parameter. The default value is 0 A.

#### **Magnetization inductance initial magnetic flux**

The value of the magnetic flux in the magnetization inductance  $L_m$  at time zero. This parameter is visible only when you select **Magnetic flux** for the **Specify magnetization inductance initial state by** parameter. The default is 0 Wb.

#### **Magnetization inductance initial magnetic flux density**

The value of magnetic flux density at time zero. This parameter is visible only when you select **Magnetic flux density versus magnetic field strength characteristic with hysteresis** for the **Magnetization inductance parameterized by** parameter on the **Magnetization** tab. The default is 0 T.

#### **Magnetization inductance initial field strength**

The value of magnetic field strength at time zero. This parameter is visible only when you select **Magnetic flux density versus magnetic field strength characteristic with hysteresis** for the **Magnetization inductance parameterized by** parameter on the **Magnetization** tab. The default is 0 A/m.

## **Parasitics Tab**

#### **Combined leakage inductance parasitic parallel conductance**

Use this parameter to represent small parasitic effects in parallel to the combined leakage inductance  $L_{eq}$ . A small parallel conductance may be required for the simulation of some circuit topologies. This parameter is visible only when you select **Combined primary and secondary values** for the **Winding parameterized by** parameter on the **Main** tab. The default value is  $1e-9$  1/Ω.

### **Primary leakage inductance parasitic parallel conductance**

Use this parameter to represent small parasitic effects in parallel to the primary leakage inductance  $L1$ . A small parallel conductance may be required for the simulation of some circuit topologies. This parameter is visible only when you select **Separate primary and secondary values** for the **Winding parameterized by** parameter on the **Main** tab. The default value is  $1e-9$   $1/\Omega$ .

### **Secondary leakage inductance parasitic parallel conductance**

Use this parameter to represent small parasitic effects in parallel to the secondary leakage inductance  $L2$ . A small parallel conductance may be required for the simulation of some circuit topologies. This parameter is visible only when you select **Separate primary and secondary values** for the **Winding parameterized by** parameter on the **Main** tab. The default value is  $1e-9$   $1/\Omega$ .

## **Ports**

The block has four electrical conserving ports. Polarity is indicated by the + and - signs. Primary and secondary windings are indicated by numbers 1 and 2, respectively.

## **Examples**

For comparison of nonlinear transformer behavior with different parameterization options, see the Nonlinear Transformer Characteristics example.

## **See Also**

Ideal Transformer | Nonlinear Inductor

# NPN Bipolar Transistor

Model NPN bipolar transistor using enhanced Ebers-Moll equations



## Library

Semiconductor Devices

## Description

The NPN Bipolar Transistor block uses a variant of the Ebers-Moll equations to represent an NPN bipolar transistor. The Ebers-Moll equations are based on two exponential diodes plus two current-controlled current sources. The NPN Bipolar Transistor block provides the following enhancements to that model:

- Early voltage effect
- Optional base, collector, and emitter resistances.
- Optional fixed base-emitter and base-collector capacitances.

The collector and base currents are:

$$I_C = IS \left[ \left( e^{qV_{BE}/(kT_{m1})} - e^{qV_{BC}/(kT_{m1})} \right) \left( 1 - \frac{V_{BC}}{V_A} \right) - \frac{1}{\beta_R} \left( e^{qV_{BC}/(kT_{m1})} - 1 \right) \right]$$

$$I_B = IS \left[ \frac{1}{\beta_F} \left( e^{qV_{BE}/(kT_{m1})} - 1 \right) + \frac{1}{\beta_R} \left( e^{qV_{BC}/(kT_{m1})} - 1 \right) \right]$$

Where:

- $I_B$  and  $I_C$  are base and collector currents, defined as positive into the device.
- $IS$  is the saturation current.
- $V_{BE}$  is the base-emitter voltage and  $V_{BC}$  is the base-collector voltage.
- $\beta_F$  is the ideal maximum forward current gain BF
- $\beta_R$  is the ideal maximum reverse current gain BR
- $V_A$  is the forward Early voltage VAF
- $q$  is the elementary charge on an electron (1.602176e-19 Coulombs).
- $k$  is the Boltzmann constant (1.3806503e-23 J/K).
- $T_{m1}$  is the transistor temperature, as defined by the **Measurement temperature** parameter value.

You can specify the transistor behavior using datasheet parameters that the block uses to calculate the parameters for these equations, or you can specify the equation parameters directly.

If  $qV_{BC} / (kT_{m1}) > 40$  or  $qV_{BE} / (kT_{m1}) > 40$ , the corresponding exponential terms in the equations are replaced with  $(qV_{BC} / (kT_{m1}) - 39)e^{40}$  and  $(qV_{BE} / (kT_{m1}) - 39)e^{40}$ , respectively. This helps prevent numerical issues associated with the steep gradient of the exponential function  $e^x$  at large values of  $x$ . Similarly, if  $qV_{BC} / (kT_{m1}) < -39$  or  $qV_{BE} / (kT_{m1}) < -39$  then the corresponding exponential terms in the equations are replaced with  $(qV_{BC} / (kT_{m1}) + 40)e^{-39}$  and  $(qV_{BE} / (kT_{m1}) + 40)e^{-39}$ , respectively.

Optionally, you can specify parasitic fixed capacitances across the base-emitter and base-collector junctions. You also have the option to specify base, collector, and emitter connection resistances.

## Modeling Temperature Dependence

The default behavior is that dependence on temperature is not modeled, and the device is simulated at the temperature for which you provide block parameters. You can optionally include modeling the dependence of the transistor static behavior on temperature during simulation. Temperature dependence of the junction capacitances is not modeled, this being a much smaller effect.

When including temperature dependence, the transistor defining equations remain the same. The measurement temperature value,  $T_{m1}$ , is replaced with the simulation temperature,  $T_s$ . The saturation current,  $IS$ , and the forward and reverse gains ( $\beta_F$  and  $\beta_R$ ) become a function of temperature according to the following equations:

$$IS_{T_s} = IS_{T_{m1}} \cdot (T_s / T_{m1})^{XTI} \cdot \exp\left(-\frac{EG}{kT_s}(1 - T_s / T_{m1})\right)$$

$$\beta_{F_s} = \beta_{F_{m1}} \left(\frac{T_s}{T_{m1}}\right)^{XTB}$$

$$\beta_{R_s} = \beta_{R_{m1}} \left(\frac{T_s}{T_{m1}}\right)^{XTB}$$

where:

- $T_{m1}$  is the temperature at which the transistor parameters are specified, as defined by the **Measurement temperature** parameter value.
- $T_s$  is the simulation temperature.
- $IS_{T_{m1}}$  is the saturation current at the measurement temperature.
- $IS_{T_s}$  is the saturation current at the simulation temperature. This is the saturation current value used in the bipolar transistor equations when temperature dependence is modeled.
- $\beta_{F_{m1}}$  and  $\beta_{R_{m1}}$  are the forward and reverse gains at the measurement temperature.
- $\beta_{F_s}$  and  $\beta_{R_s}$  are the forward and reverse gains at the simulation temperature. These are the values used in the bipolar transistor equations when temperature dependence is modeled.
- $EG$  is the energy gap for the semiconductor type measured in Joules. The value for silicon is usually taken to be 1.11 eV, where 1 eV is 1.602e-19 Joules.
- $XTI$  is the saturation current temperature exponent.
- $XTB$  is the forward and reverse gain temperature coefficient.
- $k$  is the Boltzmann constant (1.3806503e-23 J/K).

Appropriate values for  $XTI$  and  $EG$  depend on the type of transistor and the semiconductor material used. In practice, the values of  $XTI$ ,  $EG$ , and  $XTB$  need tuning to model the exact behavior of a particular transistor. Some manufacturers quote these tuned values in a SPICE Netlist, and you can read off the appropriate values. Otherwise you can determine values for  $XTI$ ,  $EG$ , and  $XTB$  by using a datasheet-defined data at a higher temperature  $T_{m2}$ . The block provides a datasheet parameterization option for this.

You can also tune the values of  $XTI$ ,  $EG$ , and  $XTB$  yourself, to match lab data for your particular device. You can use Simulink Design Optimization software to help tune the values.

## Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal port H on the block icon, and adds the **Thermal Port** tab to the block dialog box.

Use the thermal port to simulate the effects of generated heat and device temperature. For more information on using thermal ports and on the **Thermal Port** tab parameters, see “Simulating Thermal Effects in Semiconductors”.

## Basic Assumptions and Limitations

The NPN Bipolar Transistor model has the following limitations:

- The block does not account for temperature-dependent effects on the junction capacitances.
- You may need to use nonzero ohmic resistance and junction capacitance values to prevent numerical simulation issues, but the simulation may run faster with these values set to zero.

## Parameters

- “Main Tab” on page 1-1000
- “Ohmic Resistance Tab” on page 1-1002
- “Capacitance Tab” on page 1-1003
- “Temperature Dependence Tab” on page 1-1003

## Main Tab

### Parameterization

Select one of the following methods for block parameterization:

- **Specify from a datasheet** — Provide parameters that the block converts to equations that describe the transistor. The block calculates the forward Early voltage  $VAF$  as  $I_c/h_{oe}$ , where  $I_c$  is the **Collector current at which h-parameters are defined** parameter value, and  $h_{oe}$  is the **Output admittance  $h_{oe}$**  parameter value [1]. The block sets  $BF$  to the small-signal **Forward current transfer ratio  $h_{fe}$**  value. The block calculates the saturation current  $IS$  from the specified **Voltage  $V_{be}$**  value and the corresponding **Current  $I_b$  for voltage  $V_{be}$**  value when  $I_c$  is zero. This is the default method.
- **Specify using equation parameters directly** — Provide equation parameters  $IS$ ,  $BF$ , and  $VAF$ .

### **Forward current transfer ratio $h_{fe}$**

Small-signal current gain. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is 100.

### **Output admittance $h_{oe}$**

Derivative of the collector current with respect to the collector-emitter voltage for a fixed base current. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is  $5e-05$  1/ $\Omega$ .

### **Collector current at which h-parameters are defined**

The h-parameters vary with operating point, and are defined for this value of the collector current. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is 1 mA.

### **Collector-emitter voltage at which h-parameters are defined**

The h-parameters vary with operating point, and are defined for this value of the collector-emitter voltage. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is 5 V.

### **Voltage $V_{be}$**

Base-emitter voltage when the base current is  $I_b$ . The [  $V_{be}$   $I_b$  ] data pair must be quoted for when the transistor is in the normal active region, that is, not in the saturated region. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is 0.55 V.

### **Current $I_b$ for voltage $V_{be}$**

Base current when the base-emitter voltage is  $V_{be}$ . The [  $V_{be}$   $I_b$  ] data pair must be quoted for when the transistor is in the normal active region, that is, not in the saturated region. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is 0.5 mA.

### **Forward current transfer ratio BF**

Ideal maximum forward current gain. This parameter is only visible when you select **Specify using equation parameters directly** for the **Parameterization** parameter. The default value is 100.

### **Saturation current IS**

Transistor saturation current. This parameter is only visible when you select **Specify using equation parameters directly** for the **Parameterization** parameter. The default value is 1e-14 A.

### **Forward Early voltage VAF**

In the standard Ebers-Moll equations, the gradient of the  $I_c$  versus  $V_{ce}$  curve is zero in the normal active region. The additional forward Early voltage term increases this gradient. The intercept on the  $V_{ce}$ -axis is equal to  $-VAF$  when the linear region is extrapolated. This parameter is only visible when you select **Specify using equation parameters directly** for the **Parameterization** parameter. The default value is 200 V.

### **Reverse current transfer ratio BR**

Ideal maximum reverse current gain. This value is often not quoted in manufacturer datasheets, because it is not significant when the transistor is biased to operate in the normal active region. When the value is not known and the transistor is not to be operated on the inverse region, use the default value of 1.

### **Measurement temperature**

Temperature  $T_{m1}$  at which  $V_{be}$  and  $I_b$ , or  $IS$ , are measured. The default value is 25 °C.

## **Ohmic Resistance Tab**

### **Collector resistance RC**

Resistance at the collector. The default value is 0.01 Ω.

### **Emitter resistance RE**

Resistance at the emitter. The default value is 1e-4 Ω.

### **Zero bias base resistance RB**

Resistance at the base at zero bias. The default value is 1 Ω.



## Capacitance Tab

### Base-collector junction capacitance

Parasitic capacitance across the base-collector junction. The default value is 5 pF.

### Base-emitter junction capacitance

Parasitic capacitance across the base-emitter junction. The default value is 5 pF.

### Total forward transit time

Represents the mean time for the minority carriers to cross the base region from the emitter to the collector, and is often denoted by the parameter TF [1]. The default value is 0  $\mu$ s.

### Total reverse transit time

Represents the mean time for the minority carriers to cross the base region from the collector to the emitter, and is often denoted by the parameter TR [1]. The default value is 0  $\mu$ s.

## Temperature Dependence Tab

### Parameterization

Select one of the following methods for temperature dependence parameterization:

- **None** – Simulate at parameter measurement temperature – Temperature dependence is not modeled, or the model is simulated at the measurement temperature  $T_{m1}$  (as specified by the **Measurement temperature** parameter on the **Main** tab). This is the default method.
- **Model temperature dependence** – Provide a value for simulation temperature, to model temperature-dependent effects. You also have to provide a set of additional parameters depending on the block parameterization method. If you parameterize the block from a datasheet, you have to provide values for a second [ *Vbe Ib* ] data pair and  $h_{fe}$  at second measurement temperature. If you parameterize by directly specifying equation parameters, you have to provide the values for *XTI*, *EG*, and *XTB*.

### Forward current transfer ratio, $h_{fe}$ , at second measurement temperature

Small-signal current gain at the second measurement temperature. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter on the **Main** tab. It must be quoted at the same

collector-emitter voltage and collector current as for the **Forward current transfer ratio  $h_{fe}$**  parameter on the **Main** tab. The default value is 125.

#### **Voltage $V_{be}$ at second measurement temperature**

Base-emitter voltage when the base current is  $I_b$  and the temperature is set to the second measurement temperature. The [ $V_{be} I_b$ ] data pair must be quoted for when the transistor is in the normal active region, that is, not in the saturated region. This parameter is only visible when you select **Specify** from a datasheet for the **Parameterization** parameter on the **Main** tab. The default value is 0.45 V.

#### **Current $I_b$ for voltage $V_{be}$ at second measurement temperature**

Base current when the base-emitter voltage is  $V_{be}$  and the temperature is set to the second measurement temperature. The [ $V_{be} I_b$ ] data pair must be quoted for when the transistor is in the normal active region, that is, not in the saturated region. This parameter is only visible when you select **Specify** from a datasheet for the **Parameterization** parameter on the **Main** tab. The default value is 0.5 mA.

#### **Second measurement temperature**

Second temperature  $T_{m2}$  at which  $h_{fe}$ ,  $V_{be}$ , and  $I_b$  are measured. This parameter is only visible when you select **Specify** from a datasheet for the **Parameterization** parameter on the **Main** tab. The default value is 125 °C.

#### **Current gain temperature coefficient, XT<sub>B</sub>**

Current gain temperature coefficient value. This parameter is only visible when you select **Specify using equation parameters directly** for the **Parameterization** parameter on the **Main** tab. The default value is 0.

#### **Energy gap, EG**

Energy gap value. This parameter is only visible when you select **Specify using equation parameters directly** for the **Parameterization** parameter on the **Main** tab. The default value is 1.11 eV.

#### **Saturation current temperature exponent, XT<sub>I</sub>**

Saturation current temperature coefficient value. This parameter is only visible when you select **Specify using equation parameters directly** for the **Parameterization** parameter on the **Main** tab. The default value is 3.

#### **Device simulation temperature**

Temperature  $T_s$  at which the device is simulated. The default value is 25 °C.

## Ports

The block has the following ports:

B

Electrical conserving port associated with the transistor base terminal

C

Electrical conserving port associated with the transistor collector terminal

E

Electrical conserving port associated with the transistor emitter terminal

## Examples

See the NPN Bipolar Transistor Characteristics example.

## References

- [1] G. Massobrio and P. Antognetti. *Semiconductor Device Modeling with SPICE*. 2nd Edition, McGraw-Hill, 1993.
- [2] H. Ahmed and P.J. Spreadbury. *Analogue and digital electronics for engineers*. 2nd Edition, Cambridge University Press, 1984.

## See Also

Diode | PNP Bipolar Transistor

## On-Off Delay

Boolean-signal delay

**Library:** Simscape / Electrical / Power Systems / Control /  
General Control



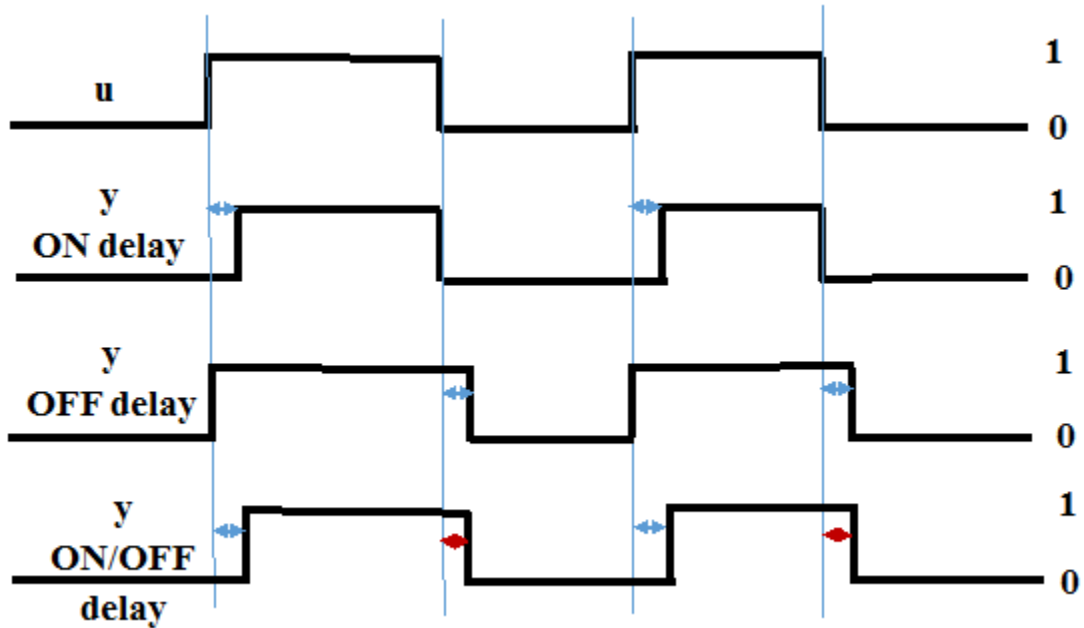
## Description

This block applies a delay on the Boolean input signal.

A time delay is added when a transition is detected in the input signal. This block allows you to add a time delay to the input signal when:

- An ON transition (input change from 0 to 1) is detected,
- An OFF transition (input signal change from 1 to 0) is detected, or
- Either transition is detected.

The operation of the on-off delay is represented in the following figure:



## Ports

### Input

**u** – Input signal

0 | 1

Input Boolean signal.

Data Types: Boolean

### Output

**y** – Output signal

0 | 1

Output signal with delays.

Data Types: Boolean

## Parameters

### **ON delay time (s) — Input ON delay time**

0.01 (default) | 0 | positive scalar

Specify delay time when input is ON.

### **OFF delay time (s) — Input OFF delay time**

0 (default) | positive scalar

Specify delay time when input is OFF.

### **Initial state — Signal initial state**

0 (default) | 1

Specify initial state.

### **Sample time — Block sample time**

0.001 (default) | 0 | positive scalar

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For discrete-time operation, set the sample time to a positive value. For continuous-time operation, set the sample time to 0.

If this block is in a masked subsystem, or other variant subsystem that allows either continuous and discrete operation, promote the sample time parameter. Promoting the sample time parameter ensures correct switching between the continuous and discrete implementations of the block. For more information, see “Promote Parameter to Mask” (Simulink).

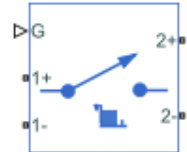
## **See Also**

**Introduced in R2018b**

# One-Quadrant Chopper

One-quadrant chopper

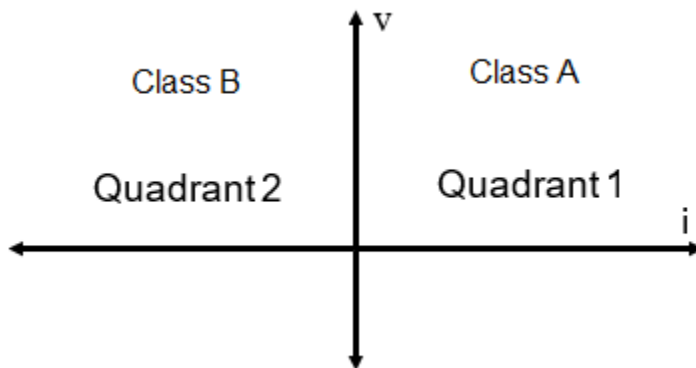
**Library:** Simscape / Electrical / Power Systems / Semiconductors



## Description

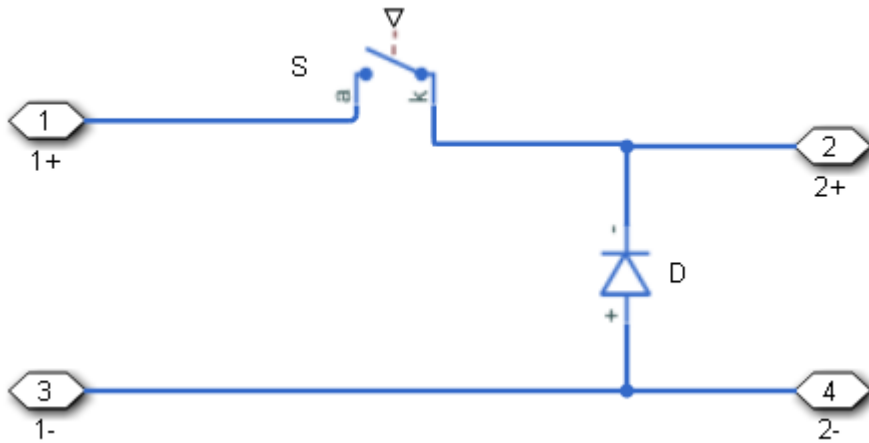
The One-Quadrant Chopper block represents a one-quadrant controlled chopper for converting a fixed DC input to a variable DC output.

The circuit topology and quadrant depend on the class of chopper that you specify.

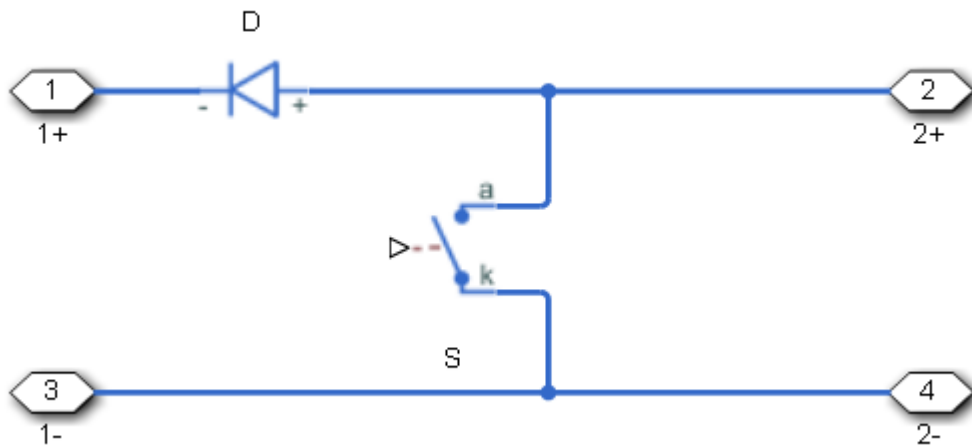


A first-quadrant or class A chopper contains a power switch and a diode.





A second-quadrant or class B chopper also contains a power switch and a diode.



For either topology, the switch  $S$  can be a fully controlled switching device (for example, an IGBT or GTO) or a partially controlled switching device (for example, a thyristor).

Options for the switching device type are:

- GTO — Gate turn-off thyristor. For information on the I-V characteristic of the device, see GTO.
- Ideal semiconductor switch — For information on the I-V characteristic of the device, see Ideal Semiconductor Switch.
- IGBT — Insulated-gate bipolar transistor. For information on the I-V characteristic of the device, see IGBT.
- MOSFET — N-channel metal-oxide-semiconductor field-effect transistor. For information on the I-V characteristic of the device, see MOSFET.
- Thyristor — For information on the I-V characteristic of the device, see Thyristor.

## Model

Each of two model variants for the One-Quadrant Chopper block corresponds to a **Block choice** option. To access the block choices, in the model window, right-click the block, and then use either of these methods:

- From the context menu, select **Simscape > Block choices**.
- On the Simulink Editor menu bar, select **View > Property Inspector**. In the **Property Inspector** window, click the value of the **Block choice**.

The model variants are:

- PS control port — Chopper with a physical signal port. This block choice is the default.
- Electrical control ports — Chopper with one positive and one negative electrical conserving port. To control switching device gates using Simscape Electrical Electronics and Mechatronics blocks, select this option.

## Protection

An inductive load can produce a high reverse-voltage spike when the semiconductor device suddenly switches off the voltage supply to the load. To protect the semiconductor device, an integral protection diode provides a conduction path for reverse current.

To include and configure the internal protection diode block for the S switching device, use the **Diode** parameters. This table shows how to set the **Model dynamics** parameter based on your goals.

Goals		Value to Select	Integral Protection Diode
Do not include protection.		None	None
Include protection.	Prioritize simulation speed.	Protection diode with no dynamics	The Diode block
	Prioritize model fidelity by precisely specifying reverse-mode charge dynamics.	Protection diode with charge dynamics	The Commutation Diode block

You can also include a snubber circuit for each switching device. Snubber circuits contain a series-connected resistor and capacitor. They protect switching devices against high voltages that inductive loads produce when the device turns off the voltage supply to the load. Snubber circuits also prevent excessive rates of current change when a switching device turns on.

To include and configure a snubber circuit for each switching device, use the **Snubbers** parameters.

## Gate Control

You can connect gate-control voltage signals to the gate ports of the switching devices.

- For the PS control port model:
  - 1 Convert a Simulink gate-control voltage signal to a physical signal using a Simulink-PS Converter block.
  - 2 Connect the Simulink-PS Converter block to the **G** port.
- For the electrical control ports model:
  - 1 Connect a Simscape electrical-domain positive DC voltage signal to the **G+** port.
  - 2 Connect the Simscape electrical-domain negative DC voltage signal to the **G-** port.
- For the synchronous converter model:
  - 1 Convert each Simulink gate-control voltage signal to a physical signal using Simulink-PS Converter blocks.

- 2 Multiplex the converted gate-control signals into a single vector using a Two-Pulse Gate Multiplexer.
- 3 Connect the vector signal to the **G** port.

## Ports

### Input

#### **G** — Switching device gate control

physical signal | scalar

Physical signal port associated with the gate terminals of the switching device.

#### **Dependencies**

This port is enabled only for the `PS control port` block choice.

Data Types: `double`

### Conserving

#### **G+** — Switching device gate control positive terminal

electrical | scalar

Positive electrical conserving port associated with the positive gate terminal of the switching device.

#### **Dependencies**

This port is enabled only for the `Electrical control ports` block choice.

Data Types: `double`

#### **G-** — Switching device gate control negative terminal

electrical | scalar

Negative electrical conserving port associated with the negative gate terminal of the switching device.

#### **Dependencies**

This port is enabled only for the `Electrical control ports` block choice.

Data Types: double

**1+ — Positive DC voltage 1**

electrical | scalar

Electrical conserving port associated with the positive terminal of the first DC voltage.

Data Types: double

**1- — Negative DC voltage 1**

electrical | scalar

Electrical conserving port associated with the negative terminal of the first DC voltage.

Data Types: double

**2+ — Positive DC voltage 2**

electrical | scalar

Electrical conserving port associated with the positive terminal of the second DC voltage.

Data Types: double

**2- — Negative DC voltage 2**

electrical | scalar

Electrical conserving port associated with the negative terminal of the second DC voltage.

Data Types: double

## Parameters

### Switching Devices

This table shows how the visibility of **Switching Devices** parameters depends on the **Switching device** that you select. To learn how to read the table, see “Parameter Dependencies” on page A-2.

**Switching Devices Parameter Dependencies**

<b>Switching Device</b>				
<b>Chopper type</b> — Choose Class A - first quadrant or Class B - second quadrant.				
<b>Switching device</b> — Choose Ideal Semiconductor Switch, GTO, IGBT, MOSFET, or Thyristor.				
<b>Ideal Semiconductor Switch</b>	<b>GTO</b>	<b>IGBT</b>	<b>MOSFET</b>	<b>Thyristor</b>
On-state resistance	Forward voltage	Forward voltage	Drain-source on resistance	Forward voltage
Off-state conductance	On-state resistance	On-state resistance	Off-state conductance	On-state resistance
Threshold voltage	Off-state conductance	Off-state conductance	Threshold voltage	Off-state conductance
	Gate trigger voltage, Vgt	Threshold voltage		Gate trigger voltage, Vgt
	Gate turn-off voltage, Vgt_off			Gate turn-off voltage, Vgt_off
	Holding current			Holding current

**Chopper type — Chopper class**

Class A - first quadrant (default) | Class B - second quadrant

Chopper class.

**Switching device — Switch type**

Ideal Semiconductor Switch (default) | GTO | IGBT | MOSFET | Thyristor

Switching device type for the converter.

**Dependencies**

See the Switching Devices Parameter Dependencies table.

**Forward voltage — Voltage**

0.8 Ohm (default) | scalar

For the different switching device types, the **Forward voltage** is taken as:

- GTO — Minimum voltage required across the anode and cathode block ports for the gradient of the device I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**
- IGBT — Minimum voltage required across the collector and emitter block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**
- Thyristor — Minimum voltage required for the device to turn on

#### Dependencies

See the Switching Devices Parameter Dependencies table.

#### On-state resistance — Resistance

0.001 Ohm (default) | scalar

For the different switching device types, the **On-state resistance** is taken as:

- GTO — Rate of change of voltage versus current above the forward voltage
- Ideal semiconductor switch — Anode-cathode resistance when the device is on
- IGBT — Collector-emitter resistance when the device is on
- Thyristor — Anode-cathode resistance when the device is on

#### Dependencies

See the Switching Devices Parameter Dependencies table.

#### Drain-source on resistance — Resistance

0.001 Ohm (default) | scalar

Resistance between the drain and the source, which also depends on the gate-to-source voltage.

#### Dependencies

See the Switching Devices Parameter Dependencies table.

#### Off-state conductance — Conductance

1e-5 1/Ohm (default) | scalar

Conductance when the device is off. The value must be less than  $1/R$ , where  $R$  is the value of **On-state resistance**.

For the different switching device types, the **On-state resistance** is taken as:

- GTO — Anode-cathode conductance
- Ideal semiconductor switch — Anode-cathode conductance
- IGBT — Collector-emitter conductance
- MOSFET — Drain-source conductance
- Thyristor — Anode-cathode conductance

**Dependencies**

See the Switching Devices Parameter Dependencies table.

**Threshold voltage — Voltage threshold**

6 V (default) | scalar

Gate voltage threshold. The device turns on when the gate voltage is above this value. For the different switching device types, the device voltage of interest is:

- Ideal semiconductor switch — Gate-emitter voltage
- IGBT — Gate-cathode voltage
- MOSFET — Gate-source voltage

**Dependencies**

See the Switching Devices Parameter Dependencies table.

**Gate trigger voltage,  $V_{gt}$  — Voltage threshold**

1 V (default) | scalar

Gate-cathode voltage threshold. The device turns on when the gate-cathode voltage is above this value.

**Dependencies**

See the Switching Devices Parameter Dependencies table.

**Gate turn-off voltage,  $V_{gt\_off}$  — Voltage threshold**

-1 V (default) | scalar



Gate-cathode voltage threshold. The device turns off when the gate-cathode voltage is below this value.

**Dependencies**

See the Switching Devices Parameter Dependencies table.

**Holding current – Current threshold**

1 A (default) | scalar

Gate current threshold. The device stays on when the current is above this value, even when the gate-cathode voltage falls below the gate trigger voltage.

**Dependencies**

See the Switching Devices Parameter Dependencies table.

**Diode**

The visibility of **Diode** parameters depends on how you configure the protection diode **Model dynamics** and **Reverse recovery time parameterization** parameters. To learn how to read this table, see “Parameter Dependencies” on page A-2.

**Diode Parameter Dependencies**

<b>Diode</b>			
<b>Model dynamics</b> — Choose Diode with no dynamics or Diode with charge dynamics.			
<b>Diode with no dynamics</b>	<b>Diode with charge dynamics</b>		
Forward voltage	Forward voltage		
On resistance	On resistance		
Off conductance	Off conductance		
	Junction capacitance		
	Peak reverse current, $i_{RM}$		
	Initial forward current when measuring $i_{RM}$		
	Rate of change of current when measuring $i_{RM}$		
	<b>Reverse recovery time parameterization</b> — Choose Specify stretch factor, Specify reverse recovery time directly, or Specify reverse recovery charge.		
	<b>Specify stretch factor</b>	<b>Specify reverse recovery time directly</b>	<b>Specify reverse recovery charge</b>
	Reverse recovery time stretch factor	Reverse recovery time, $t_{rr}$	Reverse recovery charge, $Q_{rr}$

**Model dynamics — Diode model**

Protection diode with no dynamics (default) | Protection diode with charge dynamics

Diode type. The options are:

- **Diode with no dynamics** — Select this option to prioritize simulation speed using the Diode block.
- **Diode with charge dynamics** — Select this option to prioritize model fidelity in terms of reverse mode charge dynamics using the Commutation Diode block.

**Dependencies**

See the Diode Parameter Dependencies table.

**Forward voltage — Voltage**

0.8 V (default) | scalar

Minimum voltage required across the positive and negative block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**.

**On resistance — Resistance**

0.001 Ohm (default) | scalar

Rate of change of voltage versus current above the **Forward voltage**.

**Off conductance — Conductance**

1e-5 1/Ohm (default) | scalar

Conductance of the reverse-biased diode.

**Junction capacitance — Capacitance**

50 nF (default) | scalar

Diode junction capacitance.

**Dependencies**

See the Diode Parameter Dependencies table.

**Peak reverse current,  $i_{RM}$  — Current**

-235 A (default) | scalar less than 0

Peak reverse current measured by an external test circuit.

**Dependencies**

See the Diode Parameter Dependencies table.

**Initial forward current when measuring  $i_{RM}$  — Current**

300 A (default) | scalar greater than 0

Initial forward current when measuring peak reverse current. This value must be greater than zero.

**Dependencies**

See the Diode Parameter Dependencies table.

**Rate of change of current when measuring  $i_{RM}$  — Current change rate**

-50 A/us (default) | scalar

Rate of change of current when measuring peak reverse current.

**Dependencies**

See the Diode Parameter Dependencies table.

**Reverse recovery time parameterization — Recovery-time model**

Specify stretch factor (default) | Specify reverse recovery time directly  
| Specify reverse recovery charge

Model for parameterizing the recovery time. When you select `Specify stretch factor` or `Specify reverse recovery charge`, you can specify a value that the block uses to derive the reverse recovery time. For more information on these options, see “Alternatives to Specifying `trr` Directly” on page 1-339.

**Dependencies**

See the Diode Parameter Dependencies table.

**Reverse recovery time stretch factor — Stretch factor**

3 (default) | scalar greater than 1

Value that the block uses to calculate **Reverse recovery time, `trr`**. Specifying the stretch factor is an easier way to parameterize the reverse recovery time than specifying the reverse recovery charge. The larger the value of the stretch factor, the longer it takes for the reverse recovery current to dissipate.

**Dependencies**

See the Diode Parameter Dependencies table.

**Reverse recovery time, `trr` — Time**

15  $\mu$ s (default) | scalar

Interval between the time when the current initially goes to zero (when the diode turns off) and the time when the current falls to less than 10 percent of the peak reverse current.

The value of the **Reverse recovery time, `trr`** parameter must be greater than the value of the **Peak reverse current, `iRM`** parameter divided by the value of the **Rate of change of current when measuring `iRM`** parameter.

**Dependencies**

See the Diode Parameter Dependencies table.

**Reverse recovery charge, Q<sub>rr</sub> – Charge**

1500 s\*μA (default) | scalar

Value that the block uses to calculate **Reverse recovery time, trr**. Use this parameter if the data sheet for your diode device specifies a value for the reverse recovery charge instead of a value for the reverse recovery time.

The reverse recovery charge is the total charge that continues to dissipate when the

diode turns off. The value must be less than  $-\frac{i_{RM}^2}{2a}$ ,

where:

- $i_{RM}$  is the value specified for **Peak reverse current, i<sub>RM</sub>**.
- $a$  is the value specified for **Rate of change of current when measuring i<sub>RM</sub>**.

**Dependencies**

See the Diode Parameter Dependencies table.

**Snubbers**

The table summarizes the **Snubbers** parameter dependencies. To learn how to read the table, see “Parameter Dependencies” on page A-2.

**Snubbers Parameter Dependencies**

<b>Snubbers</b>	
<b>Snubber</b> – Choose None or RC Snubber.	
<b>None</b>	<b>RC Snubber</b>
	Snubber resistance
	Snubber capacitance

**Snubber – Snubber model**

None (default) | RC snubber

Switching device snubber.

**Dependencies**

See the Snubbers Parameter Dependencies table.

**Snubber resistance — Resistance**

0.1 Ohm (default) | scalar

Resistance of the switching device snubber.

**Dependencies**

See the Snubbers Parameter Dependencies table.

**Snubber capacitance — Capacitance**

1e-7 (default) | F | scalar

Capacitance of the switching device snubber.

**Dependencies**

See the Snubbers Parameter Dependencies table.

**References**

[1] Trzynadlowski, A. M. *Introduction to Modern Power Electronics*. 2nd Ed. Hoboken, NJ: John Wiley & Sons Inc., 2010.

**See Also**

Average-Value Chopper | Four-Quadrant Chopper | Two-Quadrant Chopper

**Introduced in R2018b**

# Open Circuit

Three-phase connection that draws no current



## Library

Simscape / Electrical / Power Systems / Connections

## Description

The Open Circuit block models a three-phase connection that draws no current on any of the three phases. In Simscape, physical network block diagrams do not allow unconnected conserving ports. Therefore, use the Open Circuit block to terminate three-phase electrical ports on other blocks that you want to leave open-circuit.

## Ports

The block has the following ports:

~

Expandable three-phase port

## See Also

### Simscape Blocks

Floating Neutral | Grounded Neutral | Neutral Port

## **Topics**

“Three-Phase Three-Level PWM Generator”

“Three-Phase Bridge Cycloconverter”

“Three-Phase Voltage-Sourced Converter (FLB)”

**Introduced in R2013b**



# Operational Transconductance Amplifier

Model behavioral representation of operational transconductance amplifier



## Library

Integrated Circuits

## Description

The Operational Transconductance Amplifier block provides a behavioral representation of an operational transconductance amplifier. A transconductance amplifier converts an input voltage into an output current. Applications include variable frequency oscillators, variable gain amplifiers and current-controlled filters. These applications exploit the fact that the transconductance gain is a function of current flowing into the control current pin.

To support faster simulation, the behavioral representation does not model the detailed transistor implementation. Therefore, the model is only valid when operating in the linear region, that is, where the device input resistance, output resistance, and transconductance gain all depend linearly on the control current, and are independent of input signal amplitude. The dynamics are approximated by a first-order lag, based on the value you specify for the block parameter **Bandwidth**.

## Control Current

The control current pin C is maintained at the voltage that you specify for the **Minimum output voltage**. In practice, the **Minimum output voltage** equals the negative supply voltage plus the transistor collector-emitter voltage drop. For example, if the **Minimum output voltage** for a supply voltage of +15V is -14.5, then to achieve a control current of 500 $\mu$ A, a resistor connected between the +15V rail and the control current pin must have a value of  $(15 - (-14.5)) / 500e-6 = 59k\Omega$ .

## Transconductance

The relationship between input voltage,  $v$ , and transconductance current,  $i_{gm}$ , is:

$$\begin{aligned}v &= v_+ - v_- \\i_{gm} &= g_m \cdot v \\g_m &= \frac{g_{m0} \cdot i_c}{i_{c0}}\end{aligned}$$

where:

- $v_+$  is the voltage presented at the block + pin.
- $v_-$  is the voltage presented at the block - pin.
- $g_m$  is the transconductance.
- $i_c$  is the control current flowing into the control current pin C.
- $i_{c0}$  is the reference control current, that is, the control current at which transconductance is quoted on the datasheet.
- $g_{m0}$  is the transconductance measured at the reference control current  $i_{c0}$ .

Therefore, increasing control current increases the transconductance.

## Output Resistance and Determining Output Current

The output resistance,  $R_{out}$ , is defined by:

$$\begin{aligned}i_{gm} + i_o &= \frac{v_o}{R_{out}} \\R_{out} &= \frac{R_{out0} \cdot i_{c0}}{i_c}\end{aligned}$$

where:

- $i_{gm}$  is the transconductance current.
- $i_o$  is the output current, defined as positive if flowing into the transconductance amplifier output pin.

- $i_c$  is the control current flowing into the control current pin C.
- $i_{c0}$  is the reference control current, that is, the control current at which output resistance is quoted on the datasheet.
- $R_{out0}$  is the output resistance measured at the reference control current  $i_{c0}$ .

Therefore, increasing control current reduces output resistance.

## Input Resistance

The relationship between input voltage,  $v$ , across the + and - pins and the current flowing,  $i$ , is:

$$\frac{v}{i} = R_{in}$$

$$R_{in} = \frac{R_{in0} \cdot i_{c0}}{i_c}$$

where:

- $i_c$  is the control current flowing into the control current pin C.
- $R_{in}$  is the input resistance for the current control current value,  $i_c$ .
- $i_{c0}$  is the reference control current, that is, the control current at which input resistance is quoted on the datasheet.
- $R_{in0}$  is the input resistance measured at the reference control current  $i_{c0}$ .

Therefore, increasing control current reduces input resistance.

## Limits

Because of the physical construction of an operational transconductance amplifier based on current mirrors, the transconductance current  $i_{gm}$  cannot exceed the control current. Hence the value of  $i_{gm}$  is limited by:

$$-i_c \leq i_{gm} \leq i_c$$

The output voltage is also limited by the supply voltage:

$$V_{min} \leq v_o \leq V_{max}$$

where  $V_{\min}$  is the **Minimum output voltage**, and  $V_{\max}$  is the **Maximum output voltage**. Output voltage limiting is implemented by adding a low resistance to the output when the voltage limit is exceeded. The value of this resistance is set by the **Additional output resistance at voltage swing limits** parameter.

The transconductance current is also slew-rate limited, a value for slew rate limiting typically being given on datasheets:

$$-\mu \leq \frac{di_{gm}}{dt} \leq \mu$$

where  $\mu$  is the **Maximum current slew rate**.

## Parameters

- “Nominal Measurements Tab” on page 1-1030
- “Dynamics Tab” on page 1-1031
- “Limits Tab” on page 1-1031

## Nominal Measurements Tab

### Transconductance

The transconductance,  $g_m$ , when the control current is equal to the **Reference control current**. This is the ratio of the transconductance current,  $i_{gm}$ , to the voltage difference,  $v$ , across the + and - pins. The default value is 9600  $\mu\text{S}$ .

### Input resistance

The input resistance,  $R_{in}$ , when the control current is equal to the **Reference control current**. The input resistance is the ratio of the voltage difference,  $v$ , across the + and - pins to the current flowing from the + to the - pin. The default value is 25 kOhm.

### Output resistance

The output resistance,  $R_{out}$ , when the control current is equal to the **Reference control current**. See above for the equation defining output resistance. The default value is 3 MOhm.

### Reference control current

The control current at which the **Transconductance**, **Input resistance**, and **Output resistance** are quoted. The default value is 500  $\mu\text{A}$ .

## Dynamics Tab

### Dynamics

Select one of the following options:

- **No lag** — Do not model the dynamics of the relationship between output current and input voltage. This is the default.
- **Finite bandwidth with slew rate limiting** — Model the dynamics of the relationship between output current and input voltage using a first-order lag. If you select this option, the **Bandwidth**, **Maximum current slew rate**, and **Initial current** parameters appear on the **Dynamics** tab.

### Bandwidth

The bandwidth of the first-order lag used to model the dynamics of the relationship between output current and input voltage. The default value is 2 MHz.

### Maximum current slew rate

The maximum rate-of-change of transconductance current when there is no feedback around the device. Note that datasheets sometimes quote slew rate as a maximum rate of change of voltage. In this case, the value depends on the particular test circuit. To get an accurate value for **Maximum current slew rate**, reproduce the test circuit in a Simscape Electrical model, and tune the parameter value to match the datasheet value. If the test circuit is open-loop, and the load resistance is quoted, you can obtain an approximate value for the **Maximum current slew rate** by dividing the voltage slew rate by the load resistance. The default value is 2 A/ $\mu\text{s}$ .

### Initial current

The initial transconductance current (note, not the initial output current). This is the transconductance current sinking to both the internal output resistance,  $R_{\text{out}}$ , and the output pin. The default value is 0 A.

## Limits Tab

### Minimum output voltage

The output voltage is limited to be greater than the value of this parameter. The default value is -15 V.

### **Maximum output voltage**

The output voltage is limited to be less than the value of this parameter. The default value is 15 V.

### **Additional output resistance at voltage swing limits**

To limit the output voltage swing, an additional output resistance is applied between output and the power rail when the output voltage exceeds the limit. The value of this resistance should be low compared to the output resistance and circuit load resistance. The default value is 1 Ohm.

### **Minimum control current for simulation**

The control current measured at the control current pin C is limited to be greater than the value of this parameter. This prevents a potential divide-by-zero when calculating input and output resistance values based on the value of the control current. The default value is 0.001  $\mu$ A.

## **Ports**

The block has the following ports:

- +  
Positive electrical voltage
- Negative electrical voltage
- C  
Control current
- OUT  
Output current

## **See Also**

[Band-Limited Op-Amp](#) | [Finite-Gain Op-Amp](#) | [Op-Amp](#)

# Optocoupler

Model optocoupler as LED, current sensor, and controlled current source



## Library

Semiconductor Devices

## Description

This block represents an optocoupler using a model that consists of the following components:

- An exponential light-emitting diode in series with a current sensor on the input side
- A controlled current source on the output side

The output-side current flows from the collector junction to the emitter junction. It has a value of  $CTR \cdot I_d$ , where  $CTR$  is the **Current transfer ratio** parameter value and  $I_d$  is the diode current.

Use the Optocoupler block to interface two electrical circuits without making a direct electrical connection. A common reason for doing this is that the two circuits work at very different voltage levels.

---

**Note** Each electrical circuit must have its own Electrical Reference block.

---

If the output circuit is a phototransistor, typical values for the **Current transfer ratio** parameter are 0.1 to 0.5. If the output stage consists of a Darlington pair, the parameter value can be much higher than this. The **Current transfer ratio** value also varies with the light-emitting diode current, but this effect is not modeled by the Photodiode block.

Some manufacturers provide a maximum data rate for optocouplers. In practice, the maximum data rate depends on the following factors:

- The capacitance of the photodiode and the type of the driving circuit
- The construction of the phototransistor and its associated capacitance

The Optocoupler block only lets you define the capacitance on the light-emitting diode. You can use the **Junction capacitance** parameter to add your own capacitance across the collector and emitter connections.

The Optocoupler block lets you model temperature dependence of the underlying diode. For details, see the Diode reference page.

### Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal port H on the block icon, and adds the **Thermal Port** tab to the block dialog box.

Use the thermal port to simulate the effects of generated heat and device temperature. For more information on using thermal ports and on the **Thermal Port** tab parameters, see “Simulating Thermal Effects in Semiconductors”.

### Variables

Use the **Variables** section of the block interface to set the priority and initial target values for the block variables prior to simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

## Basic Assumptions and Limitations

The Optocoupler block has the following limitations:

- The output side is modeled as a controlled current source. As such, it only correctly approximates a bipolar transistor operating in its normal active region. To create a more detailed model, connect the Optocoupler output directly to the base of an NPN Bipolar Transistor block, and set the parameters to maintain a correct overall value for



the current transfer ratio. If you need to connect optocouplers in series, use this approach to avoid the invalid topology of two current sources in series.

- The temperature dependence of the forward current transfer ratio is not modeled. Typically the temperature dependence of this parameter is much less than that of the optical diode I-V characteristic.
- You may need to use nonzero ohmic resistance and junction capacitance values to prevent numerical simulation issues, but the simulation may run faster with these values set to zero.

## Parameters

- “Main” on page 1-1035
- “Ohmic Resistance” on page 1-1036
- “Junction Capacitance” on page 1-1036
- “Temperature Dependence” on page 1-1037

## Main

### Current transfer ratio

The output current flowing from the transistor collector to emitter junctions is equal to the product of the current transfer ratio and the current flowing the light-emitting diode. The default value is 0.2.

### Diode parameterization

Select one of the following methods for model parameterization:

- Use I-V curve data points — Specify measured data at two points on the diode I-V curve. This is the default method.
- Use parameters  $IS$  and  $N$  — Specify saturation current and emission coefficient.

### Currents [I1 I2]

A vector of the current values at the two points on the diode I-V curve that the block uses to calculate  $IS$  and  $N$ . This parameter is only visible when you select Use I-V curve data points for the **Diode parameterization** parameter. The default value is [ 0.001 0.015 ] A.

### **Voltages [V1 V2]**

A vector of the voltage values at the two points on the diode I-V curve that the block uses to calculate  $I_S$  and  $N$ . This parameter is only visible when you select **Use I-V curve data points** for the **Diode parameterization** parameter. The default value is [ 0.9 1.05 ] V.

### **Saturation current $I_S$**

The magnitude of the current that the ideal diode equation approaches asymptotically for very large reverse bias levels. This parameter is only visible when you select **Use parameters  $I_S$  and  $N$**  for the **Diode parameterization** parameter. The default value is  $1e-10$  A.

### **Measurement temperature**

The temperature at which  $I_S$  or the I-V curve was measured. The default value is 25 °C.

### **Emission coefficient $N$**

The diode emission coefficient or ideality factor. This parameter is only visible when you select **Use parameters  $I_S$  and  $N$**  for the **Diode parameterization** parameter. The default value is 2.

## **Ohmic Resistance**

### **Ohmic resistance $R_S$**

The series diode connection resistance. The default value is 0.1  $\Omega$ .

## **Junction Capacitance**

### **Junction capacitance**

Select one of the following options for modeling the diode junction capacitance:

- **Fixed or zero junction capacitance** — Model the junction capacitance as a fixed value.
- **Use C-V curve data points** — Specify measured data at three points on the diode C-V curve.
- **Use parameters  $C_{J0}$ ,  $V_J$ ,  $M$  &  $FC$**  — Specify zero-bias junction capacitance, junction potential, grading coefficient, and forward-bias depletion capacitance coefficient.

**Zero-bias junction capacitance  $C_{J0}$** 

The value of the capacitance placed in parallel with the exponential diode term. This parameter is only visible when you select **Fixed** or **zero junction capacitance** or **Use parameters  $C_{J0}$ ,  $V_J$ ,  $M$  &  $FC$**  for the **Junction capacitance** parameter. The default value is 5 pF.

**Junction potential  $V_J$** 

The junction potential. This parameter is only visible when you select **Use parameters  $C_{J0}$ ,  $V_J$ ,  $M$  &  $FC$**  for the **Junction capacitance** parameter. The default value is 1 V.

**Grading coefficient  $M$** 

The coefficient that quantifies the grading of the junction. This parameter is only visible when you select **Use parameters  $C_{J0}$ ,  $V_J$ ,  $M$  &  $FC$**  for the **Junction capacitance** parameter. The default value is 0.5.

**Reverse bias voltages [ $VR1$   $VR2$   $VR3$ ]**

A vector of the reverse bias voltage values at the three points on the diode C-V curve that the block uses to calculate  $C_{J0}$ ,  $V_J$ , and  $M$ . This parameter is only visible when you select **Use C-V curve data points** for the **Junction capacitance** parameter. The default value is [ 0.1 10 100 ] V.

**Corresponding capacitances [ $C1$   $C2$   $C3$ ]**

A vector of the capacitance values at the three points on the diode C-V curve that the block uses to calculate  $C_{J0}$ ,  $V_J$ , and  $M$ . This parameter is only visible when you select **Use C-V curve data points** for the **Junction capacitance** parameter. The default value is [ 3.5 1 0.4 ] pF.

**Capacitance coefficient  $FC$** 

Fitting coefficient that quantifies the decrease of the depletion capacitance with applied voltage. This parameter is only visible when you select **Use C-V curve data points** or **Use parameters  $C_{J0}$ ,  $V_J$ ,  $M$  &  $FC$**  for the **Junction capacitance** parameter. The default value is 0.5.

**Temperature Dependence****Parameterization**

Select one of the following methods for temperature dependence parameterization:

- **None – Simulate at parameter measurement temperature –**  
Temperature dependence is not modeled, or the model is simulated at the

measurement temperature  $T_{m1}$  (as specified by the **Measurement temperature** parameter on the **Main** tab). This is the default method.

- Use an I-V data point at second measurement temperature T2 — If you select this option, you specify a second measurement temperature  $T_{m2}$ , and the current and voltage values at this temperature. The model uses these values, along with the parameter values at the first measurement temperature  $T_{m1}$ , to calculate the energy gap value.
- Specify saturation current at second measurement temperature T2 — If you select this option, you specify a second measurement temperature  $T_{m2}$ , and saturation current value at this temperature. The model uses these values, along with the parameter values at the first measurement temperature  $T_{m1}$ , to calculate the energy gap value.
- Specify the energy gap EG — Specify the energy gap value directly.

#### **Current I1 at second measurement temperature**

Specify the diode current  $I1$  value when the voltage is  $V1$  at the second measurement temperature. This parameter is only visible when you select Use an I-V data point at second measurement temperature T2 for the **Parameterization** parameter. The default value is 0.029 A.

#### **Voltage V1 at second measurement temperature**

Specify the diode voltage  $V1$  value when the current is  $I1$  at the second measurement temperature. This parameter is only visible when you select Use an I-V data point at second measurement temperature T2 for the **Parameterization** parameter. The default value is 1.05 V.

#### **Saturation current, IS, at second measurement temperature**

Specify the saturation current  $IS$  value at the second measurement temperature. This parameter is only visible when you select Specify saturation current at second measurement temperature T2 for the **Parameterization** parameter. The default value is 1.8e-8 A.

#### **Second measurement temperature**

Specify the value for the second measurement temperature. This parameter is only visible when you select either Use an I-V data point at second measurement temperature T2 or Specify saturation current at second measurement temperature T2 for the **Parameterization** parameter. The default value is 125 °C.

### Energy gap parameterization

This parameter is only visible when you select **Specify the energy gap EG** for the **Parameterization** parameter. It lets you select a value for the energy gap from a list of predetermined options, or specify a custom value:

- Use nominal value for silicon (EG=1.11eV) — This is the default.
- Use nominal value for 4H-SiC silicon carbide (EG=3.23eV)
- Use nominal value for 6H-SiC silicon carbide (EG=3.00eV)
- Use nominal value for germanium (EG=0.67eV)
- Use nominal value for gallium arsenide (EG=1.43eV)
- Use nominal value for selenium (EG=1.74eV)
- Use nominal value for Schottky barrier diodes (EG=0.69eV)
- Specify a custom value — If you select this option, the **Energy gap, EG** parameter appears in the dialog box, to let you specify a custom value for *EG*.

### Energy gap, EG

Specify a custom value for the energy gap, *EG*. This parameter is only visible when you select **Specify a custom value** for the **Energy gap parameterization** parameter. The default value is 1.11 eV.

### Saturation current temperature exponent parameterization

Select one of the following options to specify the saturation current temperature exponent value:

- Use nominal value for pn-junction diode (XTI=3) — This is the default.
- Use nominal value for Schottky barrier diode (XTI=2)
- Specify a custom value — If you select this option, the **Saturation current temperature exponent, XTI** parameter appears in the dialog box, to let you specify a custom value for *XTI*.

### Saturation current temperature exponent, XTI

Specify a custom value for the saturation current temperature exponent, *XTI*. This parameter is only visible when you select **Specify a custom value** for the **Saturation current temperature exponent parameterization** parameter. The default value is 3.

### **Device simulation temperature**

Specify the value for the temperature  $T_s$ , at which the device is to be simulated. The default value is 25 °C.

## **Ports**

The block has the following ports:

+

Electrical conserving port associated with the diode positive terminal

-

Electrical conserving port associated with the diode negative terminal

C

Electrical conserving port associated with the transistor collector terminal

E

Electrical conserving port associated with the transistor emitter terminal

## **References**

- [1] G. Massobrio and P. Antognetti. *Semiconductor Device Modeling with SPICE*. 2nd Edition, McGraw-Hill, 1993.
- [2] H. Ahmed and P.J. Spreadbury. *Analogue and digital electronics for engineers*. 2nd Edition, Cambridge University Press, 1984.

## **See Also**

Controlled Current Source | Diode | NPN Bipolar Transistor

## P-Channel JFET

Model P-Channel JFET

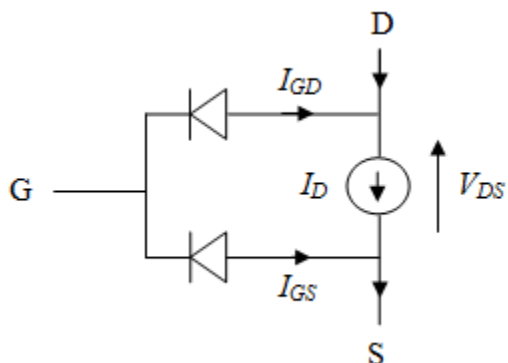


## Library

Semiconductor Devices

## Description

The P-Channel JFET block uses the Shichman and Hodges equations to represent a P-Channel JFET using a model with the following structure:



G is the transistor gate, D is the transistor drain and S is the transistor source. The drain current,  $I_D$ , depends on the region of operation and whether the transistor is operating in normal or inverse mode.

- In normal mode ( $-V_{DS} \geq 0$ ), the block provides the following relationship between the drain current  $I_D$  and the drain-source voltage  $V_{DS}$ .

Region	Applicable Range of $V_{GS}$ and $V_{DS}$ Values	Corresponding $I_D$ Equation
Off	$-V_{GS} \leq -V_{t0}$	$I_D = 0$
Linear	$0 < -V_{DS} < -V_{GS} + V_{t0}$	$I_D = \beta V_{DS}(2(-V_{GS} + V_{t0}) + V_{DS})(1 - \lambda V_{DS})$
Saturated	$0 < -V_{GS} + V_{t0} \leq -V_{DS}$	$I_D = -\beta (-V_{GS} + V_{t0})^2 (1 - \lambda V_{DS})$

- In inverse mode ( $-V_{DS} < 0$ ), the block provides the following relationship between the drain current  $I_D$  and the drain-source voltage  $V_{DS}$ .

Region	Applicable Range of $V_{GS}$ and $V_{DS}$ Values	Corresponding $I_D$ Equation
Off	$-V_{GD} \leq -V_{t0}$	$I_D = 0$
Linear	$0 < V_{DS} < -V_{GD} + V_{t0}$	$I_D = \beta V_{DS}(2(-V_{GD} + V_{t0}) - V_{DS})(1 + \lambda V_{DS})$
Saturated	$0 < -V_{GD} + V_{t0} \leq V_{DS}$	$I_D = \beta (-V_{GD} + V_{t0})^2 (1 + \lambda V_{DS})$

In the preceding equations:

- $V_{GS}$  is the gate-source voltage.
- $V_{GD}$  is the gate-drain voltage.
- $V_{t0}$  is the threshold voltage. If you select **Specify using equation parameters directly** for the **Parameterization** parameter,  $V_{t0}$  is the **Threshold voltage** parameter value. Otherwise, the block calculates  $V_{t0}$  from the datasheet parameters you specify.
- $\beta$  is the transconductance parameter. If you select **Specify using equation parameters directly** for the **Parameterization** parameter,  $\beta$  is the **Transconductance parameter** parameter value. Otherwise, the block calculates  $\beta$  from the datasheet parameters you specify.
- $\lambda$  is the channel-length modulation parameter. If you select **Specify using equation parameters directly** for the **Parameterization** parameter,  $\lambda$  is the **Channel-length modulation** parameter value. Otherwise, the block calculates  $\lambda$  from the datasheet parameters you specify.

The currents in each of the diodes satisfy the exponential diode equation



$$I_{GD} = -IS \cdot \left( e^{-qV_{GD}/kT_{m1}} - 1 \right)$$

$$I_{GS} = -IS \cdot \left( e^{-qV_{GS}/kT_{m1}} - 1 \right)$$

where:

- $IS$  is the saturation current. If you select **Specify using equation parameters directly** for the **Parameterization** parameter,  $IS$  is the **Saturation current** parameter value. Otherwise, the block calculates  $IS$  from the datasheet parameters you specify.
- $q$  is the elementary charge on an electron (1.602176e-19 Coulombs).
- $k$  is the Boltzmann constant (1.3806503e-23 J/K).
- $T_{m1}$  is the measurement temperature. The value comes from the **Measurement temperature** parameter.

The block models gate junction capacitance as a fixed gate-drain capacitance  $C_{GD}$  and a fixed gate-source capacitance  $C_{GS}$ . If you select **Specify using equation parameters directly** for the **Parameterization** parameter, you specify these values directly using the **Gate-drain junction capacitance** and **Gate-source junction capacitance** parameters. Otherwise, the block derives them from the **Input capacitance Ciss** and **Reverse transfer capacitance Crss** parameter values. The two parameterizations are related as follows:

- $C_{GD} = Crss$
- $C_{GS} = Ciss - Crss$

## Modeling Temperature Dependence

The default behavior is that dependence on temperature is not modeled, and the device is simulated at the temperature for which you provide block parameters. You can optionally include modeling the dependence of the transistor static behavior on temperature during simulation. Temperature dependence of the junction capacitances is not modeled, this being a much smaller effect.

When including temperature dependence, the transistor defining equations remain the same. The measurement temperature value,  $T_{m1}$ , is replaced with the simulation temperature,  $T_s$ . The transconductance,  $\beta$ , and the threshold voltage,  $V_{t0}$ , become a function of temperature according to the following equations:

$$\beta_{T_s} = \beta_{T_{m1}} \left( \frac{T_s}{T_{m1}} \right)^{BEX}$$

$$V_{t0s} = V_{t01} + \alpha (T_s - T_{m1})$$

where:

- $T_{m1}$  is the temperature at which the transistor parameters are specified, as defined by the **Measurement temperature** parameter value.
- $T_s$  is the simulation temperature.
- $\beta_{T_{m1}}$  is JFET transconductance at the measurement temperature.
- $\beta_{T_s}$  is JFET transconductance at the simulation temperature. This is the transconductance value used in the JFET equations when temperature dependence is modeled.
- $V_{t01}$  is the threshold voltage at measurement temperature.
- $V_{t0s}$  is the threshold voltage at simulation temperature. This is the threshold voltage value used in the JFET equations when temperature dependence is modeled.
- $BEX$  is the mobility temperature exponent. A typical value of  $BEX$  is -1.5.
- $\alpha$  is the gate threshold voltage temperature coefficient,  $dV_{th}/dT$ .

For most JFETS, you can use the default value of -1.5 for  $BEX$ . Some datasheets quote the value for  $\alpha$ , but most typically they provide the temperature dependence for the saturated drain current,  $I_{dss}$ . Depending on the block parameterization method, you have two ways of specifying  $\alpha$ :

- If you parameterize the block from a datasheet, you have to provide  $I_{dss}$  at a second measurement temperature. The block then calculates the value for  $\alpha$  based on this data.
- If you parameterize by specifying equation parameters, you have to provide the value for  $\alpha$  directly.

If you have more data comprising drain current as a function of gate-source voltage for fixed drain-source voltage plotted at more than one temperature, then you can also use Simulink Design Optimization software to help tune the values for  $\alpha$  and  $BEX$ .

In addition, the saturation current term,  $IS$ , in the gate-drain and gate-source current equations depends on temperature

$$IS_{T_s} = IS_{T_{m1}} \cdot (T_s / T_{m1})^{XTI} \cdot \exp\left(-\frac{EG}{kT_s}(1 - T_s / T_{m1})\right)$$

where:

- $IS_{T_{m1}}$  is the saturation current at the measurement temperature.
- $IS_{T_s}$  is the saturation current at the simulation temperature. This is the saturation current value used in the bipolar transistor equations when temperature dependence is modeled.
- $EG$  is the energy gap.
- $k$  is the Boltzmann constant (1.3806503e-23 J/K).
- $XTI$  is the saturation current temperature exponent.

Similar to  $\alpha$ , you have two ways of specifying  $EG$  and  $XTI$ :

- If you parameterize the block from a datasheet, you have to specify the gate reverse current,  $I_{gss}$ , at a second measurement temperature. The block then calculates the value for  $EG$  based on this data and assuming a p-n junction nominal value of 3 for  $XTI$ .
- If you parameterize by specifying equation parameters, you have to provide the values for  $EG$  and  $XTI$  directly. This option gives you most flexibility to match device behavior, for example, if you have a graph of  $I_{gss}$  as a function of temperature. With this data you can use Simulink Design Optimization software to help tune the values for  $EG$  and  $XTI$ .

## Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal port H on the block icon, and adds the **Thermal Port** tab to the block dialog box.

Use the thermal port to simulate the effects of generated heat and device temperature. For more information on using thermal ports and on the **Thermal Port** tab parameters, see “Simulating Thermal Effects in Semiconductors”.

## Basic Assumptions and Limitations

The model is based on the following assumptions:

- This block does not allow you to specify initial conditions on the junction capacitances. If you select the **Start simulation from steady state** option in the Solver Configuration block, the block solves the initial voltages to be consistent with the calculated steady state. Otherwise, voltages are zero at the start of the simulation.
- You may need to use nonzero ohmic resistance and junction capacitance values to prevent numerical simulation issues, but the simulation may run faster with these values set to zero.
- The block does not account for temperature-dependent effects on the junction capacitances.
- When you specify  $I_{dss}$  at a second measurement temperature, it must be quoted for the same working point (that is, the same drain current and gate-source voltage) as for the  $I_{dss}$  value on the **Main** tab. Inconsistent values for  $I_{dss}$  at the higher temperature will result in unphysical values for  $\alpha$  and unrepresentative simulation results.
- You may need to tune the value of  $BEX$  to replicate the  $I_D$ - $V_{GS}$  relationship (if available) for a given device. The value of  $BEX$  affects whether the  $I_D$ - $V_{GS}$  curves for different temperatures cross each other, or not, for the ranges of  $I_D$  and  $V_{GS}$  considered.

## Parameters

- “Main Tab” on page 1-1046
- “Ohmic Resistance Tab” on page 1-1048
- “Junction Capacitance Tab” on page 1-1048
- “Temperature Dependence Tab” on page 1-1049

## Main Tab

### Parameterization

Select one of the following methods for block parameterization:

- **Specify from a datasheet** — Provide parameters that the block converts to equations that describe the transistor. This is the default method.

- Specify using equation parameters directly — Provide equation parameters  $\beta$ ,  $IS$ ,  $V_{t0}$ , and  $\lambda$ .

### Gate reverse current, $I_{gss}$

The reverse current that flows in the diode when the drain and source are short-circuited and a large positive gate-source voltage is applied. This parameter is only visible when you select **Specify** from a datasheet for the **Parameterization** parameter. The default value is 5 nA.

### Saturated drain current, $I_{dss}$

The current that flows when a large negative drain-source voltage is applied for a specified gate-source voltage. For a depletion-mode device, this gate-source voltage may be zero, in which case  $I_{dss}$  may be referred to as the zero-gate voltage drain current. This parameter is only visible when you select **Specify** from a datasheet for the **Parameterization** parameter. The default value is -3 mA.

### $I_{dss}$ measurement point, [V<sub>gs</sub> V<sub>ds</sub>]

A vector of the values of  $V_{GS}$  and  $V_{DS}$  at which  $I_{dss}$  is measured. Normally  $V_{GS}$  is zero.  $V_{DS}$  should be greater than zero. This parameter is only visible when you select **Specify** from a datasheet for the **Parameterization** parameter. The default value is [ 0 -15 ] V.

### Small-signal parameters, [g<sub>fs</sub> g<sub>os</sub>]

A vector of the values of  $g_{fs}$  and  $g_{os}$ .  $g_{fs}$  is the forward transfer conductance, that is, the conductance for a fixed drain-source voltage.  $g_{os}$  is the output conductance, that is, the conductance for a fixed gate-source voltage. This parameter is only visible when you select **Specify** from a datasheet for the **Parameterization** parameter. The default value is [ 2.5e+3 75 ] uS.

### Small-signal measurement point [V<sub>gs</sub> V<sub>ds</sub>]

A vector of the values of  $V_{GS}$  and  $V_{DS}$  at which  $g_{fs}$  and  $g_{os}$  are measured.  $V_{DS}$  should be greater than zero. For depletion-mode devices,  $V_{GS}$  is typically zero. This parameter is only visible when you select **Specify** from a datasheet for the **Parameterization** parameter. The default value is [ 0 -15 ] V.

### Transconductance parameter

The derivative of drain current with respect to gate voltage. This parameter is only visible when you select **Specify using equation parameters directly** for the **Parameterization** parameter. The default value is  $1e-04$  A/V<sup>2</sup>.

### Saturation current

The magnitude of the current that the ideal diode equation approaches asymptotically for very large reverse bias levels. This parameter is only visible when you select

Specify using equation parameters directly for the **Parameterization** parameter. The default value is  $1e-14$  A.

### **Threshold voltage**

The gate-source voltage above which the transistor produces a nonzero drain current. For an enhancement device,  $V_{t0}$  should be negative. For a depletion mode device,  $V_{t0}$  should be positive. This parameter is only visible when you select **Specify using equation parameters directly** for the **Parameterization** parameter. The default value is 2 V.

### **Channel-length modulation**

The channel-length modulation. This parameter is only visible when you select **Specify using equation parameters directly** for the **Parameterization** parameter. The default value is  $0$  1/V.

### **Measurement temperature**

The temperature for which the datasheet parameters are quoted. The default value is 25 °C.

## **Ohmic Resistance Tab**

### **Source ohmic resistance**

The transistor source resistance. The default value is  $1e-4$   $\Omega$ . The value must be greater than or equal to  $0$ .

### **Drain ohmic resistance**

The transistor drain resistance. The default value is  $0.01$   $\Omega$ . The value must be greater than or equal to  $0$ .

## **Junction Capacitance Tab**

### **Parameterization**

Select one of the following methods for block parameterization:

- **Specify from a datasheet** — Provide parameters that the block converts to junction capacitance values. This is the default method.
- **Specify using equation parameters directly** — Provide junction capacitance parameters directly.

**Input capacitance, Ciss**

The gate-source capacitance with the drain shorted to the source. This parameter is only visible when you select **Specify** from a datasheet for the **Model junction capacitance** parameter. The default value is 4.5 pF.

**Reverse transfer capacitance, Crss**

The drain-gate capacitance with the source connected to ground. This parameter is only visible when you select **Specify** from a datasheet for the **Model junction capacitance** parameter. The default value is 1.5 pF.

**Gate-source junction capacitance**

The value of the capacitance placed between the gate and the source. This parameter is only visible when you select **Specify** using equation parameters directly for the **Model junction capacitance** parameter. The default value is 3 pF.

**Gate-drain junction capacitance**

The value of the capacitance placed between the gate and the drain. This parameter is only visible when you select **Specify** using equation parameters directly for the **Model junction capacitance** parameter. The default value is 1.5 pF.

**Temperature Dependence Tab****Parameterization**

Select one of the following methods for temperature dependence parameterization:

- **None** – Simulate at parameter measurement temperature – Temperature dependence is not modeled. This is the default method.
- **Model temperature dependence** – Model temperature-dependent effects. You also have to provide a set of additional parameters depending on the block parameterization method. If you parameterize the block from a datasheet, you have to provide values for  $I_{gss}$  and  $I_{dss}$  at second measurement temperature. If you parameterize by directly specifying equation parameters, you have to provide the values for  $EG$ ,  $XTI$ , and the gate threshold voltage temperature coefficient,  $dV_{t0}/dT$ . Regardless of the block parameterization method, you also have to provide values for  $BEX$  and for the simulation temperature,  $T_s$ .

**Gate reverse current, I\_gss, at second measurement temperature**

The value of the gate reverse current,  $I_{gss}$ , at the second measurement temperature. This parameter is only visible when you select **Specify** from a datasheet for the **Parameterization** parameter on the **Main** tab. It must be quoted for the same

working point (drain current and gate-source voltage) as the **Drain-source on resistance, R\_DS(on)** parameter on the **Main** tab. The default value is 950 nA.

**Saturated drain current, I<sub>dss</sub>, at second measurement temperature**

The value of the saturated drain current,  $I_{dss}$ , at the second measurement temperature, and when the  $I_{dss}$  measurement point is the same as defined by the **I<sub>dss</sub> measurement point, [V<sub>gs</sub> V<sub>ds</sub>]** parameter on the **Main** tab. This parameter is only visible when you select **Specify** from a datasheet for the **Parameterization** parameter on the **Main** tab. The default value is -2.3 mA.

**Second measurement temperature**

Second temperature  $T_{m2}$  at which **Gate reverse current, I<sub>gss</sub>, at second measurement temperature** and **Saturated drain current, I<sub>dss</sub>, at second measurement temperature** are measured. This parameter is only visible when you select **Specify** from a datasheet for the **Parameterization** parameter on the **Main** tab. The default value is 125 °C.

**Energy gap, EG**

Energy gap value. This parameter is only visible when you select **Specify** using equation parameters directly for the **Parameterization** parameter on the **Main** tab. The default value is 1.11 eV.

**Saturation current temperature exponent, XTI**

Saturation current temperature coefficient value. This parameter is only visible when you select **Specify** using equation parameters directly for the **Parameterization** parameter on the **Main** tab. The default value is 3.

**Gate threshold voltage temperature coefficient, dVt0/dT**

The rate of change of gate threshold voltage with temperature. This parameter is only visible when you select **Specify** using equation parameters directly for the **Parameterization** parameter on the **Main** tab. The default value is 1 mV/K.

**Mobility temperature exponent, BEX**

Mobility temperature coefficient value. You can use the default value for most JFETs. See the “Basic Assumptions and Limitations” on page 1-1046 section for additional considerations. The default value is -1.5.

**Device simulation temperature**

Temperature  $T_s$  at which the device is simulated. The default value is 25 °C.



## Ports

The block has the following ports:

G

Electrical conserving port associated with the transistor gate terminal

D

Electrical conserving port associated with the transistor drain terminal

S

Electrical conserving port associated with the transistor source terminal

## References

- [1] H. Shichman and D. A. Hodges, *Modeling and simulation of insulated-gate field-effect transistor switching circuits*. IEEE J. Solid State Circuits, SC-3, 1968.
- [2] G. Massobrio and P. Antognetti. *Semiconductor Device Modeling with SPICE*. 2nd Edition, McGraw-Hill, 1993. Chapter 2.

## See Also

N-Channel JFET

## P-Channel LDMOS FET

Model P-Channel LDMOS or VDMOS transistors suitable for high voltage



### Library

Semiconductor Devices

### Description

The P-Channel LDMOS FET block lets you model LDMOS (or VDMOS) transistors suitable for high voltage. The model is based on surface potential and includes effects due to an extended drain (drift) region:

- Nonlinear capacitive effects associated with the drift region
- Surface scattering and velocity saturation in the drift region
- Velocity saturation and channel-length modulation in the channel region
- Charge conservation inside the model, so you can use the model for charge sensitive simulations
- The intrinsic body diode
- Reverse recovery in the body diode model
- Temperature scaling of physical parameters
- For the thermal variant (see “Thermal Port” on page 1-1054), dynamic self-heating

For information on physical background and defining equations, see the N-Channel LDMOS FET block reference page. Both the p-type and n-type versions of the LDMOS model use the same underlying code with appropriate voltage transformations, to account for the different device types.

The charge model is similar to that of the surface-potential-based MOSFET model, with additional expressions to account for the charge in the drift region. The block uses the

derived equations as described in [1], which include both inversion and accumulation in the drift region.

## Modeling Body Diode

The block models the body diode as an ideal, exponential diode with both junction and diffusion capacitances:

$$I_{dio} = I_s \left[ \exp\left(-\frac{V_{BD}}{n\phi_T}\right) - 1 \right]$$

$$C_j = \frac{C_{j0}}{\sqrt{1 + \frac{V_{BD}}{V_{bi}}}}$$

$$C_{diff} = \frac{\tau I_s}{n\phi_T} \exp\left(-\frac{V_{BD}}{n\phi_T}\right)$$

where:

- $I_{dio}$  is the current through the diode.
- $I_s$  is the reverse saturation current.
- $V_{BD}$  is the body-drain voltage.
- $n$  is the ideality factor.
- $\phi_T$  is the thermal voltage.
- $C_j$  is the junction capacitance of the diode.
- $C_{j0}$  is the zero-bias junction capacitance.
- $V_{bi}$  is the built-in voltage.
- $C_{diff}$  is the diffusion capacitance of the diode.
- $\tau$  is the transit time.

The capacitances are defined through an explicit calculation of charges, which are then differentiated to give the capacitive expressions above. The block computes the capacitive diode currents as time derivatives of the relevant charges, similar to the computation in the surface-potential-based MOSFET model.

## Modeling Temperature Dependence

The default behavior is that dependence on temperature is not modeled, and the device is simulated at the temperature for which you provide block parameters. To model the dependence on temperature during simulation, select **Model temperature dependence** for the **Parameterization** parameter on the **Temperature Dependence** tab.

The model includes temperature effects on the capacitance characteristics, as well as modeling the dependence of the transistor static behavior on temperature during simulation.

The **Measurement temperature** parameter on the **Main** tab specifies temperature  $T_{m1}$  at which the other device parameters have been extracted. The **Temperature Dependence** tab provides the simulation temperature,  $T_s$ , and the temperature-scaling coefficients for the other device parameters. For more information, see “Temperature Dependence Tab” on page 1-1059.

## Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal port H on the block icon, and adds the **Thermal Port** tab to the block dialog box.

Use the thermal port to simulate the effects of generated heat and device temperature. For more information on using thermal ports and on the **Thermal Port** tab parameters, see “Simulating Thermal Effects in Semiconductors”.

The thermal variant of the block includes dynamic self-heating, that is, lets you simulate the effect of self-heating on the electrical characteristics of the device.

## Parameters

- “Main Tab” on page 1-1055
- “Ohmic Resistance Tab” on page 1-1057
- “Capacitances Tab” on page 1-1058
- “Body Diode Tab” on page 1-1058

- “Temperature Dependence Tab” on page 1-1059

## Main Tab

### Gain, [channel drift\_region]

The gain,  $\beta$ , of the MOSFET regions. The parameter value is a two-element vector, with the first element corresponding to the channel, and the second — to the drift region. This parameter primarily defines the linear region of operation on an  $I_D$ - $V_{DS}$  characteristic. The values of both elements must be greater than 0. The default value is [11.6, 0.01] A/V<sup>2</sup>.

### Flatband voltage, [channel drift\_region]

The flatband voltage,  $V_{FB}$ , defines the gate bias that must be applied in order to achieve the flatband condition at the surface of the silicon. The parameter value is a two-element vector, with the first element corresponding to the channel, and the second — to the drift region. The default value is [-1.05, -0.1] V. You can also use this parameter to arbitrarily shift the threshold voltage due to material work function differences, and to trapped interface or oxide charges. In practice, however, it is usually recommended to modify the threshold voltage by using the **Body factor** and **Surface potential at strong inversion** parameters first, and only use this parameter for fine-tuning.

The threshold voltage for the channel region, for a short-circuited source-bulk connection, is approximately

$$-V_T = V_{FB} + 2\phi_B + 2\phi_T + \gamma\sqrt{2\phi_B + 2\phi_T}$$

where  $2\phi_B$  is the surface potential at strong inversion and  $\gamma$  is the body factor, both at the channel region.

### Body factor, [channel drift\_region]

Body factor,  $\gamma$ , in the surface-potential equation. The parameter value is a two-element vector, with the first element corresponding to the channel, and the second — to the drift region. The default value is [3.4, 2.5] V<sup>1/2</sup>.

For the channel region, the body factor is

$$\gamma = \frac{\sqrt{2q\epsilon_{Si}N_A}}{C_{ox}}$$

See the N-Channel MOSFET block reference page for details on this equation. The drift region equation is similar, except that  $N_A$  is replaced by the doping density,  $N_D$ . The channel-region parameter value primarily impacts the threshold voltage. For the drift region, this parameter primarily affects the charge model, and also has a minor effect on the pinch-off behavior of the bulk current through the drift region.

**Surface potential at strong inversion, [channel drift\_region]**

The  $2\phi_B$  term in the surface-potential equation. The parameter value is a two-element vector, with the first element corresponding to the channel, and the second — to the drift region. The default value is [0.95, 0.95] V.

The channel-region parameter value also primarily impacts the threshold voltage. For the drift region, this parameter affects the charge model only.

**Velocity saturation factor, [channel drift\_region]**

Velocity saturation,  $\theta_{\text{sat}}$ , in the drain-current equation. Use this parameter in cases where a good fit to linear operation leads to a saturation current that is too high. By increasing this parameter value, you reduce the saturation current. The parameter value is a two-element vector, with the first element corresponding to the channel, and the second — to the drift region. The default value is [0.0, 0.1] 1/V, which means that velocity saturation in the channel region is off by default.

**Drift region surface scattering factor**

Surface scattering factor,  $\theta_{\text{surf}}$ , in the drain-current equation. This parameter applies to the drift region only and accounts for scattering in the accumulation layer due to the vertical electric field. The default value is 0 1/V.

**Channel-length modulation factor**

The factor,  $\alpha$ , multiplying the logarithmic term in the  $G_{\Delta L}$  equation. See the N-Channel MOSFET block reference page for details on this equation. This parameter describes the onset of channel-length modulation. For device characteristics that exhibit a positive conductance in saturation, increase the parameter value to fit this behavior. This parameter applies to the channel region only. The default value is 0, which means that channel-length modulation is off by default.

**Channel-length modulation voltage**

The voltage  $V_p$  in the  $G_{\Delta L}$  equation. See the N-Channel MOSFET block reference page for details on this equation. This parameter controls the drain-voltage at which channel-length modulation starts to become active. This parameter applies to the channel region only. The default value is 50 mV.

**Linear-to-saturation transition coefficient**

This parameter controls how smoothly the MOSFET transitions from linear into saturation, particularly when velocity saturation is enabled. This parameter can usually be left at its default value, but you can use it to fine-tune the knee of the  $I_D$ - $V_{DS}$  characteristic. This parameter applies both to the channel and drift regions. The expected range for this parameter value is between 2 and 8. The default value is 8.

**Measurement temperature**

Temperature  $T_{m1}$  at which the block parameters are measured. If the **Device simulation temperature** parameter on the **Temperature Dependence** tab differs from this value, then device parameters will be scaled from their defined values according to the simulation and reference temperatures. For more information, see “Temperature Dependence Tab” on page 1-1059. The default value is 25 °C.

**Ohmic Resistance Tab****Source ohmic resistance**

The transistor source resistance, that is, the series resistance associated with the source contact. The default value is  $1e-4 \Omega$ . The value must be greater than or equal to 0.

**Drain ohmic resistance**

The transistor drain resistance, that is, the series resistance associated with the drain contact and with the LOCOS part of the drift region, which is not heavily impacted by the applied gate voltage. The default value is  $0.07 \Omega$ . The value must be greater than or equal to 0.

**Gate ohmic resistance**

The transistor gate resistance, that is, the series resistance associated with the gate contact. The default value is  $8.4 \Omega$ . The value must be greater than or equal to 0.

**Drift region low-bias resistance for gated region**

Resistance  $R_D$  in the drain-current equation. It represents the resistance of the bulk part of the drift region in the absence of depletion from the top and bottom interfaces. The default value is  $0.1 \Omega$ . The value must be greater than or equal to 0.

**Drift region depletion layer thickness factor**

Parameter  $\lambda_D$  in the drain-current equation. It is the ratio of vertical depths  $y_1$  and  $y_2$  at zero bias, where  $y_1$  represents the space-charge region and  $y_2$  represents the undepleted part of the drift region. See the N-Channel LDMOS FET block reference page for an illustration. The default value is 0.2.

## Capacitances Tab

### Oxide capacitance

The parallel plate gate-channel and gate-drift-region capacitance. The parameter value is a two-element vector, with the first element corresponding to the channel, and the second — to the drift region. The default value is [1600.0, 1000.0] pF.

### Gate-source overlap capacitance

The fixed, linear capacitance associated with the overlap of the gate electrode with the source well. The default value is 15 pF.

### Gate-drain overlap capacitance

The fixed, linear capacitance associated with the overlap of the gate electrode with the drain well. The default value is 15 pF.

## Body Diode Tab

### Reverse saturation current

The current designated by the  $I_s$  symbol in the body-diode equations. The default value is 1e-13 A.

### Built-in voltage

The built-in voltage of the diode, designated by the  $V_{bi}$  symbol in the body-diode equations. The default value is 0.6 V.

### Ideality factor

The factor designated by the  $n$  symbol in the body-diode equations. The default value is 1.

### Zero-bias junction capacitance

The capacitance between the drain and bulk contacts at zero-bias due to the body diode alone. It is designated by the  $C_{j0}$  symbol in the body-diode equations. The default value is 1800 pF.

### Transit time

The time designated by the  $\tau$  symbol in the body-diode equations. The default value is 50 ns.



## Temperature Dependence Tab

### Parameterization

Select one of the following methods for temperature dependence parameterization:

- None – Simulate at parameter measurement temperature – Temperature dependence is not modeled. This is the default method.
- Model temperature dependence – Model temperature-dependent effects. Provide a value for the device simulation temperature,  $T_s$ , and the temperature-scaling coefficients for other block parameters.

### Device simulation temperature

Temperature  $T_s$  at which the device is simulated. The default value is 25 °C.

### Gain temperature exponent, [channel drift\_region]

The parameter value is a two-element vector, with the first element corresponding to the channel, and the second — to the drift region. Both in the channel and the drift region, the MOSFET gain,  $\beta$ , is assumed to scale exponentially with temperature,  $\beta = \beta_{m1}(T_{m1}/T_s)^{\eta_\beta}$ .  $\beta_{m1}$  is the value of the channel or drift region gain, as specified by the **Gain, [channel drift\_region]** parameter from the **Main** tab.  $\eta_\beta$  is the corresponding element of the **Gain temperature exponent, [channel drift\_region]** parameter. The default value is [1.3, 1.3].

### Flatband voltage temperature coefficient, [channel drift\_region]

The parameter value is a two-element vector, with the first element corresponding to the channel, and the second — to the drift region. The flatband voltage,  $V_{FB}$ , is assumed to scale linearly with temperature,  $V_{FB} = V_{FBm1} + (T_s - T_{m1})S_{TV_{FB}}$ .  $V_{FBm1}$  is the value of the channel or drift region flatband voltage, as specified by the **Flatband voltage, [channel drift\_region]** parameter from the **Main** tab.  $S_{TV_{FB}}$  is the corresponding element of the **Flatband voltage temperature coefficient, [channel drift\_region]** parameter. The default value is [0.0005, 0.0005] V/K.

### Surface potential at strong inversion temperature coefficient

The surface potential at strong inversion,  $2\phi_B$ , is assumed to scale linearly with temperature,  $2\phi_B = 2\phi_{Bm1} + (T_s - T_{m1})S_{T\phi_B}$ .  $2\phi_{Bm1}$  is the value of the **Surface potential at strong inversion** parameter from the **Main** tab and  $S_{T\phi_B}$  is the **Surface potential at strong inversion temperature coefficient**. The default value is -8.5e-4 V/K.

### Velocity saturation temperature exponent, [channel drift\_region]

The parameter value is a two-element vector, with the first element corresponding to the channel, and the second — to the drift region. The velocity saturation,  $\theta_{sat}$ , is

assumed to scale exponentially with temperature,  $\theta_{\text{sat}} = \theta_{\text{sat},m1}(T_{m1}/T_s)^{\eta_{\theta}}$ .  $\theta_{\text{sat},m1}$  is the value of the channel or drift region velocity saturation factor, as specified by the **Velocity saturation factor, [channel drift\_region]** parameter from the **Main** tab.  $\eta_{\theta}$  is the corresponding element of the **Velocity saturation temperature exponent, [channel drift\_region]** parameter. The default value is [1.04, 1.04].

### Ohmic resistance temperature exponent

The series resistances are assumed to correspond to semiconductor resistances. Therefore, they decrease exponentially with increasing temperature.  $R_i = R_{i,m1}(T_{m1}/T_s)^{\eta_R}$ , where  $i$  is S, D, or G, for the source, drain, or gate series resistance, respectively.  $R_{i,m1}$  is the value of the corresponding series resistance parameter from the **Ohmic Resistance** tab and  $\eta_R$  is the **Ohmic resistance temperature exponent**. The default value is 0.95.

### Drift region low-bias resistance temperature exponent for gated portion

Resistance  $R_D$ , the low-bias resistance of the bulk part of the drift region, scales similarly to the other series resistances. A separate value of the temperature exponent for this resistance provides an extra degree of freedom. The default value is 0.95.

### Body diode reverse saturation current temperature exponent

The reverse saturation current for the body diode is assumed to be proportional to the square of the intrinsic carrier concentration,  $n_i = N_C \exp(-E_G/2k_B T)$ .  $N_C$  is the temperature-dependent effective density of states and  $E_G$  is the temperature-dependent bandgap for the semiconductor material. To avoid introducing another temperature-scaling parameter, the block neglects the temperature dependence of the bandgap and uses the bandgap of silicon at 300K (1.12eV) for all device types. Therefore, the temperature-scaled reverse saturation current is given by

$$I_s = I_{s,m1} \left( \frac{T_s}{T_{m1}} \right)^{\eta_{Is}} \cdot \exp \left( \frac{E_G}{k_B} \cdot \left( \frac{1}{T_{m1}} - \frac{1}{T_s} \right) \right).$$

$I_{s,m1}$  is the value of the **Reverse saturation current** parameter from the **Body Diode** tab,  $k_B$  is Boltzmann's constant (8.617x10-5eV/K), and  $\eta_{Is}$  is the **Body diode reverse saturation current temperature exponent**. The default value is 3, because  $N_C$  for silicon is roughly proportional to  $T^{3/2}$ . You can remedy the effect of neglecting the temperature-dependence of the bandgap by a pragmatic choice of  $\eta_{Is}$ .

## Ports

The block has the following ports:

G

Electrical conserving port associated with the transistor gate terminal

D

Electrical conserving port associated with the transistor drain terminal

S

Electrical conserving port associated with the transistor source terminal

## References

- [1] Aarts, A., N. D'Halleweyn, and R. Van Langevelde. "A Surface-Potential-Based High-Voltage Compact LDMOS Transistor Model." *IEEE Transactions on Electron Devices*. 52(5):999 - 1007. June 2005.
- [2] Van Langevelde, R., A. J. Scholten, and D. B .M. Klaassen. "Physical Background of MOS Model 11. Level 1101." *Nat.Lab. Unclassified Report 2003/00239*. April 2003.
- [3] Oh, S-Y., D. E. Ward, and R. W. Dutton. "Transient analysis of MOS transistors." *IEEE J. Solid State Circuits*. SC-15, pp. 636-643, 1980.

## See Also

N-Channel LDMOS FET | N-Channel MOSFET

## Topics

Interactive Generation of LDMOS Characteristics

**Introduced in R2016b**

## P-Channel MOSFET

Model P-Channel MOSFET using either Shichman-Hodges equation or surface-potential-based model



## Library

Semiconductor Devices

## Description

The P-Channel MOSFET block provides two main modeling variants:

- Based on threshold voltage — Uses the Shichman-Hodges equation to represent the device. This modeling approach, based on threshold voltage, has the benefits of simple parameterization and simple current-voltage expressions. However, these models have difficulty in accurately capturing transitions across the threshold voltage and lack some important effects, such as velocity saturation. For details, see “Threshold-Based Model” on page 1-1063.
- Based on surface potential — Uses the surface-potential equation to represent the device. This modeling approach provides a greater level of model fidelity than the simple square-law (threshold-voltage-based) models can provide. The trade-off is that there are more parameters that require extraction. For details, see “Surface-Potential-Based Model” on page 1-1066.

Together with the thermal port variants (see “Thermal Port” on page 1-1070), the block therefore provides you with four choices. To select the desired variant, right-click the block in your model. From the context menu, select **Simscape > Block choices**, and then one of the following options:

- **Threshold-based** — Basic model, which represents the device using the Shichman-Hodges equation (based on threshold voltage) and does not simulate thermal effects. This is the default.
- **Threshold-based with thermal** — Model based on threshold voltage and with exposed thermal port.
- **Surface-potential-based** — Model based on surface potential. This model does not simulate thermal effects.
- **Surface-potential-based with thermal** — Thermal variant of the model based on surface potential.

## Threshold-Based Model

The threshold-based variant of the block uses the Shichman and Hodges equations [1] for an insulated-gate field-effect transistor to represent a P-Channel MOSFET.

The drain-source current,  $I_{DS}$ , depends on the region of operation:

- In the off region ( $-V_{GS} < -V_{th}$ ) the drain-source current is:

$$I_{DS} = 0$$

- In the linear region ( $0 < -V_{DS} < -V_{GS} + V_{th}$ ) the drain-source current is:

$$I_{DS} = -K \left( (V_{GS} - V_{th})V_{DS} - V_{DS}^2 / 2 \right) (1 + \lambda |V_{DS}|)$$

- In the saturated region ( $0 < -V_{GS} + V_{th} < -V_{DS}$ ) the drain-source current is:

$$I_{DS} = -(K / 2)(V_{GS} - V_{th})^2 (1 + \lambda |V_{DS}|)$$

In the preceding equations:

- $K$  is the transistor gain.
- $V_{DS}$  is the negative drain-source voltage.
- $V_{GS}$  is the gate-source voltage.
- $V_{th}$  is the threshold voltage.
- $\lambda$  is the channel modulation.

## Charge Model for Threshold-Based Variant

The block models junction capacitances either by fixed capacitance values, or by tabulated values as a function of the drain-source voltage. In either case, you can either directly specify the gate-source and gate-drain junction capacitance values, or let the block derive them from the input and reverse transfer capacitance values. Therefore, the **Parameterization** options for charge model on the **Junction Capacitance** tab are:

- **Specify fixed input, reverse transfer and output capacitance** — Provide fixed parameter values from datasheet and let the block convert the input and reverse transfer capacitance values to junction capacitance values, as described below. This is the default method.
- **Specify fixed gate-source, gate-drain and drain-source capacitance** — Provide fixed values for junction capacitance parameters directly.
- **Specify tabulated input, reverse transfer and output capacitance** — Provide tabulated capacitance and drain-source voltage values based on datasheet plots. The block converts the input and reverse transfer capacitance values to junction capacitance values, as described below.
- **Specify tabulated gate-source, gate-drain and drain-source capacitance** — Provide tabulated values for junction capacitances and drain-source voltage.

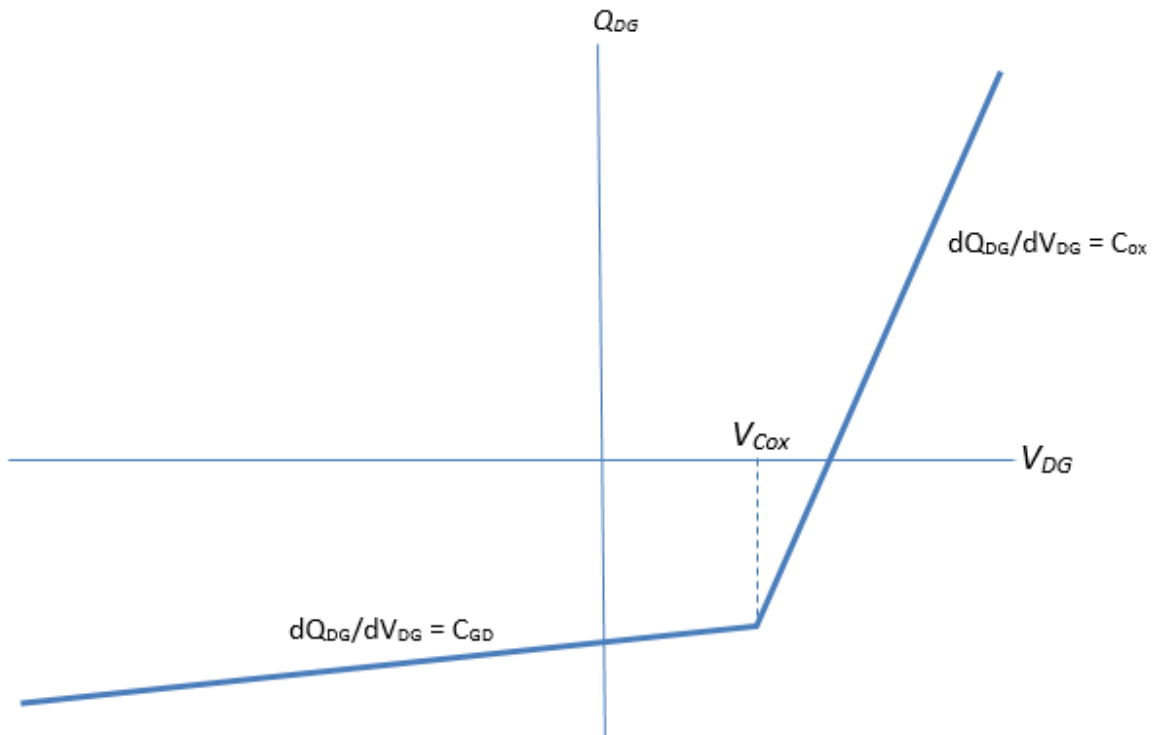
Use one of the tabulated capacitance options (**Specify tabulated input, reverse transfer and output capacitance** or **Specify tabulated gate-source, gate-drain and drain-source capacitance**) when the datasheet provides a plot of junction capacitances as a function of drain-source voltage. Using tabulated capacitance values gives more accurate dynamic characteristics and avoids the need for iterative tuning of parameters to fit the dynamics.

If you use the **Specify fixed gate-source, gate-drain and drain-source capacitance** or **Specify tabulated gate-source, gate-drain and drain-source capacitance** option, the **Junction Capacitance** tab lets you specify the **Gate-drain junction capacitance**, **Gate-source junction capacitance**, and **Drain-source junction capacitance** parameter values (fixed or tabulated) directly. Otherwise, the block derives them from the **Input capacitance**, **C<sub>iss</sub>**, **Reverse transfer capacitance**, **Cr<sub>ss</sub>**, and **Output capacitance**, **C<sub>oss</sub>** parameter values. These two parameterization methods are related as follows:

- $C_{GD} = Cr_{ss}$

- $C_{GS} = C_{iss} - C_{rss}$
- $C_{DS} = C_{oss} - C_{rss}$

The two fixed capacitance options (Specify fixed input, reverse transfer and output capacitance or Specify fixed gate-source, gate-drain and drain-source capacitance) let you model gate junction capacitance as a fixed gate-source capacitance  $C_{GS}$  and either a fixed or a nonlinear gate-drain capacitance  $C_{GD}$ . If you select the Gate-drain charge function is nonlinear option for the **Charge-voltage linearity** parameter, then the gate-drain charge relationship is defined by the piecewise-linear function shown in the following figure.



**Nonlinear gate-drain charge function**

For instructions on how to map a time response to device capacitance values, see the N-Channel IGBT block reference page. However, this mapping is only approximate because the Miller voltage typically varies more from the threshold voltage than in the case for the IGBT.

---

**Note** Because this block implementation includes a charge model, you must model the impedance of the circuit driving the gate to obtain representative turn-on and turn-off dynamics. Therefore, if you are simplifying the gate drive circuit by representing it as a controlled voltage source, you must include a suitable series resistor between the voltage source and the gate.

---

## Surface-Potential-Based Model

The surface-potential-based variant of the block provides a greater level of model fidelity than the simple square-law (threshold-voltage-based) model. The surface-potential-based block variant includes the following effects:

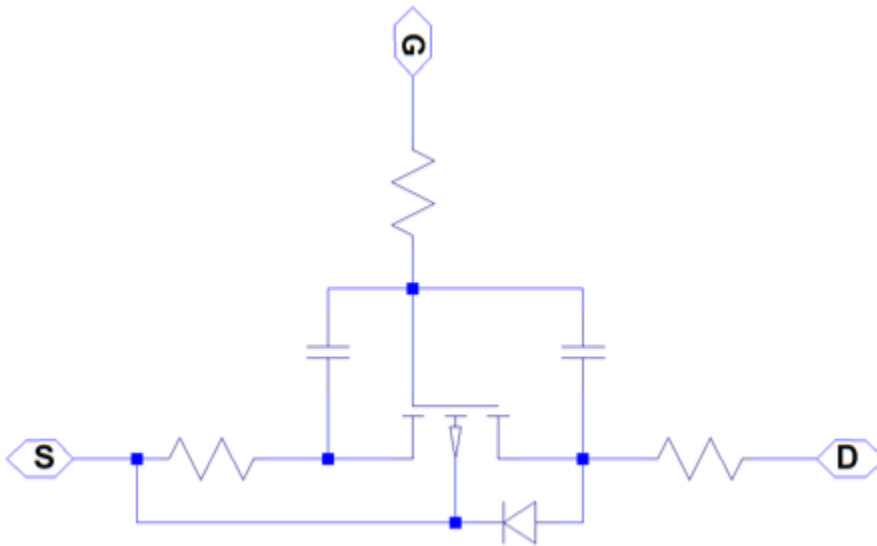
- Fully nonlinear capacitance model (including the nonlinear Miller capacitance)
- Charge conservation inside the model, so you can use the model for charge sensitive simulations
- Velocity saturation and channel-length modulation
- The intrinsic body diode
- Reverse recovery in the body diode model
- Temperature scaling of physical parameters
- For the thermal variant, dynamic self-heating (that is, you can simulate the effect of self-heating on the electrical characteristics of the device)

This model is a minimal version of the world-standard PSP model (see <https://nsti.org/Nanotech2005/WCM2005/WCM2005-GGildenblat.pdf>), including only certain effects from the PSP model to strike a balance between model fidelity and complexity. For details of the physical background to the phenomena included in this model, see [2].

The surface-potential equation is derived similar to the way described on the N-Channel MOSFET block reference page, with all voltages, charges, and currents multiplied by -1.

The overall model consists of an intrinsic MOSFET defined by the surface-potential formulation, a body diode, series resistances, and fixed overlap capacitances, as shown in the schematic.





## Modeling Body Diode

The block models the body diode as an ideal, exponential diode with both junction and diffusion capacitances:

$$I_{dio} = I_s \left[ \exp\left(-\frac{V_{BD}}{n\phi_T}\right) - 1 \right]$$

$$C_j = \frac{C_{j0}}{\sqrt{1 + \frac{V_{BD}}{V_{bi}}}}$$

$$C_{diff} = \frac{\tau I_s}{n\phi_T} \exp\left(-\frac{V_{BD}}{n\phi_T}\right)$$

where:

- $I_{\text{dio}}$  is the current through the diode.
- $I_s$  is the reverse saturation current.
- $V_{\text{BD}}$  is the body-drain voltage.
- $n$  is the ideality factor.
- $\phi_T$  is the thermal voltage.
- $C_j$  is the junction capacitance of the diode.
- $C_{j0}$  is the zero-bias junction capacitance.
- $V_{\text{bi}}$  is the built-in voltage.
- $C_{\text{diff}}$  is the diffusion capacitance of the diode.
- $\tau$  is the transit time.

The capacitances are defined through an explicit calculation of charges, which are then differentiated to give the capacitive expressions above. The block computes the capacitive diode currents as time derivatives of the relevant charges, similar to the computation in the surface-potential-based MOSFET model.

## Modeling Temperature Dependence

The default behavior is that dependence on temperature is not modeled, and the device is simulated at the temperature for which you provide block parameters. To model the dependence on temperature during simulation, select **Model temperature dependence** for the **Parameterization** parameter on the **Temperature Dependence** tab.

### Threshold-Based Model

For threshold-based variant, you can include modeling the dependence of the transistor static behavior on temperature during simulation. Temperature dependence of the junction capacitances is not modeled, this being a much smaller effect.

When including temperature dependence, the transistor defining equations remain the same. The gain,  $K$ , and the threshold voltage,  $V_{\text{th}}$ , become a function of temperature according to the following equations:

$$K_{T_s} = K_{T_{m1}} \left( \frac{T_s}{T_{m1}} \right)^{BEX}$$

$$V_{\text{ths}} = V_{\text{th1}} + \alpha (T_s - T_{\text{m1}})$$

where:

- $T_{\text{m1}}$  is the temperature at which the transistor parameters are specified, as defined by the **Measurement temperature** parameter value.
- $T_s$  is the simulation temperature.
- $K_{T_{\text{m1}}}$  is the transistor gain at the measurement temperature.
- $K_{T_s}$  is the transistor gain at the simulation temperature. This is the transistor gain value used in the MOSFET equations when temperature dependence is modeled.
- $V_{\text{th1}}$  is the threshold voltage at the measurement temperature.
- $V_{\text{ths}}$  is the threshold voltage at the simulation temperature. This is the threshold voltage value used in the MOSFET equations when temperature dependence is modeled.
- $BEX$  is the mobility temperature exponent. A typical value of  $BEX$  is -1.5.
- $\alpha$  is the gate threshold voltage temperature coefficient,  $dV_{\text{th}}/dT$ .

For most MOSFETS, you can use the default value of -1.5 for  $BEX$ . Some datasheets quote the value for  $\alpha$ , but most typically they provide the temperature dependence for drain-source on resistance,  $R_{\text{DS}}(\text{on})$ . Depending on the block parameterization method, you have two ways of specifying  $\alpha$ :

- If you parameterize the block from a datasheet, you have to provide  $R_{\text{DS}}(\text{on})$  at a second measurement temperature. The block then calculates the value for  $\alpha$  based on this data.
- If you parameterize by specifying equation parameters, you have to provide the value for  $\alpha$  directly.

If you have more data comprising drain current as a function of gate-source voltage for more than one temperature, then you can also use Simulink Design Optimization software to help tune the values for  $\alpha$  and  $BEX$ .

### Surface-Potential-Based Model

The surface-potential-based model includes temperature effects on the capacitance characteristics, as well as modeling the dependence of the transistor static behavior on temperature during simulation.

The **Measurement temperature** parameter on the **Main** tab specifies temperature  $T_{\text{m1}}$  at which the other device parameters have been extracted. The **Temperature**

**Dependence** tab provides the simulation temperature,  $T_s$ , and the temperature-scaling coefficients for the other device parameters. For more information, see “Temperature Dependence Tab (Surface-Potential-Based Variant)” on page 1-1081.

## Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and select the appropriate block variant:

- For a model based on threshold voltage and with exposed thermal port, select **Simscape > Block choices > Threshold-based with thermal**.
- For a thermal variant of the model based on surface potential, select **Simscape > Block choices > Surface-potential-based with thermal**.

This action displays the thermal port H on the block icon, and adds the **Thermal Port** tab to the block dialog box.

Use the thermal port to simulate the effects of generated heat and device temperature. For more information on using thermal ports and on the **Thermal Port** tab parameters, see “Simulating Thermal Effects in Semiconductors”.

## Basic Assumptions and Limitations

When modeling temperature dependence for threshold-based block variant, consider the following:

- The block does not account for temperature-dependent effects on the junction capacitances.
- When you specify  $R_{DS(on)}$  at a second measurement temperature, it must be quoted for the same working point (that is, the same drain current and gate-source voltage) as for the other  $R_{DS(on)}$  value. Inconsistent values for  $R_{DS(on)}$  at the higher temperature will result in unphysical values for  $\alpha$  and unrepresentative simulation results. Typically  $R_{DS(on)}$  increases by a factor of about 1.5 for a hundred degree increase in temperature.
- You may need to tune the values of  $BEX$  and threshold voltage,  $V_{th}$ , to replicate the  $I_{DS}-V_{GS}$  relationship (if available) for a given device. Increasing  $V_{th}$  moves the  $I_{DS}-V_{GS}$  plots to the right. The value of  $BEX$  affects whether the  $I_{DS}-V_{GS}$  curves for different temperatures cross each other, or not, for the ranges of  $V_{DS}$  and  $V_{GS}$  considered.

Therefore, an inappropriate value can result in the different temperature curves appearing to be reordered. Quoting  $R_{DS(on)}$  values for higher currents, preferably close to the current at which it will operate in your circuit, will reduce sensitivity to the precise value of  $BEX$ .

## Parameters

- “Main Tab (Threshold-Based Variant)” on page 1-1071
- “Main Tab (Surface-Potential-Based Variant)” on page 1-1073
- “Ohmic Resistance Tab” on page 1-1074
- “Junction Capacitance Tab” on page 1-1075
- “Channel Capacitances Tab” on page 1-1078
- “Body Diode Tab” on page 1-1078
- “Temperature Dependence Tab (Threshold-Based Variant)” on page 1-1079
- “Temperature Dependence Tab (Surface-Potential-Based Variant)” on page 1-1081

### Main Tab (Threshold-Based Variant)

This configuration of the **Main** tab corresponds to the threshold-based block variant, which is the default. If you are using the surface-potential-based variant of the block, see “Main Tab (Surface-Potential-Based Variant)” on page 1-1073.

#### Parameterization

Select one of the following methods for block parameterization:

- **Specify from a datasheet** — Provide the drain-source on resistance and the corresponding drain current and gate-source voltage. The block calculates the transistor gain for the Shichman and Hodges equations from this information. This is the default method.
- **Specify using equation parameters directly** — Provide the transistor gain.

#### Drain-source on resistance, $R_{DS(on)}$

The ratio of the drain-source voltage to the drain current for specified values of drain current and gate-source voltage.  $R_{DS(on)}$  should have a positive value. This parameter

is only visible when you select **Specify** from a datasheet for the **Parameterization** parameter. The default value is  $0.167 \Omega$ .

#### **Drain current, $I_{DS}$ , for $R_{DS(on)}$**

The drain current the block uses to calculate the value of the drain-source resistance.  $I_{DS}$  should have a negative value. This parameter is only visible when you select **Specify** from a datasheet for the **Parameterization** parameter. The default value is  $-2.5 \text{ A}$ .

#### **Gate-source voltage, $V_{GS}$ , for $R_{DS(on)}$**

The gate-source voltage the block uses to calculate the value of the drain-source resistance.  $V_{GS}$  should have a negative value. This parameter is only visible when you select **Specify** from a datasheet for the **Parameterization** parameter. The default value is  $-4.5 \text{ V}$ .

#### **Gain, $K$**

Positive constant gain coefficient for the Shichman and Hodges equations. This parameter is only visible when you select **Specify** using equation parameters directly for the **Parameterization** parameter. The default value is  $2 \text{ A/V}^2$ .

#### **Gate-source threshold voltage, $V_{th}$**

Gate-source threshold voltage  $V_{th}$  in the Shichman and Hodges equations. For an enhancement device,  $V_{th}$  should be negative. For a depletion mode device,  $V_{th}$  should be positive. The default value is  $-1.4 \text{ V}$ .

#### **Channel modulation, $L$**

The channel-length modulation, usually denoted by the mathematical symbol  $\lambda$ . When in the saturated region, it is minus the rate of change of drain current with drain-source voltage. The effect on drain current is typically small, and the effect is neglected if calculating transistor gain  $K$  from drain-source on-resistance,  $R_{DS(on)}$ . A typical value is  $0.02$ , but the effect can be ignored in most circuit simulations. However, in some circuits a small nonzero value may help numerical convergence. The default value is  $0 \text{ 1/V}$ .

#### **Measurement temperature**

Temperature  $T_{m1}$  at which **Drain-source on resistance,  $R_{DS(on)}$**  is measured. The default value is  $25 \text{ }^\circ\text{C}$ .

## Main Tab (Surface-Potential-Based Variant)

This configuration of the **Main** tab corresponds to the surface-potential-based block variant. If you are using the threshold-based variant of the block, based on the Shichman and Hodges equations, see “Main Tab (Threshold-Based Variant)” on page 1-1071.

### Gain

The MOSFET gain,  $\beta$ . This parameter primarily defines the linear region of operation on an  $I_D$ - $V_{DS}$  characteristic. The value must be greater than 0. The default value is 18 A/V<sup>2</sup>.

### Flatband voltage

The flatband voltage,  $V_{FB}$ , defines the gate bias that must be applied in order to achieve the flatband condition at the surface of the silicon. The default value is -1.1 V. You can also use this parameter to arbitrarily shift the threshold voltage due to material work function differences, and to trapped interface or oxide charges. In practice, however, it is usually recommended to modify the threshold voltage by using the **Body factor** and **Surface potential at strong inversion** parameters first, and only use this parameter for fine-tuning.

### Body factor

Body factor,  $\gamma$ , in the surface-potential equation. This parameter primarily impacts the threshold voltage. The default value is 3.5 V<sup>1/2</sup>.

### Surface potential at strong inversion

The  $2\phi_B$  term in the surface-potential equation. This parameter also primarily impacts the threshold voltage. The default value is 1 V.

### Velocity saturation factor

Velocity saturation,  $\theta_{sat}$ , in the drain-current equation. Use this parameter in cases where a good fit to linear operation leads to a saturation current that is too high. By increasing this parameter value, you reduce the saturation current. For high-voltage devices, it is often the case that a good fit to linear operation leads to a saturation current that is too low. In such a case, either increase both the gain and the drain ohmic resistance or use a P-Channel LDMOS FET block instead. The default value is 0.4 1/V.

### Channel-length modulation factor

The factor,  $\alpha$ , multiplying the logarithmic term in the  $G_{AL}$  equation. This parameter describes the onset of channel-length modulation. For device characteristics that exhibit a positive conductance in saturation, increase the parameter value to fit this

behavior. The default value is 0, which means that channel-length modulation is off by default.

### **Channel-length modulation voltage**

The voltage  $V_p$  in the  $G_{\Delta L}$  equation. This parameter controls the drain-voltage at which channel-length modulation starts to become active. The default value is 50 mV.

### **Surface roughness scattering factor**

Indicates the strength of the mobility reduction. The mobility is  $\mu = \mu_0/G_{mob}$ , where  $\mu_0$  is the low-field mobility without the effect of surface scattering. The mobility

reduction factor,  $G_{mob}$ , is given by  $G_{mob} = \sqrt{1 + (\theta_{sr} V_{eff})^2}$ , where  $\theta_{sr}$  is the surface roughness scattering factor and  $V_{eff}$  is a voltage that is indicative of the effective vertical electric field strength in the channel,  $E_{eff}$ . For high vertical electric fields, the mobility is roughly proportional to  $E_{eff}$  for holes. The default parameter value is 0 1/V.

### **Linear-to-saturation transition coefficient**

This parameter controls how smoothly the MOSFET transitions from linear into saturation, particularly when velocity saturation is enabled. This parameter can usually be left at its default value, but you can use it to fine-tune the knee of the  $I_D$ - $V_{DS}$  characteristic. The expected range for this parameter value is between 2 and 8. The default value is 8.

### **Measurement temperature**

Temperature  $T_{m1}$  at which the block parameters are measured. If the **Device simulation temperature** parameter on the **Temperature Dependence** tab differs from this value, then device parameters will be scaled from their defined values according to the simulation and reference temperatures. For more information, see “Temperature Dependence Tab (Surface-Potential-Based Variant)” on page 1-1081. The default value is 25 °C.

## **Ohmic Resistance Tab**

### **Source ohmic resistance**

The transistor source resistance, that is, the series resistance associated with the source contact. The value must be greater than or equal to 0. The default value for threshold-based variants is 1e-4  $\Omega$ . The default value for surface-potential-based variants is 2e-3  $\Omega$ .



**Drain ohmic resistance**

The transistor drain resistance, that is, the series resistance associated with the drain contact. The value must be greater than or equal to 0. The default value for threshold-based variants is 0.01  $\Omega$ . The default value for surface-potential-based variants is 0.17  $\Omega$ .

**Gate ohmic resistance**

The transistor gate resistance, that is, the series resistance associated with the gate contact. This parameter is visible only for the surface-potential-based block variants. The value must be greater than or equal to 0. The default value is 8.4  $\Omega$ .

**Junction Capacitance Tab**

This tab is visible only for the threshold-based variant of the block.

**Parameterization**

Select one of the following methods for capacitance parameterization:

- **Specify fixed input, reverse transfer and output capacitance** — Provide fixed parameter values from datasheet and let the block convert the input, output, and reverse transfer capacitance values to junction capacitance values, as described in “Charge Model for Threshold-Based Variant” on page 1-1064. This is the default method.
- **Specify fixed gate-source, gate-drain and drain-source capacitance** — Provide fixed values for junction capacitance parameters directly.
- **Specify tabulated input, reverse transfer and output capacitance** — Provide tabulated capacitance and source-drain voltage values based on datasheet plots. The block converts the input, output, and reverse transfer capacitance values to junction capacitance values, as described in “Charge Model for Threshold-Based Variant” on page 1-1064.
- **Specify tabulated gate-source, gate-drain and drain-source capacitance** — Provide tabulated values for junction capacitances and source-drain voltage.

**Input capacitance, Ciss**

The gate-source capacitance with the drain shorted to the source. This parameter is visible only for the following two values for the **Parameterization** parameter:

- If you select **Specify fixed input, reverse transfer and output capacitance**, the default value is 182 pF.

- If you select Specify tabulated input, reverse transfer and output capacitance, the default value is [225 210 200 185 175 170] pF.

### **Reverse transfer capacitance, Crss**

The drain-gate capacitance with the source connected to ground. This parameter is visible only for the following two values for the **Parameterization** parameter:

- If you select Specify fixed input, reverse transfer and output capacitance, the default value is 24 pF.
- If you select Specify tabulated input, reverse transfer and output capacitance, the default value is [75 60 50 35 25 20] pF.

### **Output capacitance, Coss**

The drain-source capacitance with the gate and source shorted. This parameter is visible only for the following two values for the **Parameterization** parameter:

- If you select Specify fixed input, reverse transfer and output capacitance, the default value is 0 pF.
- If you select Specify tabulated input, reverse transfer and output capacitance, the default value is [180 160 125 80 60 45] pF.

### **Gate-source junction capacitance**

The value of the capacitance placed between the gate and the source. This parameter is visible only for the following two values for the **Parameterization** parameter:

- If you select Specify fixed gate-source, gate-drain and drain-source capacitance, the default value is 158 pF.
- If you select Specify tabulated gate-source, gate-drain and drain-source capacitance, the default value is [150 150 150 150 150 150] pF.

### **Gate-drain junction capacitance**

The value of the capacitance placed between the gate and the drain. This parameter is visible only for the following two values for the **Parameterization** parameter:

- If you select Specify fixed gate-source, gate-drain and drain-source capacitance, the default value is 24 pF.
- If you select Specify tabulated gate-source, gate-drain and drain-source capacitance, the default value is [75 60 50 35 25 20] pF.

### Drain-source junction capacitance

The value of the capacitance placed between the drain and the source. This parameter is visible only for the following two values for the **Parameterization** parameter:

- If you select **Specify fixed gate-source, gate-drain and drain-source capacitance**, the default value is 0 pF.
- If you select **Specify tabulated gate-source, gate-drain and drain-source capacitance**, the default value is [105 100 75 45 35 25] pF.

### Corresponding source-drain voltages

The source-drain voltages corresponding to the tabulated capacitance values. This parameter is visible only for tabulated capacitance models (**Specify tabulated input, reverse transfer and output capacitance** or **Specify tabulated gate-source, gate-drain and output capacitance**). The default value is [0.1 0.3 1 3 10 30] V.

### Gate-source voltage, $V_{gs}$ , for tabulated capacitances

For tabulated capacitance models, this parameter controls the voltage dependence of the **Reverse transfer capacitance,  $C_{rss}$**  or the **Gate-drain junction capacitance** parameter (depending on the selected parameterization option). These capacitances are a function of the drain-gate voltage. The block calculates drain-gate voltages by subtracting this gate-source voltage value from the negative of the values specified for the **Corresponding source-drain voltages** parameter ( $V_{dg} = -V_{sd} - V_{gs}$ ). The default value is 0 V. This parameter is visible only for tabulated capacitance models (**Specify tabulated input, reverse transfer and output capacitance** or **Specify tabulated gate-source, gate-drain and output capacitance**).

### Charge-voltage linearity

The two fixed capacitance options (**Specify fixed input, reverse transfer and output capacitance** or **Specify fixed gate-source, gate-drain and drain-source capacitance**) let you model gate junction capacitance as a fixed gate-source capacitance  $C_{GS}$  and either a fixed or a nonlinear gate-drain capacitance  $C_{GD}$ . Select whether the gate-drain capacitance is fixed or nonlinear:

- **Gate-drain capacitance is constant** — The capacitance value is constant and defined according to the selected parameterization option, either directly or derived from a datasheet. This is the default method.
- **Gate-drain charge function is nonlinear** — The gate-drain charge relationship is defined according to the piecewise-nonlinear function described in

“Charge Model for Threshold-Based Variant” on page 1-1064. Two additional parameters appear to let you define the gate-drain charge function.

### **Gate-drain oxide capacitance**

The gate-drain capacitance when the drain-gate voltage is less than the **Drain-gate voltage at which oxide capacitance becomes active** parameter value. This parameter is only visible when you select Gate-drain charge function is nonlinear for the **Charge-voltage linearity** parameter. The default value is 200 pF.

### **Drain-gate voltage at which oxide capacitance becomes active**

The drain-gate voltage at which the drain-gate capacitance switches between off-state ( $C_{GD}$ ) and on-state ( $C_{ox}$ ) capacitance values. This parameter is only visible when you select Gate-drain charge function is nonlinear for the **Charge-voltage linearity** parameter. The default value is 0.5 V.

## **Channel Capacitances Tab**

This tab is visible only for the surface-potential-based variant of the block.

### **Oxide capacitance**

The parallel plate gate-channel capacitance. The default value is 1500 pF.

### **Gate-source overlap capacitance**

The fixed, linear capacitance associated with the overlap of the gate electrode with the source well. The default value is 100 pF.

### **Gate-drain overlap capacitance**

The fixed, linear capacitance associated with the overlap of the gate electrode with the drain well. The default value is 14 pF.

## **Body Diode Tab**

### **Reverse saturation current**

The current designated by the  $I_s$  symbol in the body-diode equations. The default value for threshold-based variant is 0 A. The default value for surface-potential-based variant is  $5.2e-13$  A.

### **Built-in voltage**

The built-in voltage of the diode, designated by the  $V_{bi}$  symbol in the body-diode equations. Built-in voltage has an impact only on the junction capacitance equation. It does not affect the conduction current. The default value is 0.6 V.

**Ideality factor**

The factor designated by the  $n$  symbol in the body-diode equations. The default value is 1.

**Zero-bias junction capacitance**

The capacitance between the drain and bulk contacts at zero-bias due to the body diode alone. It is designated by the  $C_{j0}$  symbol in the body-diode equations. The default value for threshold-based variant is 0 pF. The default value for surface-potential-based variant is 480 pF.

**Transit time**

The time designated by the  $\tau$  symbol in the body-diode equations. The default value is 50 ns.

**Temperature Dependence Tab (Threshold-Based Variant)**

This configuration of the **Temperature Dependence** tab corresponds to the threshold-based block variant, which is the default. If you are using the surface-potential-based variant of the block, see “Temperature Dependence Tab (Surface-Potential-Based Variant)” on page 1-1081

**Parameterization**

Select one of the following methods for temperature dependence parameterization:

- **None** – Simulate at parameter measurement temperature – Temperature dependence is not modeled. This is the default method.
- **Model temperature dependence** – Model temperature-dependent effects. Provide a value for simulation temperature,  $T_s$ , a value for  $BEX$ , and a value for the measurement temperature  $T_{m1}$  (using the **Measurement temperature** parameter on the **Main** tab). You also have to provide a value for  $\alpha$  using one of two methods, depending on the value of the **Parameterization** parameter on the **Main** tab. If you parameterize the block from a datasheet, you have to provide  $R_{DS(on)}$  at a second measurement temperature, and the block will calculate  $\alpha$  based on that. If you parameterize by specifying equation parameters, you have to provide the value for  $\alpha$  directly.

**Drain-source on resistance, R\_DS(on), at second measurement temperature**

The ratio of the drain-source voltage to the drain current for specified values of drain current and gate-source voltage at second measurement temperature. This parameter is only visible when you select **Specify** from a datasheet for the

**Parameterization** parameter on the **Main** tab. It must be quoted for the same working point (drain current and gate-source voltage) as the **Drain-source on resistance, R\_DS(on)** parameter on the **Main** tab. The default value is  $0.25 \Omega$ .

### Second measurement temperature

Second temperature  $T_{m2}$  at which **Drain-source on resistance, R\_DS(on), at second measurement temperature** is measured. This parameter is only visible when you select **Specify** from a datasheet for the **Parameterization** parameter on the **Main** tab. The default value is  $125 \text{ }^\circ\text{C}$ .

### Gate threshold voltage temperature coefficient, dVth/dT

The rate of change of gate threshold voltage with temperature. This parameter is only visible when you select **Specify using equation parameters directly** for the **Parameterization** parameter on the **Main** tab. The default value is  $2 \text{ mV/K}$ .

### Mobility temperature exponent, BEX

Mobility temperature coefficient value. You can use the default value for most MOSFETs. See the “Basic Assumptions and Limitations” on page 1-1070 section for additional considerations. The default value is  $-1.5$ .

### Body diode reverse saturation current temperature exponent

The reverse saturation current for the body diode is assumed to be proportional to the square of the intrinsic carrier concentration,  $n_i = N_C \exp(-E_G/2k_B T)$ .  $N_C$  is the temperature-dependent effective density of states and  $E_G$  is the temperature-dependent bandgap for the semiconductor material. To avoid introducing another temperature-scaling parameter, the block neglects the temperature dependence of the bandgap and uses the bandgap of silicon at 300K (1.12eV) for all device types. Therefore, the temperature-scaled reverse saturation current is given by

$$I_s = I_{s,m1} \left( \frac{T_s}{T_{m1}} \right)^{\eta_{I_s}} \cdot \exp \left( \frac{E_G}{k_B} \cdot \left( \frac{1}{T_{m1}} - \frac{1}{T_s} \right) \right)$$

$I_{s,m1}$  is the value of the **Reverse saturation current** parameter from the **Body Diode** tab,  $k_B$  is Boltzmann’s constant ( $8.617 \times 10^{-5} \text{ eV/K}$ ), and  $\eta_{I_s}$  is the **Body diode reverse saturation current temperature exponent**. The default value is 3, because  $N_C$  for silicon is roughly proportional to  $T^{3/2}$ . You can remedy the effect of neglecting the temperature-dependence of the bandgap by a pragmatic choice of  $\eta_{I_s}$ .

### Device simulation temperature

Temperature  $T_s$  at which the device is simulated. The default value is  $25 \text{ }^\circ\text{C}$ .

## Temperature Dependence Tab (Surface-Potential-Based Variant)

This configuration of the **Temperature Dependence** tab corresponds to the surface-potential-based block variant. If you are using the threshold-based variant of the block, see “Temperature Dependence Tab (Threshold-Based Variant)” on page 1-1079

### Parameterization

Select one of the following methods for temperature dependence parameterization:

- None – Simulate at parameter measurement temperature – Temperature dependence is not modeled. This is the default method.
- Model temperature dependence – Model temperature-dependent effects. Provide a value for the device simulation temperature,  $T_s$ , and the temperature-scaling coefficients for other block parameters.

### Gain temperature exponent

The MOSFET gain,  $\beta$ , is assumed to scale exponentially with temperature,  $\beta = \beta_{m1}(T_{m1}/T_s)^{\eta_\beta}$ .  $\beta_{m1}$  is the value of the **Gain** parameter from the **Main** tab and  $\eta_\beta$  is the **Gain temperature exponent**. The default value is 1.3.

### Flatband voltage temperature coefficient

The flatband voltage,  $V_{FB}$ , is assumed to scale linearly with temperature,  $V_{FB} = V_{FBm1} + (T_s - T_{m1})S_{TV_{FB}}$ .  $V_{FBm1}$  is the value of the **Flatband voltage** parameter from the **Main** tab and  $S_{TV_{FB}}$  is the **Flatband voltage temperature coefficient**. The default value is  $5e-4$  V/K.

### Surface potential at strong inversion temperature coefficient

The surface potential at strong inversion,  $2\phi_B$ , is assumed to scale linearly with temperature,  $2\phi_B = 2\phi_{Bm1} + (T_s - T_{m1})S_{T\phi_B}$ .  $2\phi_{Bm1}$  is the value of the **Surface potential at strong inversion** parameter from the **Main** tab and  $S_{T\phi_B}$  is the **Surface potential at strong inversion temperature coefficient**. The default value is  $-8.5e-4$  V/K.

### Velocity saturation temperature exponent

The velocity saturation,  $\theta_{sat}$ , is assumed to scale exponentially with temperature,  $\theta_{sat} = \theta_{sat,m1}(T_{m1}/T_s)^{\eta_\theta}$ .  $\theta_{sat,m1}$  is the value of the **Velocity saturation factor** parameter from the **Main** tab and  $\eta_\theta$  is the **Velocity saturation temperature exponent**. The default value is 1.04.

### Surface roughness scattering temperature exponent

This parameter leads to a temperature-dependent reduction in the MOSFET transconductance at high gate voltage. The surface roughness scattering,  $\theta_{sr}$ , is assumed to scale exponentially with temperature,  $\theta_{sr} = \theta_{sr,m1}(T_{m1}/T_s)^{\eta_{sr}}$ .  $\theta_{sr,m1}$  is the value of the **Surface roughness scattering factor** parameter from the **Main** tab and  $\eta_{sr}$  is the **Surface roughness scattering temperature exponent**. The default value is 0.65.

### Resistance temperature exponent

The series resistances are assumed to correspond to semiconductor resistances. Therefore, they decrease exponentially with increasing temperature.  $R_i = R_{i,m1}(T_{m1}/T_s)^{\eta_R}$ , where  $i$  is S, D, or G, for the source, drain, or gate series resistance, respectively.  $R_{i,m1}$  is the value of the corresponding series resistance parameter from the **Ohmic Resistance** tab and  $\eta_R$  is the **Resistance temperature exponent**. The default value is 0.95.

### Body diode reverse saturation current temperature exponent

The reverse saturation current for the body diode is assumed to be proportional to the square of the intrinsic carrier concentration,  $n_i = N_C \exp(-E_G/2k_B T)$ .  $N_C$  is the temperature-dependent effective density of states and  $E_G$  is the temperature-dependent bandgap for the semiconductor material. To avoid introducing another temperature-scaling parameter, the block neglects the temperature dependence of the bandgap and uses the bandgap of silicon at 300K (1.12eV) for all device types. Therefore, the temperature-scaled reverse saturation current is given by

$$I_s = I_{s,m1} \left( \frac{T_s}{T_{m1}} \right)^{\eta_{Is}} \cdot \exp \left( \frac{E_G}{k_B} \cdot \left( \frac{1}{T_{m1}} - \frac{1}{T_s} \right) \right).$$

$I_{s,m1}$  is the value of the **Reverse saturation current** parameter from the **Body Diode** tab,  $k_B$  is Boltzmann's constant (8.617x10-5eV/K), and  $\eta_{Is}$  is the **Body diode reverse saturation current temperature exponent**. The default value is 3, because  $N_C$  for silicon is roughly proportional to  $T^{3/2}$ . You can remedy the effect of neglecting the temperature-dependence of the bandgap by a pragmatic choice of  $\eta_{Is}$ .

### Device simulation temperature

Temperature  $T_s$  at which the device is simulated. The default value is 25 °C.



## Ports

The block has the following ports:

G

Electrical conserving port associated with the transistor gate terminal

D

Electrical conserving port associated with the transistor drain terminal

S

Electrical conserving port associated with the transistor source terminal

## References

- [1] Shichman, H. and D. A. Hodges. "Modeling and simulation of insulated-gate field-effect transistor switching circuits." *IEEE J. Solid State Circuits*. SC-3, 1968.
- [2] Van Langevelde, R., A. J. Scholten, and D. B .M. Klaassen. "Physical Background of MOS Model 11. Level 1101." *Nat.Lab. Unclassified Report 2003/00239*. April 2003.
- [3] Oh, S-Y., D. E. Ward, and R. W. Dutton. "Transient analysis of MOS transistors." *IEEE J. Solid State Circuits*. SC-15, pp. 636-643, 1980.

## See Also

N-Channel MOSFET

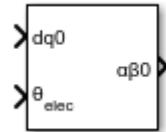
## Topics

Interactive Generation of MOSFET Characteristics

## Park to Clarke Angle Transform

Implement  $dq0$  to  $\alpha\beta0$  transform

**Library:** Simscape / Electrical / Power Systems / Control /  
Mathematical Transforms

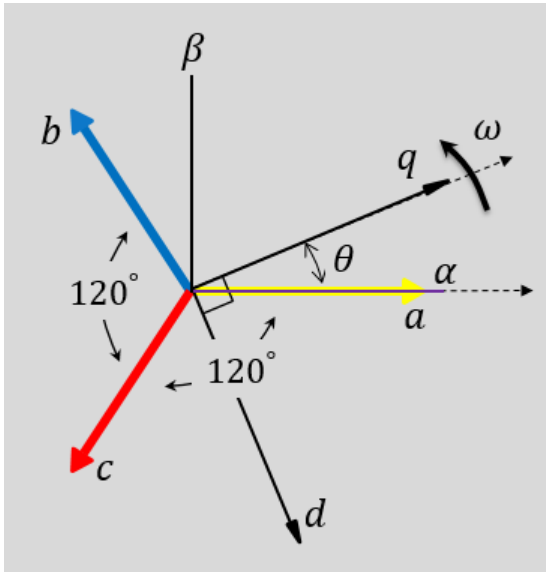


### Description

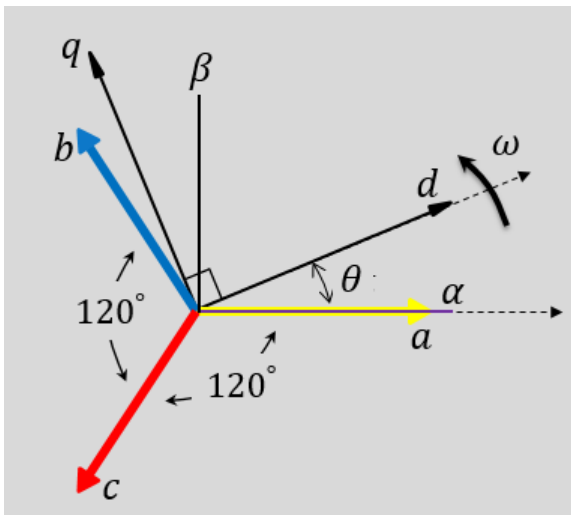
The Park to Clarke Angle Transform block converts the direct, quadrature, and zero components in a rotating reference frame to alpha, beta, and zero components in a stationary reference frame. For balanced systems, the zero components are equal to zero.

You can configure the block to align the phase  $a$ -axis of the three-phase system to either the  $q$ - or  $d$ -axis of the rotating reference frame at time,  $t = 0$ . The figures show the direction of the magnetic axes of the stator windings in the three-phase system, a stationary  $\alpha\beta0$  reference frame, and a rotating  $dq0$  reference frame where:

- The  $a$ -axis and the  $q$ -axis are initially aligned.



- The  $a$ -axis and the  $d$ -axis are initially aligned.

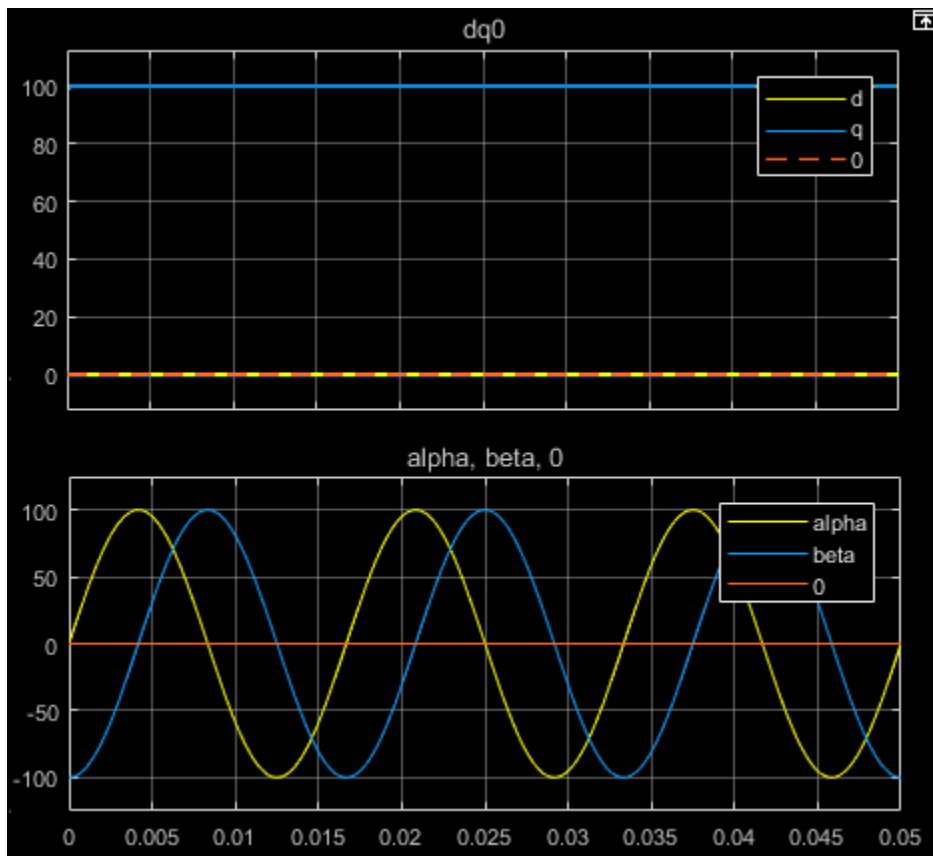


In both cases, the angle  $\theta = \omega t$ , where

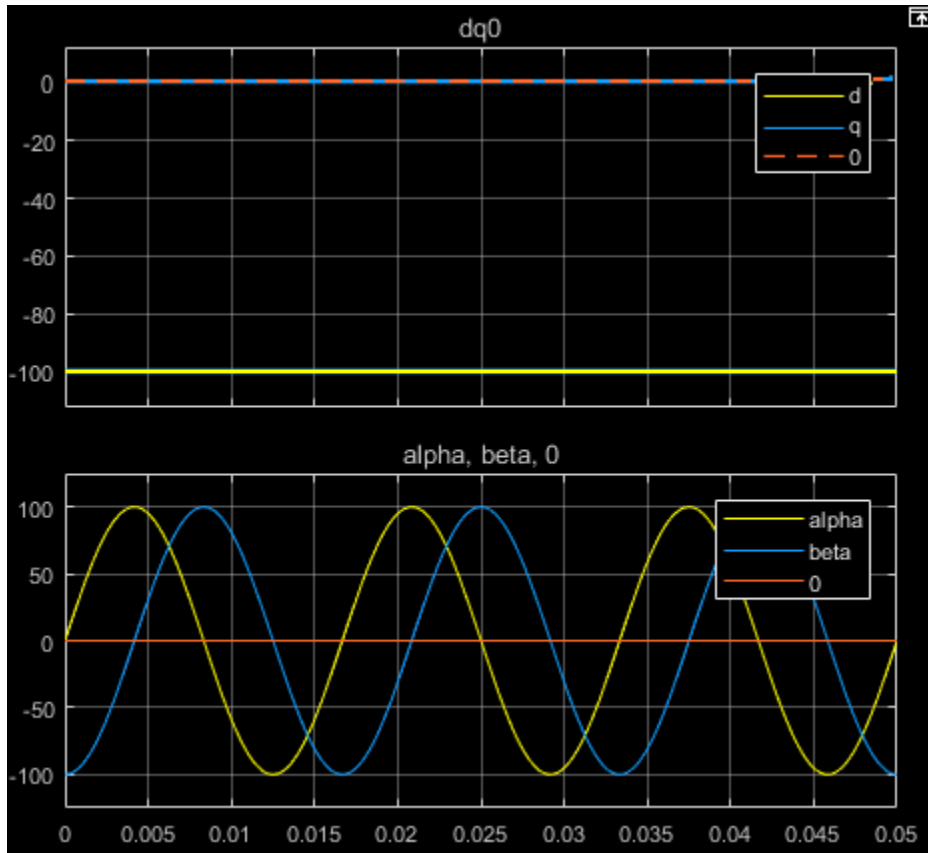
- $\theta$  is the angle between the  $a$  and  $q$  axes for the  $q$ -axis alignment or the angle between the  $a$  and  $d$  axes for the  $d$ -axis alignment.
- $\omega$  is the rotational speed of the  $d$ - $q$  reference frame.
- $t$  is the time, in s, from the initial alignment.

The figures show the time-response of the individual components of equivalent balanced  $dq0$  and  $\alpha\beta 0$  for an:

- Alignment of the  $a$ -phase vector to the  $q$ -axis



- Alignment of the  $a$ -phase vector to the  $d$ -axis



## Equations

The Park to Clarke Angle Transform block implements the transform for an  $a$ -phase to  $q$ -axis alignment as

$$\begin{bmatrix} \alpha \\ \beta \\ 0 \end{bmatrix} = \begin{bmatrix} \sin(\theta) & \cos(\theta) & 0 \\ -\cos(\theta) & \sin(\theta) & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} d \\ q \\ 0 \end{bmatrix}$$

where:

- $d$  and  $q$  are the direct-axis and quadrature-axis components of the two-axis system in the rotating reference frame.
- $0$  is the zero component.
- $\alpha$  and  $\beta$  are the alpha-axis and beta-axis components of the two-phase system in the stationary reference frame.

For an  $a$ -phase to  $d$ -axis alignment, the block implements the transform using this equation:

$$\begin{bmatrix} \alpha \\ \beta \\ 0 \end{bmatrix} = \begin{bmatrix} \cos(\theta) & -\sin(\theta) & 0 \\ \sin(\theta) & \cos(\theta) & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} d \\ q \\ 0 \end{bmatrix}$$

## Ports

### Input

#### **$dq0$ — $d$ - $q$ axis and zero components**

vector

Direct-axis and quadrature-axis components and the zero component of the system in the rotating reference frame.

Data Types: `single` | `double`

#### **$\theta_{abc}$ — Rotational angle**

scalar | in radians

Angular position of the rotating reference frame. The value of this parameter is equal to the polar distance from the vector of the  $a$ -phase in the  $abc$  reference frame to the initially aligned axis of the  $dq0$  reference frame.

Data Types: `single` | `double`

### Output

#### **$\alpha\beta0$ — $\alpha$ - $\beta$ axis and zero components**

vector

Alpha-axis component,  $\alpha$ , beta-axis component,  $\beta$ , and zero component of the two-phase system in the stationary reference frame.

Data Types: `single` | `double`

## Parameters

### Phase-a axis alignment — $dq0$ reference frame alignment

Q-axis (default) | D-axis

Align the  $a$ -phase vector of the  $abc$  reference frame to the  $d$ - or  $q$ -axis of the rotating reference frame.

## References

[1] Krause, P., O. Wasynczuk, S. D. Sudhoff, and S. Pekarek. *Analysis of Electric Machinery and Drive Systems*. Piscatawy, NJ: Wiley-IEEE Press, 2013.

## See Also

### Blocks

[Clarke Transform](#) | [Clarke to Park Angle Transform](#) | [Inverse Clarke Transform](#) | [Inverse Park Transform](#) | [Park Transform](#)

**Introduced in R2017b**

## Park Transform

Implement  $abc$  to  $dq0$  transform

**Library:** Simscape / Electrical / Power Systems / Control /  
Mathematical Transforms



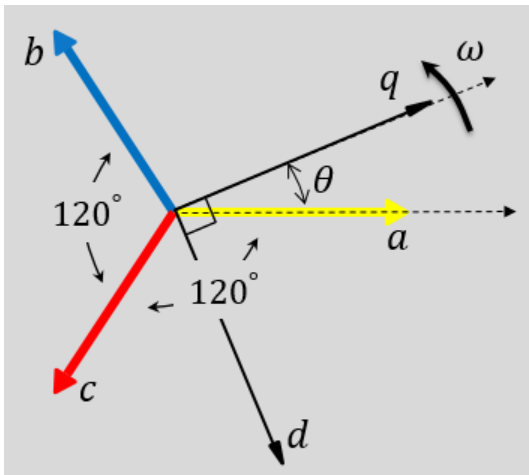
## Description

The Park Transform block converts the time-domain components of a three-phase system in an  $abc$  reference frame to direct, quadrature, and zero components in a rotating reference frame. The block can preserve the active and reactive powers with the powers of the system in the  $abc$  reference frame by implementing an invariant version of the Park transform. For a balanced system, the zero component is equal to zero.

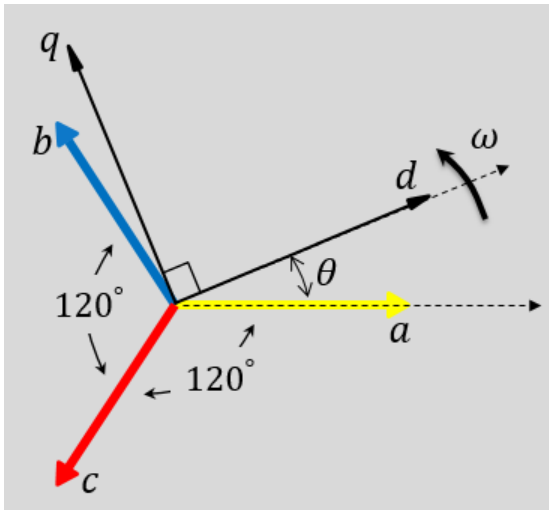
You can configure the block to align the  $a$ -axis of the three-phase system to either the  $d$ - or  $q$ -axis of the rotating reference frame at time,  $t = 0$ . The figures show the direction of the magnetic axes of the stator windings in an  $abc$  reference frame and a rotating  $dq0$  reference frame where:

- The  $a$ -axis and the  $q$ -axis are initially aligned.





- The  $a$ -axis and the  $d$ -axis are initially aligned.



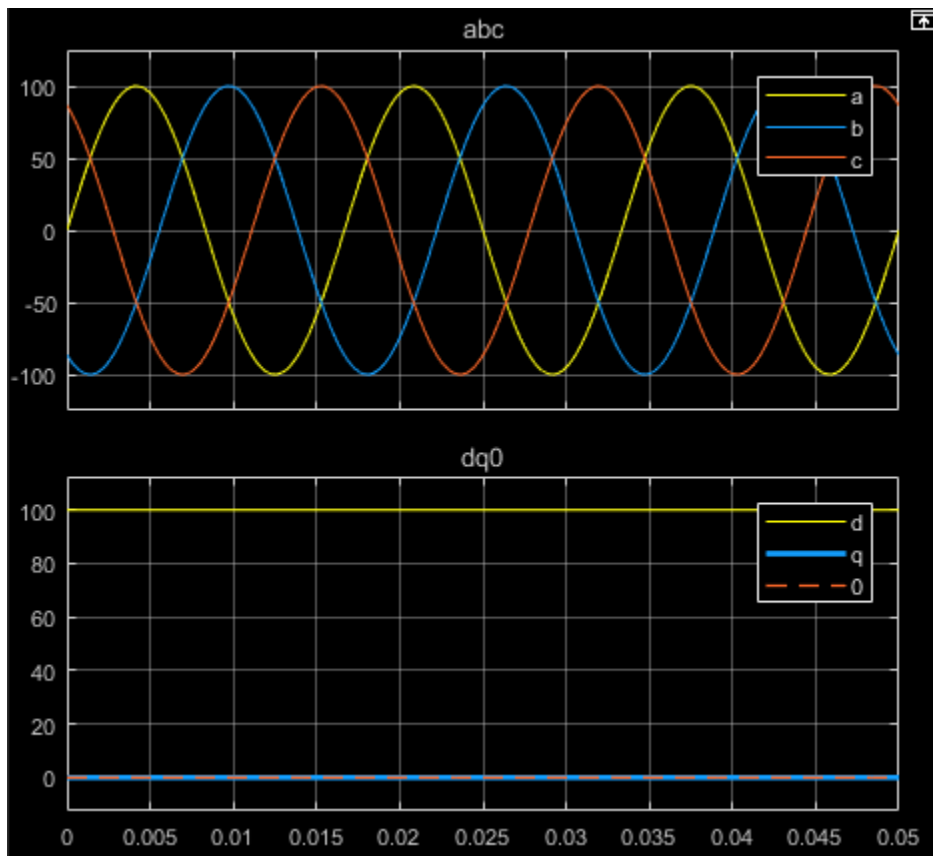
In both cases, the angle  $\theta = \omega t$ , where:

- $\theta$  is the angle between the  $a$  and  $q$  axes for the  $q$ -axis alignment or the angle between the  $a$  and  $d$  axes for the  $d$ -axis alignment.
- $\omega$  is the rotational speed of the  $d$ - $q$  reference frame.

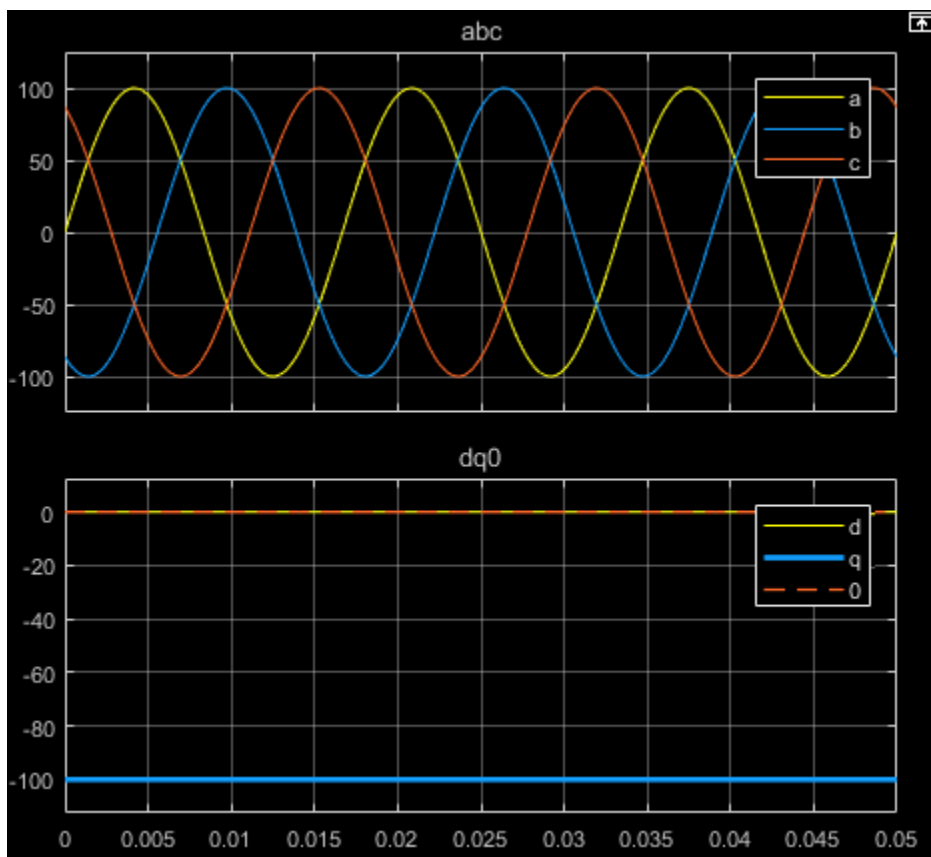
- $t$  is the time, in s, from the initial alignment.

The figures show the time-response of the individual components of equivalent balanced  $abc$  and  $dq0$  for an:

- Alignment of the  $a$ -phase vector to the  $q$ -axis



- Alignment of the  $a$ -phase vector to the  $d$ -axis



## Equations

The Park Transform block implements the transform for an  $a$ -phase to  $q$ -axis alignment as

$$\begin{bmatrix} d \\ q \\ 0 \end{bmatrix} = \frac{2}{3} \begin{bmatrix} \sin(\theta) & \sin(\theta - \frac{2\pi}{3}) & \sin(\theta + \frac{2\pi}{3}) \\ \cos(\theta) & \cos(\theta - \frac{2\pi}{3}) & \cos(\theta + \frac{2\pi}{3}) \\ \frac{1}{2} & \frac{1}{2} & \frac{1}{2} \end{bmatrix} \begin{bmatrix} a \\ b \\ c \end{bmatrix},$$

where:

- $a$ ,  $b$ , and  $c$  are the components of the three-phase system in the  $abc$  reference frame.
- $d$  and  $q$  are the components of the two-axis system in the rotating reference frame.
- $0$  is the zero component of the two-axis system in the stationary reference frame.

For a power invariant  $a$ -phase to  $q$ -axis alignment, the block implements the transform using this equation:

$$\begin{bmatrix} d \\ q \\ 0 \end{bmatrix} = \sqrt{\frac{2}{3}} \begin{bmatrix} \sin(\theta) & \sin(\theta - \frac{2\pi}{3}) & \sin(\theta + \frac{2\pi}{3}) \\ \cos(\theta) & \cos(\theta - \frac{2\pi}{3}) & \cos(\theta + \frac{2\pi}{3}) \\ \sqrt{\frac{1}{2}} & \sqrt{\frac{1}{2}} & \sqrt{\frac{1}{2}} \end{bmatrix} \begin{bmatrix} a \\ b \\ c \end{bmatrix}.$$

For an  $a$ -phase to  $d$ -axis alignment, the block implements the transform using this equation:

$$\begin{bmatrix} d \\ q \\ 0 \end{bmatrix} = \frac{2}{3} \begin{bmatrix} \cos(\theta) & \cos(\theta - \frac{2\pi}{3}) & \cos(\theta + \frac{2\pi}{3}) \\ -\sin(\theta) & -\sin(\theta - \frac{2\pi}{3}) & -\sin(\theta + \frac{2\pi}{3}) \\ \frac{1}{2} & \frac{1}{2} & \frac{1}{2} \end{bmatrix} \begin{bmatrix} a \\ b \\ c \end{bmatrix}.$$

The block implements a power invariant  $a$ -phase to  $d$ -axis alignment as

$$\begin{bmatrix} d \\ q \\ 0 \end{bmatrix} = \sqrt{\frac{2}{3}} \begin{bmatrix} \cos(\theta) & \cos(\theta - \frac{2\pi}{3}) & \cos(\theta + \frac{2\pi}{3}) \\ -\sin(\theta) & -\sin(\theta - \frac{2\pi}{3}) & -\sin(\theta + \frac{2\pi}{3}) \\ \sqrt{\frac{1}{2}} & \sqrt{\frac{1}{2}} & \sqrt{\frac{1}{2}} \end{bmatrix} \begin{bmatrix} a \\ b \\ c \end{bmatrix}.$$

## Ports

### Input

#### **$abc$ — a-, b-, and c-phase components**

vector

Components of the three-phase system in the  $abc$  reference frame.

Data Types: single | double

#### **$\theta_{abc}$ — Rotational angle**

scalar | in radians

Angular position of the rotating reference frame. The value of this parameter is equal to the polar distance from the vector of the a-phase in the  $abc$  reference frame to the initially aligned axis of the  $dq0$  reference frame.

Data Types: single | double

### Output

#### **$dq0$ — d-q axis and zero components**

vector

Direct-axis and quadrature-axis components and the zero component of the system in the rotating reference frame.

Data Types: single | double

## Parameters

#### **Power Invariant — Power invariant transform**

off (default) | on

Option to preserve the active and reactive power of the  $abc$  reference frame.

#### **Phase-a axis alignment — $dq0$ reference frame alignment**

Q-axis (default) | D-axis

Align the  $a$ -phase vector of the  $abc$  reference frame to the  $d$ - or  $q$ -axis of the rotating reference frame.

## References

- [1] Krause, P., O. Wasynczuk, S. D. Sudhoff, and S. Pekarek. *Analysis of Electric Machinery and Drive Systems*. Piscatawy, NJ: Wiley-IEEE Press, 2013.

## See Also

### Blocks

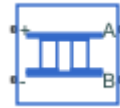
Clarke Transform | Clarke to Park Angle Transform | Inverse Clarke Transform | Inverse Park Transform | Park to Clarke Angle Transform

**Introduced in R2017b**

# Peltier Device

Converter between electrical and thermal energy

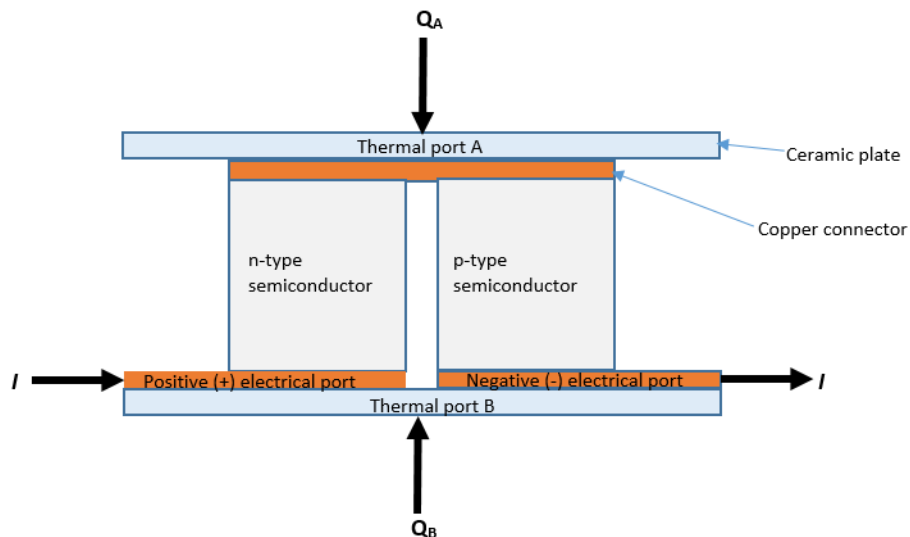
**Library:** Simscape / Electrical / Electronics and Mechatronics / Sources



## Description

The Peltier Device block represents a converter between the electrical and thermal energy:

- With no current flowing, if the temperature presented at thermal port **B** is greater than the temperature presented at thermal port **A**, then there is a positive potential difference measured from the positive to the negative electrical port.
- When the block acts as a cooling device, a positive current causes heat to flow from port **A** to port **B**, cooling port **A** relative to port **B**.



The defining equations are:

$$Q_A = \alpha T_A I - \frac{1}{2} I^2 R + K(T_A - T_B)$$

$$Q_B = -\alpha T_B I - \frac{1}{2} I^2 R + K(T_B - T_A)$$

$$W = VI$$

$$W + Q_A + Q_B = 0$$

where:

- $Q_A$  is heat flow into port **A**.
- $Q_B$  is heat flow into port **B**.
- $T_A$  is port **A** temperature.
- $T_B$  is port **B** temperature.
- $W$  is electrical power (positive when flowing into the block).
- $V$  is potential difference across the **+** and **-** ports.
- $I$  is electrical current, positive from **+** to **-** port.
- $R$  is total electrical resistance.
- $\alpha$  is Seebeck coefficient.
- $K$  is thermal conductance.

Substituting for powers and dividing all terms by current gives the electrical equation:

$$V = \alpha(T_B - T_A) + IR$$

The block has a logging variable `power_dissipated` (**Dissipated power**). This variable reports the DC electrical power, that is, the average electrical power over one AC cycle if you drive the device with an AC source. In terms of equations it is equal to the instantaneous value of  $I^2 R$  and is useful in a cooling application to indicate the nonproductive portion of the heat flow. In a heating application, the interpretation of dissipated power is less useful.



## Ports

### Conserving

**+ — Positive electrical port**  
electrical

Electrical conserving port associated with the positive terminal.

**- — Negative electrical port**  
electrical

Electrical conserving port associated with the negative terminal.

**A — Thermal port**  
thermal

Thermal conserving port. When the block acts as a cooling device, a positive current causes heat to flow from port **A** to port **B**, cooling port **A** relative to port **B**.

**B — Thermal port**  
thermal

Thermal conserving port.

## Parameters

### Electrical

**Seebeck coefficient — Thermoelectric sensitivity coefficient**  
220e-6 V/K (default)

Measure of the magnitude of an induced thermoelectric voltage in response to a temperature difference across that material.

**Total electrical resistance — Resistance between the electrical ports**  
0.02  $\Omega$  (default)

Total electrical resistance between the + and - ports.

## **Thermal**

### **Thermal conductance — Conductance between the thermal ports**

1.5e-3 W/K (default)

Thermal conductance between the **A** and **B** ports.

## **See Also**

**Introduced in R2018b**

# Permanent Magnet Synchronous Motor

Permanent magnet synchronous motor with sinusoidal flux distribution

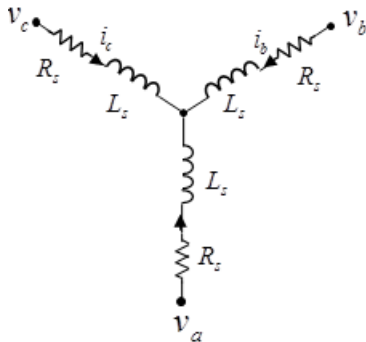


## Library

Simscape / Electrical / Power Systems / Machines / Permanent Magnet Rotor

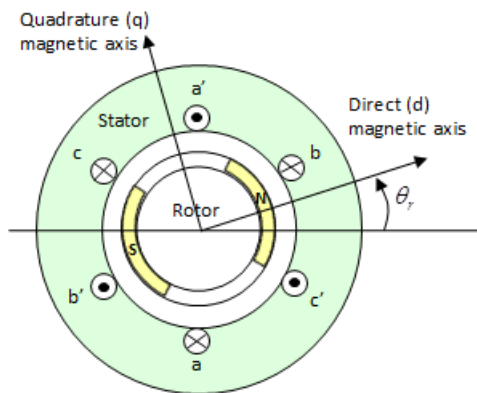
## Description

The Permanent Magnet Synchronous Motor block models a permanent magnet synchronous motor with a three-phase wye-wound stator. The figure shows the equivalent electrical circuit for the stator windings.



## Motor Construction

This figure shows the motor construction with a single pole-pair on the rotor.



Permanent magnets generate a rotor magnetic field that creates a sinusoidal rate of change of flux with rotor angle.

For the axes convention in the preceding figure, the  $a$ -phase and permanent magnet fluxes are aligned when rotor mechanical angle,  $\theta_r$ , is zero. The block supports a second rotor axis definition in which rotor mechanical angle is defined as the angle between the  $a$ -phase magnetic axis and the rotor  $q$ -axis.

## Electrical Defining Equations

Voltages across the stator windings are defined by:

$$\begin{bmatrix} v_a \\ v_b \\ v_c \end{bmatrix} = \begin{bmatrix} R_s & 0 & 0 \\ 0 & R_s & 0 \\ 0 & 0 & R_s \end{bmatrix} \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix} + \begin{bmatrix} \frac{d\psi_a}{dt} \\ \frac{d\psi_b}{dt} \\ \frac{d\psi_c}{dt} \end{bmatrix},$$

where:

- $v_a$ ,  $v_b$ , and  $v_c$  are the individual phase voltages across the stator windings.
- $R_s$  is the equivalent resistance of each stator winding.
- $i_a$ ,  $i_b$ , and  $i_c$  are the currents flowing in the stator windings.

•

$\frac{d\psi_a}{dt}$ ,  $\frac{d\psi_b}{dt}$ , and  $\frac{d\psi_c}{dt}$  are the rates of change of magnetic flux in each stator winding.

The permanent magnet and the three windings contribute to the total flux linking each winding. The total flux is defined by:

$$\begin{bmatrix} \psi_a \\ \psi_b \\ \psi_c \end{bmatrix} = \begin{bmatrix} L_{aa} & L_{ab} & L_{ac} \\ L_{ba} & L_{bb} & L_{bc} \\ L_{ca} & L_{cb} & L_{cc} \end{bmatrix} \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix} + \begin{bmatrix} \psi_{am} \\ \psi_{bm} \\ \psi_{cm} \end{bmatrix},$$

where:

- $\psi_a$ ,  $\psi_b$ , and  $\psi_c$  are the total fluxes linking each stator winding.
- $L_{aa}$ ,  $L_{bb}$ , and  $L_{cc}$  are the self-inductances of the stator windings.
- $L_{ab}$ ,  $L_{ac}$ ,  $L_{ba}$ , and so on, are the mutual inductances of the stator windings.
- $\psi_{am}$ ,  $\psi_{bm}$ , and  $\psi_{cm}$  are the permanent magnet fluxes linking the stator windings.

The inductances in the stator windings are functions of rotor electrical angle, defined by:

$$\theta_e = N \theta_r,$$

$$L_{aa} = L_s + L_m \cos(2\theta_e),$$

$$L_{bb} = L_s + L_m \cos(2(\theta_e - 2\pi/3)),$$

$$L_{cc} = L_s + L_m \cos(2(\theta_e + 2\pi/3)),$$

$$L_{ab} = L_{ba} = -M_s - L_m \cos(2(\theta_e + \pi/6)),$$

$$L_{bc} = L_{cb} = -M_s - L_m \cos(2(\theta_e + \pi/6 - 2\pi/3)),$$

and

$$L_{ca} = L_{ac} = -M_s - L_m \cos(2(\theta_e + \pi/6 + 2\pi/3)),$$

where:

- $\theta_r$  is the rotor mechanical angle.
- $\theta_e$  is the rotor electrical angle.
- $L_s$  is the stator self-inductance per phase. This value is the average self-inductance of each of the stator windings.
- $L_m$  is the stator inductance fluctuation. This value is the amplitude of the fluctuation in self-inductance and mutual inductance with changing rotor angle.
- $M_s$  is the stator mutual inductance. This value is the average mutual inductance between the stator windings.

The permanent magnet flux linking winding  $a$  is a maximum when  $\theta_e = 0^\circ$  and zero when  $\theta_e = 90^\circ$ . Therefore, the linked motor flux is defined by:

$$\begin{bmatrix} \psi_{am} \\ \psi_{bm} \\ \psi_{cm} \end{bmatrix} = \begin{bmatrix} \psi_m \cos \theta_e \\ \psi_m \cos(\theta_e - 2\pi/3) \\ \psi_m \cos(\theta_e + 2\pi/3) \end{bmatrix}.$$

where  $\psi_m$  is the permanent magnet flux linkage.

## Simplified Electrical Equations

Applying Park's transformation to the block electrical equations produces an expression for torque that is independent of the rotor angle.

Park's transformation is defined by:

$$P = 2/3 \begin{bmatrix} \cos \theta_e & \cos(\theta_e - 2\pi/3) & \cos(\theta_e + 2\pi/3) \\ -\sin \theta_e & -\sin(\theta_e - 2\pi/3) & -\sin(\theta_e + 2\pi/3) \\ 0.5 & 0.5 & 0.5 \end{bmatrix}.$$

where  $\theta_e$  is the electrical angle defined as  $N\theta_r$ ,  $N$  is the number of pole pairs.

Using Park's transformation on the stator winding voltages and currents transforms them to the dq0 frame, which is independent of the rotor angle:

$$\begin{bmatrix} v_d \\ v_q \\ v_0 \end{bmatrix} = P \begin{bmatrix} v_a \\ v_b \\ v_c \end{bmatrix}$$

and

$$\begin{bmatrix} i_d \\ i_q \\ i_0 \end{bmatrix} = P \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix}.$$

Applying Park's transformation to the first two electrical equations produces the following equations that define the block behavior:

$$v_d = R_s i_d + L_d \frac{di_d}{dt} - N \omega i_q L_q,$$

$$v_q = R_s i_q + L_q \frac{di_q}{dt} + N \omega (i_d L_d + \psi_m),$$

$$v_0 = R_s i_0 + L_0 \frac{di_0}{dt},$$

and

$$T = \frac{3}{2} N (i_q (i_d L_d + \psi_m) - i_d i_q L_q),$$

where:

- $L_d = L_s + M_s + 3/2 L_m$ .  $L_d$  is the stator  $d$ -axis inductance.
- $L_q = L_s + M_s - 3/2 L_m$ .  $L_q$  is the stator  $q$ -axis inductance.
- $L_0 = L_s - 2M_s$ .  $L_0$  is the stator zero-sequence inductance.
- $\omega$  is the rotor mechanical rotational speed.
- $N$  is the number of rotor permanent magnet pole pairs.
- $T$  is the rotor torque. Torque flows from the motor case (block physical port C) to the motor rotor (block physical port R).

The PMSM block uses the original, non-orthogonal implementation of the Park transform. If you try to apply the alternative implementation, you get different results for the dq0 voltage and currents.

## Alternative Flux Linkage Parameterization

You can parameterize the motor using the back EMF or torque constants which are more commonly given on motor datasheets by using the **Permanent magnet flux linkage** option.

The back EMF constant is defined as the peak voltage induced by the permanent magnet in each of the phases per unit rotational speed. It is related to peak permanent magnet flux linkage by:

$$k_e = N\psi_m.$$

From this definition, it follows that the back EMF  $e_{ph}$  for one phase is given by:

$$e_{ph} = k_e\omega.$$

The torque constant is defined as the peak torque induced by each of the phases per unit current. It is numerically identical in value to the back EMF constant when both are expressed in SI units:

$$k_t = N\psi_m.$$

When  $L_d=L_q$ , and when the currents in all three phases are balanced, it follows that the combined torque  $T$  is given by:

$$T = \frac{3}{2}k_t i_q = \frac{3}{2}k_t I_{pk},$$

where  $I_{pk}$  is the peak current in any of the three windings.

The factor 3/2 follows from this being the steady-state sum of the torques from all phases. Therefore the torque constant  $k_t$  could also be defined as:

$$k_t = \frac{2}{3} \left( \frac{T}{I_{pk}} \right),$$



where  $T$  is the measured total torque when testing with a balanced three-phase current with peak line voltage  $I_{pk}$ . Writing in terms of RMS line voltage:

$$k_t = \sqrt{\frac{2}{3}} \left( \frac{T}{i_{line,rms}} \right).$$

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Ports

~

Expandable three-phase port

n

Electrical conserving port associated with the neutral phase

R

Mechanical rotational conserving port associated with the motor rotor

C

Mechanical rotational conserving port associated with the motor case

## Parameters

### Main

#### Number of pole pairs

Number of permanent magnet pole pairs on the rotor. The default value is 6.

### **Permanent magnet flux linkage parameterization**

Choose Specify flux linkage, the default value, Specify torque constant, or Specify back EMF constant.

### **Permanent magnet flux linkage**

Peak permanent magnet flux linkage with any of the stator windings. This parameter is visible only if you set **Permanent magnet flux linkage** to Specify flux linkage. The default value is 0.03 Wb.

### **Torque constant**

Torque constant with any of the stator windings. This parameter is visible only if you set **Permanent magnet flux linkage** to Specify torque constant. The default value is 0.18 N\*m/A.

### **Back EMF constant**

Back EMF constant with any of the stator windings. This parameter is visible only if you set **Permanent magnet flux linkage** to Specify back EMF constant. The default value is 0.18 V\*s/rad.

### **Stator parameterization**

Choose Specify Ld, Lq, and L0, the default value, or Specify Ls, Lm, and Ms.

### **Stator d-axis inductance, Ld**

Direct-axis inductance. This parameter is visible only if you set **Stator parameterization** to Specify Ld, Lq, and L0. The default value is 0.00019 H.

### **Stator q-axis inductance, Lq**

Quadrature-axis inductance. This parameter is visible only if you set **Stator parameterization** to Specify Ld, Lq, and L0. The default value is 0.00025 H.

### **Stator zero-sequence inductance, L0**

Zero-sequence inductance. This parameter is visible only if you set **Stator parameterization** to Specify Ld, Lq, and L0. The default value is 0.00016 H.

### **Stator self-inductance per phase, Ls**

Average self-inductance of each of the three stator windings. This parameter is visible only if you set **Stator parameterization** to Specify Ls, Lm, and Ms. The default value is 0.0002 H.

**Stator inductance fluctuation, Lm**

Amplitude of the fluctuation in self-inductance and mutual inductance of the stator windings with rotor angle. This parameter is visible only if you set **Stator parameterization** to Specify Ls, Lm, and Ms. The default value is  $-0.00002$  H.

**Stator mutual inductance, Ms**

Average mutual inductance between the stator windings. This parameter is visible only if you set **Stator parameterization** to Specify Ls, Lm, and Ms. The default value is  $0.00002$  H.

**Stator resistance per phase, Rs**

Resistance of each of the stator windings. The default value is  $0.013$  Ohm.

**Zero sequence**

Option to include or exclude zero-sequence terms.

- **Include** — Include zero-sequence terms. To prioritize model fidelity, use this default setting. Using this option results in an error for simulations that use the Partitioning solver. For more information, see “Increase Simulation Speed Using the Partitioning Solver” (Simscape).
- **Exclude** — Exclude zero-sequence terms. To prioritize simulation speed for desktop simulation or real-time deployment, select this option.

**Rotor angle definition**

Reference point for the rotor angle measurement. The default value is **Angle between the a-phase magnetic axis and the d-axis**. This definition is shown in the “Motor Construction” on page 1-1101 figure. When you select this value, the rotor and *a*-phase fluxes are aligned when the rotor angle is zero.

The other value you can choose for this parameter is **Angle between the a-phase magnetic axis and the q-axis**. When you select this value, the *a*-phase current generates maximum torque when the rotor angle is zero.

**Mechanical****Rotor Inertia**

Inertia of the rotor attached to mechanical translational port **R**. The default value is  $0.01$  kg\*m<sup>2</sup>. The value can be zero.

**Rotor damping**

Rotary damping. The default value is  $0$  N\*m/(rad/s).

## References

- [1] Kundur, P. *Power System Stability and Control*. New York, NY: McGraw Hill, 1993.
- [2] Anderson, P. M. *Analysis of Faulted Power Systems*. Hoboken, NJ: Wiley-IEEE Press, 1995.

## See Also

### Simscape Blocks

Brushless DC Motor | Hybrid Excitation Synchronous Machine

### Blocks

BLDC Commutation Logic | BLDC Current Controller | BLDC Current Controller with PWM Generation

## Topics

- “Expand and Collapse Three-Phase Ports on a Block”
- “Electric Power Assisted Steering”
- “Three-Phase PMSM Drive”
- “Parameterize a Permanent Magnet Synchronous Motor”

### Introduced in R2013b

# Phase Permute

Permute phases of three-phase system



## Library

Simscape / Electrical / Power Systems / Connections

## Description

The Phase Permute block cyclically permutes (changes the order of) the phases of a three-phase system.

The block has two three-phase connections associated with its terminals. If you consider the side of the block labeled ~123 (**a1,b1,c1** in expanded view) as side 1 and the side of the block labeled ~231 (**a2,b2,c2**) as side 2, then the block connects phases as shown in the table.

Side 1 Phase	Connects to Side 2 Phase
a1	c2
b1	a2
c1	b2

## Ports

The block has the following ports:

~123

Expandable three-phase port

~231

Expandable three-phase port

## **See Also**

### **Simscape Blocks**

Phase Splitter

### **Topics**

“Three-Phase Asynchronous Machine Starting”

“Three-Phase Asynchronous Wind Turbine Generator”

“Marine Full Electric Propulsion Power System”

### **Introduced in R2013b**

# Phase Splitter

Expand composite three-phase port



## Library

Simscape / Electrical / Power Systems / Connections

## Description

The Phase Splitter block expands a composite three-phase port into its constituent phases.

The expanded output ports are electrical conserving ports. Therefore, you can connect the output ports to electrical components from the Simscape and Simscape Electrical Electronics and Mechatronics libraries.

## Ports

The block has the following ports:

~

Composite three-phase port

a, b, c

Constituent phases of the expanded three-phase port

## **See Also**

### **Simscape Blocks**

Phase Permute

### **Topics**

“Comparison of Three-Phase Port Types”

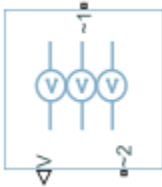
“AC Cable with Bonded Sheaths”

**Introduced in R2013b**



# Phase Voltage Sensor

Measure phase voltages in three-phase system



## Library

Simscape / Electrical / Power Systems / Sensors

## Description

The Phase Voltage Sensor block represents an ideal three-phase voltage sensor. It measures the voltages across the three-phase ports ~1 and ~2 and outputs a single three-element, physical signal vector. Each element of the physical signal output vector is equal to the voltage in the respective phase.

## Ports

The block has the following ports:

~1

Expandable three-phase port

~2

Expandable three-phase port

V

Three-element physical signal vector output port associated with the phase voltages

## **See Also**

Line Voltage Sensor

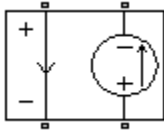
## **Topics**

“Expand and Collapse Three-Phase Ports on a Block”

**Introduced in R2013b**

# PCCCS

Model polynomial current-controlled current source



## Library

SPICE-Compatible Components/Sources

## Description

The PCCCS (Polynomial Current-Controlled Current Source) block represents a current source whose output current value is a polynomial function of the current through the input ports. The following equations describe the current through the source as a function of time:

- If you specify an  $n$ -element vector of polynomial coefficients for the **Polynomial coefficients** parameter:

$$I_{out} = p(0) + p(1) * I_{in} + \dots + p(n-1) * I_{in}^{n-1} + p(n) * I_{in}^n$$

- If you specify a scalar coefficient for the **Polynomial coefficients** parameter:

$$I_{out} = p * I_{in}$$

where:

- $I_{in}$  is the current through the input ports.
- $p$  is the **Polynomial coefficients** parameter value.

The block uses a small conductance internally to prevent numerical simulation issues. The conductance connects the output ports of the device and has a conductance  $GMIN$ :

- By default, *GMIN* matches the **Minimum conductance GMIN** parameter of the SPICE Environment Parameters block, whose default value is 1e-12.
- To change *GMIN*, add a SPICE Environment Parameters block to your model and set the **Minimum conductance GMIN** parameter to the desired value.

## Parameters

### Polynomial coefficients

The polynomial coefficients that relate the input current to the output current, as described in the preceding section. The default value is [ 0 1 ].

## Ports

The block has the following ports:

+

Positive electrical input voltage.

-

Negative electrical input voltage.

N+

Positive electrical output voltage.

N-

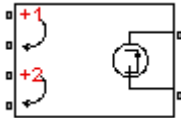
Negative electrical output voltage.

## See Also

, PCCCS2PCCVS, PVCCS, and PVCVS

## PCCCS2

Model polynomial current-controlled current source with two controlling inputs



## Library

SPICE-Compatible Components/Sources

## Description

The PCCCS2 (Two-Input Polynomial Current-Controlled Current Source) block represents a current source whose output current value is a polynomial function of the currents through the pairs of controlling input ports. The following equations describes the current through the source as a function of time:

$$I_{out} = p_1 + p_2 * I_{in1} + p_3 * I_{in2} + p_4 * I_{in1}^2 + p_5 I_{in1} * I_{in2} + p_6 * I_{in2}^2 + \dots$$

where:

- $I_{in1}$  is the current across the first pair of input ports.
- $I_{in2}$  is the current across the second pair of input ports.
- $p$  is the **Polynomial coefficients** parameter value.

The block uses a small conductance internally to prevent numerical simulation issues. The conductance connects the output ports of the device and has a conductance  $GMIN$ :

- By default,  $GMIN$  matches the **Minimum conductance GMIN** parameter of the SPICE Environment Parameters block, whose default value is 1e-12.
- To change  $GMIN$ , add a SPICE Environment Parameters block to your model and set the **Minimum conductance GMIN** parameter to the desired value.

## Parameters

### Polynomial coefficients

The polynomial coefficients that relate the input current to the output current, as described in the preceding section. The default value is [ 0 1 1 ].

## Ports

The block has the following ports:

+1

Positive electrical input voltage of first controlling source.

-1

Negative electrical input voltage of first controlling source.

+2

Positive electrical input voltage of second controlling source.

-2

Negative electrical input voltage of second controlling source.

N+

Positive electrical output voltage.

N-

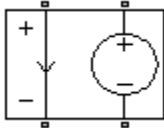
Negative electrical output voltage.

## See Also

PCCCS, PCCVS2, PVCCS2, and PVCVS2

## PCCVS

Model polynomial current-controlled voltage source



## Library

SPICE-Compatible Components/Sources

## Description

The PCCVS (Polynomial Current-Controlled Voltage Source) block represents a voltage source whose output voltage value is a polynomial function of the current through the input ports. The following equations describe the voltage across the source as a function of time:

- If you specify an  $n$ -element vector of polynomial coefficients for the **Polynomial coefficients** parameter:

$$V_{out} = p(0) + p(1) * I_{in} + \dots + p(n-1) * I_{in}^{n-1} + p(n) * I_{in}^n$$

- If you specify a scalar coefficient for the **Polynomial coefficients** parameter:

$$V_{out} = p * I_{in}$$

where:

- $I_{in}$  is the current through the input ports.
- $p$  is the **Polynomial coefficients** parameter value.

## Parameters

### Polynomial coefficients

The polynomial coefficients that relate the input current to the output voltage, as described in the preceding section. The default value is [ 0 1 ].

## Ports

The block has the following ports:

+

Positive electrical input voltage.

-

Negative electrical input voltage.

N+

Positive electrical output voltage.

N-

Negative electrical output voltage.

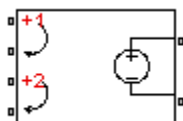
## See Also

PCCCS, PCCVS2, PVCCS, and PVCVS



## PCCVS2

Model polynomial current-controlled voltage source with two controlling inputs



## Library

SPICE-Compatible Components/Sources

## Description

The PCCVS2 (Two-Input Polynomial Current-Controlled Voltage Source) block represents a voltage source whose output voltage value is a polynomial function of the currents through the pairs of controlling input ports. The following equations describes the voltage across the source as a function of time:

$$V_{out} = p_1 + p_2 * I_{in1} + p_3 * I_{in2} + p_4 * I_{in1}^2 + p_5 * I_{in1} * I_{in2} + p_6 * I_{in2}^2 + \dots$$

where:

- $I_{in1}$  is the current across the first pair of input ports.
- $I_{in2}$  is the current across the second pair of input ports.
- $p$  is the **Polynomial coefficients** parameter value.

The block uses a small conductance internally to prevent numerical simulation issues. The conductance connects the output ports of the device and has a conductance *GMIN*:

- By default, *GMIN* matches the **Minimum conductance GMIN** parameter of the SPICE Environment Parameters block, whose default value is 1e-12.
- To change *GMIN*, add a SPICE Environment Parameters block to your model and set the **Minimum conductance GMIN** parameter to the desired value.

## Parameters

### Polynomial coefficients

The polynomial coefficients that relate the input current to the output voltage, as described in the preceding section. The default value is [ 0 1 1 ].

## Ports

The block has the following ports:

+1

Positive electrical input voltage of first controlling source.

-1

Negative electrical input voltage of first controlling source.

+2

Positive electrical input voltage of second controlling source.

-2

Negative electrical input voltage of second controlling source.

N+

Positive electrical output voltage.

N-

Negative electrical output voltage.

## See Also

PCCCS2, PCCVS, PVCCS2, and PVCVS2

# Photodiode

Model photodiode as parallel controlled current source and exponential diode



## Library

Sensors

## Description

The Photodiode block represents a photodiode as a controlled current source and an exponential diode connected in parallel. The controlled current source produces a current  $I_p$  that is proportional to the radiant flux density:

$$I_p = DeviceSensitivity \cdot RadiantFluxDensity$$

where:

- *DeviceSensitivity* is the ratio of the current produced to the incident radiant flux density.
  - If you select Specify measured current for given flux density for the **Sensitivity parameterization** parameter, the block calculates this variable by converting the **Measured current** parameter value to units of amps and dividing it by the **Flux density** parameter values.
  - If you select Specify current per unit flux density for the **Sensitivity parameterization** parameter, this variable is defined by the **Device sensitivity** parameter value.
- *RadiantFluxDensity* is the incident radiant flux density.

To model dynamic response time, use the **Junction capacitance** parameter to include the diode junction capacitance in the model.

The exponential diode model provides the following relationship between the diode current  $I$  and the diode voltage  $V$ :

$$I = IS \cdot \left( e^{\frac{qV}{NkT_{m1}}} - 1 \right)$$

where:

- $q$  is the elementary charge on an electron (1.602176e-19 Coulombs).
- $k$  is the Boltzmann constant (1.3806503e-23 J/K).
- $N$  is the emission coefficient.
- $IS$  is the saturation current, which is equal to the **Dark current** parameter value.
- $T_{m1}$  is the temperature at which the diode parameters are specified, as defined by the **Measurement temperature** parameter value.

When  $(qV / NkT_{m1}) > 80$ , the block replaces  $e^{\frac{qV}{NkT_{m1}}}$  with  $(qV / NkT_{m1} - 79)e^{80}$ , which matches the gradient of the diode current at  $(qV / NkT_{m1}) = 80$  and extrapolates linearly.

When  $(qV / NkT_{m1}) < -79$ , the block replaces  $e^{\frac{qV}{NkT_{m1}}}$  with  $(qV / NkT_{m1} + 80)e^{-79}$ , which also matches the gradient and extrapolates linearly. Typical electrical circuits do not reach these extreme values. The block provides this linear extrapolation to help convergence when solving for the constraints during simulation.

When you select Use dark current and  $N$  for the **Diode parameterization** parameter, you specify the diode in terms of the **Dark current** and **Emission coefficient**  $N$  parameters. When you select Use dark current plus a forward bias I-V data point for the **Diode parameterization** parameter, you specify the **Dark current** parameter and a voltage and current measurement point on the diode I-V curve. The block calculates  $N$  from these values as follows:

$$N = V_F / (V_t \log(I_F / IS + 1))$$

where:

- $V_F$  is the **Forward voltage VF** parameter value.

- $V_t = kT_{m1} / q$ .
- $I_F$  is the **Current IF at forward voltage VF** parameter value.

The exponential diode model provides the option to include a junction capacitance:

- When you select **Fixed** or **zero** junction capacitance for the **Junction capacitance** parameter, the capacitance is fixed.
- When you select **Use parameters CJO, VJ, M & FC** for the **Junction capacitance** parameter, the block uses the coefficients *CJO*, *VJ*, *M*, and *FC* to calculate a junction capacitance that depends on the junction voltage.
- When you select **Use C-V curve data points** for the **Junction capacitance** parameter, the block uses three capacitance values on the C-V capacitance curve to estimate *CJO*, *VJ*, and *M* and uses these values with the specified value of *FC* to calculate a junction capacitance that depends on the junction voltage. The block calculates *CJO*, *VJ*, and *M* as follows:

- $$CJO = C_1 ((V_{R2} - V_{R1}) / (V_{R2} - V_{R1} (C_2 / C_1)^{-1/M}))^M$$
- $$VJ = -(-V_{R2} (C_1 / C_2)^{-1/M} + V_{R1}) / (1 - (C_1 / C_2)^{-1/M})$$
- $$M = \log(C_3 / C_2) / \log(V_{R2} / V_{R3})$$

where:

- $V_{R1}$ ,  $V_{R2}$ , and  $V_{R3}$  are the values in the **Reverse bias voltages [VR1 VR2 VR3]** vector.
- $C_1$ ,  $C_2$ , and  $C_3$  are the values in the **Corresponding capacitances [C1 C2 C3]** vector.

It is not possible to estimate *FC* reliably from tabulated data, so you must specify its value using the **Capacitance coefficient FC** parameter. In the absence of suitable data for this parameter, use a typical value of 0.5.

The reverse bias voltages (defined as positive values) should satisfy  $V_{R3} > V_{R2} > V_{R1}$ . This means that the capacitances should satisfy  $C_1 > C_2 > C_3$  as reverse bias widens the depletion region and hence reduces capacitance. Violating these inequalities results in an error. Voltages  $V_{R2}$  and  $V_{R3}$  should be well away from the Junction potential *VJ*. Voltage  $V_{R1}$  should be less than the Junction potential *VJ*, with a typical value for  $V_{R1}$  being 0.1 V.

The voltage-dependent junction is defined in terms of the capacitor charge storage  $Q_j$  as:

- For  $V < FC \cdot VJ$ :

$$Q_j = CJ0 \cdot (VJ / (M - 1)) \cdot ((1 - V / VJ)^{1-M} - 1)$$

- For  $V \geq FC \cdot VJ$ :

$$Q_j = CJ0 \cdot F_1 + (CJ0 / F_2) \cdot (F_3 \cdot (V - FC \cdot VJ) + 0.5(M / VJ) \cdot (V^2 - (FC \cdot VJ)^2))$$

where:

- $F_1 = (VJ / (1 - M)) \cdot (1 - (1 - FC)^{1-M})$
- $F_2 = (1 - FC)^{1+M}$
- $F_3 = 1 - FC \cdot (1 + M)$

These equations are the same as used in [2], except that the temperature dependence of  $VJ$  and  $FC$  is not modeled. This model does not include the diffusion capacitance term that affects performance for high frequency switching applications.

The Photodiode block contains several options for modeling the dependence of the diode current-voltage relationship on the temperature during simulation. Temperature dependence of the junction capacitance is not modeled, this being a much smaller effect. For details, see the Diode reference page.

## Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal port H on the block icon, and adds the **Thermal Port** tab to the block dialog box.

Use the thermal port to simulate the effects of generated heat and device temperature. For more information on using thermal ports and on the **Thermal Port** tab parameters, see “Simulating Thermal Effects in Semiconductors”.

## Variables

Use the **Variables** section of the block interface to set the priority and initial target values for the block variables prior to simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

## Basic Assumptions and Limitations

The Photodiode block has the following limitations:

- When you select `Use dark current plus a forward bias I-V curve data point` for the **Diode parameterization** parameter, choose a voltage near the diode turn-on voltage. Typically this will be in the range from 0.05 to 1 Volt. Using a value outside of this region may lead to a poor estimate for  $N$ .
- You may need to use nonzero ohmic resistance and junction capacitance values to prevent numerical simulation issues, but the simulation may run faster with these values set to zero.

## Parameters

- “Main” on page 1-1129
- “Junction Capacitance” on page 1-1131
- “Temperature Dependence” on page 1-1132

## Main

### Sensitivity parameterization

Select one of the following methods for sensitivity parameterization:

- `Specify measured current for given flux density` — Specify the measured current and the corresponding flux density. This is the default method.
- `Specify current per unit flux density` — Specify the device sensitivity directly.

## Measured current

The current the block uses to calculate the device sensitivity. This parameter is only visible when you select **Specify measured current for given flux density** for the **Sensitivity parameterization** parameter. The default value is 25  $\mu\text{A}$ .

## Flux density

The flux density the block uses to calculate the device sensitivity. This parameter is only visible when you select **Specify measured current for given flux density** for the **Sensitivity parameterization** parameter. The default value is 5  $\text{W}/\text{m}^2$ .

## Device sensitivity

The current per unit flux density. This parameter is only visible when you select **Specify current per unit flux density** for the **Sensitivity parameterization** parameter. The default value is  $5\text{e-}06 \text{ m}^2\text{*A}/\text{W}$ .

## Diode parameterization

Select one of the following methods for diode model parameterization:

- Use dark current plus a forward bias I-V data point — Specify the dark current and a point on the diode I-V curve. This is the default method.
- Use dark current and N — Specify dark current and emission coefficient.

## Current I1

The current at the forward-biased point on the diode I-V curve that the block uses to calculate  $I_S$  and  $N$ . This parameter is only visible when you select **Use dark current plus a forward bias I-V data point** for the **Diode parameterization** parameter. The default value is 0.1 A.

## Voltage V1

The corresponding voltage at the forward-biased point on the diode I-V curve that the block uses to calculate  $I_S$  and  $N$ . This parameter is only visible when you select and **Use dark current plus a forward bias I-V data point** for the **Diode parameterization** parameter. The default value is 1.3 V.

## Dark current

The current through the diode when it is not exposed to light. The default value is  $5\text{e-}9 \text{ A}$ .



**Emission coefficient, N**

The diode emission coefficient or ideality factor. This parameter is only visible when you select Use dark current and N for the **Diode parameterization** parameter. The default value is 3.

**Ohmic resistance, RS**

The series diode connection resistance. The default value is 0.1  $\Omega$ .

**Measurement temperature**

The temperature at which the I-V curve or dark current was measured. The default value is 25 °C.

## Junction Capacitance

**Junction capacitance**

Select one of the following options for modeling the junction capacitance:

- Fixed or zero junction capacitance — Model the junction capacitance as a fixed value.
- Use C-V curve data points — Specify measured data at three points on the diode C-V curve.
- Use parameters CJ0, VJ, M & FC — Specify zero-bias junction capacitance, junction potential, grading coefficient, and forward-bias depletion capacitance coefficient.

**Zero-bias junction capacitance, CJ0**

The value of the capacitance placed in parallel with the exponential diode term. This parameter is only visible when you select Fixed or zero junction capacitance or Use parameters CJ0, VJ, M & FC for the **Junction capacitance** parameter. The default value is 60 pF. When you select Fixed or zero junction capacitance for the **Junction capacitance** parameter, a value of zero omits junction capacitance.

**Reverse bias voltages [VR1 VR2 VR3]**

A vector of the reverse bias voltage values at the three points on the diode C-V curve that the block uses to calculate CJ0, VJ, and M. This parameter is only visible when you select Use C-V curve data points for the **Junction capacitance** parameter. The default value is [ 0.1 10 100 ] V.

### Corresponding capacitances [C1 C2 C3]

A vector of the capacitance values at the three points on the diode C-V curve that the block uses to calculate  $CJ0$ ,  $VJ$ , and  $M$ . This parameter is only visible when you select Use C-V curve data points for the **Junction capacitance** parameter. The default value is [ 45 30 6 ] pF.

### Junction potential, VJ

The junction potential. This parameter is only visible when you select Use parameters  $CJ0$ ,  $VJ$ ,  $M$  &  $FC$  for the **Junction capacitance** parameter. The default value is 1 V.

### Grading coefficient, M

The grading coefficient. This parameter is only visible when you select Use parameters  $CJ0$ ,  $VJ$ ,  $M$  &  $FC$  for the **Junction capacitance** parameter. The default value is 0.5.

### Capacitance coefficient, FC

Fitting coefficient that quantifies the decrease of the depletion capacitance with applied voltage. This parameter is only visible when you select Use C-V curve data points or Use parameters  $CJ0$ ,  $VJ$ ,  $M$  &  $FC$  for the **Junction capacitance** parameter. The default value is 0.5.

## Temperature Dependence

### Parameterization

Select one of the following methods for temperature dependence parameterization:

- None – Simulate at parameter measurement temperature – Temperature dependence is not modeled, or the model is simulated at the measurement temperature  $T_{m1}$  (as specified by the **Measurement temperature** parameter on the **Main** tab). This is the default method.
- Use an I-V data point at second measurement temperature T2 – If you select this option, you specify a second measurement temperature  $T_{m2}$ , and the current and voltage values at this temperature. The model uses these values, along with the parameter values at the first measurement temperature  $T_{m1}$ , to calculate the energy gap value.
- Specify saturation current at second measurement temperature T2 – If you select this option, you specify a second measurement temperature  $T_{m2}$ , and saturation current value at this temperature. The model uses these values,

along with the parameter values at the first measurement temperature  $T_{m1}$ , to calculate the energy gap value.

- Specify the energy gap EG — Specify the energy gap value directly.

### **Current $I1$ at second measurement temperature**

Specify the diode current  $I1$  value when the voltage is  $V1$  at the second measurement temperature. This parameter is only visible when you select Use an I-V data point at second measurement temperature  $T2$  for the **Parameterization** parameter. The default value is 0.07 A.

### **Voltage $V1$ at second measurement temperature**

Specify the diode voltage  $V1$  value when the current is  $I1$  at the second measurement temperature. This parameter is only visible when you select Use an I-V data point at second measurement temperature  $T2$  for the **Parameterization** parameter. The default value is 1.3 V.

### **Saturation current, $IS$ , at second measurement temperature**

Specify the saturation current  $IS$  value at the second measurement temperature. This parameter is only visible when you select Specify saturation current at second measurement temperature  $T2$  for the **Parameterization** parameter. The default value is  $2.5e-7$  A.

### **Second measurement temperature**

Specify the value for the second measurement temperature. This parameter is only visible when you select either Use an I-V data point at second measurement temperature  $T2$  or Specify saturation current at second measurement temperature  $T2$  for the **Parameterization** parameter. The default value is 125 °C.

### **Energy gap parameterization**

This parameter is only visible when you select Specify the energy gap EG for the **Parameterization** parameter. It lets you select a value for the energy gap from a list of predetermined options, or specify a custom value:

- Use nominal value for silicon (EG=1.11eV) — This is the default.
- Use nominal value for 4H-SiC silicon carbide (EG=3.23eV)
- Use nominal value for 6H-SiC silicon carbide (EG=3.00eV)
- Use nominal value for germanium (EG=0.67eV)
- Use nominal value for gallium arsenide (EG=1.43eV)

- Use nominal value for selenium (EG=1.74eV)
- Use nominal value for Schottky barrier diodes (EG=0.69eV)
- Specify a custom value — If you select this option, the **Energy gap, EG** parameter appears in the dialog box, to let you specify a custom value for *EG*.

### **Energy gap, EG**

Specify a custom value for the energy gap, *EG*. This parameter is only visible when you select Specify a custom value for the **Energy gap parameterization** parameter. The default value is 1.11 eV.

### **Saturation current temperature exponent parameterization**

Select one of the following options to specify the saturation current temperature exponent value:

- Use nominal value for pn-junction diode (XTI=3) — This is the default.
- Use nominal value for Schottky barrier diode (XTI=2)
- Specify a custom value — If you select this option, the **Saturation current temperature exponent, XTI** parameter appears in the dialog box, to let you specify a custom value for *XTI*.

### **Saturation current temperature exponent, XTI**

Specify a custom value for the saturation current temperature exponent, *XTI*. This parameter is only visible when you select Specify a custom value for the **Saturation current temperature exponent parameterization** parameter. The default value is 3.

### **Device simulation temperature**

Specify the value for the temperature  $T_s$ , at which the device is to be simulated. The default value is 25 °C.

## **Ports**

The block has the following ports:

D

Physical port representing incident flux

+

Electrical conserving port associated with the diode positive terminal

-

Electrical conserving port associated with the diode negative terminal

## References

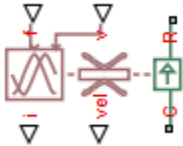
- [1] MH. Ahmed and P.J. Spreadbury. *Analogue and digital electronics for engineers*. 2nd Edition, Cambridge University Press, 1984.
- [2] G. Massobrio and P. Antognetti. *Semiconductor Device Modeling with SPICE*. 2nd Edition, McGraw-Hill, 1993.

## See Also

Diode | Light-Emitting Diode | Optocoupler

## Piezo Linear Motor

Model force-speed characteristics of linear piezoelectric traveling wave motor



## Library

Translational Actuators

## Description

The Piezo Linear Motor block represents the force-speed characteristics of a linear piezoelectric traveling wave motor. The block represents the force-speed relationship of the motor at a level that is suitable for system-level modeling. To simulate the motor, the block uses the following models:

- “Mass and Friction Model for Unpowered Motor” on page 1-1136
- “Resonant Circuit Model for Powered Motor” on page 1-1137

### Mass and Friction Model for Unpowered Motor

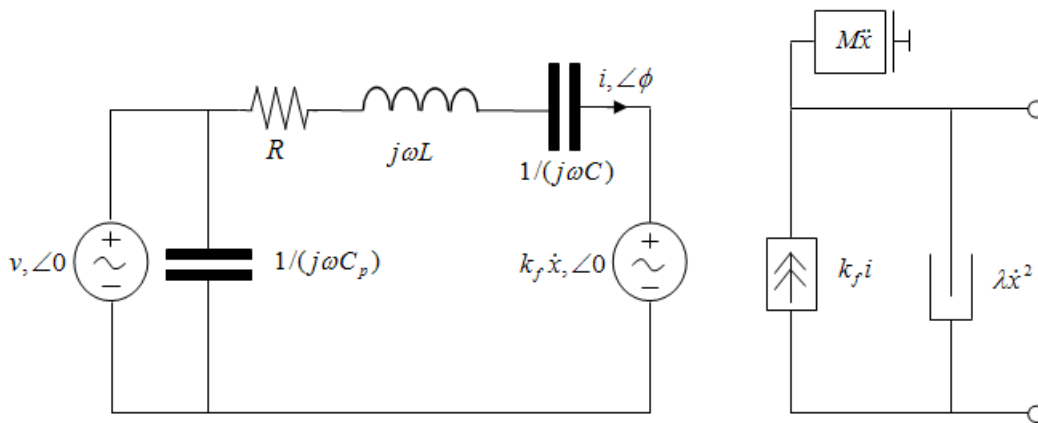
The motor is unpowered when the physical signal input  $v$  is zero. This corresponds to applying zero RMS volts to the motor. In this scenario, the block models the motor using the following elements:

- A mass whose value is the **Plunger mass** parameter value.
- A friction whose characteristics you specify using the parameter values in the **Motor-Off Friction** tab.

The block uses a Simscape Translational Friction block to model the friction component. For detailed information about the friction model, see the Translational Friction block reference page.

## Resonant Circuit Model for Powered Motor

When the motor is active, Piezo Linear Motor block represents the motor characteristics using the following equivalent circuit model.



In the preceding figure:

- The AC voltage source represents the block's physical signal input of frequency  $f$  and magnitude  $v$ .
- The resistor  $R$  provides the main electrical and mechanical damping term.
- The inductor  $L$  represents the rotor vibration inertia.
- The capacitor  $C$  represents the piezo crystal stiffness.
- The capacitor  $C_p$  represents the phase capacitance. This is the electrical capacitance associated with each of the two motor phases.
- The force constant  $k_f$  relates the RMS current  $i$  to the resulting mechanical force.
- The quadratic mechanical damping term,  $\lambda \dot{x}^2$ , shapes the force-speed curve predominantly at speeds close to maximum RPM.  $\dot{x}$  is the linear speed.
- The term  $M\ddot{x}$  represents the plunger inertia.

At model initialization, the block calculates the model parameters  $R$ ,  $L$ ,  $C$ ,  $k_t$  and  $\lambda$  to ensure that the steady-state force-speed curve matches the values for the following user-specified parameters:

- **Rated force**
- **Rated speed**
- **No-load maximum speed**
- **Maximum (stall) force**

These parameter values are defined for the **Rated RMS voltage** and **Motor natural frequency** (or rated frequency) parameter values.

The quadratic mechanical damping term produces a quadratic force-speed curve. Piezoelectric motors force-speed curves can typically be approximated more accurately using a quadratic function than a linear one because the force-speed gradient becomes steeper as the motor approaches the maximum speed.

If the plunger mass  $M$  is not specified on the datasheet, you can select a value that provides a good match to the quoted response time. The response time is often defined as the time for the rotor to reach maximum speed when starting from rest, under no-load conditions.

The quality factor that you specify using the **Resonance quality factor** parameter relates to the equivalent circuit model parameters as follows:

$$Q = \frac{1}{R} \sqrt{\frac{L}{C}}$$

This term is not usually provided on a datasheet. You can calculate its value by matching the sensitivity of force to driving frequency.

To reverse the motor direction of operation, make the physical signal input  $v$  negative.

## Basic Assumptions and Limitations

The block has the following limitations:

- When the motor is powered, the model is valid only between zero and maximum speed, for the following reasons:



- Datasheets do not provide information for operation outside of normal range.
- Piezoelectric motors are not designed to operate in the powered braking and generating regions.

The block behaves as follows outside the valid operating region:

- Below zero speed, the model maintains a constant force with a zero speed value. The zero speed value is the **Maximum (stall) force** parameter value if the RMS input voltage equals the **Rated RMS voltage** parameter value, and the frequency input equals the **Motor natural frequency** parameter value.
- Above maximum speed, the model produces the negative force predicted by the equivalent circuit model, but limits the absolute value of the force to the zero-speed maximum force.
- The force-speed characteristics are most representative when operating the model close to the rated voltage and resonant frequency.

## Parameters

- “Electrical Force Tab” on page 1-1139
- “Mechanical Tab” on page 1-1140
- “Motor-Off Friction Tab” on page 1-1140

### Electrical Force Tab

#### Motor natural frequency

Frequency at which the piezoelectric crystal naturally resonates. For most applications, set the input signal at port *f* to this frequency. To slow down the motor, for example in a closed-loop speed control, use a frequency slightly less than the motor natural frequency. The default value is 92 kHz.

#### Rated RMS voltage

Voltage at which the motor is designed to operate. The default value is 5.7 V.

#### Rated force

Force the motor delivers at the rated RMS voltage. The default value is 0.1 N.

#### Rated speed

Motor speed when the motor drives a load at the rated force. The default value is 50 mm/s.

### **No-load maximum speed**

Motor speed when driving no load and powered at the rated voltage and driving frequency. The default value is 150 mm/s.

### **Maximum (stall) force**

Maximum force the motor delivers when actively driving a load and powered at the rated voltage and frequency. The default value is 0.15 N.

---

**Note** The **Holding force** parameter value, the load force the motor holds when stationary, may be greater than the **Maximum (stall) force** parameter value.

---

### **Resonance quality factor**

Quality factor  $Q$  that specifies how force varies as a function of driving frequency. Increasing the quality factor results in a much more rapid decrease in force as driving frequency is moved away from the natural frequency. The default value is 100.

### **Capacitance per phase**

Electrical capacitance associated with each of the two motor phases. The default value is 5 nF.

## **Mechanical Tab**

### **Plunger mass**

Mass of the moving part of the motor. The default value is 0.3 g.

### **Initial rotor speed**

Rotor speed at the start of the simulation. The default value is 0 mm/s.

## **Motor-Off Friction Tab**

### **Holding force**

The sum of the Coulomb and the static frictions. It must be greater than or equal to the **Coulomb friction force** parameter value. The default value is 0.3 N.

### **Coulomb friction force**

The friction that opposes rotation with a constant force at any velocity. The default value is 0.15 N.

**Viscous friction coefficient**

Proportionality coefficient between the friction force and the relative velocity. The parameter value must be greater than or equal to zero. The default value is  $1e-05$  s\*N/mm.

**Transition approximation coefficient**

The parameter sets the coefficient value that is used to approximate the transition between the static and the Coulomb frictions. For detailed information about the coefficient,  $c_v$ , see the Simscape Translational Friction block reference page. The default value is  $0.1$  s/mm.

**Linear region velocity threshold**

The parameter sets the small vicinity near zero velocity, within which friction force is considered to be linearly proportional to the relative velocity. MathWorks recommends that you use values between  $1e-6$  and  $1e-4$  mm/s. The default value is  $0.1$  mm/s.

**Ports**

The block has the following ports:

f

Physical signal input value specifying the motor driving frequency in Hz.

v

Physical signal input magnitude specifying the RMS supply voltage, and sign specifying the direction of rotation. If v is positive, then a positive force acts from port C to port R.

i

Physical signal output value that is the RMS phase current.

vel

Physical signal output value that is the linear speed of the rotor.

C

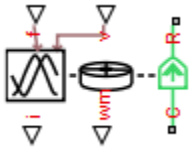
Mechanical translational conserving port.

R

Mechanical translational conserving port.

## Piezo Rotary Motor

Model torque-speed characteristics of rotary piezoelectric traveling wave motor



### Library

Rotational Actuators

### Description

The Piezo Rotary Motor block represents the torque-speed characteristics of a piezoelectric traveling wave motor. The block represents the torque-speed relationship of the motor at a level that is suitable for system-level modeling. To simulate the motor, the block uses the following models:

- “Inertia and Friction Model for Unpowered Motor” on page 1-1142
- “Resonant Circuit Model for Powered Motor” on page 1-1143

### Inertia and Friction Model for Unpowered Motor

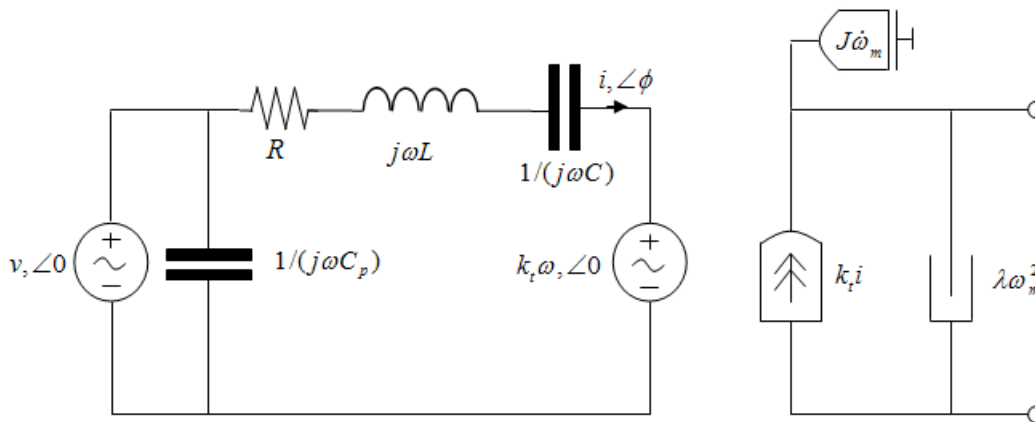
The motor is unpowered when the physical signal input  $v$  is zero. This corresponds to applying zero RMS volts to the motor. In this scenario, the block models the motor using the following elements:

- An inertia whose value is the **Rotor inertia** parameter value.
- A friction whose characteristics are determined by the parameter values in the **Motor-Off Friction** tab.

The block uses a Simscape Rotational Friction block to model the friction component. For detailed information about the friction model, see the Rotational Friction block reference page.

## Resonant Circuit Model for Powered Motor

When the motor is active, Piezo Rotary Motor block represents the motor characteristics using the following equivalent circuit model.



In the preceding figure:

- The AC voltage source represents the block's physical signal input of frequency  $f$  and magnitude  $v$ .
- The resistor  $R$  provides the main electrical and mechanical damping term.
- The inductor  $L$  represents the rotor vibration inertia.
- The capacitor  $C$  represents the piezo crystal stiffness.
- The capacitor  $C_p$  represents the phase capacitance. This is the electrical capacitance associated with each of the two motor phases.
- The torque constant  $k_t$  relates the RMS current  $i$  to the resulting mechanical torque.
- The quadratic mechanical damping term,  $\lambda\omega_m^2$ , shapes the torque-speed curve predominantly at speeds close to maximum RPM.  $\omega_m$  is the mechanical rotational speed.
- The term  $J\dot{\omega}_m$  represents the rotor inertia.

At model initialization, the block calculates the model parameters  $R$ ,  $L$ ,  $C$ ,  $k_t$  and  $\lambda$  to ensure that the steady-state torque-speed curve matches the values of the following user-specified parameter values:

- **Rated torque**
- **Rated rotational speed**
- **No-load maximum rotational speed**
- **Maximum torque**

These parameter values are defined for the **Rated RMS voltage** and **Motor natural frequency** (or rated frequency) parameter values.

The quadratic mechanical damping term produces a quadratic torque-speed curve. Piezoelectric motors torque-speed curves can typically be approximated more accurately using a quadratic function than a linear one because the torque-speed gradient becomes steeper as the motor approaches the maximum speed.

If the rotor inertia  $J$  is not specified on the datasheet, you can select a value that provides a good match to the quoted response time. The response time is often defined as the time for the rotor to reach maximum speed when starting from rest, under no-load conditions.

The quality factor that you specify using the **Resonance quality factor** parameter relates to the equivalent circuit model parameters as follows:

$$Q = \frac{1}{R} \sqrt{\frac{L}{C}}$$

This term is not usually provided on a datasheet. You can calculate its value by matching the sensitivity of torque to driving frequency.

To reverse the motor direction of operation, make the physical signal input  $v$  negative.

## Basic Assumptions and Limitations

The block has the following limitations:

- When the motor is powered, the model is valid only between zero and maximum speed, for the following reasons:

- Datasheets do not provide information for operation outside of normal range.
- Piezoelectric motors are not designed to operate in the powered braking and generating regions.

The block behaves as follows outside the valid operating region:

- Below zero speed, the model maintains a constant torque that is the zero rpm torque value. The zero rpm torque value is the **Maximum torque** parameter value if the RMS input voltage equals the **Rated RMS voltage** parameter value, and the frequency input equals the **Motor natural frequency** parameter value.
- Above maximum speed, the model produces the negative torque predicted by the equivalent circuit model, but limits the absolute value of the torque to the zero-speed maximum torque.
- The torque-speed characteristics are most representative when operating the model close to the rated voltage and resonant frequency.

## Parameters

- “Electrical Torque Tab” on page 1-1145
- “Mechanical Tab” on page 1-1146
- “Motor-Off Friction Tab” on page 1-1146

### Electrical Torque Tab

#### Motor natural frequency

Frequency at which the piezoelectric crystal naturally resonates. For most applications, set the input signal at port *f* to this frequency. To slow down the motor, for example in a closed-loop speed control, use a frequency slightly less than the motor natural frequency. The default value is 40 kHz.

#### Rated RMS voltage

Voltage at which the motor is designed to operate. The default value is 130 V.

#### Rated torque

Torque the motor delivers at the rated RMS voltage. The default value is 0.5 N\*m.

#### Rated rotational speed

Motor speed when the motor drives a load at the rated torque. The default value is 100 rpm.

### **No-load maximum rotational speed**

Motor rotational speed when driving no load and powered at the rated voltage and driving frequency. The default value is 160 rpm.

### **Maximum torque**

Maximum torque that the motor delivers when actively driving a load and powered at the rated voltage and frequency. The default value is 1 N\*m.

---

**Note** The **Holding torque** parameter value, the load torque the motor holds when stationary, may be greater than the **Maximum torque** parameter value.

---

### **Resonance quality factor**

Quality factor  $Q$  that specifies how torque varies as a function of driving frequency. Increasing the quality factor results in a much more rapid decrease in torque as driving frequency is moved away from the natural frequency. The default value is 100.

### **Capacitance per phase**

Electrical capacitance associated with each of the two motor phases. The default value is 5 nF.

## **Mechanical Tab**

### **Rotor inertia**

Rotor resistance to change in motor motion. The default value is 200 g\*cm<sup>2</sup>.

### **Initial rotor speed**

Rotor speed at the start of the simulation. The default value is 0 rpm.

## **Motor-Off Friction Tab**

### **Holding torque**

The sum of the Coulomb and the static frictions. It must be greater than or equal to the **Coulomb friction torque** parameter value. The default value is 1.5 N\*m.

### **Coulomb friction torque**

The friction that opposes rotation with a constant torque at any velocity. The default value is 1 N\*m.



**Viscous friction coefficient**

Proportionality coefficient between the friction torque and the relative angular velocity. The parameter value must be greater than or equal to zero. The default value is  $0.001 \text{ N}\cdot\text{m}/(\text{rad}\cdot\text{s})$ .

**Transition approximation coefficient**

The parameter sets the coefficient value that is used to approximate the transition between the static and the Coulomb frictions. For detailed information about the coefficient,  $c_v$ , see the Simscape Rotational Friction block reference page. The default value is  $10 \text{ s/rad}$ .

**Linear region velocity threshold**

The parameter sets the small vicinity near zero velocity, within which friction torque is considered to be linearly proportional to the relative velocity. MathWorks recommends that you use values in the range between  $1\text{e-}5$  and  $1\text{e-}3 \text{ rad/s}$ . The default value is  $1\text{e-}04 \text{ rad/s}$ .

## Ports

The block has the following ports:

f

Physical signal input value specifying the motor driving frequency in Hz.

v

Physical signal input magnitude specifying the RMS supply voltage, and sign specifying the direction of rotation. If v is positive, then a positive torque acts from port C to port R.

i

Physical signal output value that is the RMS phase current.

wm

Physical signal output value that is the rotational speed of the rotor.

C

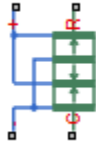
Mechanical rotational conserving port.

R

Mechanical rotational conserving port.

## Piezo Stack

Model electrical and force characteristics of piezoelectric stacked actuator



## Library

Translational Actuators

## Description

The Piezo Stack block represents the electrical and force characteristics of a piezoelectric stacked actuator using the following equations:

$$S = s^E T + d^T E$$

$$D = dT + \epsilon^T E$$

where

- $S$  is the strain tensor.
- $T$  is the stress tensor.
- $E$  is the electric field vector.
- $D$  is the electric displacement vector.
- $s^E$  is the elastic compliance matrix when subjected to a constant electric field.
- $d$  is the piezoelectric constant matrix.
- $\epsilon^T$  is the permittivity measured at a constant stress.

---

**Note** The block models one-dimensional lumped parameter behavior, so  $S$ ,  $T$ ,  $E$  and  $D$  are all scalar values.

---

You can specify the block parameters that determine static force using either datasheet parameters or material properties, as determined by the value of the **Parameterization** parameter on the **Static Force** tab of the block dialog box.

The **Dynamic Forces** tab of the block dialog box lets you include optional effective mass and mechanical damping effects.

- If you specify a nonzero value for the **Effective mass** parameter or a finite value for the **Resonant frequency at constant field** parameter, the block attaches a lumped mass to the mechanical R port. When you specify a finite resonant frequency, the block calculates the effective mass to achieve the correct resonant frequency.
- If you specify a nonzero value for the **Damping** parameter or a finite value for the **Mechanical quality factor** parameter, the block adds a damping term across the R and C mechanical ports. When you specify a mechanical quality factor,  $Q_m$ , the block

calculates the damping from this parameter value as  $\sqrt{Mk}/Q_m$ , where  $k$  is the short-circuit device stiffness, or equivalently the stiffness at constant field.

A positive voltage across the electrical + to - ports creates a positive displacement acting from the mechanical C to R ports.

## Basic Assumptions and Limitations

The model does not include hysteresis effects.

## Parameters

- “Static Force Tab” on page 1-1150
- “Dynamic Forces Tab” on page 1-1151
- “Initial Conditions Tab” on page 1-1152

## Static Force Tab

### Parameterization

Select one of the following methods for static force parameterization:

- **Specify from a datasheet** — Provide datasheet parameters that the block converts to static force values. This is the default method.
- **Specify material properties** — Provide material properties that the block converts to static force values.

### Stack area

Cross-sectional area of the stack. The default value is 100 mm<sup>2</sup>.

### Stack length

Stack length when no load and no electrical potential are applied. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is 36 mm.

### No-load displacement at V0 volts

Unconstrained displacement of the stack when a voltage of V0 volts is applied. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is 0.038 mm.

### Blocking force at V0 volts

Force the stack produces when a voltage of V0 volts is applied and the stack is physically prevented from expanding. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is 3.8e+03 N.

### Test voltage V0

Voltage used to determine the no-load displacement and blocking force. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is 120 V.

### Capacitance

This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is 13 uF.

### Piezo layer thickness

Thickness of each layer in the piezo stack. This parameter is only visible when you select **Specify material properties** for the **Parameterization** parameter. The default value is 0.3 mm.

**Number of layers**

Number of layers in the piezo stack. This parameter is only visible when you select **Specify material properties** for the **Parameterization** parameter. The default value is 50.

**Piezoelectric charge constant**

Mechanical strain per unit electric field applied. This parameter is only visible when you select **Specify material properties** for the **Parameterization** parameter. The default value is  $5e-10$  m/V.

**Dielectric constant**

Permittivity or dielectric displacement per unit electric field measured at constant stress. This parameter is only visible when you select **Specify material properties** for the **Parameterization** parameter. The default value is  $2.124e-08$  F/m.

**Elastic compliance**

Strain produced in a piezoelectric material per unit of stress applied. This parameter is only visible when you select **Specify material properties** for the **Parameterization** parameter. The default value is  $1.9e-11$  m<sup>2</sup>/N.

**Dynamic Forces Tab****Parameterization**

Select one of the following methods for dynamic force parameterization:

- **Specify from a datasheet** — Provide datasheet parameters that the block converts to dynamic force values. This is the default method.
- **Specify material properties** — Provide material properties that the block converts to dynamic force values.

**Resonant frequency at constant field**

Frequency at which the actuator naturally resonates if mechanically perturbed with the electrical ports shorted. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is Inf kHz.

**Mechanical quality factor**

Factor that affects the damping across the R and C mechanical ports. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is Inf.

### **Damping**

Translational damping term. This parameter is only visible when you select **Specify material properties** for the **Parameterization** parameter. The default value is 0 N/(m/s).

### **Effective mass**

Mass that approximates the distributed dynamics of the device and causes the stack to resonate at the correct frequency when attached to the mechanical R port. This mass is usually about one third of the actual stack mass. This parameter is only visible when you select **Specify material properties** for the **Parameterization** parameter. The default value is 0 g.

## **Initial Conditions Tab**

### **Initial stack deflection**

Stack deflection at time zero. If you have an external Ideal Translational Motion Sensor block attached across the Piezo Stack block, you must use the same initial deflection parameter for both blocks. The default value is 0 mm.

### **Initial voltage**

Stack voltage at time zero. The default value is 0 V.

## **Ports**

The block has the following ports:

+

Positive electrical port

-

Negative electrical port

C

Mechanical translational conserving port

R

Mechanical translational conserving port

# PMSM Current Controller

Permanent magnet synchronous machine current controller

**Library:** Simscape / Electrical / Power Systems / Control / PMSM Control



## Description

The PMSM Current Controller block implements a discrete-time PI-based PMSM current controller in the rotor d-q reference frame.

You typically use this block in a series of blocks making up a control structure.

- You can generate a current reference in the d-q frame to be used as an input to this block with a PMSM Current Reference Generator.
- You can obtain a voltage reference in the abc domain by converting the output of this block using an Inverse Park Transform block.

You can see an example of a full control structure, from machine measurements to machine inputs, in the PMSM Field-Oriented Control block.

## Equations

The block is discretized using the backward Euler method due to its first-order simplicity and its stability.

Two PI current controllers implemented in the rotor reference frame produce the reference voltage vector:

$$v_d^{ref} = \left( K_{p\_id} + K_{i\_id} \frac{T_s z}{z-1} \right) (i_d^{ref} - i_d) + v_{d\_FF},$$

and

$$v_q^{ref} = \left( K_{p\_iq} + K_{i\_iq} \frac{T_s z}{z-1} \right) (i_q^{ref} - i_q) + v_{q\_FF},$$

where:

- $v_d^{ref}$  and  $v_q^{ref}$  are the  $d$ -axis and  $q$ -axis reference voltages, respectively.
- $i_d^{ref}$  and  $i_q^{ref}$  are the  $d$ -axis and  $q$ -axis reference currents, respectively.
- $i_d$  and  $i_q$  are the  $d$ -axis and  $q$ -axis currents, respectively.
- $K_{p\_id}$  and  $K_{p\_iq}$  are the proportional gains for the  $d$ -axis and  $q$ -axis controllers, respectively.
- $K_{i\_id}$  and  $K_{i\_iq}$  are the integral gains for the  $d$ -axis and  $q$ -axis controllers, respectively.
- $v_{d\_FF}$  and  $v_{q\_FF}$  are the feedforward voltages for the  $d$ -axis and  $q$ -axis, respectively, obtained from the machine mathematical equations and provided as inputs.
- $T_s$  is the sample time of the discrete controller.

## Zero Cancellation

Using PI control results in a zero in the closed-loop transfer function, which can result in undesired overshoot in the closed-loop response. This zero can be canceled by introducing a zero-cancellation block in the feedforward path. The zero cancellation transfer functions in discrete time are:

$$G_{ZC\_id}(z) = \frac{\frac{T_s K_{i\_id}}{K_{p\_id}}}{z + \left( \frac{T_s - \frac{K_{p\_id}}{K_{i\_id}}}{\frac{K_{p\_id}}{K_{i\_id}}} \right)},$$

and



$$G_{ZC\_iq}(z) = \frac{\frac{T_s K_{i\_iq}}{K_{p\_iq}}}{z + \left( \frac{T_s - \frac{K_{p\_iq}}{K_{i\_iq}}}{\frac{K_{p\_iq}}{K_{i\_iq}}} \right)}$$

## Voltage Saturation

Saturation must be imposed when the stator voltage vector exceeds the voltage phase limit  $V_{ph\_max}$ :

$$\sqrt{v_d^2 + v_q^2} \leq V_{ph\_max},$$

where  $v_d$  and  $v_q$  are the  $d$ -axis and  $q$ -axis voltages, respectively.

In the case of axis prioritization, the voltages  $v_1$  and  $v_2$  are introduced, where:

- $v_1 = v_d$  and  $v_2 = v_q$  for  $d$ -axis prioritization.
- $v_1 = v_q$  and  $v_2 = v_d$  for  $q$ -axis prioritization.

The constrained (saturated) voltages  $v_1^{sat}$  and  $v_2^{sat}$  are obtained as follows:

$$v_1^{sat} = \min\left(\max\left(v_1^{unsat}, -V_{ph\_max}\right), V_{ph\_max}\right)$$

and

$$v_2^{sat} = \min\left(\max\left(v_2^{unsat}, -V_{2\_max}\right), V_{2\_max}\right),$$

where:

- $v_1^{unsat}$  and  $v_2^{unsat}$  are the unconstrained (unsaturated) voltages.

- $v_{2\_max}$  is the maximum value of  $v_2$  that does not exceed the voltage phase limit, given

$$\text{by } v_{2\_max} = \sqrt{(V_{ph\_max})^2 - (v_1^{sat})^2}.$$

In the case that the direct and quadrature axes have the same priority (d-q equivalence) the constrained voltages are obtained as follows:

$$v_d^{sat} = \min \left( \max \left( v_d^{unsat}, -V_{d\_max} \right), V_{d\_max} \right)$$

and

$$v_q^{sat} = \min \left( \max \left( v_q^{unsat}, -V_{q\_max} \right), V_{q\_max} \right),$$

where

$$V_{d\_max} = \frac{V_{ph\_max} |v_d^{unsat}|}{\sqrt{(v_d^{unsat})^2 + (v_q^{unsat})^2}}$$

and

$$V_{q\_max} = \frac{V_{ph\_max} |v_q^{unsat}|}{\sqrt{(v_d^{unsat})^2 + (v_q^{unsat})^2}}.$$

## Integral Anti-Windup

An anti-windup mechanism is employed to avoid saturation of integrator output. In such a situation, the integrator gains become:

$$K_{i\_id} + K_{aw\_id} \left( v_d^{sat} - v_d^{unsat} \right)$$

and

$$K_{i\_iq} + K_{aw\_iq} \left( v_q^{sat} - v_q^{unsat} \right),$$

where  $K_{aw\_id}$  and  $K_{aw\_iq}$  are the anti-windup gains for the  $d$ -axis and  $q$ -axis, respectively.

## Assumptions

- The plant model for direct and quadrature axis can be approximated with a first-order system.
- This control solution is used only for permanent magnet synchronous motors with sinusoidal flux distribution and field windings.

## Ports

### Input

#### **idqRef** — Reference currents

vector

Desired  $d$ - and  $q$ -axis currents for control of a PMSM, in A.

Data Types: `single` | `double`

#### **idq** — Measured currents

vector

Actual  $d$ - and  $q$ -axis currents of the controlled PMSM, in A.

Data Types: `single` | `double`

#### **vdqFF** — Feedforward voltages

vector

Feedforward pre-control voltages, in V.

Data Types: `single` | `double`

#### **VphMax** — Maximum phase voltage

scalar

Maximum allowable voltage in each phase, in V.

Data Types: `single` | `double`

## **Reset — External reset**

scalar

External reset signal (rising edge) for integrators.

Data Types: `single` | `double`

## **Output**

### **vdqRef — Reference voltages**

vector

Desired  $d$ - and  $q$ -axis voltages for control of a PMSM, in V.

Data Types: `single` | `double`

## **Parameters**

### **Control Parameters**

#### **D-axis current proportional gain — D-axis proportional gain**

1 (default) | positive number

Proportional gain of the PI controller used for direct-axis current control.

#### **D-axis current integral gain — D-axis integral gain**

100 (default) | positive number

Integrator gain of the PI controller used for direct-axis current control.

#### **D-axis current anti-windup gain — D-axis anti-windup gain**

1 (default) | positive number

Anti-windup gain of the PI controller used for direct-axis current control.

#### **Q-axis current proportional gain — Q-axis proportional gain**

1 (default) | positive number

Proportional gain of the PI controller used for quadrature-axis current control.

#### **Q-axis current integral gain — Q-axis integral gain**

100 (default) | positive number

Integrator gain of the PI controller used for quadrature-axis current control.

**Q-axis current anti-windup gain — Q-axis anti-windup gain**

1 (default) | positive number

Anti-windup gain of the PI controller used for quadrature-axis current control.

**Sample time (-1 for inherited) — Block sample time**

-1 (default) | -1 or positive number

Sample time for the block (-1 for inherited). If you use this block inside a triggered subsystem, set the sample time to -1. If you use this block in a continuous variable-step model, you can specify the sample time explicitly.

**Axis prioritization — Axis prioritization for voltage limiter**

q-axis (default) | d-axis | d-q equivalence

Prioritize or maintain the ratio between the  $d$ - and  $q$ -axes when the block limits voltage.

**Enable zero cancellation — Feedforward zero-cancellation**

off (default) | on

Enable or disable zero-cancellation on the feedforward path.

**Enable pre-control voltage — Pre-control voltage**

on (default) | off

Enable or disable pre-control voltage.

## References

- [1] Bernardes, T., V. F. Montagner, H. A. Gründling, and H. Pinheiro. "Discrete-time sliding mode observer for sensorless vector control of permanent magnet synchronous machine." *IEEE Transactions on Industrial Electronics*. Vol. 61, Number 4, 2014, pp. 1679-1691.
- [2] Carpiuc, S., and C. Lazar. "Fast real-time constrained predictive current control in permanent magnet synchronous machine-based automotive traction drives." *IEEE Transactions on Transportation Electrification*. Vol.1, Number 4, 2015, pp. 326-335.

## **See Also**

### **Blocks**

PMSM Current Controller with Pre-Control | PMSM Current Reference Generator

**Introduced in R2017b**

# PMSM Current Controller with Pre-Control

Permanent magnet synchronous machine current controller with pre-control

**Library:** Simscape / Electrical / Power Systems / Control / PMSM Control



## Description

The PMSM Current Controller with Pre-Control block implements a discrete-time PI-based PMSM current controller in the rotor d-q reference frame with internal feedforward pre-control.

You typically use this block in a series of blocks making up a control structure.

- You can generate a current reference in the d-q frame to be used as an input to this block with a PMSM Current Reference Generator.
- You can obtain a voltage reference in the abc domain by converting the output of this block using an Inverse Park Transform block.

You can see an example of a full control structure, from machine measurements to machine inputs, in the PMSM Field-Oriented Control block.

## Equations

The block is discretized using the backward Euler method due to its first-order simplicity and its stability.

Two PI current controllers implemented in the rotor reference frame produce the reference voltage vector:

$$v_d^{ref} = \left( K_{p\_id} + K_{i\_id} \frac{T_s z}{z-1} \right) (i_d^{ref} - i_d) + v_{d\_FF},$$

and

$$v_q^{ref} = \left( K_{p\_iq} + K_{i\_iq} \frac{T_s z}{z-1} \right) (i_q^{ref} - i_q) + v_{q\_FF},$$

where:

- $v_d^{ref}$  and  $v_q^{ref}$  are the  $d$ -axis and  $q$ -axis reference voltages, respectively.
- $i_d^{ref}$  and  $i_q^{ref}$  are the  $d$ -axis and  $q$ -axis reference currents, respectively.
- $i_d$  and  $i_q$  are the  $d$ -axis and  $q$ -axis currents, respectively.
- $K_{p\_id}$  and  $K_{p\_iq}$  are the proportional gains for the  $d$ -axis and  $q$ -axis controllers, respectively.
- $K_{i\_id}$  and  $K_{i\_iq}$  are the integral gains for the  $d$ -axis and  $q$ -axis controllers, respectively.
- $T_s$  is the sample time of the discrete controller.
- $v_{d\_FF}$  and  $v_{q\_FF}$  are the feedforward voltages for the  $d$ -axis and  $q$ -axis, respectively.

The feedforward voltages are obtained from the machine mathematical equations:

$$v_{d\_FF} = -\omega_e L_q i_q,$$

and

$$v_{q\_FF} = \omega_e (L_d i_d + \psi_m),$$

where:

- $\omega_e$  is the rotor electrical velocity.
- $L_d$  and  $L_q$  are the  $d$ -axis and  $q$ -axis inductances, respectively.
- $\psi_m$  is the permanent magnet flux linkage.



## Zero Cancellation

Using PI control results in a zero in the closed-loop transfer function, which can result in undesired overshoot in the closed-loop response. This zero can be canceled by introducing a zero-cancellation block in the feedforward path. The zero cancellation transfer functions in discrete time are:

$$G_{ZC\_id}(z) = \frac{\frac{T_s K_{i\_id}}{K_{p\_id}}}{z + \left( \frac{T_s - \frac{K_{p\_id}}{K_{i\_id}}}{\frac{K_{p\_id}}{K_{i\_id}}} \right)},$$

and

$$G_{ZC\_iq}(z) = \frac{\frac{T_s K_{i\_iq}}{K_{p\_iq}}}{z + \left( \frac{T_s - \frac{K_{p\_iq}}{K_{i\_iq}}}{\frac{K_{p\_iq}}{K_{i\_iq}}} \right)}.$$

## Voltage Saturation

Saturation must be imposed when the stator voltage vector exceeds the voltage phase limit  $V_{ph\_max}$ :

$$\sqrt{v_d^2 + v_q^2} \leq V_{ph\_max},$$

where  $v_d$  and  $v_q$  are the  $d$ -axis and  $q$ -axis voltages, respectively.

In the case of axis prioritization, the voltages  $v_1$  and  $v_2$  are introduced, where:

- $v_1 = v_d$  and  $v_2 = v_q$  for  $d$ -axis prioritization.

- $v_1 = v_q$  and  $v_2 = v_d$  for  $q$ -axis prioritization.

The constrained (saturated) voltages  $v_1^{sat}$  and  $v_2^{sat}$  are obtained as follows:

$$v_1^{sat} = \min\left(\max\left(v_1^{unsat}, -V_{ph\_max}\right), V_{ph\_max}\right)$$

and

$$v_2^{sat} = \min\left(\max\left(v_2^{unsat}, -V_{2\_max}\right), V_{2\_max}\right),$$

where:

- $v_1^{unsat}$  and  $v_2^{unsat}$  are the unconstrained (unsaturated) voltages.
- $v_{2\_max}$  is the maximum value of  $v_2$  that does not exceed the voltage phase limit, given

$$\text{by } v_{2\_max} = \sqrt{\left(V_{ph\_max}\right)^2 - \left(v_1^{sat}\right)^2}.$$

In the case that the direct and quadrature axes have the same priority (d-q equivalence), the constrained voltages are obtained as follows:

$$v_d^{sat} = \min\left(\max\left(v_d^{unsat}, -V_{d\_max}\right), V_{d\_max}\right)$$

and

$$v_q^{sat} = \min\left(\max\left(v_q^{unsat}, -V_{q\_max}\right), V_{q\_max}\right),$$

where:

$$V_{d\_max} = \frac{V_{ph\_max} |v_d^{unsat}|}{\sqrt{\left(v_d^{unsat}\right)^2 + \left(v_q^{unsat}\right)^2}}$$

and

$$V_{q\_max} = \frac{V_{ph\_max} |v_q^{unsat}|}{\sqrt{(v_d^{unsat})^2 + (v_q^{unsat})^2}}.$$

## Integral Anti-Windup

An anti-windup mechanism is employed to avoid saturation of integrator output. In such a situation, the integrator gains become:

$$K_{i\_id} + K_{aw\_id} (v_d^{sat} - v_d^{unsat})$$

and

$$K_{i\_iq} + K_{aw\_iq} (v_q^{sat} - v_q^{unsat}),$$

where  $K_{aw\_id}$ ,  $K_{aw\_iq}$ , and  $K_{aw\_if}$  are the anti-windup gains for the  $d$ -axis,  $q$ -axis, and field controllers, respectively.

## Assumptions

- The plant model for direct and quadrature axis can be approximated with a first-order system.
- This control solution is used only for permanent magnet synchronous motors with sinusoidal flux distribution and field windings.

## Ports

### Input

#### **idqRef** — Reference currents

vector

Desired  $d$ - and  $q$ -axis currents for control of a PMSM, in A.

Data Types: single | double

## **idq — Measured currents**

vector

Actual  $d$ - and  $q$ -axis currents of the controlled PMSM, in A.

Data Types: `single` | `double`

## **wElectrical — Measured electrical velocity**

vector

Rotor electrical velocity used for feedforward pre-control, in rad/s.

Data Types: `single` | `double`

## **VphMax — Maximum phase voltage**

scalar

Maximum allowable voltage in each phase, in V.

Data Types: `single` | `double`

## **Reset — External reset**

scalar

External reset signal (rising edge) for integrators.

Data Types: `single` | `double`

## **Output**

### **vdqRef — Reference voltages**

vector

Desired  $d$ - and  $q$ -axis voltages for control of a PMSM, in V.

Data Types: `single` | `double`

## **Parameters**

### **Control Parameters**

#### **D-axis current proportional gain — D-axis proportional gain**

1 (default) | positive number

Proportional gain of the PI controller used for direct-axis current control.

**D-axis current integral gain — D-axis integral gain**

100 (default) | positive number

Integrator gain of the PI controller used for direct-axis current control.

**D-axis current anti-windup gain — D-axis anti-windup gain**

1 (default) | positive number

Anti-windup gain of the PI controller used for direct-axis current control.

**Q-axis current proportional gain — Q-axis proportional gain**

1 (default) | positive number

Proportional gain of the PI controller used for quadrature-axis current control.

**Q-axis current integral gain — Q-axis integral gain**

100 (default) | positive number

Integrator gain of the PI controller used for quadrature-axis current control.

**Q-axis current anti-windup gain — Q-axis anti-windup gain**

1 (default) | positive number

Anti-windup gain of the PI controller used for quadrature-axis current control.

**Sample time (-1 for inherited) — Block sample time**

-1 (default) | -1 or positive number

Sample time for the block (-1 for inherited). If you use this block inside a triggered subsystem, set the sample time to -1. If you use this block in a continuous variable-step model, you can specify the sample time explicitly.

**Axis prioritization — Axis prioritization for voltage limiter**

q-axis (default) | d-axis | d-q equivalence

Prioritize or maintain the ratio between  $d$ - and  $q$ -axes when the block limits voltage.

**Enable zero cancellation — Feedforward zero-cancellation**

off (default) | on

Enable or disable zero-cancellation on the feedforward path.

## **Enable pre-control voltage — Pre-control voltage**

on (default) | off

Enable or disable pre-control voltage.

## **Pre-Control Parameters**

### **D-axis current vector, $i_d$ (A) — D-axis current breakpoint vector**

$[-200, 0, 200]$ A (default) | monotonically increasing vector

Direct-axis current vector used in the lookup tables for parameters determination. For constant machine parameters, do not change the default.

### **Q-axis current vector, $i_q$ (A) — Q-axis current breakpoint vector**

$[-200, 0, 200]$ A (default) | monotonically increasing vector

Quadrature-axis current vector used in the lookup tables used to determine parameters. For constant machine parameters, do not change the default.

### **$L_d$ matrix, $L_d(i_d, i_q)$ (H) — D-axis inductance lookup data**

$0.0002 * \text{ones}(3, 3)$ H (default) | positive matrix

$L_d$  matrix used as lookup-table data. For constant machine parameters change only the constant factor, for example,  $L_d * \text{ones}(3, 3)$ .

### **$L_q$ matrix, $L_q(i_d, i_q)$ (H) — Q-axis inductance lookup data**

$0.0002 * \text{ones}(3, 3)$ H (default) | positive matrix

$L_q$  matrix used as lookup-table data. For constant machine parameters change only the constant factor, e.g.,  $L_q * \text{ones}(3, 3)$ .

### **Permanent magnet flux linkage matrix, $PM(i_d, i_q)$ (Wb) — Flux linkage lookup data**

$0.04 * \text{ones}(3, 3)$ Wb (default) | real matrix

Permanent magnet flux linkage matrix used in the lookup table. For constant machine parameters change only the constant factor, for example  $psim * \text{ones}(3, 3)$ .

## **References**

- [1] Bernardes, T., V. F. Montagner, H. A. Gründling, and H. Pinheiro. "Discrete-time sliding mode observer for sensorless vector control of permanent magnet synchronous

machine." *IEEE Transactions on Industrial Electronics*. Vol. 61, Number 4, 2014, pp. 1679-1691.

- [2] Carpiuc, S., and C. Lazar. "Fast real-time constrained predictive current control in permanent magnet synchronous machine-based automotive traction drives." *IEEE Transactions on Transportation Electrification*. Vol.1, Number 4, 2015, pp. 326-335.

## **See Also**

### **Blocks**

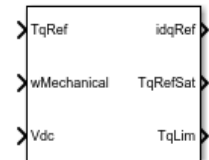
PMSM Current Controller | PMSM Current Reference Generator

**Introduced in R2017b**

# PMSM Current Reference Generator

Permanent magnet synchronous machine current reference generator

**Library:** Simscape / Electrical / Power Systems / Control / PMSM Control



## Description

The PMSM Current Reference Generator block implements a current reference generator for PMSM current control in the rotor d-q reference frame.

You typically use this block in a series of blocks making up a control structure.

- You can generate a voltage reference in the d-q frame by placing this block before a PMSM Current Control or PMSM Current Control with Pre-Control block.
- You can implement velocity control by placing this block after a Velocity Controller block.

You can see an example of a full control structure, from machine measurements to machine inputs, in the PMSM Field-Oriented Control block.

## Equations

The PMSM Current Reference Generator block can obtain the current reference using one of these methods:

- Zero  $d$ -axis control (ZDAC)
- User defined lookup tables
- Automatically generated lookup tables

For the ZDAC method, the block sets the  $d$ -axis current reference  $i_d^{ref}$  to zero and determines the  $q$ -axis current reference  $i_q^{ref}$  using the torque equation:



$$i_d^{ref} = 0,$$

and

$$i_q^{ref} = \frac{2T_{ref}}{3p\psi_m},$$

where:

- $T_{ref}$  is the reference torque input.
- $p$  is the number of pole pairs.
- $\psi_m$  is the permanent magnet flux linkage.

For operation below the base speed of the synchronous machine, ZDAC is a suitable method. Above base speed, a field weakening controller is required to adjust the  $d$ -axis reference.

To pregenerate optimal current references for several operating points offline, define two lookup tables using the user-defined lookup table approach:

$$i_d^{ref} = f(n_m, T_{ref}, v_{dc}),$$

and

$$i_q^{ref} = g(n_m, T_{ref}, v_{dc}),$$

where:

- $n_m$  is the rotor angular velocity.
- $v_{dc}$  is the DC-link voltage of the converter.

To let the block create the lookup tables, choose the automatically generated lookup table approach. The block generates the lookup table using two strategies:

- Maximum torque per ampere
- Field weakening

The selection between the two strategies is based on the modulation factor, which can be computed as follows:

$$M_f = \frac{V_s}{V_{ph\_max}},$$

where  $V_s$  is the stator voltage amplitude and  $V_{ph\_max}$  is the maximum allowable phase voltage. In the case that the modulation factor is greater than 1, the block generates current references using the field weakening procedure. Otherwise, current references are computed using the maximum torque per ampere procedure.

## Maximum Torque Per Ampere

You can generate current references in the constant torque region (occurring below rated speed) by using the maximum torque per ampere (MTPA) strategy.

The direct and quadrature components of the stator current are written in terms of angle and magnitude as:

$$i_d = -I_s \sin\beta,$$

and

$$i_q = I_s \cos\beta,$$

where:

- $\beta$  is the angle of the stator current vector.
- $I_s$  is the stator current amplitude.

Using the angle-magnitude variant of the d-q currents, the PMSM torque equation is written as:

$$T_e = \frac{3p}{2} \psi_m I_s \cos\beta + \frac{3p}{4} (L_q - L_d) I_s^2 \sin 2\beta,$$

where  $L_d$  and  $L_q$  are the direct and quadrature inductances, respectively.

To obtain fast transient response and maximize torque with the smallest possible stator current amplitude, MTPA imposes  $(dT_e)/d\beta = 0$  to the torque equation, which yields

$$-\frac{3p}{2}\psi_m I_s \sin\beta + \frac{3p}{2}(L_q - L_d) I_s^2 (\cos^2\beta - \sin^2\beta) = 0.$$

The MTPA  $d$ -axis current  $i_{d\_mtpa}$  is written in terms of the  $q$ -axis component  $i_{q\_mtpa}$  by substituting the d-q currents back from their angle and magnitude variants:

$$i_{d\_mtpa} = \frac{\psi_m}{2(L_q - L_d)} - \sqrt{\frac{\psi_m^2}{4(L_q - L_d)^2} + i_{q\_mtpa}^2}.$$

Finally, by plugging the previous equation into the d-q variant of the PMSM torque equation, the following polynomial is obtained:

$$9p^2(L_q - L_d)^2 i_{q\_mtpa}^4 + 6T_{ref} p \psi_m i_{q\_mtpa} - 4T_{ref}^2 = 0.$$

The  $q$ -axis component is obtained by solving this polynomial.

## Field Weakening

You can generate current references in the above rated speed region by using the field weakening (FW) strategy.

Above the rated speed, the stator voltage is limited by the power converter and the available DC-link voltage. The maximum stator voltage is:

$$V_s = \sqrt{v_d^2 + v_q^2} \leq V_{ph\_max},$$

where  $V_{ph\_max}$  is the maximum available stator phase voltage.

The steady-state voltage equations for PMSMs are

$$v_d = R_s i_d - \omega_e L_q i_q,$$

and

$$v_q = R_s i_q + \omega_e (L_d i_d + \psi_m).$$

For rotor speeds above rated, the stator resistance is negligible, and the field weakening  $d$ -axis current component  $i_{d\_fw}$  is obtained in terms of the  $q$ -axis component  $i_{q\_fw}$  from the  $v_q$  steady-state equation:

$$i_{d\_fw} = -\frac{\psi_m}{L_d} + \frac{1}{L_d} \sqrt{\frac{V_{ph\_max}^2}{\omega_e^2} - (L_q i_{q\_fw})^2},$$

Finally, by plugging the  $i_{d\_fw}$  equation into the PMSM torque equation, the following polynomial is obtained:

$$9p^2(L_d - L_q)^2 L_q^2 \omega_e^2 i_{q\_fw}^4 + \left( 9p^2 \psi_m^2 L_q^2 \omega_e^2 - 9p^2 (L_d - L_q)^2 V_{ph\_max}^2 \right) i_{q\_fw}^2 - 12T_{ref} p \psi_m L_d L_q \omega_e^2 i_{q\_fw} + 4T_{ref}^2 = 0$$

The  $q$ -axis component is obtained by solving this polynomial.

## Assumptions

The machine parameters are constants.

## Limitations

The automatically generated current references introduce latency in the presimulation phase. For medium-power PMSM drives the latency is around 300 ms.

## Ports

### Input

#### **TqRef** — Reference torque

scalar

Desired mechanical torque produced by the PMSM, in N\*m.

Data Types: `single` | `double`

#### **wMechanical** — Rotor mechanical speed

scalar

Mechanical angular velocity of the rotor, obtained via direct measurement of the PMSM, in rad/s.

Data Types: `single` | `double`

### **Vdc — DC-link voltage**

scalar

DC-link voltage of the converter, in V. For the ZDAC method, this value is used to limit the output reference torque and torque limit. For the lookup table method, this value is used as an input to the lookup tables.

Data Types: `single` | `double`

## **Output**

### **idqRef — Reference currents**

vector

Reference  $d$ - and  $q$ -currents to be given as inputs to a PMSM current controller, in A.

Data Types: `single` | `double`

### **TqRefSat — Reference torque**

scalar

Reference torque saturated by the calculated torque limit **TqLim**, in N\*m.

Data Types: `single` | `double`

### **TqLim — Torque limit**

scalar

Torque limit imposed by both the electrical and mechanical constraints of the system, in N\*m.

Data Types: `single` | `double`

## Parameters

### General Parameters

#### **Nominal dc-link voltage (V) — Rated DC voltage**

300V (default) | positive number

Nominal DC-link voltage of the electrical source.

#### **Maximum power (W) — Rated power**

30000W (default) | positive number

Maximum allowable PMSM power.

#### **Maximum torque (N\*m) — Rated torque**

250N\*m (default) | positive number

Maximum allowable PMSM torque.

#### **Sample time (-1 for inherited) — Block sample time**

-1 (default) | -1 or positive number

Sample time for the block (-1 for inherited). If you use this block inside a triggered subsystem, set the sample time to -1. If you use this block in a continuous variable-step model, you can specify the sample time explicitly.

### Reference Generation Strategy

#### **Current references — Current reference strategy**

Zero d-axis control (default) | Lookup-table based | Automatically generated lookup-table

Select the strategy for determining current references.

#### **Mechanical speed vector, wMechanical (rpm) — Rotor speed lookup vector**

[0, 3000] rpm (default) | positive monotonically increasing vector

Speed vector used in the lookup tables for determining current references.

#### **Torque reference vector, TqRef (N\*m) — Torque reference lookup vector**

[-100, 0, 100]N\*m (default) | positive monotonically increasing vector

Torque vector used in the lookup tables for determining current references.

**DC-link voltage vector,  $V_{dc}$  (V) — DC-link voltage lookup vector**

[300, 350]V (default) | positive monotonically increasing vector

DC-link voltage vector used in the lookup tables for determining current references.

**D-axis current reference matrix,  $i_d(\omega_{Mechanical}, T_qRef, V_{dc})$  (A) — Reference d-axis current values**

zeros(2,3,2)A (default) | real matrix

Direct-axis current reference lookup data.

**Q-axis current reference matrix,  $i_q(\omega_{Mechanical}, T_qRef, V_{dc})$  (A) — Reference q-axis current values**

zeros(2,3,2)A (default) | real matrix

Quadrature-axis current reference lookup data.

**Number of pole pairs — Pole pairs**

8 (default) | positive integer

Number of permanent magnet pole pairs on the rotor.

**Permanent magnet flux linkage (Wb) — PM Flux Linkage**

0.04Wb (default) | positive scalar

Peak permanent magnet flux linkage.

**D-axis inductance (H) — Inductance of d-axis**

0.00024 (default) | positive scalar

Direct-axis inductance.

**Q-axis inductance (H) — Inductance of q-axis**

0.00029 (default) | positive scalar

Quadrature-axis inductance.

**Stator resistance (Ohm) — Resistance of stator**

0.01 (default) | positive scalar

Stator resistance per phase.

## References

- [1] Haque, M. E., L. Zhong, and M. F. Rahman. "Improved trajectory control for an interior permanent magnet synchronous motor drive with extended operating limit." *Journal of Electrical & Electronics Engineering*. Vol. 22, Number 1, 2003, p. 49.
- [2] Yang, N., G. Luo, W. Liu, and K. Wang. "Interior permanent magnet synchronous motor control for electric vehicle using look-up table." *In 7th International Power Electronics and Motion Control Conference*. Vol. 2, 2012, pp. 1015-1019.
- [3] Carpiuc, S., C. Lazar, and D. I. Patrascu. "Optimal Torque Control of the Externally Excited Synchronous Machine." *Control Engineering and Applied Informatics*. Vol. 14, Number 2, 2012, pp. 80-88.

## See Also

### Blocks

PMSM Current Controller | PMSM Current Controller with Pre-Control

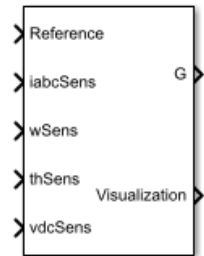
**Introduced in R2017b**



# PMSM Field-Oriented Control

Permanent magnet synchronous machine field-oriented control

**Library:** Simscape / Electrical / Power Systems / Control / PMSM Control



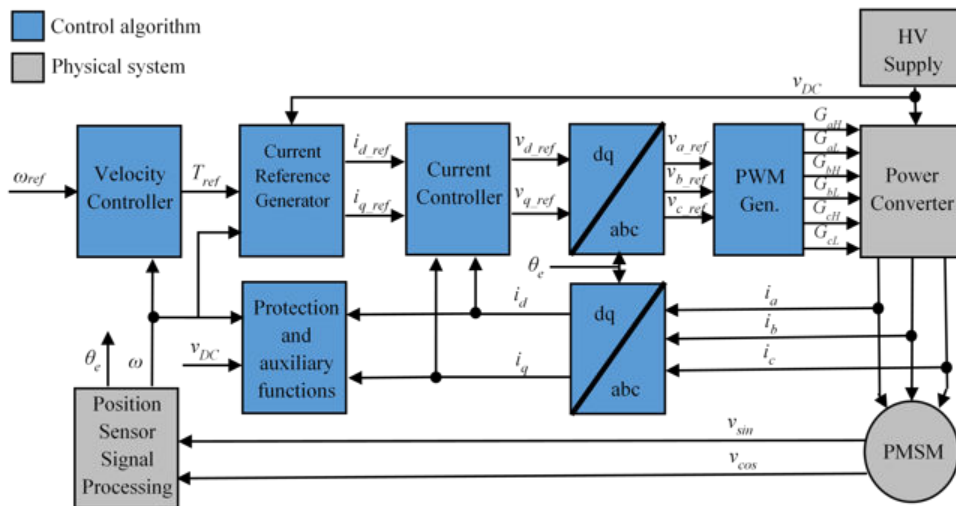
## Description

The PMSM Field-Oriented Control block implements a PMSM field-oriented control structure. Field Oriented Control (FOC) is a performant AC motor control strategy that decouples torque and flux by transforming the stationary phase currents to a rotating frame. Use FOC when rotor speed and position are known and your application requires:

- High torque and low current at startup.
- High efficiency.

## Equations

The PMSM FOC structure decouples the torque and flux by using the rotor d-q reference frame. This diagram shows the overall architecture of the block.



In the diagram:

- $\omega$  and  $\omega_{ref}$  are the measured and reference angular velocities, respectively.
- $T_{ref}$  is the reference electromagnetic torque.
- $i$  and  $v$  are stator currents and voltages and subscripts  $d$  and  $q$  represent the  $d$ -axis and  $q$ -axis, and subscripts  $a$ ,  $b$ , and  $c$ , represent the three stator windings.
- $\theta_e$  is the rotor electrical angle.
- $G$  is a gate pulse, subscripts  $H$  and  $L$ , represent high and low, and subscripts  $a$ ,  $b$ , and  $c$  represent the three stator windings.

You can choose to implement either velocity or torque control with the `Control mode` parameter. The block implements velocity control exactly as shown in the diagram. The block implements torque control by removing the Velocity Controller block and accepting the reference torque directly.

## Assumptions

The machine parameters are known.

## Limitations

The control structure is implemented with a single sample rate.

## Ports

### Input

#### **Reference — System reference**

scalar

System reference specified as torque reference in N\*m or velocity reference in rad/s, depending on the control mode selected.

Data Types: `single` | `double`

#### **iabcSens — Measured phase currents**

vector

Measured stator phase currents, in A.

Data Types: `single` | `double`

#### **wSens — Rotor speed**

scalar

Measured mechanical angular velocity of rotor, in rad/s.

Data Types: `single` | `double`

#### **thSens — Rotor angle**

scalar

Measured mechanical angle of rotor, in rad.

Data Types: `single` | `double`

#### **vdcSens — DC-link voltage**

scalar

Measured DC-link voltage, in V.

Data Types: single | double

## Output

### **G — Gate pulses**

vector

Six pulse waveforms that determine switching behavior in the attached power converter.

Data Types: single | double

### **Visualization — Visualization signals**

bus

Bus containing signals for visualization, including:

- Reference
- wElectrical
- iabc
- theta
- Vdc
- PwmEnable
- TqRef
- TqLim
- idqRef
- idq
- vdqRef
- modWave

Data Types: single | double

## Parameters

### **General**

#### **Control Mode — Control mode strategy**

Torque control (default) | Velocity control

Specify either a torque control or velocity control strategy.

**Nominal dc-link voltage (V) — Rated DC voltage**

300 V (default) | positive number

Nominal DC-link voltage of the electrical source.

**Maximum power (W) — Maximum power**

35000 W (default) | positive number

Maximum machine power.

**Maximum torque (N\*m) — Maximum torque**

250 N\*m (default) | positive number

Maximum machine torque.

**Number of rotor pole pairs — Pole pairs**

8 (default) | positive integer

Number of permanent magnet pole pairs on the rotor.

**Inverter dc-link voltage threshold (V) — DC-link voltage threshold**

100 V (default) | positive number

Voltage threshold to activate the power inverter.

**Fundamental sample time (s) — Block sample time**

5e-6 (default) | positive number

Fundamental sample time for the block.

**Control sample time (s) — Control sample time**

1e-4 (default) | positive number

Sample time for the control system.

**Outer Loop****Control Type — Control type strategy**

PI control (default) | P control | P-PI control

Specify the type of the control strategy.

**Controller proportional gain — Proportional gain of PI controller**

1 (default) | positive number

Proportional gain of the PI controller.

**Controller integral gain — Integral gain of PI controller**

1 (default) | positive number

Integral gain of the PI controller.

**P controller proportional gain — Proportional gain of P controller**

1 (default) | positive number

Proportional gain of P controller.

**Integral anti-windup gain — Anti-windup gain**

1 (default) | positive number

Anti-windup gain of the PI controller.

**Current references — Current reference strategy**

Zero d-axis control (default) | Lookup-table based | Automatically generated lookup-table

Select the current reference strategy.

**Mechanical speed vector,  $w_{\text{Mechanical}}$  (rpm) — Rotor speed lookup vector**

[0, 3000] rpm (default) | positive monotonically increasing vector

Speed vector used in the lookup tables for determining current references.

**Torque reference vector,  $T_q\text{Ref}$  (N\*m) — Torque reference lookup vector**

[-100, 0, 100] N\*m (default) | positive monotonically increasing vector

Torque vector used in the lookup tables for determining current references.

**DC-link voltage vector,  $V_{dc}$  (V) — DC-link voltage lookup vector**

[300, 350] V (default) | positive monotonically increasing vector

: DC-link voltage vector used in the lookup tables for determining current references.

**D-axis current reference matrix,  $id(w_{\text{Mechanical}}, Tq_{\text{Ref}}, V_{\text{dc}})$  (A) — Reference d-axis current values**

zeros(2,3,2) A (default) | real matrix

Direct-axis current reference lookup data.

**Q-axis current reference matrix,  $iq(w_{\text{Mechanical}}, Tq_{\text{Ref}}, V_{\text{dc}})$  (A) — Reference q-axis current values**

zeros(2,3,2) A (default) | real matrix

Quadrature-axis current reference lookup data.

**Permanent magnet flux linkage (Wb) — PM Flux Linkage**

0.04 Wb (default) | positive scalar

Peak permanent magnet flux linkage.

**D-axis inductance (H) — Inductance of d-axis**

0.00024 (default) | positive scalar

Direct-axis inductance.

**Q-axis inductance (H) — Inductance of q-axis**

0.00029 (default) | positive scalar

Quadrature-axis inductance.

**Stator resistance (Ohm) — Resistance of stator**

0.01 (default) | positive scalar

Stator resistance per phase.

**Inner Loop****D-axis current proportional gain — D-axis proportional gain**

1 (default) | positive number

Proportional gain of the PI controller used for direct-axis current control.

**D-axis current integral gain — D-axis integral gain**

100 (default) | positive number

Integrator gain of the PI controller used for direct-axis current control.

**D-axis current anti-windup gain — D-axis anti-windup gain**

1 (default) | positive number

Anti-windup gain of the PI controller used for direct-axis current control.

**Q-axis current proportional gain — Q-axis proportional gain**

1 (default) | positive number

Proportional gain of the PI controller used for quadrature-axis current control.

**Q-axis current integral gain — Q-axis integral gain**

100 (default) | positive number

Integrator gain of the PI controller used for quadrature-axis current control.

**Q-axis current anti-windup gain — Q-axis anti-windup gain**

1 (default) | positive number

Anti-windup gain of the PI controller used for quadrature-axis current control.

**Axis prioritization — Axis prioritization for voltage limiter**

q-axis (default) | d-axis | d-q equivalence

Prioritize or maintain ratio between  $d$ - and  $q$ -axis when the block limits voltage.

**Enable zero cancellation — Feedforward zero cancellation**

off (default) | on

Enable or disable zero cancellation on the feedforward path.

**Enable pre-control voltage — Precontrol voltage**

on (default) | off

Enable or disable precontrol voltage.

**Machine parameters — Machine parameterization**

Constant parameters (default) | Lookup table based parameters

Specify how to parameterize the machine.

- **Constant parameters** — Specify machine parameters that are constant throughout the simulation.



- Lookup table based parameters — Specify machine parameters as lookup tables that depend on current.

**D-axis inductance for feed-forward pre-control (H) — Feedforward d-axis inductance**

0.00024 (default) | positive scalar

Direct-axis inductance for feedforward precontrol.

**Q-axis inductance for feedforward precontrol (H) — Feedforward q-axis inductance**

0.00029 (default) | positive scalar

Quadrature-axis inductance for feed-forward pre-control.

**Permanent magnet flux linkage for feedforward pre-control (H) — Feedforward flux linkage**

0.04 (default) | scalar

Permanent magnet flux linkage for feedforward pre-control.

**D-axis current vector,  $i_d$  (A) — D-axis current breakpoint vector**

[-200, 0, 200] A (default) | monotonically increasing vector

Direct-axis current vector used in the lookup tables for parameters determination. For constant machine parameters, do not change the default.

**Q-axis current vector,  $i_q$  (A) — Q-axis current breakpoint vector**

[-200, 0, 200] A (default) | monotonically increasing vector

Quadrature-axis current vector used in the lookup tables for parameters determination. For constant machine parameters, do not change the default.

**$L_d$  matrix,  $L_d(i_d, i_q)$  (H) — D-axis inductance lookup data**

0.0002 \* ones(3, 3) H (default) | positive matrix

$L_d$  matrix used as lookup table data. For constant machine parameters change only the constant factor, for example,  $L_d * \text{ones}(3, 3)$ .

**$L_q$  matrix,  $L_q(i_d, i_q)$  (H) — Q-axis inductance lookup data**

0.0002 \* ones(3, 3) H (default) | positive matrix

$L_q$  matrix used as lookup table data. For constant machine parameters change only the constant factor, for example,  $L_q * \text{ones}(3, 3)$ .

### **Permanent magnet flux linkage matrix, PM(id,iq) (Wb) — Flux linkage lookup data**

0.04 \* ones(3, 3) Wb (default) | real matrix

Permanent magnet flux linkage matrix used in the lookup table. For constant machine parameters change only the constant factor, for example,  $psim * \text{ones}(3, 3)$ .

### **PWM**

#### **PWM method — Pulse width modulation method**

SVM: space vector modulation (default) | SPWM: sinusoidal PWM

Specify the waveform technique.

#### **Sampling mode — Wave-sampling method**

Natural (default) | Asymmetric | Symmetric

Specify whether the block samples the modulation waveform when the waves intersect or when the carrier wave is at one or both of its boundary conditions.

#### **Switching frequency (Hz) — Switching rate**

1000 (default) | positive integer

Specify the rate at which you want the switches in the power converter to switch.

## **References**

- [1] Bernardes, T., V. F. Montagner, H. A. Gründling, and H. Pinheiro. "Discrete-time sliding mode observer for sensorless vector control of permanent magnet synchronous machine." *IEEE Transactions on Industrial Electronics*. Vol. 61, Number 4, 2014, pp. 1679-1691.
- [2] Carpiuc, S., and C. Lazar. "Fast real-time constrained predictive current control in permanent magnet synchronous machine-based automotive traction drives." *IEEE Transactions on Transportation Electrification*. Vol.1, Number 4, 2015, pp. 326-335.
- [3] Haque, M. E., L. Zhong, and M. F. Rahman. "Improved trajectory control for an interior permanent magnet synchronous motor drive with extended operating

limit." *Journal of Electrical & Electronics Engineering*. Vol. 22, Number 1, 2003, p. 49.

[4] Yang, N., G. Luo, W. Liu, and K. Wang. "Interior permanent magnet synchronous motor control for electric vehicle using look-up table." *In 7th International Power Electronics and Motion Control Conference*. Vol. 2, 2012, pp. 1015-1019.

## See Also

### Blocks

PMSM Current Controller | PMSM Current Controller with Pre-Control | PMSM Current Reference Generator

**Introduced in R2017b**

# PMSM Field-Weakening Controller

Permanent magnet synchronous machine field-weakening controller

**Library:** Simscape / Electrical / Power Systems / Control / PMSM Control



## Description

The PMSM Field-Weakening Controller block implements a field-weakening controller for PMSM machines.

Use this block to enforce phase voltage constraints on a current-controlled PMSM. The block decreases the PMSM phase voltage by adjusting the angle of the reference current vector when the voltage vector magnitude exceeds its limit. The block does not adjust the amplitude of the current vector.

You can use this block as part of a PMSM control system:

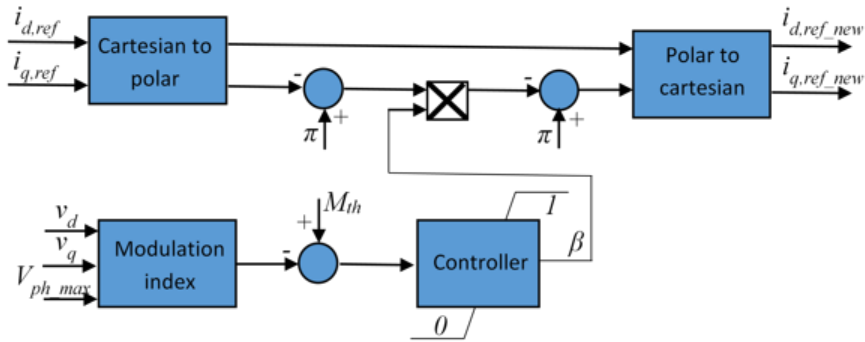
- Use the zero  $d$ -axis control technique to generate an unconstrained current reference vector to drive the PMSM. You can implement this strategy with the PMSM Current Reference Generator block.
- Use this block to adjust the angle of the current reference vector in order to satisfy voltage phase constraints.
- Use a PMSM Current Controller to generate a voltage reference vector to drive the PMSM.

## Equations

An internal integral controller outputs a factor  $\beta \in [0,1]$ , which is determined by how closely the required stator voltage approaches the saturated voltage value at any instant in time:

- When the required stator voltage exceeds the limit,  $\beta$  tends to 0, decreasing the  $q$ -axis current.
- When the required stator voltage is within its limit,  $\beta$  tends to 1 and the angle remains unchanged.

This diagram shows the structure of the field-weakening controller.



In the diagram, you provide the modulation index threshold  $M_{th}$  as an input parameter to the block, and the block computes the modulation index  $M$  as the ratio between the actual phase voltage and the maximum available phase voltage  $V_{ph\_max}$ :

$$M = \frac{\sqrt{v_d^2 + v_q^2}}{V_{ph\_max}},$$

where  $v_d$  and  $v_q$  are the  $d$ -axis and  $q$ -axis components of the voltage vector.

## Ports

### Input

#### **idqRef – Reference currents**

vector

Desired  $d$ - and  $q$ -axis currents for control of permanent magnet synchronous motor, in A.

Data Types: single | double

**vdq — Voltages**

vector

Direct and quadrature axis voltages of permanent magnet synchronous motor, in V.

Data Types: single | double

**VphMax — Maximum phase voltage**

scalar

Maximum allowable voltage in each phase, in V.

Data Types: single | double

## Output

**idqRefFW — Field-weakening reference currents**

vector

Field-weakening reference direct and quadrature axis currents, in A.

Data Types: single | double

## Parameters

**Modulation index threshold — Modulation index threshold**

1 (default) | positive number

Reference modulation index.

**Field-weakening controller integral gain — Integral gain**

100 (default) | positive number

Integrator gain of the field-weakening controller.

**Integral anti-windup gain — Anti-windup gain**

10 (default) | positive number

Anti-windup gain of the field-weakening controller.

**Sample time (-1 for inherited) – Block sample time**

-1 (default) | -1 or positive number

Sample time for the block (-1 for inherited). If you use this block inside a triggered subsystem, set the sample time to -1. If you use this block in a continuous variable-step model, set the sample time explicitly.

**References**

- [1] Wai, J., and T. M. Jahns. "A new control technique for achieving wide constant power speed operation with an interior PM alternator machine." *In Industry Applications Conference*. Vol. 2, 2001, pp. 807-814.

**See Also****Blocks**

PMSM Current Controller | PMSM Current Reference Generator

**Introduced in R2017b**

## PMSM Torque Estimator

Permanent magnet synchronous machine torque estimator

**Library:** Simscape / Electrical / Power Systems / Control / PMSM Control



### Description

The PMSM Torque Estimator block implements a torque estimator for permanent magnet synchronous machines (PMSM).

Use this block to estimate the mechanical torque of a motor when it is not directly measurable. The block estimates torque using known machine parameters and the measured phase current vector in the  $dq0$  reference frame.

Use the Park Transform block to convert the measured phase current vector in the  $abc$  reference frame to the  $dq0$  reference frame.

### Equations

The block estimates the mechanical torque  $T_e$  of the PMSM using the torque equation in the  $d$ - $q$  rotor reference frame:

$$T_e = \frac{3p}{2} (\psi_m i_q + (L_d - L_q) i_d i_q),$$

where

- $p$  is the number of pole pairs of the PMSM.
- $\psi_m$  is the flux linkage of the permanent magnet.
- $L_d$  and  $L_q$  are the  $d$ - and  $q$ -axis inductances of the PMSM.
- $i_d$  and  $i_q$  are the  $d$ - and  $q$ -axis currents of the PMSM.



In practice, the machine parameters are not constants and depend on some physical phenomena. You can choose to define these parameters simply as constants or, more realistically, as functions of currents by using lookup tables.

## Assumptions

The machine parameters are known.

## Ports

### Input

#### **idq — Stator currents**

vector

Stator direct and quadrature currents of the PMSM, in A.

Data Types: `single` | `double`

### Output

#### **TqEst — Torque estimate**

scalar

Estimated mechanical torque value of the PMSM, in N\*m.

Data Types: `single` | `double`

## Parameters

#### **Machine parameters — Parameter selection strategy**

Constant parameters (default) | Lookup table based parameters

Specify the type of machine parameters, which can be in the form of constant values or tabulated data.

#### **Number of pole pairs — Pole pairs**

8 (default) | positive integer

Number of permanent magnet pole pairs on the rotor.

**D-axis current vector,  $i_d$  (A) — D-axis current breakpoint vector**

$[-200, 0, 200]$ A (default) | monotonically increasing vector

Direct-axis current vector used in the lookup tables for parameters determination.

**Q-axis current vector,  $i_q$  (A) — Q-axis current breakpoint vector**

$[-200, 0, 200]$ A (default) | monotonically increasing vector

Quadrature-axis current vector used in the lookup tables for parameters determination.

**Ld matrix,  $L_d(i_d, i_q)$  (H) — D-axis inductance lookup data**

$0.0002 * \text{ones}(3, 3)$ H (default) | positive matrix

$L_d$  matrix used as lookup table data.

**Lq matrix,  $L_q(i_d, i_q)$  (H) — Q-axis inductance lookup data**

$0.0002 * \text{ones}(3, 3)$ H (default) | positive matrix

$L_q$  matrix used as lookup table data.

**Permanent magnet flux linkage matrix,  $PM(i_d, i_q)$  (Wb) — Flux linkage lookup data**

$0.04 * \text{ones}(3, 3)$ Wb (default) | real matrix

Permanent magnet flux linkage matrix used in the lookup table.

**D-axis inductance (H) — Inductance of d-axis**

$0.0002$  (default) | positive scalar

Direct-axis inductance.

**Q-axis inductance (H) — Inductance of q-axis**

$0.0002$  (default) | positive scalar

Quadrature-axis inductance.

**Permanent magnet flux linkage (Wb) — PM Flux Linkage**

$0.04$ Wb (default) | positive scalar

Peak permanent magnet flux linkage.

**Sample time (-1 for inherited) – Block sample time**

-1 (default) | -1 or positive number

Sample time for the block (-1 for inherited). If you use this block inside a triggered subsystem, set the sample time to -1. If you use this block in a continuous variable-step model, set the sample time explicitly.

## See Also

**Blocks**

PMSM Current Controller | PMSM Current Controller with Pre-Control | PMSM Current Reference Generator

**Introduced in R2017b**

## PNP Bipolar Transistor

Model PNP bipolar transistor using enhanced Ebers-Moll equations



### Library

Semiconductor Devices

### Description

The PNP Bipolar Transistor block uses a variant of the Ebers-Moll equations to represent an PNP bipolar transistor. The Ebers-Moll equations are based on two exponential diodes plus two current-controlled current sources. The PNP Bipolar Transistor block provides the following enhancements to that model:

- Early voltage effect
- Optional base, collector, and emitter resistances.
- Optional fixed base-emitter and base-collector capacitances.

The collector and base currents are [1]:

$$I_C = -IS \left[ \left( e^{-qV_{BE}/(kT_{m1})} - e^{-qV_{BC}/(kT_{m1})} \right) \left( 1 + \frac{V_{BC}}{V_A} \right) - \frac{1}{\beta_R} \left( e^{-qV_{BC}/(kT_{m1})} - 1 \right) \right]$$

$$I_B = -IS \left[ \frac{1}{\beta_F} \left( e^{-qV_{BE}/(kT_{m1})} - 1 \right) + \frac{1}{\beta_R} \left( e^{-qV_{BC}/(kT_{m1})} - 1 \right) \right]$$

Where:

- $I_B$  and  $I_C$  are base and collector currents, defined as positive into the device.
- $IS$  is the saturation current.
- $V_{BE}$  is the base-emitter voltage and  $V_{BC}$  is the base-collector voltage.
- $\beta_F$  is the ideal maximum current gain BF
- $\beta_R$  is the ideal maximum current gain BR
- $V_A$  is the forward Early voltage VAF
- $q$  is the elementary charge on an electron (1.602176e-19 Coulombs).
- $k$  is the Boltzmann constant (1.3806503e-23 J/K).
- $T_{m1}$  is the transistor temperature, as defined by the **Measurement temperature** parameter value.

You can specify the transistor behavior using datasheet parameters that the block uses to calculate the parameters for these equations, or you can specify the equation parameters directly.

If  $-qV_{BC} / (kT_{m1}) > 40$  or  $-qV_{BE} / (kT_{m1}) > 40$ , the corresponding exponential terms in the equations are replaced with  $(-qV_{BC} / (kT_{m1}) - 39)e^{40}$  and  $(-qV_{BE} / (kT_{m1}) - 39)e^{40}$ , respectively. This helps prevent numerical issues associated with the steep gradient of the exponential function  $e^x$  at large values of  $x$ . Similarly, if  $-qV_{BC} / (kT_{m1}) < -39$  or  $-qV_{BE} / (kT_{m1}) < -39$  then the corresponding exponential terms in the equations are replaced with  $(-qV_{BC} / (kT_{m1}) + 40)e^{-39}$  and  $(-qV_{BE} / (kT_{m1}) + 40)e^{-39}$ , respectively.

Optionally, you can specify parasitic fixed capacitances across the base-emitter and base-collector junctions. You also have the option to specify base, collector, and emitter connection resistances.

## Modeling Temperature Dependence

The default behavior is that dependence on temperature is not modeled, and the device is simulated at the temperature for which you provide block parameters. You can optionally include modeling the dependence of the transistor static behavior on temperature during simulation. Temperature dependence of the junction capacitances is not modeled, this being a much smaller effect.

When including temperature dependence, the transistor defining equations remain the same. The measurement temperature value,  $T_{m1}$ , is replaced with the simulation temperature,  $T_s$ . The saturation current,  $IS$ , and the forward and reverse gains ( $\beta_F$  and  $\beta_R$ ) become a function of temperature according to the following equations:

$$IS_{T_s} = IS_{T_{m1}} \cdot (T_s / T_{m1})^{XTI} \cdot \exp\left(-\frac{EG}{kT_s}(1 - T_s / T_{m1})\right)$$

$$\beta_{F_s} = \beta_{F_{m1}} \left(\frac{T_s}{T_{m1}}\right)^{XTB}$$

$$\beta_{R_s} = \beta_{R_{m1}} \left(\frac{T_s}{T_{m1}}\right)^{XTB}$$

where:

- $T_{m1}$  is the temperature at which the transistor parameters are specified, as defined by the **Measurement temperature** parameter value.
- $T_s$  is the simulation temperature.
- $IS_{T_{m1}}$  is the saturation current at the measurement temperature.
- $IS_{T_s}$  is the saturation current at the simulation temperature. This is the saturation current value used in the bipolar transistor equations when temperature dependence is modeled.
- $\beta_{F_{m1}}$  and  $\beta_{R_{m1}}$  are the forward and reverse gains at the measurement temperature.
- $\beta_{F_s}$  and  $\beta_{R_s}$  are the forward and reverse gains at the simulation temperature. These are the values used in the bipolar transistor equations when temperature dependence is modeled.
- $EG$  is the energy gap for the semiconductor type measured in Joules. The value for silicon is usually taken to be 1.11 eV, where 1 eV is 1.602e-19 Joules.
- $XTI$  is the saturation current temperature exponent.
- $XTB$  is the forward and reverse gain temperature coefficient.
- $k$  is the Boltzmann constant (1.3806503e-23 J/K).

Appropriate values for  $XTI$  and  $EG$  depend on the type of transistor and the semiconductor material used. In practice, the values of  $XTI$ ,  $EG$ , and  $XTB$  need tuning to model the exact behavior of a particular transistor. Some manufacturers quote these tuned values in a SPICE Netlist, and you can read off the appropriate values. Otherwise you can determine values for  $XTI$ ,  $EG$ , and  $XTB$  by using a datasheet-defined data at a higher temperature  $T_{m2}$ . The block provides a datasheet parameterization option for this.

You can also tune the values of  $XTI$ ,  $EG$ , and  $XTB$  yourself, to match lab data for your particular device. You can use Simulink Design Optimization software to help tune the values.

## Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal port H on the block icon, and adds the **Thermal Port** tab to the block dialog box.

Use the thermal port to simulate the effects of generated heat and device temperature. For more information on using thermal ports and on the **Thermal Port** tab parameters, see “Simulating Thermal Effects in Semiconductors”.

## Basic Assumptions and Limitations

The PNP Bipolar Transistor model has the following limitations:

- The block does not account for temperature-dependent effects on the junction capacitances.
- You may need to use nonzero ohmic resistance and junction capacitance values to prevent numerical simulation issues, but the simulation may run faster with these values set to zero.

## Parameters

- “Main Tab” on page 1-1201
- “Ohmic Resistance Tab” on page 1-1203
- “Capacitance Tab” on page 1-1204
- “Temperature Dependence Tab” on page 1-1204

## Main Tab

### Parameterization

Select one of the following methods for block parameterization:

- **Specify from a datasheet** — Provide parameters that the block converts to equations that describe the transistor. The block calculates the forward Early voltage  $VAF$  as  $I_c/h_{oe}$ , where  $I_c$  is the **Collector current at which h-parameters are defined** parameter value, and  $h_{oe}$  is the **Output admittance  $h_{oe}$**  parameter value [2]. The block sets  $BF$  to the small-signal **Forward current transfer ratio  $h_{fe}$**  value. The block calculates the saturation current  $IS$  from the specified **Voltage  $V_{be}$**  value and the corresponding **Current  $I_b$  for voltage  $V_{be}$**  value when  $I_c$  is zero. This is the default method.
- **Specify using equation parameters directly** — Provide equation parameters  $IS$ ,  $BF$ , and  $VAF$ .

### **Forward current transfer ratio $h_{fe}$**

Small-signal current gain. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is 100.

### **Output admittance $h_{oe}$**

Derivative of the collector current with respect to the collector-emitter voltage for a fixed base current. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is  $5e-5$  1/ $\Omega$ .

### **Collector current at which h-parameters are defined**

The h-parameters vary with operating point, and are defined for this value of the collector current. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is -1 mA.

### **Collector-emitter voltage at which h-parameters are defined**

The h-parameters vary with operating point, and are defined for this value of the collector-emitter voltage. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is -5 V.

### **Voltage $V_{be}$**

Base-emitter voltage when the base current is  $I_b$ . The [  $V_{be}$   $I_b$  ] data pair must be quoted for when the transistor is in the normal active region, that is, not in the saturated region. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is -0.55 V.

### **Current $I_b$ for voltage $V_{be}$**

Base current when the base-emitter voltage is  $V_{be}$ . The [  $V_{be}$   $I_b$  ] data pair must be quoted for when the transistor is in the normal active region, that is, not in the saturated region. This parameter is only visible when you select **Specify from a datasheet** for the **Parameterization** parameter. The default value is -0.5 mA.



**Forward current transfer ratio BF**

Ideal maximum forward current gain. This parameter is only visible when you select **Specify using equation parameters directly** for the **Parameterization** parameter. The default value is 100.

**Saturation current IS**

Transistor saturation current. This parameter is only visible when you select **Specify using equation parameters directly** for the **Parameterization** parameter. The default value is 1e-14 A.

**Forward Early voltage VAF**

In the standard Ebers-Moll equations, the gradient of the  $I_c$  versus  $V_{ce}$  curve is zero in the normal active region. The additional forward Early voltage term increases this gradient. The intercept on the  $V_{ce}$ -axis is equal to  $-VAF$  when the linear region is extrapolated. This parameter is only visible when you select **Specify using equation parameters directly** for the **Parameterization** parameter. The default value is 200 V.

**Reverse current transfer ratio BR**

Ideal maximum reverse current gain. This value is often not quoted in manufacturer datasheets because it is not significant when the transistor is biased to operate in the normal active region. When the value is not known and the transistor is not to be operated on the inverse region, use the default value of 1.

**Measurement temperature**

Temperature  $T_{m1}$  at which  $V_{be}$  and  $I_b$ , or  $IS$ , are measured. The default value is 25 °C.

**Ohmic Resistance Tab****Collector resistance RC**

Resistance at the collector. The default value is 0.01  $\Omega$ .

**Emitter resistance RE**

Resistance at the emitter. The default value is 1e-4  $\Omega$ .

**Zero bias base resistance RB**

Resistance at the base at zero bias. The default value is 1  $\Omega$ .

## Capacitance Tab

### Base-collector junction capacitance

Parasitic capacitance across the base-collector junction. The default value is 5 pF.

### Base-emitter junction capacitance

Parasitic capacitance across the base-emitter junction. The default value is 5 pF.

### Total forward transit time

Represents the mean time for the minority carriers to cross the base region from the emitter to the collector, and is often denoted by the parameter TF [1]. The default value is 0  $\mu$ s.

### Total reverse transit time

Represents the mean time for the minority carriers to cross the base region from the collector to the emitter, and is often denoted by the parameter TR [1]. The default value is 0  $\mu$ s.

## Temperature Dependence Tab

### Parameterization

Select one of the following methods for temperature dependence parameterization:

- **None** — Simulate at parameter measurement temperature — Temperature dependence is not modeled, or the model is simulated at the measurement temperature  $T_{m1}$  (as specified by the **Measurement temperature** parameter on the **Main** tab). This is the default method.
- **Model temperature dependence** — Provide a value for simulation temperature, to model temperature-dependent effects. You also have to provide a set of additional parameters depending on the block parameterization method. If you parameterize the block from a datasheet, you have to provide values for a second [ *Vbe Ib* ] data pair and  $h_{fe}$  at second measurement temperature. If you parameterize by directly specifying equation parameters, you have to provide the values for *XTI*, *EG*, and *XTB*.

### Forward current transfer ratio, $h_{fe}$ , at second measurement temperature

Small-signal current gain at second measurement temperature. This parameter is only visible when you select **Specify** from a datasheet for the **Parameterization** parameter on the **Main** tab. It must be quoted at the same collector-emitter voltage

and collector current as for the **Forward current transfer ratio  $h_{fe}$**  parameter on the **Main** tab. The default value is 125.

#### **Voltage $V_{be}$ at second measurement temperature**

Base-emitter voltage when the base current is  $I_b$  and the temperature is set to the second measurement temperature. The [ $V_{be} I_b$ ] data pair must be quoted for when the transistor is in the normal active region, that is, not in the saturated region. This parameter is only visible when you select **Specify** from a datasheet for the **Parameterization** parameter on the **Main** tab. The default value is -0.45 V.

#### **Current $I_b$ for voltage $V_{be}$ at second measurement temperature**

Base current when the base-emitter voltage is  $V_{be}$  and the temperature is set to the second measurement temperature. The [ $V_{be} I_b$ ] data pair must be quoted for when the transistor is in the normal active region, that is, not in the saturated region. This parameter is only visible when you select **Specify** from a datasheet for the **Parameterization** parameter on the **Main** tab. The default value is -0.5 mA.

#### **Second measurement temperature**

Second temperature  $T_{m2}$  at which  $h_{fe}$ ,  $V_{be}$ , and  $I_b$  are measured. This parameter is only visible when you select **Specify** from a datasheet for the **Parameterization** parameter on the **Main** tab. The default value is 125 °C.

#### **Current gain temperature coefficient, XTB**

Current gain temperature coefficient value. This parameter is only visible when you select **Specify using equation parameters directly** for the **Parameterization** parameter on the **Main** tab. The default value is 0.

#### **Energy gap, EG**

Energy gap value. This parameter is only visible when you select **Specify using equation parameters directly** for the **Parameterization** parameter on the **Main** tab. The default value is 1.11 eV.

#### **Saturation current temperature exponent, XTI**

Saturation current temperature coefficient value. This parameter is only visible when you select **Specify using equation parameters directly** for the **Parameterization** parameter on the **Main** tab. The default value is 3.

#### **Device simulation temperature**

Temperature  $T_s$  at which the device is simulated. The default value is 25 °C.

## Ports

The block has the following ports:

**B**

Electrical conserving port associated with the transistor base terminal

**C**

Electrical conserving port associated with the transistor collector terminal

**E**

Electrical conserving port associated with the transistor emitter terminal

## Examples

See the PNP Bipolar Transistor Characteristics example.

## References

- [1] G. Massobrio and P. Antognetti. *Semiconductor Device Modeling with SPICE*. 2nd Edition, McGraw-Hill, 1993.
- [2] H. Ahmed and P.J. Spreadbury. *Analogue and digital electronics for engineers*. 2nd Edition, Cambridge University Press, 1984.

## See Also

Diode | NPN Bipolar Transistor

# Positive Supply Rail

Model ideal positive supply rail



## Library

Sources

## Description

The Positive Supply Rail block represents an ideal positive supply rail. Use this block instead of the Simscape DC Voltage Source block to define the output voltage relative to the Simscape Electrical Reference block that must appear in each model.

---

**Note** Do not attach more than one Positive Supply Rail block to any connected line.

---

## Parameters

### Constant voltage

The voltage at the output port relative to the Electrical Reference block ground port.  
The default value is 1 V.

## Ports

The block has the following ports:

+

Positive electrical voltage

## **See Also**

DC Voltage Source | Negative Supply Rail

# Potentiometer

Model rotary or linear-travel potentiometer controlled by physical signal



## Library

Passive Devices

## Description

The Potentiometer block represents a rotary or linear-travel potentiometer, with the wiper position controlled by the input physical signal.

If the potentiometer resistance changes linearly based on wiper position, then the resistance between the wiper position and port L is:

$$R_{WL} = \frac{R_0}{x_{\max} - x_{\min}} (x - x_{\min})$$

where

- $R_{WL}$  is the resistance between the wiper position and port L.
- $R_0$  is the total resistance between ports L and R.
- $x$  is the wiper position.
- $x_{\min}$  is the value of the wiper position when the wiper is at port L.
- $x_{\max}$  is the value of the wiper position when the wiper is at port R.

If you specify LOG for the potentiometer resistance **Taper** parameter, then the resistance between the wiper position and port L is:

$$R_{WL} = \begin{cases} A \left( e^{\lambda(x-x_{\min})} - 1 \right) & \text{if resistance gradient is higher at R} \\ R_0 - A \left( e^{\lambda(x_{\max}-x)} - 1 \right) & \text{if resistance gradient is higher at L} \end{cases}$$

where  $A$  and  $\lambda$  are chosen such that  $R_{WL}$  at  $x_{\max}$  is  $R_0$ , and  $R_{WL}$  at  $x = (x_{\max} + x_{\min}) / 2$  is equal to  $R_{av}$ , the resistance when the wiper is centered.

---

**Note** Potentiometers widely described as LOG or logarithmic taper are, in fact, exponential taper. That is, the gradient of the resistance between wiper and left-hand port increases as the resistance increases. The Potentiometer block implements this behavior.

---

For both linear and logarithmic tapers, the resistance between the wiper position and port R is:

$$R_{WR} = R_0 - R_{WL}$$

where

- $R_{WR}$  is the resistance between the wiper position and port R.
- $R_0$  is the total resistance between ports L and R.
- $R_{WL}$  is the resistance between the wiper position and port L.

## Parameters

### Total resistance

The resistance between port L and port R when port W is open-circuit. The default value is 1000Ω.

### Residual resistance

The lower limit placed on the resistance between the wiper and the two end ports. It must be greater than zero. A typical value is 5e-3 times the total resistance. The default value is 1Ω.

### Resistance when centered

This parameter is available only if you select LOG for the **Taper** parameter. If you select Higher at R for the **Resistance gradient** parameter, then **Resistance when**



**centered** is the resistance between port L and port W when the wiper is centered. Otherwise, if you select **Higher at R** for the **Resistance gradient** parameter, then **Resistance when centered** is the resistance between port R and port W when the wiper is centered. Because the resistance taper is exponential in shape, the value of the **Resistance when centered** parameter must be less than half of the **Total resistance** parameter value. The default value is 200Ω.

#### **PS input for wiper at L**

The value of the input physical signal at port x that corresponds to the wiper being located at port L. The default value is 0.

#### **PS input for wiper at R**

The value of the input physical signal at port x that corresponds to the wiper being located at port R. The default value is 1.

#### **Taper**

Specifies the potentiometer resistance taper behavior: LIN (linear) or LOG (logarithmic). The default value is LIN.

#### **Resistance gradient**

Specifies whether the potentiometer resistance varies more rapidly at the left or the right end: **Higher at L** or **Higher at R**. This parameter is available only if you select LOG for the **Taper** parameter. The default value is **Higher at R**.

## **Ports**

The block has the following ports:

L

Electrical port representing the left pin

R

Electrical port representing the right pin

W

Electrical port representing the wiper pin

x

Physical signal input port controlling the wiper position

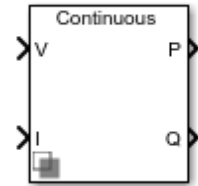
## **See Also**

Variable Resistor

# Power Measurement

Measure single-phase real and reactive power

**Library:** Simscape / Electrical / Power Systems / Control / Measurements



## Description

The Power Measurement block measures the real and reactive power of an element in a single-phase network. The block outputs the power quantities for each frequency component you specify. For three-phase measurements, consider using the Three-Phase Power Measurement block.

Use this block to measure power for both sinusoidal and nonsinusoidal periodic signals.

Set the **Sample time** parameter to 0 for continuous-time operation, or explicitly for discrete-time operation.

Specify a vector of all frequency components to include in the power output using the **Harmonic numbers** parameter:

- To output the DC component, specify 0.
- To output the component corresponding to the fundamental frequency, specify 1.
- To output components corresponding to higher-order harmonics, specify  $n > 1$ .

## Equations

For each specified harmonic  $k$ , the block calculates the real power  $P_k$  and reactive power  $Q_k$  from the phasor equation:

$$P_k + jQ_k = G(V_k e^{j\theta_{V_k}})(\overline{I_k e^{j\theta_{I_k}}}),$$

where:

- $G$  is equal to  $0.25$  for the DC component ( $k = 0$ ) and  $0.5$  for the AC components ( $k > 0$ ).
- $V_k e^{j\theta_k}$  is the phasor representation of the  $k$ -component input voltage.
- $\overline{I_k e^{j\theta_k}}$  is the complex conjugate of  $I_k e^{j\theta_k}$ , the phasor representation of the  $k$ -component input current.

The block estimates the real-time  $k$ -component voltage and current phasors using these relationships:

$$V_k e^{j\theta_k} = \frac{2}{T} \int_{t-T}^t V(t) \sin(2\pi k F t) dt + j \frac{2}{T} \int_{t-T}^t V(t) \cos(2\pi k F t) dt$$

$$I_k e^{j\theta_k} = \frac{2}{T} \int_{t-T}^t I(t) \sin(2\pi k F t) dt + j \frac{2}{T} \int_{t-T}^t I(t) \cos(2\pi k F t) dt.$$

In these phasor equations:

- $V(t)$  and  $I(t)$  are the input voltage and current, respectively.
- $T$  is the period of the input signal, or equivalently the inverse of its base frequency  $F$ .

If the input signals have a finite number of harmonics  $n$ , the total real power  $P$  and total reactive power  $Q$  can be calculated from their components:

$$P = \sum_{k=0}^n P_k$$

$$Q = \sum_{k=1}^n Q_k.$$

The summation for  $Q$  does not include the DC component ( $k = 0$ ) because this component only contributes to real power.

## Ports

### Input

**v — Input voltage**

scalar

Voltage across element from which to measure power, in V.

Data Types: `single` | `double`**i — Input current**

scalar

Current through element from which to measure power, in A.

Data Types: `single` | `double`

### Output

**P — Real power**

vector

Real power for selected frequency components, in W.

Data Types: `single` | `double`**P — Reactive power**

vector

Reactive power for selected frequency components, in var.

Data Types: `single` | `double`

## Parameters

**Base frequency (Hz) — Fundamental frequency**

60 (default) | positive number

Fundamental frequency corresponding to component  $k=1$ .

## Harmonic numbers — Frequency components

[0 1 2 3] (default) | scalar or vector

Frequency components to include in the output. Specify either a scalar value corresponding to the desired component or a vector of all desired components.

- The value  $k = 0$  corresponds to the DC component.
- The value  $k = 1$  corresponds to the fundamental frequency.
- Values  $k > 1$  correspond to higher-level harmonics.

If you specify a vector, the order of the power outputs correspond to the order of this vector.

## Sample time — Block sample time

0 (default) | positive number

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For continuous operation, set this property to 0. For discrete operation, specify the sample time explicitly as a positive number. This block does not support inherited sample time.

If this block is in a masked subsystem, or other variant subsystem that allows either continuous and discrete operation, promote the sample time parameter. Promoting the sample time parameter ensures correct switching between the continuous and discrete implementations of the block. For more information, see “Promote Parameter to Mask” (Simulink).

## See Also

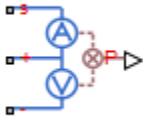
### Blocks

RMS Measurement | Sinusoidal Measurement (PLL) | Three-Phase Sinusoidal Measurement (PLL) | Three-Phase Power Measurement

**Introduced in R2017b**

# Power Sensor

Measure instantaneous or cycle-average power



## Library

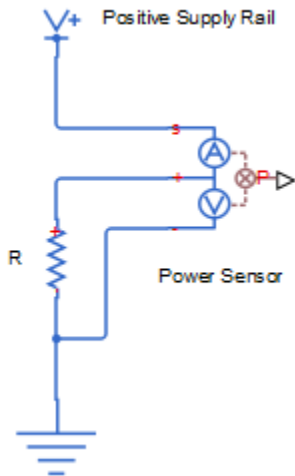
Sensors

## Description

The Power Sensor block calculates the power taken by the load connected across the + and - terminals under the assumption that only the load is connected to the + terminal. Refer to the block icon for the arrangement of internal current and voltage sensors.

The sensor can return either instantaneous power, or power averaged over a fixed time period. Use the latter option for periodic current and voltage waveforms such as those associated with PWM control.

The following figure shows how you connect the block to measure power dissipated in a resistor.



For an alternative workflow using data logging to view component powers, see the “Buck Converter” example.

## Parameters

### Measurement type

Select whether you want to measure Instantaneous power or Average power over a specified period. The default value is Instantaneous power.

### Averaging period

The fixed period of time for measuring the average power. This parameter is only visible when you select Average power over a specified period for the **Measurement type** parameter. The default value is  $1e-4$  s.

## Ports

The block has the following ports:

S

Electrical conserving port connected to the positive supply rail



+

Electrical conserving port connected to the positive terminal of the load

-

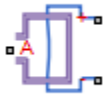
Electrical conserving port connected to the negative terminal of the load

P

Physical signal port that outputs the measured power

# Pressure Transducer

Model generic pressure transducer that turns pressure measurement into voltage



## Library

Sensors

## Description

The Pressure Transducer block models a generic pressure transducer that turns a pressure measurement into a voltage. The output voltage is linearly proportional to the pressure, and the block outputs zero volts if the pressure is less than zero. An input pressure equal to the **Pressure range** parameter value results in an output voltage equal to the **Full-scale deflection** parameter value. For higher pressures, the output voltage remains at this **Full-scale deflection** value.

You have three choices of operation mode, which let you select between vacuum, atmospheric pressure, or sealed-gauge reference pressure as the reference point for the pressure measurement.

Optionally, if you set the **Dynamics** parameter to `Model transducer bandwidth`, then the dynamics of the sensor are approximated by a first-order lag. The lag is determined by the **Bandwidth** parameter. If you select this option, you must also specify an initial condition for the lag by using the **Measured pressure** variable target.

If running your simulation with a fixed-step solver, or generating code for hardware-in-the-loop testing, MathWorks recommends that you set the **Dynamics** parameter to `No dynamics – Suitable for HIL`, because this avoids the need for a small simulation time step if the sensor bandwidth is high.

## Variables

Use the **Variables** section of the block interface to set the priority and initial target values for the block variables prior to simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

The **Measured pressure** variable target specifies the initial output for the sensor.

## Parameters

### Pressure range

The maximum pressure that the sensor can measure. The default value is 1e6 Pa.

### Operation mode

Select one of the following options to define the reference point for the pressure measurement:

- **Absolute** — The pressure measurement is with respect to zero absolute pressure, that is, vacuum. This is the default option.
- **Gauge** — The pressure measurement is with respect to atmospheric pressure. Atmospheric pressure is defined by the Gas Properties block in the Simscape Foundation library.
- **Sealed-Gauge** — The pressure measurement is referenced to an internal sealed chamber. If you select this option, use the **Reference pressure** parameter to specify the reference point for pressure measurement.

### Reference pressure

The reference pressure in the internal sealed chamber. This parameter is only visible when you select **Sealed-Gauge** for the **Operation mode** parameter. The default value is 1.01325e5 Pa.

### Full-scale deflection

The output voltage when the measured pressure is equal to, or greater than, the **Pressure range** parameter value. The default value is 5 V.

### Output resistance

The output resistance of the transducer. The default value is 200  $\Omega$ .

### Dynamics

Select one of the following options for modeling sensor dynamics:

- **No dynamics – Suitable for HIL** — Do not model sensor dynamics. Use this option when running your simulation fixed step or generating code for hardware-in-the-loop testing, because this avoids the need for a small simulation time step if the sensor bandwidth is high. This is the default option.
- **Model transducer bandwidth** — Model sensor dynamics with a first-order lag approximation, based on the **Bandwidth** parameter value. You can control the initial condition for the lag by specifying the **Measured pressure** variable target.

## **Bandwidth**

Determines the value of the sensor lag. This parameter is only visible when you select **Model transducer bandwidth** for the **Dynamics** parameter. The default value is 5 kHz.

## **Ports**

The block has the following ports:

A

Pneumatic port

+

Positive electrical port

-

Negative electrical port

# Primary Winding

(To be removed) Linear nonideal transformer winding

---

**Note** The Primary Winding block will be removed in a future release. Use the Winding, Eddy Current, and Fundamental Reluctance blocks instead. To learn how to create a custom transformer without this block, refer to “Push-Pull Buck Converter in Continuous Conduction Mode”.

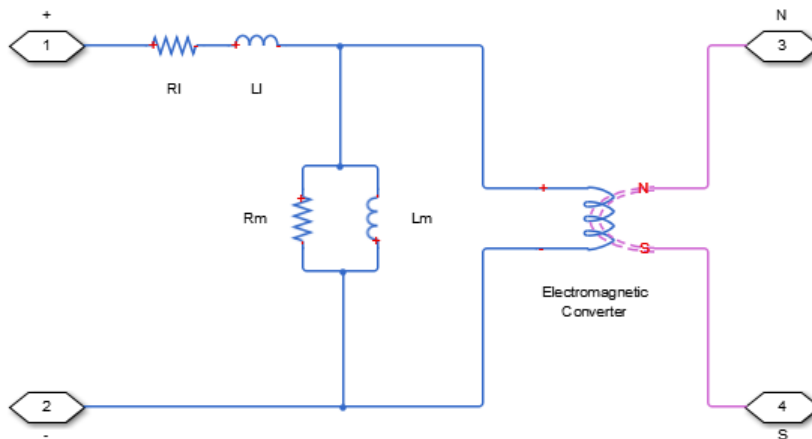
---

## Library

Simscape / Electrical / Power Systems / Passive Devices / Transformers / Fundamental Components

## Description

The Primary Winding block models linear nonideal winding of a transformer with linear winding leakage and linear core magnetization effects. Although magnetization effects occur in the magnetic core, it is common practice to place mathematically equivalent electrical components on the electrical winding and parameterize them using electrical parameters. The figure shows the equivalent circuit diagram for the primary winding.



- $R_l$  is the leakage resistance.
- $L_l$  is the leakage inductance.
- $R_m$  is the magnetization resistance.
- $L_m$  is the magnetization inductance.

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Parameters

### Main Tab

#### Number of winding turns

Number of wire turns on the transformer winding. The default value is 10.

#### Leakage resistance

Power loss in the winding. The default value is  $1e-3$  Ohm.

#### Leakage inductance

Magnetic flux loss in the winding. The default value is  $1e-3$  H.

#### Core-loss resistance

Magnetic losses in the transformer core. The default value is  $1e6$  Ohm.

#### Magnetization inductance

Magnetic effects in the transformer core when operating in its linear region. The default value is  $1e6$  H.

## Ports

The block has the following ports:

- +  
Positive electrical conserving port
- Negative electrical conserving port
- N  
North magnetic conserving port
- S  
South magnetic conserving port

## See Also

Electromagnetic Converter | Secondary Winding

## Topics

“Custom Zigzag Transformer”

“Push-Pull Buck Converter in Continuous Conduction Mode”

“Push-Pull Buck Converter in Discontinuous Conduction Mode”

**Introduced in R2013b**

# Proximity Sensor

Model simple distance sensor

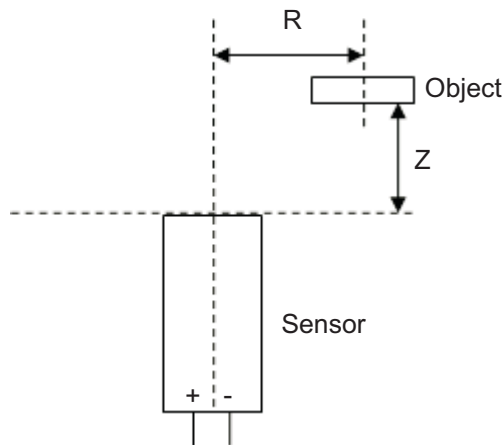


## Library

Sensors

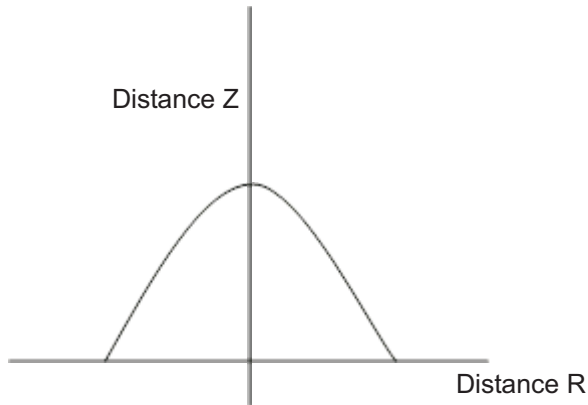
## Description

The Proximity Sensor block represents a simple proximity sensor. The sensing distance  $Z$  is defined as the distance normal to the sensor surface at which the sensor detects an object for a given radial offset  $R$ , as shown in the following figure.



A typical sensing distance curve is shown in the following figure.





The output is modeled by an electrical switch which can either be Normally Open (N.O.) or Normally Closed (N.C.) when no object is detected.

## Parameters

### Vector of radial offset distances R

Vector of distances from the sensor to the object resolved into a plane tangential to the sensor head. The default value is [ -25 -20 -15 -10 -5 0 5 10 15 20 25 ] mm.

### Corresponding sensing distances Z

Vector of distances from the sensor to the object resolved with respect to a normal vector at the sensor head. The default value is [ 0 0 5 8 9.5 10 9.5 8 5 0 0 ] mm.

### Output when not detected

Indicates whether the output is Normally Open (N.O.), meaning the output becomes closed only when the object is detected, or Normally Closed (N.C.), meaning the output becomes open only when the object is detected. The default value is Normally Open (N.O.).

### Closed resistance R\_closed

The resistance between the + and - ports when the output contacts are closed. The default value is 0.01 Ω.

**Open conductance G\_open**

The conductance between the + and - ports when the output contacts are open. The default value is  $1e-08$  1/Ω.

**Ports**

The block has the following ports:

R

Radial distance to the sensor

Z

Perpendicular distance to the sensor

+

Positive electrical voltage

-

Negative electrical voltage

# PS Sensor

Model generic linear sensor



## Library

Sensors

## Description

The PS Sensor block represents a generic linear sensor. The block converts the physical signal input  $U$  into an electrical output  $Y$  across the + and - ports. The **Output type** parameter value determines which of the following electrical outputs the block produces:

- Output voltage
- Output current
- Output resistance

$Y$  is related to  $U$  as  $Y = \max(\min(A \cdot U + B, Y_{\max}), Y_{\min})$ , where  $Y_{\min}$  and  $Y_{\max}$  are minimum and maximum limits on the output, respectively.

## Parameters

### Output type

Indicates whether the sensor output is a **Variable voltage** of  $Y$  V, a **Variable current** of  $Y$  A, or **Variable resistor** with a value of  $Y$   $\Omega$ . The default value is **Variable voltage**.

### Sensor gain, A

The sensitivity of the output  $Y$  with respect to the input  $U$ ,  $dY/dU$ . The default value is 1.

### Sensor offset, B

The output when the input  $U$  is zero. The output does not exceed the limits  $Y_{\max}$  and  $Y_{\min}$ . The default value is 0.

### Maximum output, Ymax

The upper limit on the sensor output. The following table shows the units of this parameter, which depend on the selected value of the **Output type** parameter.

Output type	Units
Variable voltage	V
Variable current	A
Variable resistor	$\Omega$

The default value is 5.

### Minimum output, Ymin

The lower limit on the sensor output. The following table shows the units of this parameter, which depend on the selected value of the **Output type** parameter.

Output type	Units
Variable voltage	V
Variable current	A
Variable resistor	$\Omega$

The default value is 0.01.

If you select `Variable resistance` for the **Output type** parameter, the minimum resistance  $Y_{\min}$  must be greater than zero.

## Ports

The block has the following ports:

U

Physical input signal

+

Positive electrical voltage

-

Negative electrical voltage

## **See Also**

Controlled Current Source | Controlled Voltage Source | Variable Resistor

## PTC Thermistor

Model switching type positive temperature coefficient (PTC) thermistor

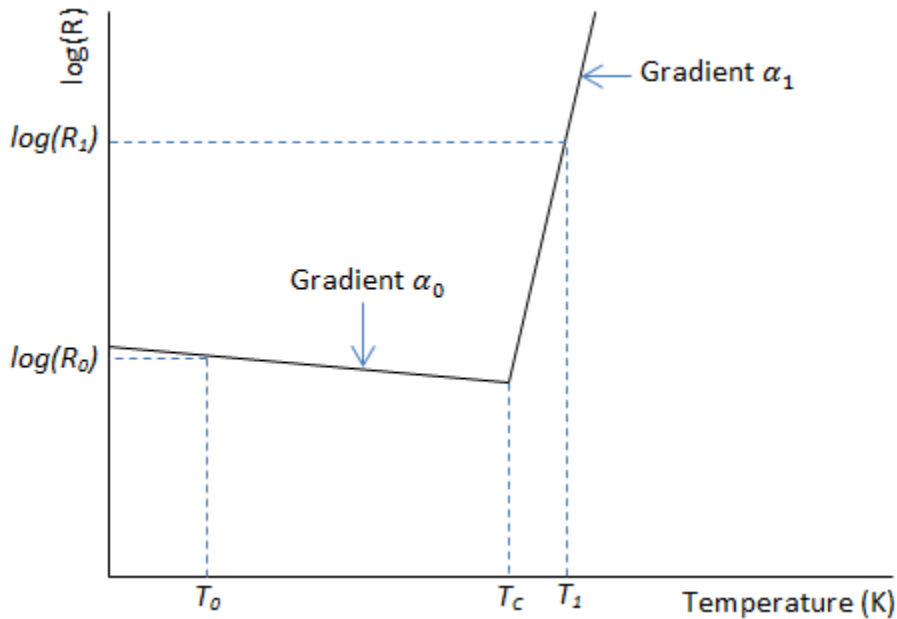


## Library

Sensors

## Description

The PTC Thermistor block represents a switching type PTC thermistor. This type of thermistor has a decreasing resistance with temperature increasing up to the Curie temperature. Above the Curie temperature the resistance increases very rapidly with increasing temperature, as shown in the following plot. The region to the right of the Curie temperature is called the Positive Temperature Coefficient (PTC) regime. To represent a non-switching linear PTC thermistor, use the Thermal Resistor block.



For a switching type PTC thermistor, the resistance  $R$  at temperature  $T$  is given by

$$R = \begin{cases} R_0 e^{\alpha_0(T-T_0)} & \text{for } T < T_c \\ R_1 e^{\alpha_1(T-T_1)} & \text{for } T \geq T_c \end{cases}$$

$$T_c = \frac{\log(R_1) - \log(R_0) + \alpha_0 T_0 - \alpha_1 T_1}{\alpha_0 - \alpha_1}$$

where:

- $T_c$  is the Curie temperature.
- $R_0$  is the resistance at nominal temperature  $T_0$ .
- $R_1$  is the resistance at reference temperature  $T_1$ .
- $T_0$  is the nominal temperature at which the resistance is quoted, usually room temperature.  $T_0$  is less than the Curie temperature  $T_c$ .

- $T_1$  is the reference temperature, equal or greater than the Curie temperature  $T_c$ , which means that at this temperature the PTC regime is in force.
- $\alpha_0$  is the temperature coefficient at nominal temperature  $T_0$ .
- $\alpha_1$  is the temperature coefficient at reference temperature  $T_1$ .

The following equation describes the thermal behavior of the block:

$$Q = K_d t_c \frac{dT}{dt}$$

where:

- $Q$  is the net heat flow into port A.
- $K_d$  is the **Dissipation factor** parameter value.
- $t_c$  is the **Thermal time constant** parameter value.
- $dT/dt$  is the rate of change of the temperature.

## Variables

Use the **Variables** section of the block interface to set the priority and initial target values for the block variables prior to simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

## Parameters

- “Electrical Tab” on page 1-1234
- “Thermal Tab” on page 1-1235

## Electrical Tab

### Nominal resistance R0 at T0

The nominal resistance of the thermistor at the nominal temperature. Many datasheets quote the nominal resistance at 25°C and list it as R25. The default value is 1000 Ω.



**Temperature coefficient alpha0 at T0**

The temperature coefficient at the nominal temperature. The value must be less than zero. The default value is  $-0.01$  1/K.

**Nominal temperature T0**

The temperature at which the nominal resistance is measured. The default value is 298.15 K.

**Reference resistance R1 at T1**

The reference resistance of the thermistor at the reference temperature. The default value is 10000  $\Omega$ .

**Temperature coefficient alpha1 at T1**

The temperature coefficient at the reference temperature. The value must be greater than zero. The default value is 1 1/K.

**Reference temperature T1**

The temperature at which the reference resistance is measured. This temperature must be in the PTC regime. The default value is 398.15 K.

**Thermal Tab****Thermal time constant**

The time it takes the sensor temperature to reach 63% of the final temperature change when a step change in ambient temperature occurs. The default value is 5 s.

**Dissipation factor**

The thermal power required to raise the thermistor temperature by one K. The default value is  $7.5e-4$  W/K.

**Ports**

The block has the following ports:

A

Thermal port

+

Positive electrical port

-

Negative electrical port

## **See Also**

Resistor | Thermal Resistor | Thermistor

# Pulse Current Source

Model periodic square pulse current source



## Library

SPICE-Compatible Components/Sources

## Description

The Pulse Current Source block represents a current source whose output current value is a periodic square pulse as a function of time and is independent of the voltage across the terminals of the source. The following equations describe the current through the source as a function of time:

$$I_{out}(0) = I1$$

$$I_{out}(TD) = I1$$

$$I_{out}(TD + TR) = I2$$

$$I_{out}(TD + TR + PW) = I2$$

$$I_{out}(TD + TR + PW + TF) = I1$$

$$I_{out}(TD + PER) = I1$$

where:

- $I1$  is the **Initial value**, **I1** parameter value.
- $I2$  is the **Pulse value**, **I2** parameter value.
- $TD$  is the **Pulse delay time**, **TD** parameter value.

- $TR$  is the **Pulse rise time**, **TR** parameter value.
- $TF$  is the **Pulse fall time**, **TF** parameter value.
- $PW$  is the **Pulse width**, **PW** parameter value.
- $PER$  is the **Pulse period**, **PER** parameter value.

The block determines the values at intermediate time points by linear interpolation.

The specified values for  $PW$  and  $PER$  have the following effect on the block output:

- If both  $PW$  and  $PER$  are infinite, the block produces a step response at time  $TD$ .
- If  $PER$  is infinite and  $PW$  is finite, the block produces a single pulse of width  $PW$  and infinite period.
- If  $PW$  is infinite and  $PER$  is finite, the block produces a step response with pulses of width  $TR$  to a value  $I1$  every  $PER$  seconds.
- If  $PW > PER$ , the block produces a step response with pulses of width  $TR$  to a value  $I1$  every  $PER$  seconds.

The block uses a small conductance internally to prevent numerical simulation issues. The conductance connects the + and - ports of the device and has a conductance  $GMIN$ :

- By default,  $GMIN$  matches the **Minimum conductance GMIN** parameter of the SPICE Environment Parameters block, whose default value is  $1e-12$ .
- To change  $GMIN$ , add a SPICE Environment Parameters block to your model and set the **Minimum conductance GMIN** parameter to the desired value.

## Parameters

### Initial value, $I1$

The value of the output current at time zero. The default value is  $0$  A.

### Pulse value, $I2$

The value of the output current when the output is high. The default value is  $0$  A.

### Pulse delay time, $TD$

The time at which the pulse first starts. The default value is  $0$  s.

**Pulse rise time, TR**

The time it takes the output current to rise from the **Initial value, I1** value to the **Pulse value, I2** value. The default value is  $1e-09$  s. The value must be greater than or equal to 0.

**Pulse fall time, TF**

The time it takes the output current to fall from the **Pulse value, I2** value to the **Initial value, I1** value. The default value is  $1e-09$  s. The value must be greater than or equal to 0.

**Pulse width, PW**

The time width of the output pulse. The default value is  $Inf$  s. The value must be greater than 0.

**Pulse period, PER**

The period of the output pulse. The default value is  $Inf$  s. This value means that the block produces a single pulse with an infinite period. The value must be greater than 0.

## Ports

The block has the following ports:

+

Positive electrical voltage.

-

Negative electrical voltage.

## See Also

Pulse Voltage Source

## Pulse Voltage Source

Model periodic square pulse voltage source



## Library

SPICE-Compatible Components/Sources

## Description

The Pulse Voltage Source block represents a voltage source whose output voltage value is a periodic square pulse as a function of time and is independent of the current through the source. The following equations describe the output voltage as a function of time:

$$V_{out}(0) = V1$$

$$V_{out}(TD) = V1$$

$$V_{out}(TD + TR) = V2$$

$$V_{out}(TD + TR + PW) = V2$$

$$V_{out}(TD + TR + PW + TF) = V1$$

$$V_{out}(TD + PER) = V1$$

where:

- $V1$  is the **Initial value**, **V1** parameter value.
- $V2$  is the **Pulse value**, **V2** parameter value.
- $TD$  is the **Pulse delay time**, **TD** parameter value.
- $TR$  is the **Pulse rise time**, **TR** parameter value.

- $TF$  is the **Pulse fall time, TF** parameter value.
- $PW$  is the **Pulse width, PW** parameter value.
- $PER$  is the **Pulse period, PER** parameter value.

The block determines the values at intermediate time points by linear interpolation.

The specified values for  $PW$  and  $PER$  have the following effect on the block output:

- If both  $PW$  and  $PER$  are infinite, the block produces a step response at time  $TD$ .
- If  $PER$  is infinite and  $PW$  is finite, the block produces a single pulse of width  $PW$  and infinite period.
- If  $PW$  is infinite and  $PER$  is finite, the block produces a step response with pulses of width  $TR$  to a value  $V1$  every  $PER$  seconds.
- If  $PW > PER$ , the block produces a step response with pulses of width  $TR$  to a value  $V1$  every  $PER$  seconds.

## Parameters

### Initial value, V1

The value of the output voltage at time zero. The default value is 0 V.

### Pulse value, V2

The value of the output voltage when the output is high. The default value is 0 V.

### Pulse delay time, TD

The time at which the pulse first starts. The default value is 0 s.

### Pulse rise time, TR

The time it takes the output voltage to rise from the **Initial Value, V1** value to the **Pulse Value, V2** value. The default value is  $1e-09$  s. The value must be greater than or equal to 0.

### Pulse fall time, TF

The time it takes the output voltage to fall from the **Pulse Value, V2** value to the **Initial Value, V1** value. The default value is  $1e-09$  s. The value must be greater than or equal to 0.

### Pulse width, PW

The time width of the output pulse. The default value is Inf s.

### **Pulse period, PER**

The period of the output pulse. The default value is Inf s. This value means that the block produces a single pulse with an infinite period.

## **Ports**

The block has the following ports:

+

Positive electrical voltage.

-

Negative electrical voltage.

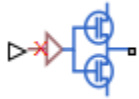
## **See Also**

Pulse Current Source



# Push-Pull Output

Represent CMOS complementary output stage behaviorally



## Library

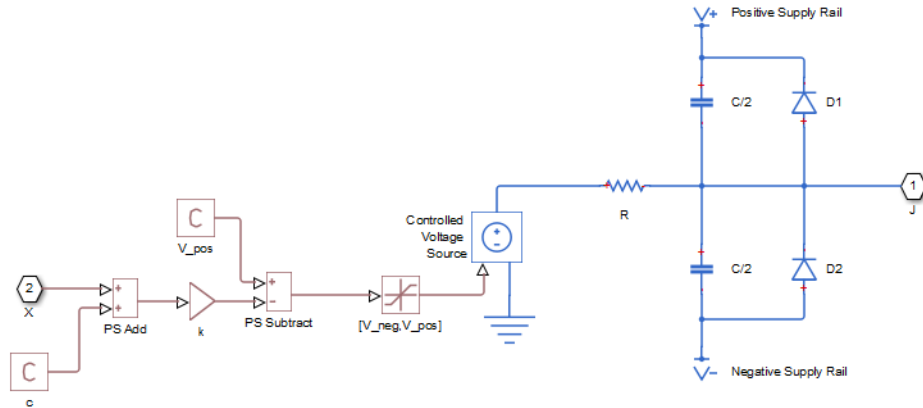
Integrated Circuits

## Description

The Push-Pull Output block represents a CMOS complementary output stage behaviorally. To improve simulation speed, the block does not model all the internal individual MOSFET devices that make up the gate. You can use this block to create a representative output current-voltage relationship when defining an integrated circuit model behavior with Physical Signal blocks from the Simscape Foundation library.

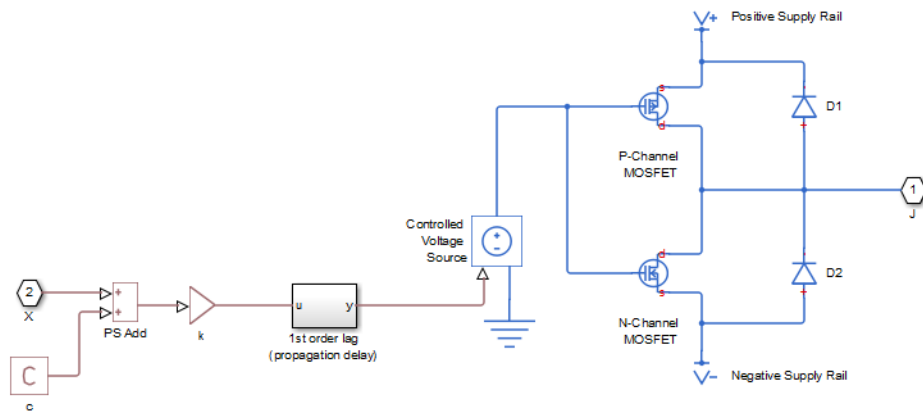
You can choose between are two output current-voltage relationships:

- **Linear** — The block represents the output as a voltage source plus series resistance and parallel capacitance, as shown in the following figure. The value you specify for the **Output resistance** parameter is assigned to the series resistance, and the capacitance values are determined by matching the RC time constant to the **Propagation delay** parameter value.



The input to the Controlled Voltage Source block is limited to be between the supply rails, and it is also inverted by subtraction from the supply voltage. The inversion makes it behave like a complementary output stage, with a high gate-source voltage resulting in a low output.

- **Quadratic** — The output stage is modeled by the two MOSFETs that constitute the complementary pair. The MOSFET parameters are derived from the output resistance values and short-circuit currents that you specify as mask parameters. The gate input demand is lagged to approximate the **Propagation delay** parameter value.



Both **Linear** and **Quadratic** output models add an offset and scale the physical input  $X$  so that the gate voltage is given by:

$$V_g = k \cdot (X + c)$$

where

- $k$  is the input signal scaling.
- $c$  is the input signal offset.

The offset and scaling can be used, for example, to match logical values for  $X$  (that is, range  $[0, 1]$ ) to  $[V_-, V_+]$  at the output pin. For example, if  $V_+ = 10V$  and  $V_- = 0$ , then to match the signal logical values to this voltage range, set  $c = -1$  and  $k = -10$ .

For both **Linear** and **Quadratic** output models, the protection diodes D1 and D2 act to limit the output voltage range. These diodes are Diode blocks from the Simscape Foundation library, that is, piecewise linear diodes defined by their forward voltage and on resistance. If the voltage across D1 rises above the forward voltage, then the diode starts to conduct, and provided that the on resistance is low, it effectively prevents the output rising above  $V_+$  plus the diode forward voltage drop. An equivalent behavior results if the output voltage drops too low.

The output model is very similar to that used for the logic blocks. For a plot of a typical output V-I characteristic when using the **Quadratic** output model, see [Selecting the Output Model for Logic Blocks](#).

---

**Note** This block is constructed out of blocks from the Simscape Physical Signals library (such as PS Add, PS Gain, and so on). Currently, the blocks in the Physical Signals library do not support unit propagation and checking. For more information, see “How to Work with Physical Units” (Simscape).

---

## Basic Assumptions and Limitations

The Push-Pull Output block has the following limitations:

- The block does not accurately model dynamic response.
- The **Quadratic** output model does not model any output capacitance effects. Add output capacitance externally to the block if required.

## Parameters

- “Input Scaling Tab” on page 1-1246
- “Output Characteristics Tab” on page 1-1246
- “Supply Voltage Tab” on page 1-1248
- “Initial Conditions Tab” on page 1-1248

### Input Scaling Tab

#### Input signal scaling, $k$

The input physical signal  $X$  is mapped to the gate voltage by  $Vg = k \cdot (X + c)$ , where  $k$  is the input signal scaling. Use this parameter in conjunction with the **Input signal offset,  $c$**  to map the range of  $X$  to the voltage range defined by the power supply. The default value is 1 V.

#### Input signal offset, $c$

The input physical signal  $X$  is mapped to the gate voltage by  $Vg = k \cdot (X + c)$ , where  $c$  is the input signal offset. Use this parameter in conjunction with the **Input signal scaling,  $k$**  to map the range of  $X$  to the voltage range defined by the power supply. The default value is 0.

### Output Characteristics Tab

#### Output current-voltage relationship

Select the output model, **Linear** or **Quadratic** output model options. If **Linear**, the output voltage drops linearly with output current. If you select **Quadratic**, then the output voltage dependency on output current is defined by the quadratic I-V characteristics of the two output MOSFET devices. The default value is **Linear**.

#### Output resistance

Defines one over the slope of the output I-V characteristic. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter. The default value is 25  $\Omega$ .

#### Power rail voltages, [V- V+], used for measurements

Defines the rail voltages for which mask data output resistances and currents are defined. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter. The default value is [ 0 5 ] V.

**Output resistance values at zero output current and at I<sub>OH</sub> when V<sub>g</sub>=V<sub>-</sub>**

A row vector [ R<sub>OH1</sub> R<sub>OH2</sub> ] of two resistance values. The first value R<sub>OH1</sub> is the gradient of the output voltage-current relationship when the complementary pair output is HIGH (V<sub>g</sub>=V<sub>-</sub>) and there is no output current. The second value R<sub>OH2</sub> is the gradient of the output voltage-current relationship when the output is HIGH and the output current is I<sub>OH</sub>. This parameter is available when you select the Quadratic option for the **Output current-voltage relationship** parameter. The default value is [ 25 250 ] Ω.

**Output current I<sub>OH</sub> when output is shorted to V<sub>-</sub> and V<sub>g</sub>=V<sub>-</sub>**

The resulting current when the output is HIGH (V<sub>g</sub>=V<sub>-</sub>), but the load forces the output voltage to the negative supply rail. This parameter is available when you select the Quadratic option for the **Output current-voltage relationship** parameter. The default value is 63 mA.

**Output resistance values at zero output current and at I<sub>OL</sub> when V<sub>g</sub>=V<sub>+</sub>**

A row vector [ R<sub>OL1</sub> R<sub>OL2</sub> ] of two resistance values. The first value R<sub>OL1</sub> is the gradient of the output voltage-current relationship when the complementary pair output is LOW (V<sub>g</sub>=V<sub>+</sub>) and there is no output current. The second value R<sub>OL2</sub> is the gradient of the output voltage-current relationship when the output is LOW and the output current is I<sub>OL</sub>. This parameter is available when you select the Quadratic option for the **Output current-voltage relationship** parameter. The default value is [ 30 800 ] Ω.

**Output current I<sub>OL</sub> when output is shorted to V<sub>+</sub> and V<sub>g</sub>=V<sub>+</sub>**

The resulting current when the output is LOW (V<sub>g</sub>=V<sub>+</sub>), but the load forces the output voltage to the positive supply voltage. This parameter is available when you select the Quadratic option for the **Output current-voltage relationship** parameter. The default value is -45 mA.

**Propagation delay**

Time it takes for the output to reach 63.2% of its final value following a step change in the input, X. For Quadratic output, it is implemented by the lagged gate input demand. The default value is 25 ns.

**Protection diode on resistance**

The gradient of the voltage-current relationship for the protection diodes when forward biased. The default value is 5 Ω.

**Protection diode forward voltage**

The voltage above which the protection diode is turned on. The default value is 0.6 V.

## Supply Voltage Tab

### Negative power rail voltage, V-

Negative power supply voltage applied to the N-channel MOSFET source pin. The default value is 0 V.

### Positive power rail voltage, V+

Positive power supply voltage applied to the P-channel MOSFET source pin. The default value is 5 V.

## Initial Conditions Tab

### Initial output voltage

This parameter is visible when you select the **Linear** option for the **Output current-voltage relationship** parameter on the **Output Characteristics** tab. The parameter is used to set the voltage on the output capacitors so that the output voltage is initialized to the parameter's value. The default value is 0 V.

### Initial input signal

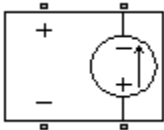
This parameter is visible when you select the **Quadratic** option for the **Output current-voltage relationship** parameter on the **Output Characteristics** tab. The parameter is used to initialize the propagation delay first-order lag such that there is no transient at time zero. The default value is 0 V.

## Ports

The block has one input physical signal port X and one electrical conserving port that outputs the resulting voltage.

# PVCCS

Model polynomial voltage-controlled current source



## Library

SPICE-Compatible Components/Sources

## Description

The PVCCS (Polynomial Voltage-Controlled Current Source) block represents a current source whose output current value is a polynomial function of the voltage across the input ports. The following equations describe the current through the source as a function of time:

- If you specify an  $n$ -element vector of polynomial coefficients for the **Polynomial coefficients** parameter:

$$I_{out} = p(0) + p(1) * V_{in} + \dots + p(n-1) * V_{in}^{n-1} + p(n) * V_{in}^n$$

- If you specify a scalar coefficient for the **Polynomial coefficients** parameter:

$$I_{out} = p * V_{in}$$

where:

- $V_{in}$  is the voltage across the input ports.
- $p$  is the **Polynomial coefficients** parameter value.

The block uses a small conductance internally to prevent numerical simulation issues. The conductance connects the output ports of the device and has a conductance  $GMIN$ :

- By default, *GMIN* matches the **Minimum conductance GMIN** parameter of the SPICE Environment Parameters block, whose default value is 1e-12.
- To change *GMIN*, add a SPICE Environment Parameters block to your model and set the **Minimum conductance GMIN** parameter to the desired value.

## Parameters

### Polynomial coefficients

The polynomial coefficients that relate the input voltage to the output current, as described in the preceding section. The default value is [ 0 1 ].

## Ports

The block has the following ports:

+

Positive electrical input voltage.

-

Negative electrical input voltage.

N+

Positive electrical output voltage.

N-

Negative electrical output voltage.

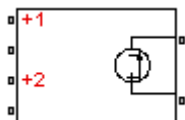
## See Also

PCCCS, PCCVS, PVCCS2, and PVCVS



## PVCCS2

Model polynomial voltage-controlled current source with two controlling inputs



## Library

SPICE-Compatible Components/Sources

## Description

The PVCCS2 (Two-Input Polynomial Voltage-Controlled Current Source) block represents a current source whose output current value is a polynomial function of the voltages across the pairs of controlling input ports. The following equations describes the current through the source as a function of time:

$$I_{out} = p_1 + p_2 * V_{in1} + p_3 * V_{in2} + p_4 * V_{in1}^2 + p_5 V_{in1} * V_{in2} + p_6 * V_{in2}^2 + \dots$$

where:

- $V_{in1}$  is the voltage across the first pair of input ports.
- $V_{in2}$  is the voltage across the second pair of input ports.
- $p$  is the **Polynomial coefficients** parameter value.

The block uses a small conductance internally to prevent numerical simulation issues. The conductance connects the output ports of the device and has a conductance  $GMIN$ :

- By default,  $GMIN$  matches the **Minimum conductance GMIN** parameter of the SPICE Environment Parameters block, whose default value is 1e-12.
- To change  $GMIN$ , add a SPICE Environment Parameters block to your model and set the **Minimum conductance GMIN** parameter to the desired value.

## Parameters

### Polynomial coefficients

The polynomial coefficients that relate the input voltage to the output current, as described in the preceding section. The default value is [ 0 1 1 ].

## Ports

The block has the following ports:

+1

Positive electrical input voltage of first controlling source.

-1

Negative electrical input voltage of first controlling source.

+2

Positive electrical input voltage of second controlling source.

-2

Negative electrical input voltage of second controlling source.

N+

Positive electrical output voltage.

N-

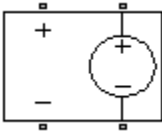
Negative electrical output voltage.

## See Also

PCCCS2, PCCVS2, PVCCS, and PVCVS2

## PVCVS

Model polynomial voltage-controlled voltage source



## Library

SPICE-Compatible Components/Sources

## Description

The PVCVS (Polynomial Voltage-Controlled Voltage Source) block represents a voltage source whose output voltage value is a polynomial function of the voltage across the input ports. The following equations describe the voltage across the source as a function of time:

- If you specify an  $n$ -element vector of polynomial coefficients for the **Polynomial coefficients** parameter:

$$V_{out} = p(0) + p(1) * V_{in} + \dots + p(n-1) * V_{in}^{n-1} + p(n) * V_{in}^n$$

- If you specify a scalar coefficient for the **Polynomial coefficients** parameter:

$$V_{out} = p * V_{in}$$

where:

- $V_{in}$  is the voltage across the input ports.
- $p$  is the **Polynomial coefficients** parameter value.

## Parameters

### Polynomial coefficients

The polynomial coefficients that relate the input voltage to the output voltage, as described in the preceding section. The default value is [ 0 1 ].

## Ports

The block has the following ports:

+

Positive electrical input voltage.

-

Negative electrical input voltage.

N+

Positive electrical output voltage.

N-

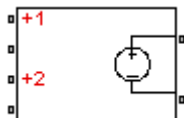
Negative electrical output voltage.

## See Also

PCCCS, PCCVS, PVCCS, and PVCVS2

## PVCVS2

Model polynomial voltage-controlled voltage source with two controlling inputs



## Library

SPICE-Compatible Components/Sources

## Description

The PVCVS2 (Two-Input Polynomial Voltage-Controlled Voltage Source) block represents a voltage source whose output voltage value is a polynomial function of the voltages across the pairs of controlling input ports. The following equations describes the voltage across the source as a function of time:

$$V_{out} = p_1 + p_2 * V_{in1} + p_3 * V_{in2} + p_4 * V_{in1}^2 + p_5 V_{in1} * V_{in2} + p_6 * V_{in2}^2 + \dots$$

where:

- $V_{in1}$  is the voltage across the first pair of input ports.
- $V_{in2}$  is the voltage across the second pair of input ports.
- $p$  is the **Polynomial coefficients** parameter value.

The block uses a small conductance internally to prevent numerical simulation issues. The conductance connects the output ports of the device and has a conductance *GMIN*:

- By default, *GMIN* matches the **Minimum conductance GMIN** parameter of the SPICE Environment Parameters block, whose default value is 1e-12.
- To change *GMIN*, add a SPICE Environment Parameters block to your model and set the **Minimum conductance GMIN** parameter to the desired value.

## Parameters

### Polynomial coefficients

The polynomial coefficients that relate the input voltage to the output voltage, as described in the preceding section. The default value is [ 0 1 1 ].

## Ports

The block has the following ports:

+1

Positive electrical input voltage of first controlling source.

-1

Negative electrical input voltage of first controlling source.

+2

Positive electrical input voltage of second controlling source.

-2

Negative electrical input voltage of second controlling source.

N+

Positive electrical output voltage.

N-

Negative electrical output voltage.

## See Also

PCCCS2, PCCVS2, PVCCS2, and PVCVS

# PWL Current Source

Model lookup table current source



## Library

SPICE-Compatible Components/Sources

## Description

The PWL Current Source block represents a current source that you specify in lookup table form using a vector of time values and a vector of the corresponding current values. You must specify at least four time-current value pairs. The block generates a time-dependent current based on these time-current values using the selected interpolation and extrapolation methods. You have a choice of three interpolation methods and two extrapolation methods. The output current is independent of the voltage across the terminals of the source.

The block uses a small conductance internally to prevent numerical simulation issues. The conductance connects the + and - ports of the device and has a conductance *GMIN*:

- By default, *GMIN* matches the **Minimum conductance GMIN** parameter of the SPICE Environment Parameters block, whose default value is 1e-12.
- To change *GMIN*, add a SPICE Environment Parameters block to your model and set the **Minimum conductance GMIN** parameter to the desired value.

## Parameters

### Time specification

The vector of time values as a tabulated 1-by-n array. The time values vector must be strictly monotonically increasing. The values can be non-uniformly spaced. The default value is [ 0 1 2 3 4 ] s.

### Current at specified time

The vector of current values as a tabulated 1-by-n array. The current values vector must be the same size as the time values vector. The default value is [ 0 0 0 0 0 ] A.

### Interpolation method

Select the method the block uses determine the output current values at intermediate time points that are not specified in the preceding vectors:

- **Linear** — Use a linear function. This is the default method.
- **Cubic** — Use the Piecewise Cubic Hermite Interpolation Polynomial (PCHIP). For more information, see [1] and the `pchip` MATLAB function.
- **Spline** — Use the cubic spline interpolation algorithm described in [2].

### Extrapolation method

Select the method the block uses determine the output current values at time points that are outside the time range specified in the preceding vectors:

- **Last point value** — Use the last specified current value at the appropriate end of the range. That is, use the last specified current value for all time values greater than the last specified time argument, and the first specified current value for all time values less than the first specified time argument. This is the default method.
- **Last 2 points** — Extrapolate using the linear method (regardless of the interpolation method specified), based on the last two current values at the appropriate end of the range. That is, use the first and second specified current values if the time value is below the specified range, and the two last specified current values if the time value is above the specified range.

## Ports

The block has the following ports:



+

Positive electrical voltage.

-

Negative electrical voltage.

## References

- [1] D. Kahaner, Cleve Moler, and Stephen Nash *Numerical Methods and Software* Prentice Hall, 1988.
- [2] W.H. Press, B.P. Flannery, S.A. Teulkolsky, and W.T. Wetterling *Numerical Recipes in C: The Art of Scientific Computing* Cambridge University Press, 1992.

## See Also

PWL Voltage Source

## PWL Voltage Source

Model lookup table voltage source



### Library

SPICE-Compatible Components/Sources

### Description

The PWL Voltage Source block represents a voltage source that you specify in lookup table form using a vector of time values and a vector of the corresponding voltage values. You must specify at least four time-current value pairs. The block generates a time-dependent voltage based on these time-voltage values using the selected interpolation and extrapolation methods. You have a choice of three interpolation methods and two extrapolation methods. The output voltage is independent of the current through the source.

### Parameters

#### Time specification

The vector of time values as a tabulated 1-by-n array. The time values vector must be strictly monotonically increasing. The values can be non-uniformly spaced. The default value is [ 0 1 2 3 4 ] s.

#### Voltage at specified time

The vector of voltage values as a tabulated 1-by-n array. The voltage values vector must be the same size as the time values vector. The default value is [ 0 0 0 0 0 ] V.

### Interpolation method

Select the method the block uses determine the output voltage values at intermediate time points that are not specified in the preceding vectors:

- **Linear** — Use a linear function. This is the default method.
- **Cubic** — Use the Piecewise Cubic Hermite Interpolation Polynomial (PCHIP). For more information, see [1] and the `pchip` MATLAB function.
- **Spline** — Use the cubic spline interpolation algorithm described in [2].

### Extrapolation method

Select the method the block uses determine the output voltage values at time points that are outside the time range specified in the preceding vectors:

- **Last point value** — Use the last specified voltage value at the appropriate end of the range. That is, use the last specified voltage value for all time values greater than the last specified time argument, and the first specified voltage value for all time values less than the first specified time argument. This is the default method.
- **Last 2 points** — Extrapolate using the linear method (regardless of the interpolation method specified), based on the last two voltage values at the appropriate end of the range. That is, use the first and second specified voltage values if the time value is below the specified range, and the two last specified voltage values if the time value is above the specified range.

## Ports

The block has the following ports:

+

Positive electrical voltage.

-

Negative electrical voltage.

## References

- [1] D. Kahaner, Cleve Moler, and Stephen Nash *Numerical Methods and Software* Prentice Hall, 1988.

[2] W.H. Press, B.P. Flannery, S.A. Teukolsky, and W.T. Wetterling *Numerical Recipes in C: The Art of Scientific Computing* Cambridge University Press, 1992.

## **See Also**

PWL Current Source

# PWM Generator

Generate pulse width modulated signal

**Library:** Simscape / Electrical / Power Systems / Control /  
Pulse Width Modulation



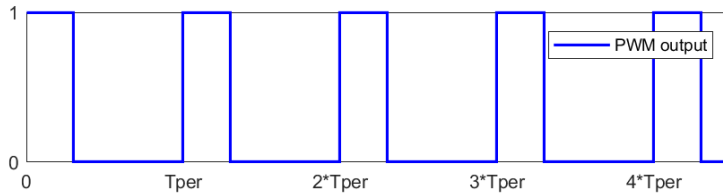
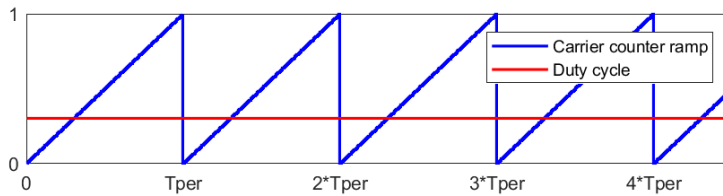
## Description

The PWM Generator block implements a PWM generator. The pulse width modulation technique controls power transfer from one electrical component to another by quickly switching between full power transfer and no power transfer.

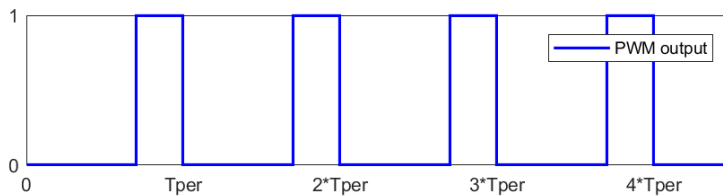
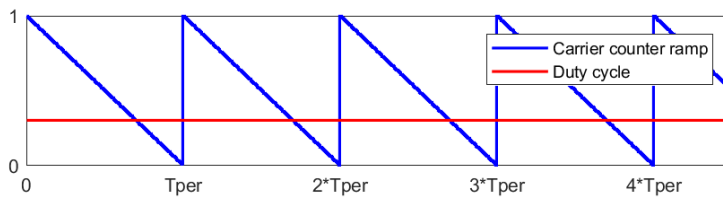
## Working Principle

The PWM generator block outputs either 1 when the duty cycle is greater than the carrier counter value, or 0 otherwise. You can set the period of each cycle by specifying the timer period  $T_{per}$ . You can change the initial output, or phase, of the PWM output by specifying one of three types of carrier counters:

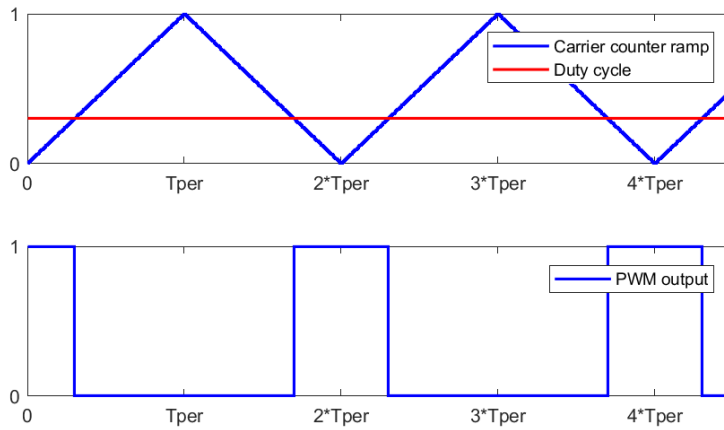
- Up counter — The PWM output signal initializes at the start of the on cycle. This graphic shows the carrier counter signal and the corresponding PWM output.



- Down counter — The PWM output signal initializes at the start of the off cycle. This graphic shows the carrier counter signal and the corresponding PWM output.



- Up-down counter — The PWM output signal initializes halfway through the on cycle. This graphic shows the carrier counter signal and the corresponding PWM output.



## Ports

### Input

#### DC — Duty cycle

scalar

Duty cycle in the range  $[0, 1]$ .

Data Types: single | double

### Output

#### PWM — PWM signal

scalar

Pulse width modulation signal.

Data Types: single | double

## Parameters

### **Carrier counter — Carrier counter strategy**

Up (default) | Down | Up-Down

Use the carrier counter strategy to change the initial behavior of the PWM output:

- Up counter — PWM output begins at the start of the on state.
- Down counter — PWM output begins at the start of the off state.
- Up-down counter — PWM output begins in the middle of the on state.

### **Timer period (s) — PWM period**

0.001 (default) | positive number

PWM timer period.

### **Sample time — Block sample time**

5e-5 (default) | positive number

Sample time for the block. For continuous-time simulation, set to zero. For discrete-time simulation, to ensure adequate resolution in the generated signal, specify a positive value that is less than or equal to  $10 * T_{per}$ , where  $T_{per}$  is the **Timer period (s)**.

## See Also

### **Blocks**

PWM Generator | PWM Generator (Three-phase, Three-level) | PWM Generator (Three-phase, Two-level) | Thyristor 6-Pulse Generator

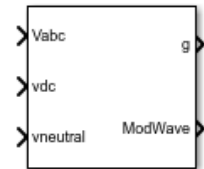
**Introduced in R2017b**



# PWM Generator (Three-phase, Three-level)

Generate three-phase, three-level pulse width modulated waveform

**Library:** Simscape / Electrical / Power Systems / Control / Pulse Width Modulation



## Description

The PWM Generator (Three-phase, Three-level) block controls switching behavior for a three-phase, three-level power converter. The block:

- 1 Calculates on- and off-gating times based on the block inputs:
  - Three sinusoidal reference voltages
  - A DC-link voltage
  - A DC-link neutral point balance control signal
- 2 Uses the gating times to generate 12 switch-controlling pulses.
- 3 Uses the gating times to generate modulation waveforms.

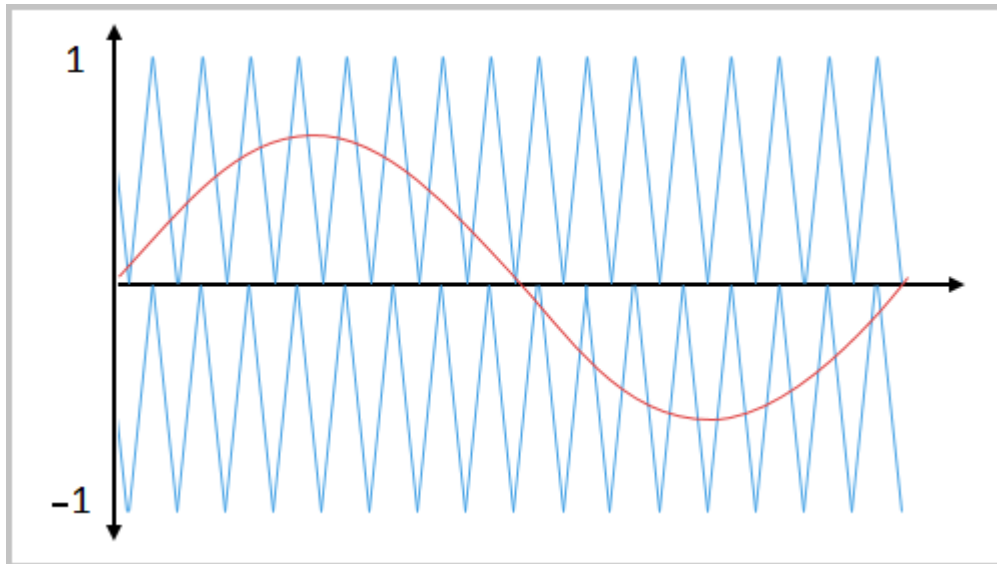
## Sampling Mode

This block allows you to choose natural, symmetric, or asymmetric sampling of the modulation wave.

The PWM Generator (Three-phase, Two-level) block does not perform carrier-based pulse width modulation (PWM). Instead, the block uses input signals to calculate gating times and then uses the gating times to generate both the switch-controlling pulses and the modulation waveforms that it outputs.

Carrier-based PWM is, however, useful for showing how the sampling mode that you select relates to the switch-on and switch-off behavior of the pulses that the block generates. A generator that uses a three-level, carrier-based PWM method:

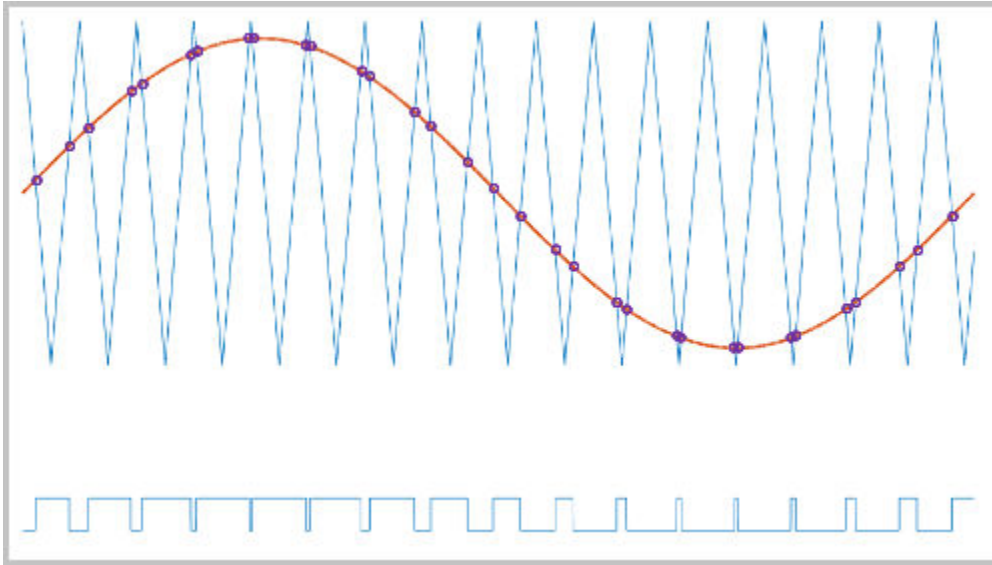
- 1 Samples a reference wave.
- 2 Compares the sample to two parallel triangle carrier waves, separated by one level.



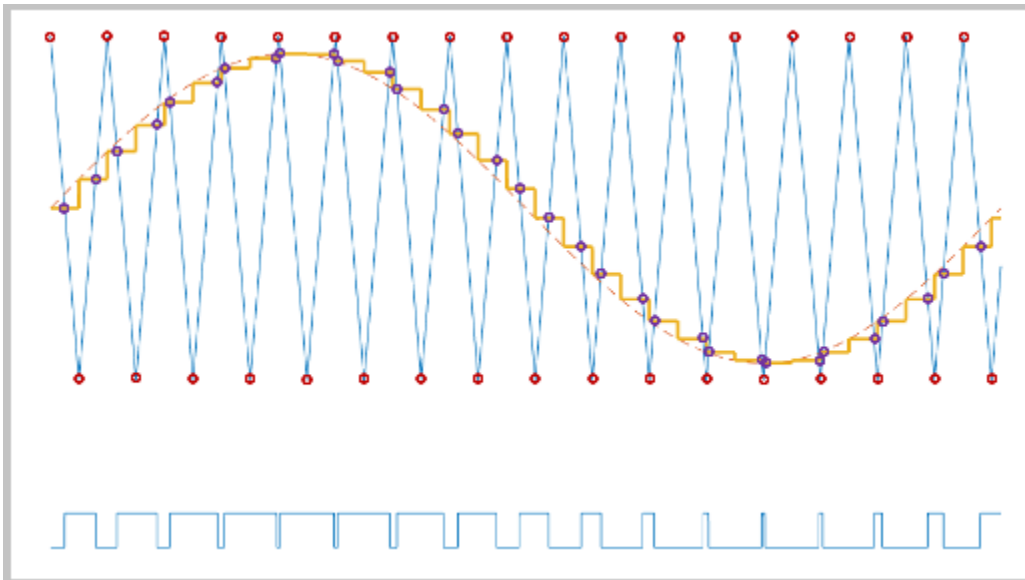
- 3 Generates a switch-on pulse if a sample is higher than the carrier signal or a switch-off pulse if a sample is lower than the carrier wave.

To determine switch-on and switch-off pulse behavior, a three-level carrier-based PWM generator uses these methods to sample each of the triangle waves:

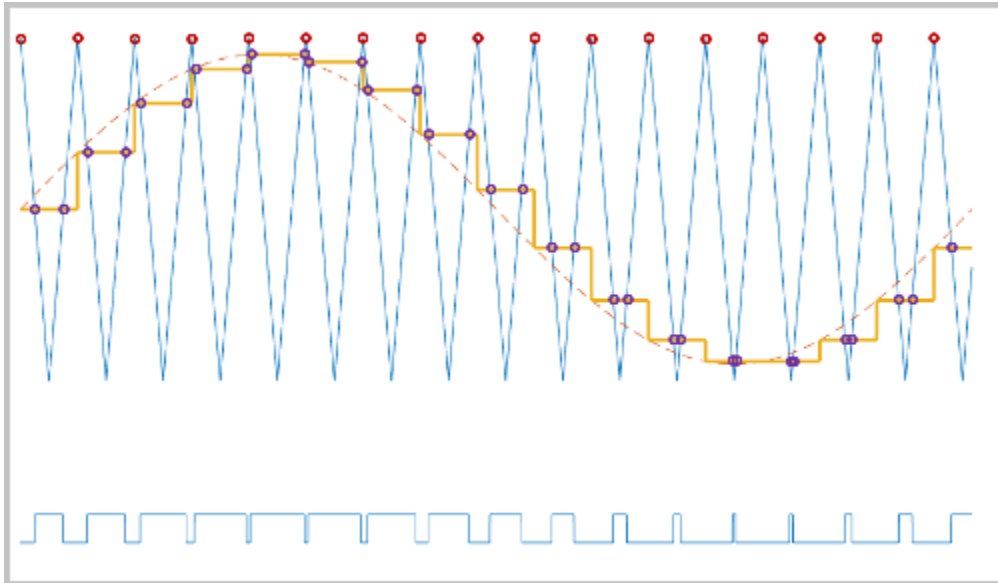
- Natural — The sampling and comparison occur at the intersection points of the modulation wave and the carrier wave.



- Asymmetric — Sampling occurs at the upper and lower boundaries of the carrier wave. The comparison occurs at the intersection that follows the sampling.



- Symmetric — Sampling occurs only at the upper boundary of the carrier wave. The comparison occurs at the intersection that follows the sampling.



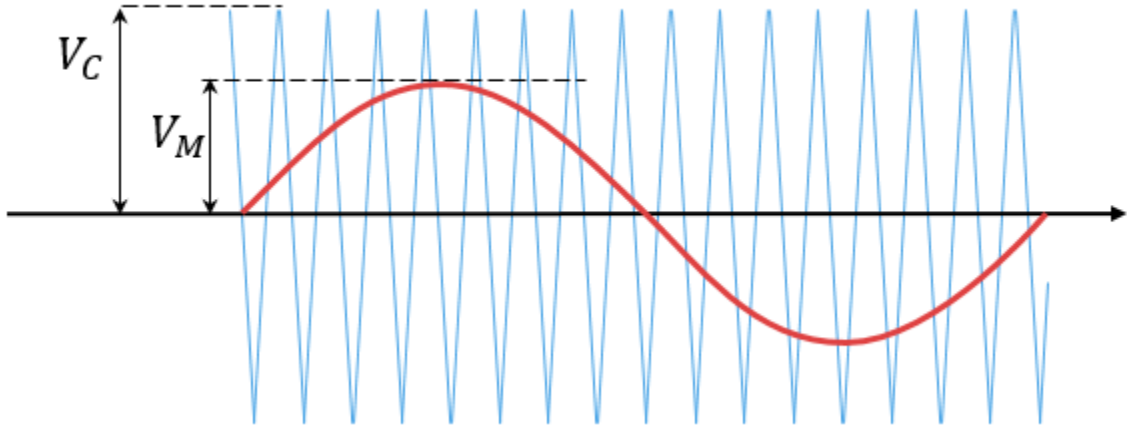
## Overmodulation

The modulation index, which measures the ability of the power converter to output a given voltage, is defined as

$$m = \frac{V_M}{V_C},$$

where

- $m$  is the modulation index.
- $V_m$  is the peak value of the modulation wave.
- $V_c$  is the peak value of the triangle carrier wave.



For three-phase SPWM,

$$V_{peak} = m \frac{v_{dc}}{2},$$

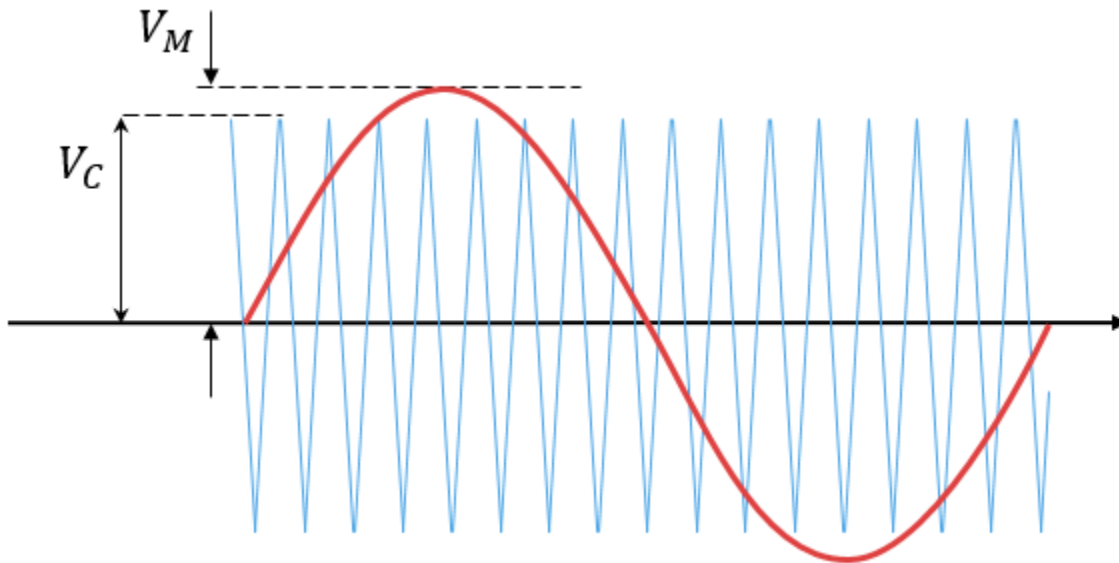
where

- $V_{peak}$  is the peak value of the fundamental component of the phase-to-neutral voltage.
- $v_{dc}$  is the DC-link voltage.

For three-phase space-vector PWM (SVM),

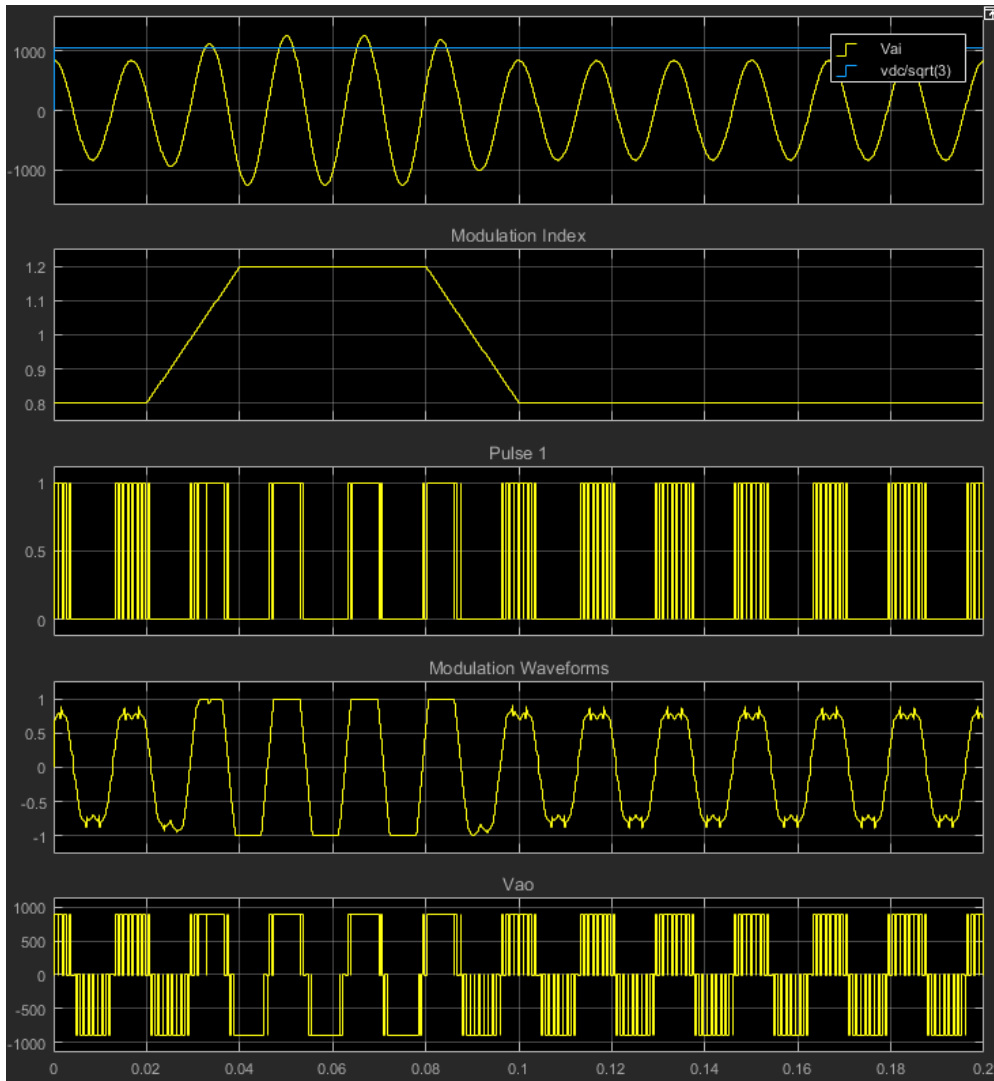
$$V_{peak} = m \frac{v_{dc}}{\sqrt{3}}.$$

For normal steady-state operation,  $0 < m \leq 1$ . If a transient, such as a load increase, causes the amplitude of  $V_m$  to exceed the amplitude of  $V_c$ , overmodulation ( $m > 1$ ) occurs



If overmodulation occurs, the output voltage of the power converter clamps to the positive or negative DC rail.

In the Three-Phase Three-Level PWM Generator example, the **Three-Level Controller** subsystem contains a 1800-V DC-link input, and a modulation index,  $m$ , of 0.8. For SVM, the maximal input voltage is  $1800 / \sqrt{3}$  V, that is 1039.23 V. To demonstrate overmodulation, a transient is added at the beginning of the simulation. The transient forces the amplitudes of the reference voltages to exceed the amplitude of  $1 / \sqrt{3}$  of the DC-link voltage. To highlight overmodulation, the scope includes simulation results for only one of the 12 output pulses and only the  $a$ -phase of the reference voltages, modulation waveforms, and output voltages.



The modulation index is greater than one between 0.03–0.09 seconds. During overmodulation:

- The pulse remains in the on or off position.
- The output voltage clamps to the positive or negative DC rail.

## Input/Output Ports

### Input

#### **Vabc — Three-phase sinusoidal reference signal**

vector

Specify the three sinusoidal voltages, one per phase, that you want the attached converter to output.

#### **vdc — DC-link voltage signal**

scalar

Specify a positive real number for the DC-link voltage of the converter.

#### **vneutral — DC-link neutral point balance control**

scalar

This signal is the output from a feedback-control loop that balances the DC supply. The value of the signal must be a real number between  $-1$  and  $+1$ .

### Output

#### **g — Gate control**

vector

12 pulse waveforms that determine switching behavior in the attached power converter.

#### **ModWave — Modulation wave**

vector

If you are generating code for a platform that has hardware with PWM capability, you can deploy the modulation wave to the hardware. Otherwise, this data is for reference only.

## Parameters

#### **Continuous PWM — Continuous pulse width modulation method**

SPWM: sinusoidal PWM (default) | SVM: space vector modulation



Specify the waveform technique.

**Sampling mode — Wave-sampling method**

Natural (default) | Asymmetric | Symmetric

The sampling mode determines whether the block samples the modulation waveform when the waves intersect or when the carrier wave is at one or both of its boundary conditions.

**Switching frequency (Hz) — Switching rate**

1e3 (default) | positive number

Specify the rate at which you want the switches in the power converter to switch.

**Sample time (s) — Block sample time**

5e-5 (default) | positive number

Specify the time interval between successive block executions (output calculations). To ensure adequate resolution in the generated signal, set this value to be less than or equal to  $1/(50 * F_{sw})$ , where  $F_{sw}$  is the **Switching frequency (Hz)**.

## References

- [1] Chung, D. W., J. S. Kim, and S. K. Sul. "Unified Voltage Modulation Technique for Real Time Three-Phase Power Conversion." *IEEE Transactions on Industry Applications*, Vol. 34, No. 2, 1998, pp. 374-380.
- [2] Seo, J. H., C. H. Choi, and D. S. Hyun. "A new simplified space-vector PWM method for three-level inverters." *IEEE Transactions on Power Electronics*, Vol. 16, No. 4, 2001, pp. 545-550.

## See Also

**Simscape Blocks**

Converter

**Blocks**

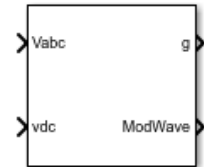
PWM Generator | PWM Generator (Three-phase, Two-level) | Thyristor 6-Pulse Generator

**Introduced in R2016b**

# PWM Generator (Three-phase, Two-level)

Generate three-phase, two-level pulse width modulated waveform

**Library:** Simscape / Electrical / Power Systems / Control / Pulse Width Modulation



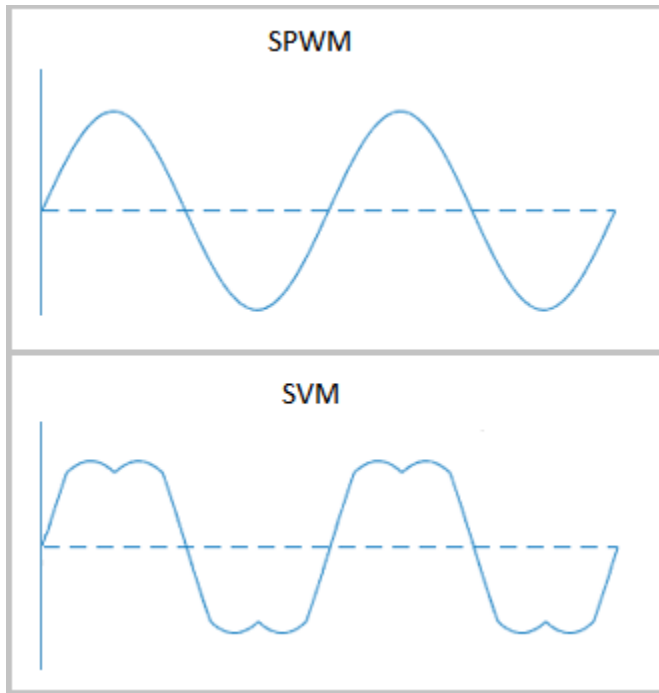
## Description

The PWM Generator (Three-phase, Two-level) block controls switching behavior for a three-phase, two-level power converter. The block:

- 1 Calculates on- and off-gating times based on the block inputs:
  - Three sinusoidal reference voltages, one per phase
  - A DC-link voltage
- 2 Uses the gating times to generate six switch-controlling pulses.
- 3 Uses the gating times to generate modulation waveforms.

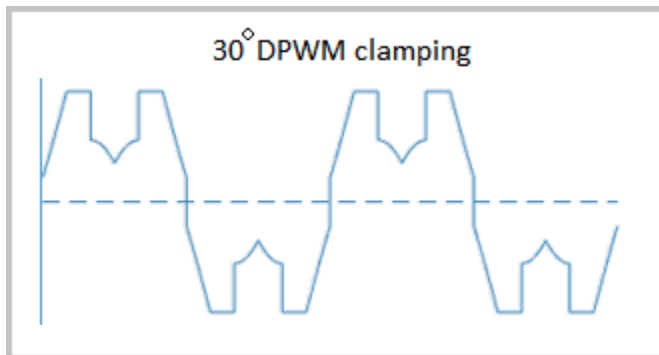
## Continuous and Discontinuous PWM

The block provides modes for both continuous and discontinuous pulse width modulation (PWM). The figure shows the general difference between continuous sinusoidal PWM (SPWM) and continuous space vector modulation (SVM) waveforms.

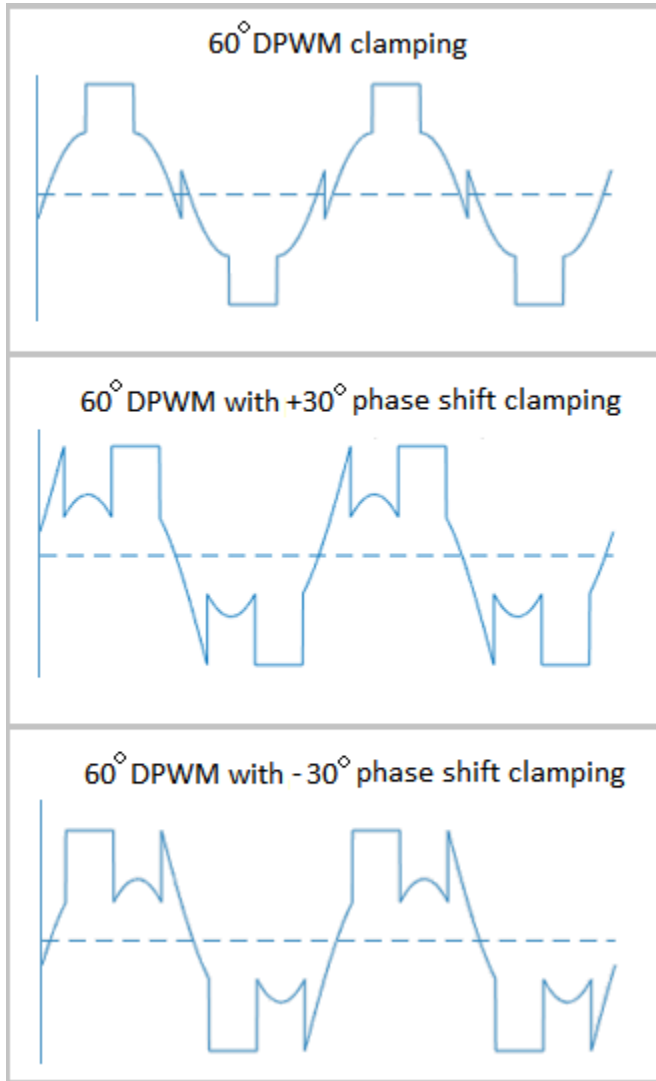


For discontinuous PWM (DPWM), the block clamps the modulation wave to the positive or negative DC rail for a total of 120 degrees during each fundamental period. During the clamping intervals, modulation discontinues.

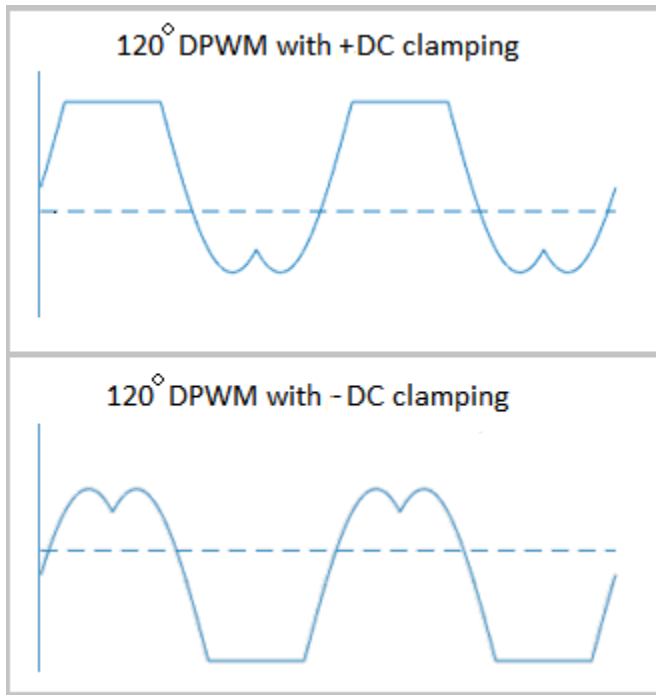
A waveform with 30-degree DPWM has four 30-degree intervals per fundamental period.



Selecting a positive or negative 30-degree phase shift affects the clamping intervals for 60-degree DPWM.



The figure shows the waveforms for positive and negative DC clamping for 120-degree DPWM.



## Sampling Mode

This block allows you to choose natural, symmetric, or asymmetric sampling of the modulation wave.

The PWM Generator (Three-phase, Two-level) block does not perform carrier-based PWM. Instead, the block uses input signals to calculate gating times and then uses the gating times to generate both the switch-controlling pulses and the modulation waveforms that it outputs.

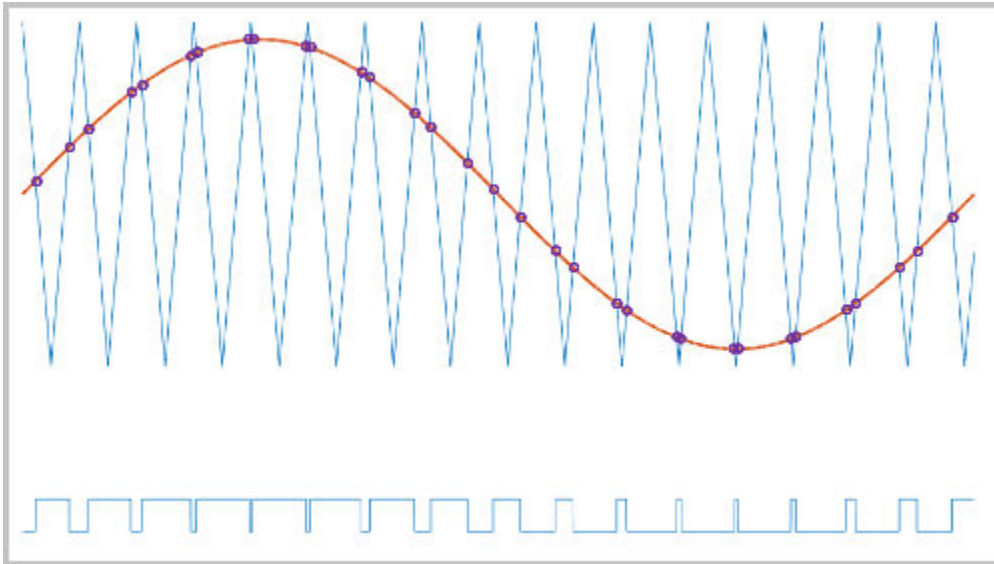
Carrier-based PWM is, however, useful for showing how the sampling mode that you select relates to the switch-on and switch-off behavior of the pulses that the block generates. A generator that uses a two-level, carrier-based PWM method:

- 1 Samples a reference wave.
- 2 Compares the sample to a triangle carrier wave.

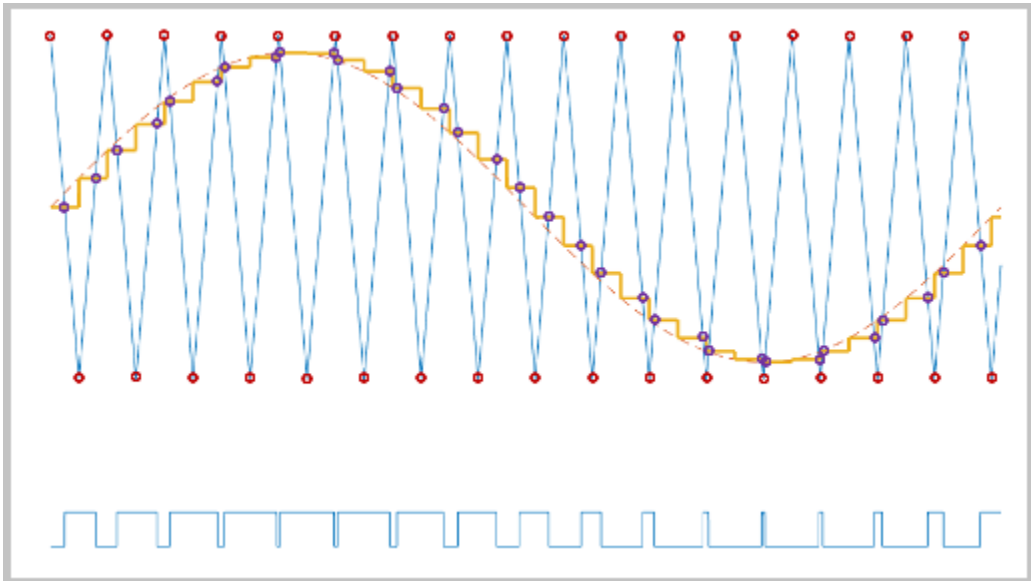
- 3 Generates a switch-on pulse if a sample is higher than the carrier signal or a switch-off pulse if a sample is lower than the carrier wave.

To determine switch-on and switch-off pulse behavior, a two-level carrier-based PWM generator uses these methods to sample the triangle wave:

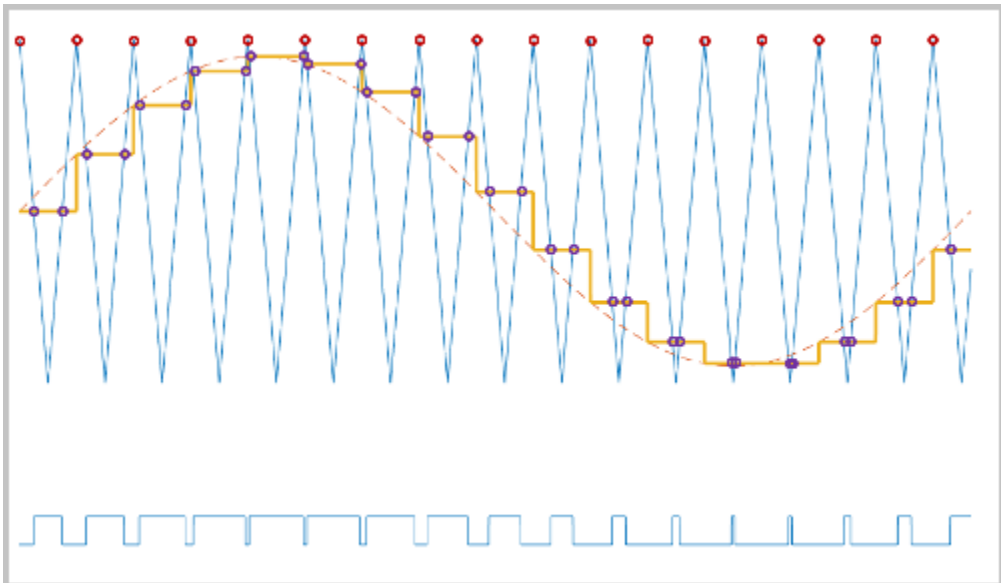
- Natural — The sampling and comparison occur at the intersection points of the modulation wave and the carrier wave.



- Asymmetric — Sampling occurs at the upper and lower boundaries of the carrier wave. The comparison occurs at the intersection that follows the sampling.



- Symmetric — Sampling occurs at only the upper boundary of the carrier wave. The comparison occurs at the intersection that follows the sampling.





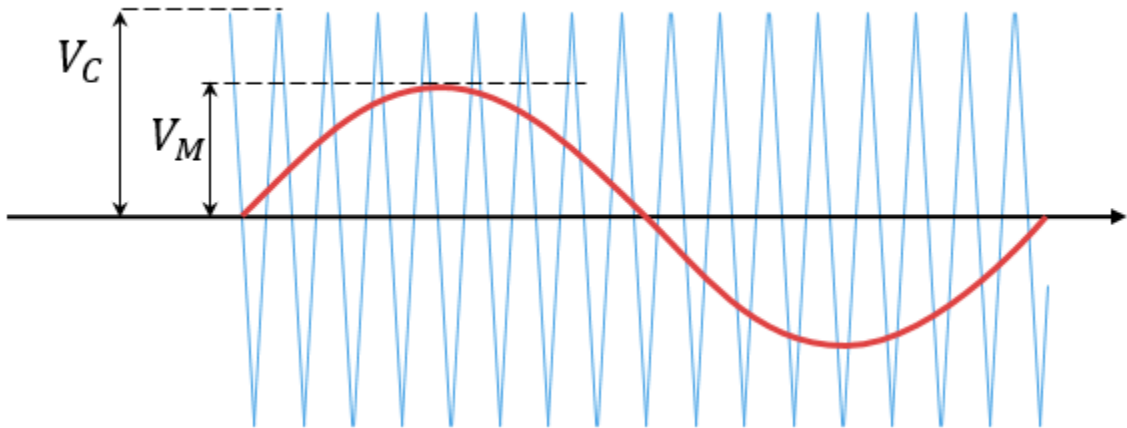
## Overmodulation

The modulation index, which measures the ability of the power converter to output a given voltage, is defined as

$$m = \frac{V_M}{V_C},$$

where

- $m$  is the modulation index.
- $V_m$  is the peak value of the modulation wave.
- $V_c$  is the peak value of the triangle carrier wave.



For three-phase SPWM,

$$V_{peak} = m \frac{v_{dc}}{2},$$

where

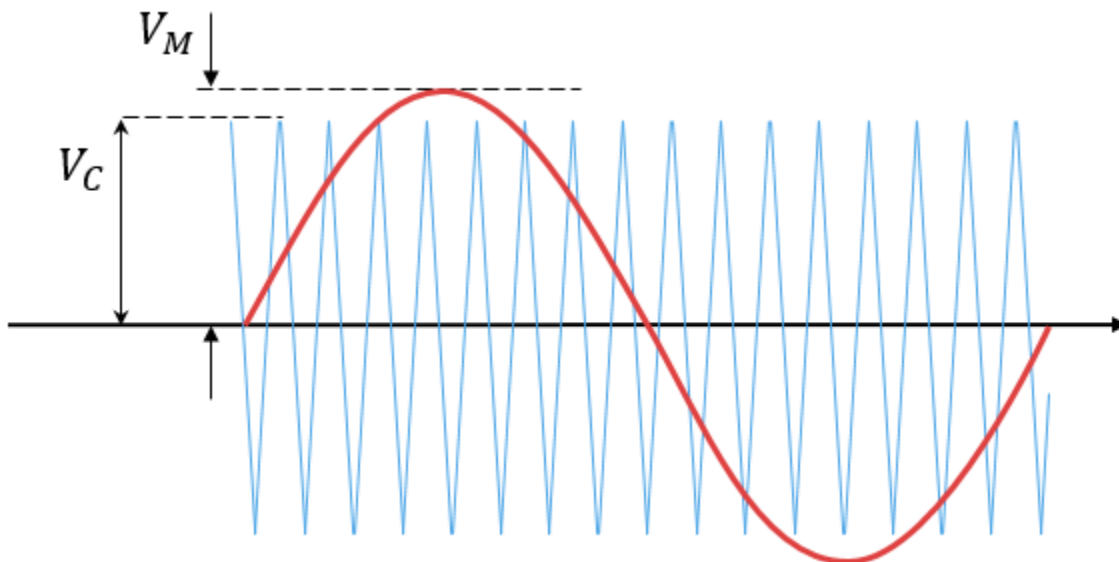
- $V_{peak}$  is the peak value of the fundamental component of the phase-to-neutral voltage.

- $v_{dc}$  is the DC-link voltage.

For three-phase space-vector PWM (SVM) and DPWM,

$$V_{peak} = m \frac{v_{dc}}{\sqrt{3}}.$$

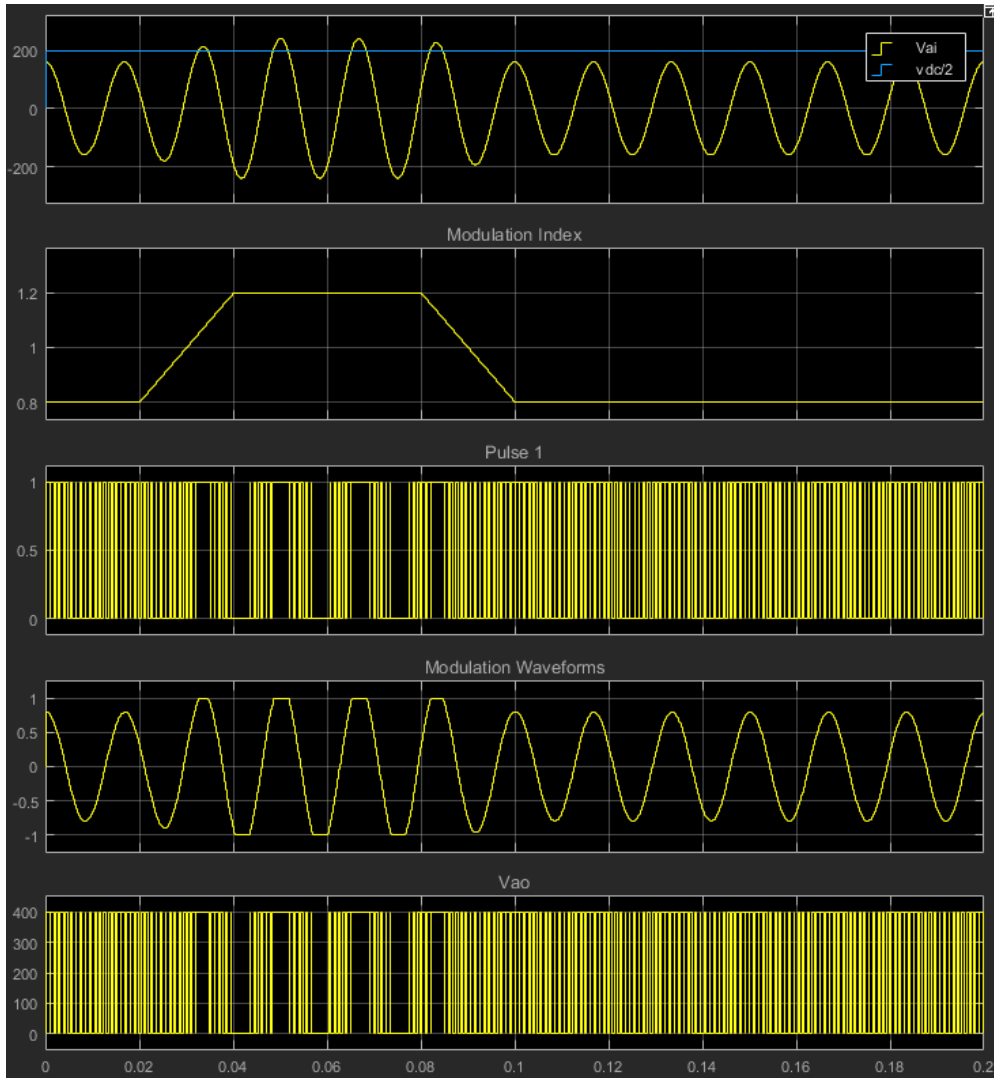
For normal steady-state operation,  $0 < m \leq 1$ . If a transient, such as a load increase, causes the amplitude of  $V_m$  to exceed the amplitude of  $V_c$ , overmodulation ( $m > 1$ ) occurs.



If overmodulation occurs, the output voltage of the power converter clamps to the positive or negative DC rail.

In the Three-Phase Two-Level PWM Generator example, the **Two-Level Controller** subsystem contains a 400-V DC-link input, and a modulation index,  $m$ , of 0.8. For SPWM, the maximal input voltage is  $400 \text{ V}/2$ , that is, 200 V. To demonstrate overmodulation, a transient is added at the beginning of the simulation. The transient forces the amplitudes of the reference voltages to exceed the amplitude of  $1/2$  of the DC-link voltage. To highlight overmodulation, the scope includes simulation results for only one of the six

output pulses and only the  $a$ -phase of the reference voltages, modulation waveforms, and output voltages.



The modulation index is greater than one between 0.03–0.09 seconds. During overmodulation:

- The pulse remains in the on or off position.
- The output voltage,  $V_{ao}$ , clamps to the positive or negative DC rail.

## Input/Output Ports

### Input

#### **Vabc — Three-phase sinusoidal reference signal**

vector

Specify the three sinusoidal voltages, one per phase, that you want the attached converter to output.

#### **vdc — DC-link voltage signal**

scalar

Specify a positive real number for the DC-link voltage of the converter.

### Output

#### **g — Gate control**

vector

Six pulse waveforms that determine switching behavior in the attached power converter.

#### **ModWave — Modulation wave**

vector

If you are generating code for a platform that has hardware with PWM capability, you can deploy the modulation wave to the hardware. Otherwise, this data is only for your reference.

## Parameters

#### **PWM mode — Pulse width modulation method**

Continuous PWM (CPWM) (default) | Discontinuous PWM (DPWM)

Discontinuous PWM clamps the waveform to the DC rail for a total of 120 degrees in each fundamental period. Continuous PWM does not.

### **Continuous PWM — Continuous pulse width modulation method**

SPWM: sinusoidal PWM (default) | SVM: space vector modulation

#### **Dependencies**

The **Continuous PWM** parameter is only available when you set the **PWM mode** parameter to Continuous PWM (CPWM).

### **Sampling mode — Wave-sampling method**

Natural (default) | Asymmetric | Symmetric

The sampling mode determines whether the block samples the modulation waveform when the waves intersect or when the carrier wave is at one or both of its boundary conditions.

### **Switching frequency (Hz) — Switching rate**

1e3 (default) | positive number

Specify the rate at which you want the switches in the power converter to switch.

### **Sample time (s) — Block sample time**

5e-5 (default) | positive number

Specify the time interval between successive block executions (output calculations). To ensure adequate resolution in the generated signal, set this value to be less than or equal to  $1/(50 * F_{sw})$ , where  $F_{sw}$  is the **Switching frequency (Hz)**.

### **Discontinuous PWM (DPWM) — Clamping method**

60 DPWM: 60 degree discontinuous PWM (default)

Specify the method for distributing the 120 degrees per period during which the block clamps the modulation wave to the DC rail. Other options are:

- 60 DPWM (+30 degree shift): +30 degree shift from 60 DPWM
- 60 DPWM (-30 degree shift): -30 degree shift from 60 DPWM
- 30 DPWM: 30 degree discontinuous PWM
- 120 DPWM: positive dc component
- 120 DPWM: negative dc component

When the wave is clamped, modulation discontinues.

## Dependencies

The **Discontinuous PWM** parameter is only available when you set the **PWM mode** parameter to Discontinuous PWM (DPWM).

## References

- [1] Chung, D. W., J. S. Kim, and S. K. Sul. "Unified Voltage Modulation Technique for Real Time Three-Phase Power Conversion." *IEEE Transactions on Industry Applications*, Vol. 34, No. 2, 1998, pp. 374-380.
- [2] Hava, A. M., R. J. Kerkman, and T. A. Lipo. "Simple Analytical and Graphical Methods for Carrier-Based PWM-VSI Drives." *IEEE Transactions on Power Electronics*, Vol. 14, No. 1, 1999, pp. 49-61.

## See Also

### Simscape Blocks

Converter

### Blocks

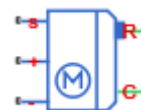
PWM Generator | PWM Generator (Three-phase, Three-level) | Thyristor 6-Pulse Generator

### Introduced in R2016b

## RC Servo

Radio control servomotor with PWM-based angular position tracking and fault modeling

**Library:** Simscape / Electrical / Electronics and Mechatronics /  
Actuators & Drivers / Rotational Actuators



## Description

The RC Servo block represents a small DC motor with a gearbox and control circuitry, commonly used in quadcopters, radio-controlled planes and helicopters, and other mechatronic devices. RC servos provide angular position control of the output shaft over a limited angle range. The angle demand is set by the pulse width of a PWM signal applied to port s.

The RC Servo block models the following effects:

- Torque-speed behavior based on DC motor equations
- Position tracking based on the input PWM signal pulse width
- Internal gear reduction ratio, including associated friction losses
- Mechanical end stops, to prevent the output shaft being driven out of range by the load
- Position measurement error
- Fault modeling

The motor equations are the same as those used by the DC Motor block, except that the inductance is not modeled. The RC Servo block determines the equation parameters using the stall torque and no-load speeds, and makes a correction to take account of the backdrive torque.

## Faults

The RC Servo block allows you to model several types of faults:

- Fail off — No electrical torque.
- Fail forward — Rotates in a positive direction to hit the upper end stop.
- Fail reverse — Rotates in a negative direction to hit the lower end stop.
- Failed winding — Torque is applied only if the motor rotor lines up with one of the two remaining functioning windings.

The block can trigger fault events:

- At a specific time
- When a current limit is exceeded for longer than a specific time interval

You can enable or disable these trigger mechanisms separately, or use them together if more than one trigger mechanism is required in a simulation. When more than one mechanism is enabled, the first mechanism to trigger the fault takes precedence. In other words, component fails no more than once per simulation.

You can choose whether to issue an assertion when a fault occurs, by using the **Reporting when a fault occurs** parameter. The assertion can take the form of a warning or an error. By default, the block does not issue an assertion.

## Variables

Use the **Variables** section of the block interface to set the priority and initial target values for the block variables prior to simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

## Basic Assumptions and Limitations

- This block has no optional thermal port.
- If you simulate the model with a fixed-step solver, for example, using a local solver, the step size must be small enough to get the required resolution of the input pulse width. MathWorks recommends that you use this block with variable step solvers for fast desktop simulation.



## Ports

### Conserving

#### **s — Shaft control**

electrical

Electrical conserving port associated with the PWM control signal. The output shaft angle demand is set by the pulse width of the voltage applied to this port.

#### **+ — Positive terminal**

electrical

Electrical conserving port associated with the motor positive terminal.

#### **- — Negative terminal**

electrical

Electrical conserving port associated with the motor negative terminal.

#### **R — Rotor**

mechanical rotational

Mechanical rotational conserving port associated with the rotor.

#### **C — Casing**

mechanical rotational

Mechanical rotational conserving port associated with the stator (casing).

## Parameters

### Electrical Torque

#### **Stall torque — Maximum load torque**

2.4 kg\*cm (default)

Maximum load torque that the RC servo can move without stalling (stopping).

**Time to travel 60 degrees (no load) — Output shaft 60 degrees turn time**

0.28 s (default)

The time for the output shaft to turn 60 degrees when the motor is driving no load.

**Corresponding nominal voltage — DC supply voltage**

4.8 V (default)

The DC supply voltage used when measuring stall torque and time to travel 60 degrees.

**Rotational range — Output shaft angular range**

[0 180] deg (default)

The output shaft angular range of the RC servo.

**Corresponding pulse widths — Input pulse widths range**

[550 2330] us (default)

The input pulse widths corresponding to the minimum and maximum output angles, as defined by the **Rotational range** parameter. Pulse widths outside of this range are clipped by the block to stay within this range.

## Control

**Pulse threshold — Threshold voltage for pulse control**

3 V (default)

The input pulse is detected as high when the voltage between the s and - ports is above this level.

**Signal input resistance — Impedance**

18 kOhm (default)

The electrical impedance measured between the s and - ports.

**Angle resolution — Minimum error allowed between the demanded and measured output shaft angle**

0.1 deg (default)

When the error between the demanded output shaft angle and measured output shaft angle drops below the angle resolution, the motor is powered off. This parameter models

the hysteresis usually incorporated into an RC servo controller to prevent chatter around a set point.

### **Angle measurement error — Model error in angle measurement**

0 deg (default)

This parameter allows you to model an angle measurement error, such as can happen due to a failing potentiometer angle sensor. For example, if you want to model the motor being powered against one of the hard stops, you can set a suitable angle measurement error to achieve this.

## **Mechanical**

### **Backdrive torque (unpowered) — Load torque required to backdrive unpowered motor**

0.5 kg\*cm (default)

Load torque required to backdrive the motor when it is unpowered. The block uses this value to determine the gear friction parameters.

### **Gear reduction ratio — Approximate reduction ratio from motor shaft to RC servo output shaft**

320 (default)

Reduction ratio from the DC motor shaft to the RC servo output shaft. This parameter affects only the impact of rotor inertia on equivalent inertia value at the output shaft. It has no impact on no-load speed. Therefore, the value does not have to be precise.

### **Rotor inertia — Inertia of DC motor and rotor gearing**

10 g\*cm<sup>2</sup> (default)

Inertia of the DC motor, plus inertia of the gearing reflected to the rotor (typically small if the gears are plastic).

### **End-stop angles — Mechanical angle range**

[-5 185] deg (default)

Mechanical end stops prevent rotation of the output shaft beyond the specified range. The range specified by the end stop angles must be larger than that specified by the **Rotational range** parameter.

**End-stop stiffness — Stiffness of mechanical end stops**

1e6 N\*m/rad (default)

Stiffness of mechanical end stops.

**End-stop damping — Damping of mechanical end stops**

0.01 N\*m/(rad/s) (default)

Damping of mechanical end stops.

## Faults

**Enable faults — Select Yes to enable faults modeling**

No (default) | Yes

Select **Yes** to enable faults modeling. The associated parameters in the **Faults** section become visible to let you select the reporting method and specify the trigger mechanism (temporal or behavioral). You can enable these trigger mechanisms separately or use them together.

**Reporting when a fault occurs — Choose whether to issue an assertion when a fault occurs**

None (default) | Warn | Error

Choose whether to issue an assertion when a fault occurs:

- **None** — The block does not issue an assertion.
- **Warn** — The block issues a warning.
- **Error** — Simulation stops with an error.

**Dependencies**

Enabled when the **Enable faults** parameter is set to **Yes**.

**Fault type — Select the type of fault**

Fail off (default) | Fail forward | Fail reverse | Failed winding

Select the type of fault:

- **Fail off** — No electrical torque.
- **Fail forward** — Rotates in a positive direction to hit the upper end stop.

- **Fail reverse** — Rotates in a negative direction to hit the lower end stop.
- **Failed winding** — Torque is applied only if the motor rotor lines up with one of the two remaining functioning windings.

**Dependencies**

Enabled when the **Enable faults** parameter is set to Yes.

**Enable temporal fault trigger — Select Yes to enable time-based fault triggering**

No (default) | Yes

Select Yes to enable time-based fault triggering. You can enable the temporal and behavioral trigger mechanisms separately or use them together.

**Dependencies**

Enabled when the **Enable faults** parameter is set to Yes.

**Simulation time for a fault event — Time before entering faulted state**

1 s (default)

Set the simulation time at which you want the block to enter the faulted state.

**Dependencies**

Enabled when the **Enable temporal fault trigger** parameter is set to Yes.

**Enable behavioral fault trigger — Select Yes to enable behavioral fault triggering**

No (default) | Yes

Select Yes to enable behavioral fault triggering. You can enable the temporal and behavioral trigger mechanisms separately or use them together.

**Dependencies**

Enabled when the **Enable faults** parameter is set to Yes.

**Maximum permissible current — Current threshold to fault transition**

1 A (default)

Specify the maximum permissible current value. If the current exceeds this value for longer than the **Time to fail when exceeding maximum permissible current** parameter value, then the block enters the faulted state.

**Dependencies**

Enabled when the **Enable behavioral fault trigger** parameter is set to Yes.

**Time to fail when exceeding current — Maximum length of time the current exceeds the threshold**

1 s (default)

Set the maximum length of time that the current can exceed the maximum permissible value without triggering the fault.

**Dependencies**

Enabled when the **Enable behavioral fault trigger** parameter is set to Yes.

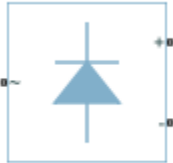
## **See Also**

DC Motor

**Introduced in R2017b**

# Rectifier

Convert three-phase AC voltage to DC voltage

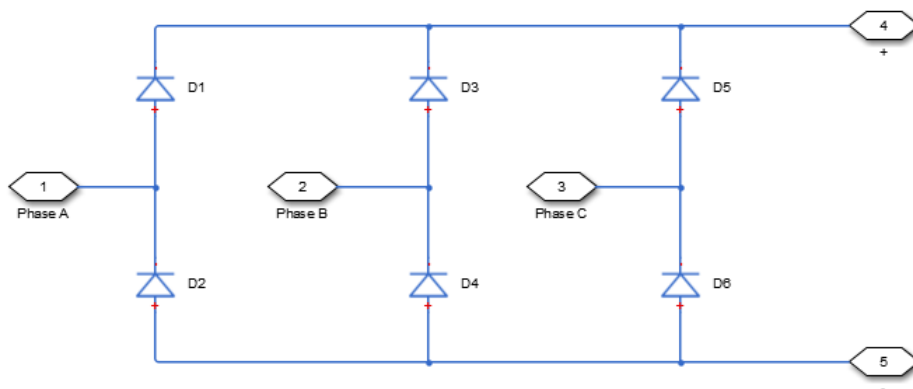


## Library

Simscape / Electrical / Power Systems / Semiconductors

## Description

The Rectifier block models a three-arm diode bridge circuit that converts a three-phase AC voltage to a DC voltage. The figure shows the equivalent circuit for the three-arm diode bridge.



Using the Charge Dynamics tab of the block dialog box, you can choose the type of diode that the three-arm bridge circuit uses. The table shows you how to set the **Model dynamics** parameter based on your goals.

<b>Goal</b>	<b>Value to Select</b>	<b>Block Behavior</b>
Prioritize simulation speed.	No dynamics	Each arm of the bridge circuit uses a copy of the Diode block. The block dialog box does not display additional parameters.
Precisely specify reverse-mode charge dynamics.	Model charge dynamics	Each arm of the bridge circuit uses a copy of the Commutation Diode block. The block dialog box shows parameters relating to the Commutation Diode block.

## Parameters

- “Main Tab” on page 1-1298
- “Charge Dynamics Tab” on page 1-1299

### Main Tab

#### Forward voltage

Minimum voltage required across the + and - ports of each diode for the gradient of the diode i-v characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**. The default forward voltage value is 0.8 V.

#### On resistance

Rate of change of voltage versus current above the forward voltage for each diode. The default value is 0.001 Ohm.

#### Off conductance

Conductance of each reverse-biased diode. The default value is  $1e-5$  1/Ohm.



## Charge Dynamics Tab

### Model dynamics

Diode charge dynamics. The default value is `No dynamics`.

The charge dynamics options you can select are:

- `No dynamics`
- `Model charge dynamics`

When you select `Model charge dynamics`, additional parameters appear.

### Additional Parameters for Model charge dynamics

#### Junction capacitance

Diode junction capacitance. The default value is `50 nF`.

#### Peak reverse current, `iRM`

Peak reverse current measured by an external test circuit. This value must be less than zero. The default value is `-235 A`.

#### Initial forward current when measuring `iRM`

Initial forward current when measuring peak reverse current. This value must be greater than zero. The default value is `300 A`.

#### Rate of change of current when measuring `iRM`

Rate of change of current when measuring peak reverse current. This value must be less than zero. The default value is `-50 A/μs`.

#### Reverse recovery time parameterization

Determines how you specify reverse recovery time in the block. The default value is `Specify reverse recovery time directly`.

If you select `Specify stretch factor` or `Specify reverse recovery charge`, you specify a value that the block uses to derive the reverse recovery time. For more information on these options, see “Alternatives to Specifying `trr` Directly” on page 1-339.

#### Reverse recovery time, `trr`

Interval between the time when the current initially goes to zero (when the diode turns off) and the time when the current falls to less than 10% of the peak reverse current. The default value is `15 μs`.

This parameter is visible only if you set **Reverse recovery time parameterization** to Specify reverse recovery time directly.

The value of the **Reverse recovery time, trr** parameter must be greater than the value of the **Peak reverse current, i<sub>RM</sub>** parameter divided by the value of the **Rate of change of current when measuring i<sub>RM</sub>** parameter.

#### **Reverse recovery time stretch factor**

Value that the block uses to calculate **Reverse recovery time, trr**. This value must be greater than 1. The default value is 3.

This parameter is visible only if you set **Reverse recovery time parameterization** to Specify stretch factor.

Specifying the stretch factor is an easier way to parameterize the reverse recovery time than specifying the reverse recovery charge. The larger the value of the stretch factor, the longer it takes for the reverse recovery current to dissipate.

#### **Reverse recovery charge, Q<sub>rr</sub>**

Value that the block uses to calculate **Reverse recovery time, trr**. Use this parameter if the data sheet for your diode device specifies a value for the reverse recovery charge instead of a value for the reverse recovery time.

The reverse recovery charge is the total charge that continues to dissipate when the

diode turns off. The value must be less than  $-\frac{i_{RM}^2}{2a}$ ,

where:

- $i_{RM}$  is the value specified for **Peak reverse current, i<sub>RM</sub>**.
- $a$  is the value specified for **Rate of change of current when measuring i<sub>RM</sub>**.

The default value is 1500 μAs.

The parameter is visible only if you set **Reverse recovery time parameterization** to Specify reverse recovery charge.

For more information on these parameters, see Commutation Diode.

## Ports

The block has the following ports:

~

Expandable three-phase port

+

Electrical conserving port associated with the positive terminal

-

Electrical conserving port associated with the negative terminal

## See Also

[Average-Value Inverter](#) | [Average-Value Rectifier](#) | [Converter](#)

## Topics

[“Expand and Collapse Three-Phase Ports on a Block”](#)

**Introduced in R2013b**

## Relay

Model switching and associated delay of relay



## Library

Passive Devices

## Description

The Relay block models a relay controlled by an external physical signal. In the steady state, the relay behaves as follows:

- When the physical signal input PS rises above the **Threshold** parameter, the relay is *energized* (meaning closed). The common port C connects to the normally open port S2.
- When the physical signal input PS falls below the **Threshold** parameter, the relay is *not energized* (meaning open). The common port C connects to the normally closed port S1.

The initial state of the block is set automatically based on the signal value at port PS at the start of simulation ( $t = 0$ ):

- If PS is greater than the threshold, the common port C connects to the S2 contact (relay is closed).
- If PS is less than or equal to the threshold, the common port C connects to the S1 contact (relay is open).

During switching, the relay behaves as follows:

- When the relay closes, the C to S1 connection breaks open after delay **Time-to-break C-S1 connection**. The C to S2 connection closes after delay **Time-to-make C-S2 connection**.
- When the relay opens, the C to S2 connection breaks open after delay **Time-to-break C-S2 connection**. The C to S1 connection closes after delay **Time-to-make C-S1 connection**.

You can specify break delays that are longer than the close delays to implement a make-before-break behavior.

## Basic Assumptions and Limitations

If the PS input changes during the switching process, the block behavior can be inaccurate. The switching delay occurs due to both mechanical inertia and the fact that modeling inertia as a delay requires approximation.

## Parameters

### Time-to-break C-S1 connection

Time it takes the connection between ports C and S1 to break apart when the relay is energized. The default value is 0 s.

### Time-to-make C-S1 connection

Time it takes the connection between ports C and S1 to close when the relay is not energized. The default value is 0 s.

### Time-to-break C-S2 connection

Time it takes the connection between ports C and S2 to break apart when the relay is not energized. The default value is 0 s.

### Time-to-make C-S2 connection

Time it takes the connection between ports C and S2 to close when the relay is not energized. The default value is 0 s.

### Connected resistance R

Resistance across closed relay contacts. The parameter value must be greater than zero. The default value is 0.01  $\Omega$ .

### **Open-circuit conductance G**

Conductance across open relay contacts. The parameter value must be greater than zero. The default value is  $1e-08$  1/Ω.

### **Threshold**

If the physical signal input rises above this value, the relay is energized. Conversely, if the physical signal input falls below this value, the relay is de-energized. The default value is 0.

## **Ports**

This block has the following ports:

PS

Physical signal that energizes and de-energizes the relay

C

Common electrical port

S1

Normally-closed electrical port

S2

Normally-open electrical port

## **See Also**

Switch

# Reluctance with Hysteresis

Nonlinear reluctance with magnetic hysteresis

**Library:** Simscape / Electrical / Electronics and Mechatronics /  
Passive Devices  
Simscape / Electrical / Power Systems / Passive  
Devices / Fundamental Components



## Description

The Reluctance with Hysteresis block models a nonlinear reluctance with magnetic hysteresis. Use this block to build custom inductances and transformers that exhibit magnetic hysteresis.

The length and area parameters in the **Geometry** section let you define the geometry for the part of the magnetic circuit that you are modeling. The block uses the geometry information to map the magnetic domain Through and Across variables to flux density and field strength, respectively:

$$B = \Phi / A_e$$

$$MMF = l_e \cdot H$$

where:

- $MMF$  is magnetomotive force (mmf) across the component.
- $\Phi$  is flux through the component.
- $B$  is flux density.
- $H$  is field strength.
- $A_e$  is the effective cross-sectional area of the section being modeled.
- $l_e$  is the effective length of the section being modeled.

The block then implements the relationship between  $B$  and  $H$  according to the Jiles-Atherton [1 on page 1-1311, 2 on page 1-1311] equations. The equation that relates  $B$  and  $H$  to the magnetization of the core is:

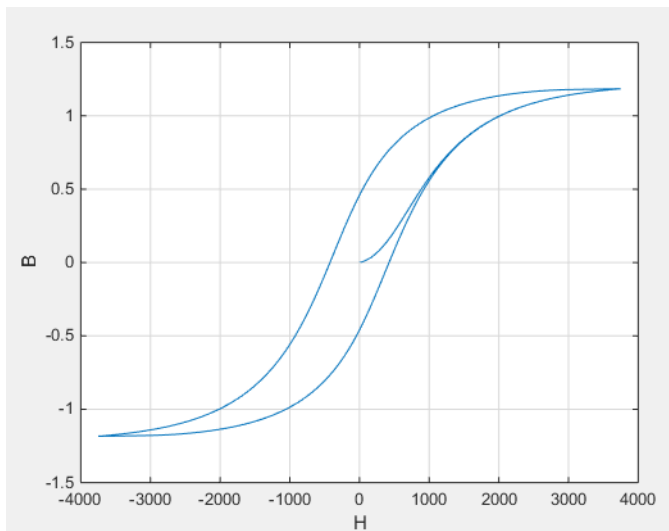
$$B = \mu_0 (H + M)$$

where:

- $\mu_0$  is the magnetic permeability constant.
- $M$  is magnetization of the core.

The magnetization acts to increase the magnetic flux density, and its value depends on both the current value and the history of the field strength  $H$ . The block uses the Jiles-Atherton equations to determine  $M$  at any given time.

The figure below shows a typical plot of the resulting relationship between  $B$  and  $H$ .



In this case, the magnetization starts as zero, and hence the plot starts at  $B = H = 0$ . As the field strength increases, the plot tends to the positive-going hysteresis curve; then on reversal the rate of change of  $H$ , it follows the negative-going hysteresis curve. The difference between positive-going and negative-going curves is due to the dependence of  $M$  on the trajectory history. Physically the behavior corresponds to magnetic dipoles in the core aligning as the field strength increases, but not then fully recovering to their original position as field strength decreases.

The starting point for the Jiles-Atherton equation is to split the magnetization effect into two parts, one that is purely a function of effective field strength ( $H_{\text{eff}}$ ) and the other an irreversible part that depends on past history:



$$M = cM_{an} + (1 - c)M_{irr}$$

The  $M_{an}$  term is called the anhysteretic magnetization because it exhibits no hysteresis. It is described by the following function of the current value of the effective field strength,  $H_{eff}$ :

$$M_{an} = M_s \left( \coth \left( \frac{H_{eff}}{\alpha} \right) - \frac{\alpha}{H_{eff}} \right)$$

This function defines a saturation curve with limiting values  $\pm M_s$  and point of saturation determined by the value of  $\alpha$ , the anhysteretic shape factor. It can be approximately thought of as describing the average of the two hysteretic curves. In the block interface, you provide values for  $dM_{an} / dH_{eff}$  when  $H_{eff} = 0$  and a point  $[H_1, B_1]$  on the anhysteretic B-H curve, and these are used to determine values for  $\alpha$  and  $M_s$ .

The parameter  $c$  is the coefficient for reversible magnetization, and dictates how much of the behavior is defined by  $M_{an}$  and how much by the irreversible term  $M_{irr}$ . The Jiles-Atherton model defines the irreversible term by a partial derivative with respect to field strength:

$$\frac{dM_{irr}}{dH} = \frac{M_{an} - M_{irr}}{K\delta - \alpha(M_{an} - M_{irr})}$$

$$\delta = \begin{cases} 1 & \text{if } H \geq 0 \\ -1 & \text{if } H < 0 \end{cases}$$

Comparison of this equation with a standard first order differential equation reveals that as increments in field strength,  $H$ , are made, the irreversible term  $M_{irr}$  attempts to track

the reversible term  $M_{an}$ , but with a variable tracking gain of  $1 / (K\delta - \alpha(M_{an} - M_{irr}))$ . The tracking error acts to create the hysteresis at the points where  $\delta$  changes sign. The main parameter that shapes the irreversible characteristic is  $K$ , which is called the bulk coupling coefficient. The parameter  $\alpha$  is called the inter-domain coupling factor, and is also used to define the effective field strength used when defining the anhysteretic curve:

$$H_{eff} = H + \alpha M$$

The value of  $\alpha$  affects the shape of the hysteresis curve, larger values acting to increase the B-axis intercepts. However, notice that for stability the term  $K\delta - \alpha(M_{an} - M_{irr})$  must be positive for  $\delta > 0$  and negative for  $\delta < 0$ . Therefore not all values of  $\alpha$  are permissible, a typical maximum value being of the order 1e-3.

## Procedure for Finding Approximate Values for Jiles-Atherton Equation Coefficients

You can determine representative parameters for the equation coefficients by using the following procedure:

- 1 Provide a value for the **Anhyseretic B-H gradient when H is zero** parameter ( $dM_{an} / dH_{eff}$  when  $H_{eff} = 0$ ) plus a data point  $[H_1, B_1]$  on the anhyseretic B-H curve. From these values, the block initialization determines values for  $\alpha$  and  $M_s$ .
- 2 Set the **Coefficient for reversible magnetization, c** parameter to achieve correct initial B-H gradient when starting a simulation from  $[H B] = [0 0]$ . The value of  $c$  is approximately the ratio of this initial gradient to the **Anhyseretic B-H gradient when H is zero**. The value of  $c$  must be greater than 0 and less than 1.
- 3 Set the **Bulk coupling coefficient, K** parameter to the approximate magnitude of  $H$  when  $B = 0$  on the positive-going hysteresis curve.
- 4 Start with  $\alpha$  very small, and gradually increase to tune the value of  $B$  when crossing  $H = 0$  line. A typical value is in the range of 1e-4 to 1e-3. Values that are too large will cause the gradient of the B-H curve to tend to infinity, which is nonphysical and generates a run-time assertion error.

Sometimes you need to iterate on these four steps to get a good match against a predefined B-H curve.

## Variables

Use the **Variables** section of the block interface to set the priority and initial target values for the block variables prior to simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

## Ports

### Conserving

#### **N – North terminal**

magnetic

Magnetic conserving port associated with the block North terminal.

#### **S – South terminal**

magnetic

Magnetic conserving port associated with the block South terminal.

## Parameters

### Geometry

#### **Effective length – Effective length of the section being modeled**

0.032 m (default)

Effective length of the section being modeled, that is, the average distance of the magnetic path.

#### **Effective cross-sectional area – Effective cross-sectional area of the section being modeled**

1.6e-5 m<sup>2</sup> (default)

Effective cross-sectional area of the section being modeled, that is, the average area of the magnetic path.

#### **Averaging period for power logging – Excitation period used for averaging**

0 s (default)

Averaging period for the hysteresis losses calculation. These losses are proportional to the area enclosed by the B-H trajectory. If the block is excited at a known, fixed frequency, you can set this value to the corresponding excitation period to calculate the hysteresis loss. In this case, the block logs the hysteresis loss once per AC cycle to the variable

`power_dissipated`. If you are using a fixed-step solver, this value must be an integer multiple of the simulation step size.

If the block is not excited at a known, fixed frequency, set this parameter to 0. In this case, the block sets `power_dissipated` to zero, and you can calculate the actual hysteresis loss by post-processing the logged variable `power_instantaneous`.

### **Dependencies**

This parameter is visible only when you select **Magnetic flux density versus magnetic field strength characteristic with hysteresis** for the **Parameterized by** parameter.

## **B-H Curve**

### **Anhyysteretic B-H gradient when H is zero — Gradient of the anhyysteretic B-H curve around zero field strength**

0.005 m\*T/A (default)

The gradient of the anhyysteretic (no hysteresis) B-H curve around zero field strength. Set it to the average gradient of the positive-going and negative-going hysteresis curves.

### **Flux density point on anhyysteretic B-H curve — Flux density of the point for field strength measurement**

1.49 T (default)

Specify a point on the anhyysteretic curve by providing its flux density value. Picking a point at high field strength where the positive-going and negative-going hysteresis curves align is the most accurate option.

### **Corresponding field strength — Field strength at measurement point**

1000 A/m (default)

The corresponding field strength for the point that you define by the **Flux density point on anhyysteretic B-H curve** parameter.

### **Coefficient for reversible magnetization, c — Proportion of magnetization that is reversible**

0.1 (default)

The proportion of the magnetization that is reversible. The value must be greater than zero and less than one.

**Bulk coupling coefficient, K – Jiles-Atherton equations parameter**

200 A/m (default)

The Jiles-Atherton parameter that primarily controls the field strength magnitude at which the B-H curve crosses the zero flux density line.

**Inter-domain coupling factor, alpha – Jiles-Atherton equations parameter**

0.0001 (default)

The Jiles-Atherton parameter that primarily affects the points at which the B-H curves intersect the zero field strength line. Typical values are in the range of 1e-4 to 1e-3.

**References**

- [1] Jiles, D. C. and D. L. Atherton. "Theory of ferromagnetic hysteresis." *Journal of Magnetism and Magnetic Materials*. Vol. 61, 1986, pp. 48-60.
- [2] Jiles, D. C. and D. L. Atherton. "Ferromagnetic hysteresis." *IEEE Transactions on Magnetism*. Vol. 19, No. 5, 1983, pp. 2183-2184.

**See Also**

Nonlinear Inductor | Nonlinear Transformer

**Introduced in R2017b**

# Resistor

Resistor model including tolerance, operational limits, fault behavior, and noise

**Library:** Simscape / Electrical / Electronics and Mechatronics /  
Passive Devices



## Description

The Resistor block represents a linear resistor, while letting you model the following effects:

- “Tolerances” on page 1-1313
- “Operating Limits” on page 1-1313
- “Faults” on page 1-1314
- “Thermal Noise” on page 1-1314
- “Thermal Port” on page 1-1315

You can turn these modeling options on and off independently of each other. When all the additional options are turned off, the component behavior is identical to the Simscape Foundation library Resistor block.

In its simplest form, the Resistor block models a linear resistor, described with the following equation:

$$i = v/R$$

where:

- $i$  is current.
- $v$  is voltage.
- $R$  is resistance.

If you set the **Noise mode** parameter to Enabled, then the defining equations are augmented by a discrete variable  $i_N$  to represent thermal noise, as described in “Thermal Noise” on page 1-1314.

## Tolerances

You can apply tolerances to the nominal value you provide for the **Resistance** parameter. Datasheets typically provide a tolerance percentage for a given resistor type. The table shows how the block applies tolerances and calculates resistance based on the selected **Tolerance application** option.

Option	Resistance Value
None – use nominal value	$R$
Random tolerance	Uniform distribution: $R \cdot (1 - tol + 2 \cdot tol \cdot rand)$  Gaussian distribution: $R \cdot (1 + tol \cdot randn / nSigma)$
Apply maximum tolerance value	$R \cdot (1 + tol)$
Apply minimum tolerance value	$R \cdot (1 - tol)$

In the table,

- $R$  is the **Resistance** parameter value, nominal resistance.
- $tol$  is fractional tolerance, calculated from the percent-based **Tolerance (%)** parameter.
- $nSigma$  is the value you provide for the **Number of standard deviations for quoted tolerance** parameter.
- `rand` and `randn` are standard MATLAB functions for generating uniform and normal distribution random numbers.

## Operating Limits

You can specify operating limits in terms of power and maximum working voltage. For the thermal variant of the block (see “Thermal Port” on page 1-1315), you can also specify operating limits in terms of temperature.

When an operating limit is exceeded, the block can either generate a warning or stop the simulation with an error. For more information, see the “Operating Limits” on page 1-1318 parameters section.

## Faults

The Resistor block allows you to model an electrical fault as an instantaneous change in resistance. The block can trigger fault events:

- At a specific time
- When a current limit is exceeded for longer than a specific time interval

You can enable or disable these trigger mechanisms separately, or use them together if more than one trigger mechanism is required in a simulation. When more than one mechanism is enabled, the first mechanism to trigger the fault takes precedence. In other words, the component fails no more than once per simulation.

When the resistor fails, its resistance is changed to the value you specify for the **Faulted zero-voltage resistance** parameter. You can also choose whether to issue an assertion when a fault occurs, by using the **Reporting when a fault occurs** parameter. The assertion can take the form of a warning or an error. By default, the block does not issue an assertion.

## Thermal Noise

The Resistor block can generate thermal noise current. If you set the **Noise mode** parameter to **Enabled**, then the defining equations are augmented by a discrete variable  $i_N$  to represent thermal noise:

$$i = v/R + i_N$$

If the sampling time is  $h$ , then the thermal noise is given by:

$$i_N = \sqrt{2kT/R} \frac{N(0,1)}{\sqrt{h}}$$

where:

- $k$  is the Boltzmann constant, 1.3806504e-23 J/K.
- $T$  is the temperature.
- $R$  is the resistance.
- $N$  is a Gaussian random number with zero mean and standard deviation of one.



- $2kT/R$  is the double-sided thermal noise power distribution (the single-sided equivalent is  $4kT/R$ ).

The block generates Gaussian noise by using the Random Number source in the Simscape Foundation library. You can control the random number seed by setting the **Repeatability** parameter:

- **Not repeatable** — Every time you simulate your model, the block resets the random seed using the MATLAB random number generator:

```
seed = randi(2^32-1);
```

- **Repeatable** — The block automatically generates a seed value and stores it inside the block, to always start the simulation with the same random number. This auto-generated seed value is set when you add a Resistor block from the block library to the model. When you make a new copy of the Resistor block from an existing one in a model, a new seed value is generated. The block sets the value using the MATLAB random number generator command shown above.
- **Specify seed** — If you select this option, the additional **Seed** parameter lets you directly specify the random number seed value.

## Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal port H on the block icon, and adds the **Thermal** tab and the **Variables** tab to the block dialog box.

Use the **Thermal** tab to specify how the resistance value changes with temperature and to set the thermal mass. Use the **Variables** tab to set the initial temperature target.

With the thermal port exposed, the generated noise uses the temperature at the thermal port when determining the instantaneous noise value. Exposing the thermal port also extends the options on the **Operating Limits** tab as follows:

- The **Power rating** parameter becomes temperature dependent. You define a temperature up to which the full power rating is available, plus a higher temperature for which the power rating is reduced to zero. It is assumed that the power rating decreases linearly with temperature between these two values.
- An additional parameter, **Operating temperature range, [Tmin Tmax]**, lets you define the valid temperature range for block operation.

## Variables

Use the **Variables** section of the block interface to set the priority and initial target values for the block variables prior to simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

This section appears only for the blocks with exposed thermal port. The **Temperature** variable lets you specify a high-priority target for the temperature at the start of simulation.

## Basic Assumptions and Limitations

Simulating with noise enabled slows down simulation. Choose the sample time ( $h$ ) so that noise is generated only at frequencies of interest, and not higher.

## Ports

### Conserving

#### + — Positive terminal

electrical

Electrical conserving port associated with the resistor positive terminal.

#### - — Negative terminal

electrical

Electrical conserving port associated with the resistor negative terminal.

#### H — Resistor thermal mass

thermal

Thermal conserving port that represents the resistor thermal mass.

### Dependencies

Enabled for the thermal variant of the block. For more information, see “Thermal Port” on page 1-1315.

## Parameters

### Main

#### Resistance — Nominal resistance value

1  $\Omega$  (default)

The nominal resistance value. Resistance value must be greater than zero.

#### Tolerance (%) — Resistor tolerance, in percent

5 (default)

The resistor tolerance as defined on the manufacturer datasheet.

#### Tolerance application — Select how to apply tolerance during simulation

None — use nominal value (default) | Random tolerance | Apply maximum tolerance value | Apply minimum tolerance value

Select how to apply tolerance during simulation:

- **None — use nominal value** — The block does not apply tolerance, uses the nominal resistance value. This is the default.
- **Random tolerance** — The block applies random offset to the resistance value, within the tolerance value limit. You can choose Uniform or Gaussian distribution for calculating the random number by using the **Tolerance distribution** parameter.
- **Apply maximum tolerance value** — The resistance is increased by the specified tolerance percent value.
- **Apply minimum tolerance value** — The resistance is decreased by the specified tolerance percent value.

#### Tolerance distribution — Select the distribution type

Uniform (default) | Gaussian

Select the distribution type for random tolerance:

- **Uniform** — Uniform distribution
- **Gaussian** — Gaussian distribution

#### Dependencies

Enabled when the **Tolerance application** parameter is set to **Random tolerance**.

**Number of standard deviations for quoted tolerance — Used for calculating the Gaussian random number**

4 (default)

Number of standard deviations for calculating the Gaussian random number.

**Dependencies**

Enabled when the **Tolerance distribution** parameter is set to **Gaussian**.

## Operating Limits

**Enable operating limits — Select Yes to enable reporting when the operational limits are exceeded**

No (default) | Yes

Select **Yes** to enable reporting when the operational limits are exceeded. The associated parameters in the **Operating Limits** section become visible to let you select the reporting method and specify the operating limits in terms of power and maximum working voltage. Parameters that specify operating limits in terms of temperature are visible only for blocks with an exposed thermal port (see “Thermal Port” on page 1-1315). The default value is **No**.

**Reporting when operating limits exceeded — Select the reporting method**

Warn (default) | Error

Select what happens when an operating limit is exceeded:

- **Warn** — The block issues a warning.
- **Error** — Simulation stops with an error.

**Dependencies**

Enabled when the **Enable operating limits** parameter is set to **Yes**.

**Maximum working voltage — Maximum voltage allowed for normal block operation**

100 V (default)

Maximum voltage magnitude allowed for normal block operation.

**Dependencies**

Enabled when the **Enable operating limits** parameter is set to Yes.

**Power rating — Maximum power allowed for normal block operation**

1 W (default)

Maximum power allowed for normal block operation.

If you expose the thermal port of the block, this parameter becomes temperature dependent. The value you specify for the **Power rating** parameter applies up to the temperature specified by the **Temperature below which full power rating is available** parameter value. Then the power rating decreases linearly with temperature, until it becomes 0 at temperature specified by the **Temperature above which power rating is reduced to zero** parameter value.

**Dependencies**

Enabled when the **Enable operating limits** parameter is set to Yes.

**Temperature below which full power rating is available — Maximum temperature where full power rating still applies**

70 °C (default)

Maximum temperature where full power rating, specified by the **Power rating** parameter value, still applies.

**Dependencies**

Enabled for the thermal variant of the block. For more information, see “Thermal Port” on page 1-1315.

**Temperature above which power rating is reduced to zero — Temperature where power rating becomes 0**

155 °C (default)

Temperature where power rating becomes 0. Above this temperature, the simulation always issues an assertion regardless of dissipated power. This parameter value must be higher than **Temperature below which full power rating is available**.

**Dependencies**

Enabled for the thermal variant of the block. For more information, see “Thermal Port” on page 1-1315.

## **Operating temperature range, [Tmin Tmax] — Minimum and maximum temperature values allowed for normal block operation**

[-50 150] °C (default)

A row vector of length 2 specifying minimum and maximum temperature values allowed for normal block operation. The first element is the lowest allowable operating temperature, and the second element is the largest allowable operating temperature.

### **Dependencies**

Enabled for the thermal variant of the block. For more information, see “Thermal Port” on page 1-1315.

## **Faults**

### **Enable faults — Select Yes to enable faults modeling**

No (default) | Yes

Select Yes to enable faults modeling. The associated parameters in the **Faults** section become visible to let you select the reporting method and specify the trigger mechanism (temporal or behavioral). You can enable these trigger mechanisms separately or use them together.

### **Reporting when a fault occurs — Choose whether to issue an assertion when a fault occurs**

None (default) | Warn | Error

Choose whether to issue an assertion when a fault occurs:

- None — The block does not issue an assertion.
- Warn — The block issues a warning.
- Error — Simulation stops with an error.

### **Dependencies**

Enabled when the **Enable faults** parameter is set to Yes.

### **Faulted resistance — Resistance when block is in faulted state**

inf  $\Omega$  (default)

Resistance between the + and - ports when the block is in the faulted state.

**Dependencies**

Enabled when the **Enable faults** parameter is set to Yes.

**Enable temporal fault trigger – Select Yes to enable time-based fault triggering**

No (default) | Yes

Select Yes to enable time-based fault triggering. You can enable the temporal and behavioral trigger mechanisms separately or use them together.

**Dependencies**

Enabled when the **Enable faults** parameter is set to Yes.

**Simulation time for fault event – Time before entering faulted state**

1 s (default)

Set the simulation time at which you want the block to enter the faulted state.

**Dependencies**

Enabled when the **Enable temporal fault trigger** parameter is set to Yes.

**Enable behavioral fault trigger – Select Yes to enable behavioral fault triggering**

No (default) | Yes

Select Yes to enable behavioral fault triggering. You can enable the temporal and behavioral trigger mechanisms separately or use them together.

**Dependencies**

Enabled when the **Enable faults** parameter is set to Yes.

**Maximum permissible current – Current threshold to fault transition**

1 A (default)

Specify the maximum permissible current value. If the current exceeds this value for longer than the **Time to fail when exceeding maximum permissible current** parameter value, then the block enters the faulted state.

**Dependencies**

Enabled when the **Enable behavioral fault trigger** parameter is set to Yes.

## **Time to fail when exceeding maximum permissible current — Maximum length of time the current exceeds the threshold**

1 s (default)

Set the maximum length of time that the current can exceed the maximum permissible value without triggering the fault.

### **Dependencies**

Enabled when the **Enable behavioral fault trigger** parameter is set to Yes.

## **Noise**

### **Noise mode — Select whether to model thermal noise current**

Disabled (default) | Enabled

Select whether to model thermal noise current:

- **Disabled** — No noise is produced by the resistor.
- **Enabled** — Resistor generates thermal noise current, and the associated parameters become visible in the **Noise** section.

### **Sample time — Rate at which the noise source is sampled**

1e-3 s (default)

Defines the rate at which the noise source is sampled. Choose it to reflect the frequencies of interest in your model. Making the sample time too small will unnecessarily slow down your simulation.

### **Dependencies**

Enabled when the **Noise mode** parameter is set to Enabled.

### **Repeatability — Select the noise control option**

Not repeatable (default) | Repeatable | Specify seed

Select the noise control option:

- **Not repeatable** — The random sequence used for noise generation is not repeatable.
- **Repeatable** — The random sequence used for noise generation is repeatable, with a system-generated seed.



- **Specify seed** — The random sequence used for noise generation is repeatable, and you control the seed by using the **Seed** parameter.

**Dependencies**

Enabled when the **Noise mode** parameter is set to Enabled.

**Auto-generated seed used for repeatable option — Auto-generated random number seed**

random real number

Random number seed stored inside the block to make the random sequence repeatable. The parameter value is automatically generated using the MATLAB random number generator command. You can modify this parameter value, but it gets overwritten by a new random value if you copy the block to another block in the model. Therefore, if you want to control the seed of the random sequence, use the **Specify seed** option for the **Repeatability** parameter and specify the desired seed value using the **Seed** parameter.

**Dependencies**

Enabled when the **Repeatability** parameter is set to Repeatable.

**Seed — Random number seed**

0 (default)

Seed used by the noise random number generator.

**Dependencies**

Enabled when the **Repeatability** parameter is set to **Specify seed**.

**Device simulation temperature — Temperature of resistor at the start of the simulation**

25 °C (default)

The temperature of the resistor at the start of the simulation.

**Dependencies**

Enabled when the **Noise mode** parameter is set to Enabled.

For blocks with an exposed thermal port, this parameter is disabled. Instead, use the **Variables** tab to set the initial temperature target. For more information, see “Variables” on page 1-1316.

## Thermal

This tab appears only for blocks with exposed thermal port. For more information, see “Thermal Port” on page 1-1315.

### **Resistance temperature coefficient — Specifies how the resistance value changes with temperature**

0.00393 1/K (default)

The coefficient  $\alpha$  in the equation that describes resistance as a function of temperature,  $R_T = R(1 + \alpha(T - T_0))$ . The default value is for copper.

### **Measurement temperature — Temperature corresponding to nominal resistance**

25 °C (default)

The temperature  $T_0$ , for which the nominal resistance  $R$  is specified.

### **Thermal mass — Thermal mass associated with port H**

100 J/K (default)

Thermal mass associated with the thermal port H. It represents the energy required to raise the temperature of the thermal port by one degree.

## See Also

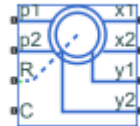
Fault | Resistor

**Introduced in R2009a**

# Resolver

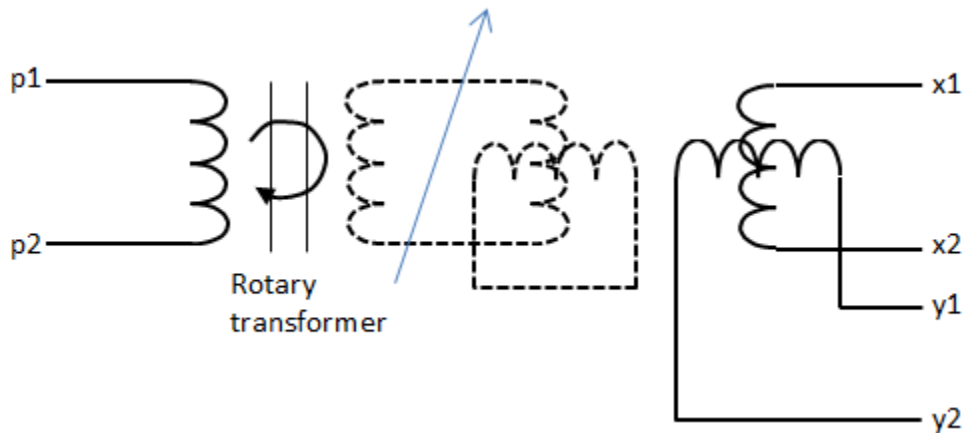
Model rotary transformer that measures motor rotation angle

**Library:** Simscape / Electrical / Electronics and Mechatronics / Sensors  
Simscape / Electrical / Power Systems / Sensors



## Description

The Resolver block models a generic resolver, which measures the electrical phase angle of a signal through electromagnetic coupling. The resolver consists of a rotary transformer that couples an AC voltage applied to the primary winding to two secondary windings. These secondary windings are physically oriented at 90 degrees to each other. As the rotor angle changes, the relative coupling between the primary and the two secondary windings varies. In the Resolver block model, the first secondary winding is oriented such that peak coupling occurs when the rotor is at zero degrees, and therefore the second secondary winding has minimum coupling when the rotor is at zero degrees.



Without loss of generality, it is assumed that the transformer between primary and rotor circuit is ideal with a ratio of 1:1. This results in the rotor current and voltage being equivalent to the primary current and voltage.

You have two options for defining the block equations:

- Omit the dynamics by neglecting the transformer inductive terms. This model is only valid if the sensor is driven by a sine wave because any DC component on the primary side will pass to the output side.
- Include the inductive terms, thereby capturing voltage amplitude loss and phase differences. This model is valid for any input waveform. Within this option, you can either specify the inductances and the peak coupling coefficient directly, or specify the transformation ratio and measured impedances, in which case the block uses these values to determine the inductive terms.

## Equations when Omitting Dynamics

The equations are based on the superposition of two ideal transformers, both with coupling coefficients that depend on rotor angle. The two ideal transformers have a common primary winding. See the Simscape Ideal Transformer block reference page for more information on modeling ideal transformers. The equations are:

$$K_x = R \cos(\theta) N$$

$$K_y = R \sin(\theta) N$$

$$v_x = K_x v_p$$

$$v_y = K_y v_p$$

$$i_p = -K_x i_x - K_y i_y$$

where:

- $v_p$  and  $i_p$  are the rotor (or equivalently primary) voltage and current, respectively.
- $v_x$  and  $i_x$  are the first secondary voltage and current, respectively.
- $v_y$  and  $i_y$  are the second secondary voltage and current, respectively.
- $K_x$  is the coupling coefficient for the first secondary winding.
- $K_y$  is the coupling coefficient for the second secondary winding.

- $R$  is the transformation ratio.
- $N$  is the number of pole pairs.
- $\theta$  is the rotor angle.

## Equations when Including Dynamics

The equations are based on the superposition of two mutual inductors, both with coupling coefficients that depend on rotor angle. The two mutual inductors have a common primary winding. See the Simscape Mutual Inductor block reference page for more information on modeling mutual inductors. The equations are:

$$v_p = R_p i_p + L_p \frac{di_p}{dt} + \sqrt{L_p L_s} k \left( \cos(N\theta) \frac{di_x}{dt} + \sin(N\theta) \frac{di_y}{dt} \right)$$

$$v_x = R_s i_x + L_s \frac{di_x}{dt} + \sqrt{L_p L_s} k \cos(N\theta) \frac{di_p}{dt}$$

$$v_y = R_s i_y + L_s \frac{di_y}{dt} + \sqrt{L_p L_s} k \sin(N\theta) \frac{di_p}{dt}$$

where:

- $v_p$  and  $i_p$  are the rotor (or equivalently primary) voltage and current, respectively.
- $v_x$  and  $i_x$  are the first secondary voltage and current, respectively.
- $v_y$  and  $i_y$  are the second secondary voltage and current, respectively.
- $R_p$  is the rotor (or primary) resistance.
- $L_p$  is the rotor (or primary) inductance.
- $R_s$  is the stator (or secondary) resistance.
- $L_s$  is the stator (or secondary) inductance.
- $N$  is the number of pole pairs.
- $k$  is the coefficient of coupling.
- $\theta$  is the rotor angle.

It is assumed that coupling between the two secondary windings is zero.

Datasheets typically do not quote the coefficient of coupling and inductance parameters, but instead give the transformation ratio  $R$  and measured impedances. If you select **Specify transformation ratio and measured impedances** for the **Parameterization** parameter, then the values you provide are used to determine values for the equation coefficients, as defined above.

## Variables

Use the **Variables** section of the block interface to set the priority and initial target values for the block variables prior to simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

## Assumptions and Limitations

The model is based on the following assumptions:

- The resolver draws no torque between the mechanical rotational ports  $R$  and  $C$ .
- The transformer between primary and rotor circuit is ideal with a ratio of 1:1.
- The coupling between the two secondary windings is zero.

## Ports

### Conserving

#### **p1 — Primary winding positive terminal**

electrical

Electrical conserving port associated with the positive terminal of the primary winding.

#### **p2 — Primary winding negative terminal**

electrical

Electrical conserving port associated with the negative terminal of the primary winding.

#### **R — Resolver rotor**

mechanical rotational

Mechanical rotational conserving port connected to the rotor.

### **C — Resolver case**

mechanical rotational

Mechanical rotational conserving port connected to the resolver case.

### **x1 — Secondary winding x positive terminal**

electrical

Electrical conserving port associated with the positive terminal of secondary winding x.

### **x2 — Secondary winding x negative terminal**

electrical

Electrical conserving port associated with the negative terminal of secondary winding x.

### **y1 — Secondary winding y positive terminal**

electrical

Electrical conserving port associated with the positive terminal of secondary winding y.

### **y2 — Secondary winding y negative terminal**

electrical

Electrical conserving port associated with the negative terminal of secondary winding y.

## **Parameters**

### **Parameterization — Resolver parameterization**

Specify transformation ratio and omit dynamics (default) | Specify transformation ratio and measured impedances | Specify equation parameters directly

Select one of the following methods for block parameterization:

- **Specify transformation ratio and omit dynamics** — Provide values for transformation ratio, number of pole pairs, and initial rotor angle only. This model neglects the transformer inductive terms, and is only valid if the sensor is driven by a sine wave. The equations are based on the superposition of two ideal transformers,

both with coupling coefficients that depend on rotor angle. For more information, see “Equations when Omitting Dynamics” on page 1-1326.

- **Specify transformation ratio and measured impedances** — Provide additional values to determine the transformer inductive terms, to model the voltage amplitude loss and phase differences. This model is valid for any input waveform. The equations are based on the superposition of two mutual inductors, both with coupling coefficients that depend on rotor angle. For more information, see “Equations when Including Dynamics” on page 1-1327.
- **Specify equation parameters directly** — Model the dynamics, but provide values for rotor and stator inductances and the peak coefficient of coupling, instead of transformation ratio and measured impedances. For more information, see “Equations when Including Dynamics” on page 1-1327. This model is valid for any input waveform.

### **Transformation ratio — Peak output to input voltage ratio**

0.5 (default) | positive number

Ratio between the peak output voltage and the peak input voltage assuming negligible secondary voltage drop due to resistance and inductance.

#### **Dependencies**

To enable this parameter, set the **Parameterization** parameter to **Specify transformation ratio and omit dynamics** or **Specify transformation ratio and measured impedances**. If you select **Specify transformation ratio and measured impedances** for the **Parameterization** parameter, then the transformation ratio takes the voltage drop due to primary winding resistance into account.

### **Rotor resistance — Primary resistance**

70 Ohm (default) | positive number

Rotor ohmic resistance. This resistance is also referred to as the primary resistance.

#### **Dependencies**

To enable this parameter, set the **Parameterization** parameter to **Specify transformation ratio and measured impedances** or **Specify equation parameters directly**.

### **Stator resistance — Secondary resistance**

180 Ohm (default) | positive number



Stator ohmic resistance. This resistance is also referred to as the secondary resistance. It is assumed that both secondaries have the same resistance.

**Dependencies**

To enable this parameter, set the **Parameterization** parameter to Specify transformation ratio and measured impedances or Specify equation parameters directly.

**Rotor reactance – Primary reactance**

100 Ohm (default) | positive number

Rotor reactance when the secondary windings are open-circuit. This reactance is also referred to as the primary reactance.

**Dependencies**

To enable this parameter, set the **Parameterization** parameter to Specify transformation ratio and measured impedances.

**Stator reactance – Secondary reactance**

300 Ohm (default) | positive number

Stator reactance when the primary winding is open-circuit. This reactance is also referred to as the secondary reactance.

**Dependencies**

To enable this parameter, set the **Parameterization** parameter to Specify transformation ratio and measured impedances.

**Frequency at which reactances and transformation ratio are specified – Sinusoidal source frequency**

10 kHz (default) | positive number

Frequency of the sinusoidal source used when measuring the reactances.

**Dependencies**

To enable this parameter, set the **Parameterization** parameter to Specify transformation ratio and measured impedances.

**Rotor inductance – Primary reactance**

0.0016 H (default) | positive number

Rotor or primary inductance,  $L_p$ .

**Dependencies**

To enable this parameter, set the **Parameterization** parameter to Specify equation parameters directly.

**Stator inductance — Secondary reactance**

0.0048 H (default) | positive number

Stator or secondary inductance,  $L_s$ .

**Dependencies**

To enable this parameter, set the **Parameterization** parameter to Specify equation parameters directly.

**Peak coefficient of coupling — Maximum coupling coefficient**

0.35 (default) | number between zero and one, exclusive

Peak coefficient of coupling between the primary and secondary windings.

**Dependencies**

To enable this parameter, set the **Parameterization** parameter to Specify equation parameters directly.

**Number of pole pairs — Rotor pole pairs**

1 (default) | positive number

Number of pole pairs on the rotor.

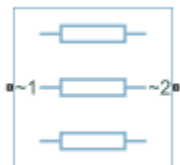
**See Also**

Incremental Shaft Encoder

**Introduced in R2017b**

## RLC

Three-phase line impedance



## Library

Simscape / Electrical / Power Systems / Passive Devices

## Description

The RLC block models a three-phase impedance with two three-phase connections. Each of the three identical impedance components can include any combination of a resistor (R), capacitor (C), and inductor (L), connected in series or in parallel.

Define the values for the R, L, and C components by specifying the appropriate block parameters. Do not set the parameter values to zero or infinity to remove terms; instead, select the correct option for the **Component structure** parameter.

For certain combinations of R, L, and C, for some circuit topologies, specify parasitic resistance or conductance values that help the simulation to converge numerically. These parasitic terms ensure that an inductor has a small parallel resistive path and that a capacitor has a small series resistance.

## Parameters

- “Main Tab” on page 1-1334
- “Parasitics Tab” on page 1-1334

- “Initial Conditions Tab” on page 1-1334

## **Main Tab**

### **Component structure**

Select the desired combination of a resistor (R), capacitor (C), and inductor (L), connected in series or in parallel. The default is R, resistor.

### **Resistance**

Resistance of each of the line impedances. This parameter is visible only when you select a component structure that includes a resistor. The default value is 1 Ohm.

### **Inductance**

Inductance of each of the line impedances. This parameter is visible only when you select a component structure that includes an inductor. The default value is 0.001 H.

### **Capacitance**

Capacitance in each of the line impedances. This parameter is visible only when you select a component structure that includes a capacitor. The default value is 1e-6 F.

## **Parasitics Tab**

### **Parasitic series resistance**

Represents small parasitic effects. The parameter value corresponds to the series resistance value added to all instances of capacitors in the load. The default value is 1e-6 Ohm.

### **Parasitic parallel conductance**

Represents small parasitic effects. The parameter value corresponds to the parallel conductance value added across all instances of inductors in the load. The default value is 1e-6 1/Ohm.

## **Initial Conditions Tab**

### **Initial inductor current [ Ia Ib Ic ]**

Initial current in the a, b, and c phase inductors, respectively. This parameter is visible only when you select a component structure that includes an inductor. The default value is [0 0 0] A.

**Initial capacitor voltage [ Va Vb Vc ]**

Initial voltage across the a, b, and c phase capacitors, respectively. This parameter is visible only when you select a component structure that includes a capacitor. The default value is [0 0 0] V.

**Block Parameterization**

The following table lists the block parameters for each of the configurations, based on the selected **Component structure** option.

<b>Component Structure</b>	<b>Main Tab</b>	<b>Parasitics Tab</b>	<b>Initial Conditions Tab</b>
R	Resistance	None	None
L	Inductance	Parasitic parallel conductance	Initial inductor current [ Ia Ib Ic ]
C	Capacitance	Parasitic series resistance	Initial capacitor voltage [ Va Vb Vc ]
Series RL	Resistance Inductance	Parasitic parallel conductance	Initial inductor current [ Ia Ib Ic ]
Series RC	Resistance Capacitance	None	Initial capacitor voltage [ Va Vb Vc ]
Series LC	Inductance Capacitance	Parasitic parallel conductance	Initial inductor current [ Ia Ib Ic ] Initial capacitor voltage [ Va Vb Vc ]
Series RLC	Resistance Inductance Capacitance	Parasitic parallel conductance	Initial inductor current [ Ia Ib Ic ] Initial capacitor voltage [ Va Vb Vc ]
Parallel RL	Resistance Inductance	None	Initial inductor current [ Ia Ib Ic ]

<b>Component Structure</b>	<b>Main Tab</b>	<b>Parasitics Tab</b>	<b>Initial Conditions Tab</b>
Parallel RC	Resistance Capacitance	Parasitic series resistance	Initial capacitor voltage [ Va Vb Vc ]
Parallel LC	Inductance Capacitance	Parasitic series resistance	Initial inductor current [ Ia Ib Ic ]  Initial capacitor voltage [ Va Vb Vc ]
Parallel RLC	Resistance Inductance Capacitance	Parasitic series resistance	Initial inductor current [ Ia Ib Ic ]  Initial capacitor voltage [ Va Vb Vc ]

## Ports

The block has two expandable three-phase ports, ~1 and ~2, representing the two terminals of the three-phase line.

## See Also

Delta-Connected Load | Wye-Connected Load

## Topics

“Three-Phase Asynchronous Machine Starting”

“Expand and Collapse Three-Phase Ports on a Block”

**Introduced in R2013b**

# RMS Measurement

Measure root-mean-square (RMS) properties of a signal

**Library:** Simscape / Electrical / Power Systems / Control / Measurements



## Description

The RMS Measurement block measures root-mean-square (RMS) properties of the input signal. You can use it to measure one of these properties:

- The total RMS of the input signal
- The RMS of the individual harmonics of the input signal that you specify.

Use the total RMS configuration with appropriate sensors to perform RMS voltage, current, or power analyses in your system.

You can use the harmonics configuration to perform total harmonic distortion analyses on systems with nonlinear loads such as:

- Converters
- Motor drives
- Inverters

## Equations

The total RMS value is calculated from the input signal  $x_{RMS}$  as:

$$x_{RMS}(t) = \sqrt{\frac{1}{T} \int_{t-T}^t x(t)^2 dt},$$

where:

- $T$  is the period of the input signal, or equivalently the inverse of its base frequency  $F$ .
- $x$  is the input signal.

Because the calculation is performed over a period of time, the block requires  $T$  seconds to respond to a step change in the input signal. This condition also applies to startup.

The harmonic RMS component  $x_{k,RMS}$  for harmonic  $k$  is calculated as:

$$x_{k,RMS}(t) = G \left( \frac{2}{T} \right) \sqrt{\left( \int_{t-T}^t x(t) \sin\left(\frac{2\pi kt}{T}\right) dt \right)^2 + \left( \int_{t-T}^t x(t) \cos\left(\frac{2\pi kt}{T}\right) dt \right)^2},$$

where  $G$  is equal to 0.5 for the DC component ( $k = 0$ ) and  $1/\sqrt{2}$  for the AC components ( $k > 0$ ).

## Ports

### Input

#### **u** — Input signal

scalar

Periodic input signal.

Data Types: single | double

### Output

#### **RMS** — Root-mean-square

scalar or vector

Estimated RMS of the input signal. If you select **Specify harmonics**, the output is a vector with each element corresponding to a specified harmonic. Otherwise, the output is a scalar representing the total RMS.

Data Types: single | double



## Parameters

### Base frequency (Hz) — Fundamental frequency

60 Hz (default) | scalar

Base frequency of the input signal corresponding to the first harmonic.

### Specify harmonics — RMS output mode

off (default) | on

Specify whether to output the total RMS of the input signals, or the individual harmonics that you specify.

### Harmonic numbers — Harmonics specification

[0 1 2] (default) | vector

Specify the harmonics for which to output an RMS.

### Dependencies

To enable this parameter, select the **Specify harmonics** parameter.

### Sample time — Block sample time

0 (default) | positive number

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For continuous operation, set this property to 0. For discrete operation, specify the sample time explicitly as a positive number. This block does not support inherited sample time.

If this block is in a masked subsystem, or other variant subsystem that allows either continuous and discrete operation, promote the sample time parameter. Promoting the sample time parameter ensures correct switching between the continuous and discrete implementations of the block. For more information, see “Promote Parameter to Mask” (Simulink).

## **See Also**

### **Blocks**

Sinusoidal Measurement (PLL) | Three-Phase Sinusoidal Measurement (PLL)

**Introduced in R2017b**

# RST Controller

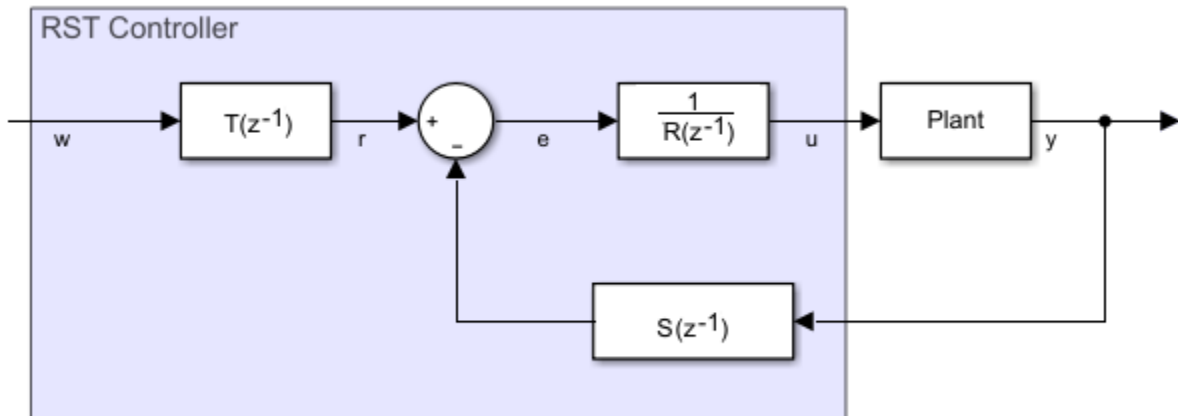
Predictive control using a polynomial representation

**Library:** Simscape / Electrical / Power Systems / Control / General Control



## Description

The RST Controller block implements a generalized predictive controller using a reference signal tracking polynomial representation. The diagram shows the equivalent circuit for the control algorithm.



## Equations

A controlled auto-regressive integrated moving average (CARIMA) model describes the plant:

$$A(z^{-1})y(k) = z^{-d}B(z^{-1})u(k-1) + \frac{e(k)C(z^{-1})}{D(z^{-1})}$$

$$A(z^{-1}) = 1 + a_1z^{-1} + \dots + a_{n_A}z^{-n_A}$$

$$B(z^{-1}) = b_0 + b_1z^{-1} + \dots + b_{n_B}z^{-n_B}$$

$$C(z^{-1}) = 1$$

$$D(z^{-1}) = 1 - z^{-1},$$

where:

- $d$  is the system dead-time.
- $y(k)$  is the plant output.
- $u(k)$  is the controller output.
- $e(k)$  is white noise with a zero-mean value.
- $A(z^{-1})$  and  $B(z^{-1})$  are the system polynomials.
- $n_A$  and  $n_B$  are the polynomials degrees.
- $C(z^{-1})$  and  $D(z^{-1})$  are the disturbance polynomials for obtaining the steady-state error.

The prediction model is given as

$$\hat{y}(k+j|k) = G_{j-d}(z^{-1})D(z^{-1})z^{-d-1}u(k+j) + \frac{H_{j-d}(z^{-1})D(z^{-1})}{C(z^{-1})}u(k-1) + \frac{F_{j-d}(z^{-1})}{C(z^{-1})}y(k)$$

and

$$j = \overline{hi, hp}$$

where:

- $hi$  is the minimum prediction.
- $hp$  is the prediction horizon.

The future control sequence, computed at time  $k$ , is

$$u(k + j - 1 | k),$$

where

$$j = \overline{1, hc}$$

and  $hc$  is the control horizon.

The predicted values of the output is

$$\hat{y}(k + j | k).$$

To determine the system polynomials,  $F_{j-d}(z^{-1})$ ,  $G_{j-d}(z^{-1})$ , and  $H_{j-d}(z^{-1})$ , the block uses two Diophantine equations. The first Diophantine equation is

$$\frac{C(z^{-1})}{A(z^{-1})D(z^{-1})} = E_{j-d}(z^{-1}) + z^{-j+d} \frac{F_{j-d}(z^{-1})}{A(z^{-1})D(z^{-1})},$$

where:

$$E_{j-d}(z^{-1}) = 1 + e_1 z^{-1} + \dots + e_{n_E} z^{-n_E}$$

$$F_{j-d}(z^{-1}) = f_0 + f_1 z^{-1} + \dots + f_{n_F} z^{-n_F}$$

$$n_E = j - d - 1$$

$$n_F = \max(n_A + n_D - 1, n_C - j + d)$$

The second Diophantine equation is

$$E_{j-d}(z^{-1})B(z^{-1}) = C(z^{-1})G_{j-d}(z^{-1}) + z^{-j+d}H_{j-d}(z^{-1}),$$

where:

$$G_{j-d}(z^{-1}) = g_0 + g_1z^{-1} + \dots + g_{n_G}z^{-n_G}$$

$$H_{j-d}(z^{-1}) = h_0 + h_1z^{-1} + \dots + h_{n_H}z^{-n_H}$$

$$n_G = j - d - 1$$

$$n_H = \max(n_C, n_B + d) - 1$$

The resulting prediction model is

$$\hat{y}(k+j|k) = G_{j-d}(z^{-1})D(z^{-1})z^{-d-1}u(k+j) + \hat{y}_0(k+j|k),$$

where

$$\hat{y}_0(k+j|k) = \frac{H_{j-d}(z^{-1})D(z^{-1})}{C(z^{-1})}u(k-1) + \frac{F_{j-d}(z^{-1})}{C(z^{-1})}y(k)$$

represents the free response of the system.

Using the matrix notation, the prediction model can be written as

$$\hat{\mathbf{y}} = \mathbf{G}u_d + \hat{\mathbf{y}}_0,$$

where:

$$\hat{\mathbf{y}} = [\hat{y}(k+hi|k), \hat{y}(k+hi+1|k), \dots, \hat{y}(k+hp|k)]^T$$

$$\mathbf{G} = \begin{bmatrix} g_{hi-d-1} & \cdots & g_0 & 0 & \cdots & 0 \\ g_{hi-d} & \cdots & g_1 & g_0 & \cdots & 0 \\ \cdots & \cdots & \cdots & \cdots & \cdots & \cdots \\ g_{hc-1} & \cdots & \cdots & \cdots & \cdots & g_0 \\ g_{hp-d-1} & \cdots & \cdots & \cdots & \cdots & g_{hp-hc-1} \end{bmatrix}$$

$$\mathbf{u}_d = \left[ D(z^{-1})u(k), \dots, D(z^{-1})u(k+hc-1) \right]^T$$

$$\hat{\mathbf{y}}_0 = \left[ \hat{y}_0(k+hi|k), \hat{y}_0(k+hi+1|k), \dots, \hat{y}_0(k+hp|k) \right]^T$$

To minimize tracking error and controller output, the block uses a cost function. To trade off between the minimization of the tracking error and the minimization of the controller output, the block uses a weighting factor,  $\lambda$ , such that

$$J = (\mathbf{G}\mathbf{u}_d + \hat{\mathbf{y}}_0 - \mathbf{w})^T (\mathbf{G}\mathbf{u}_d + \hat{\mathbf{y}}_0 - \mathbf{w}) + \lambda \mathbf{u}_d^T \mathbf{u}_d$$

for

$$D(z^{-1})u(k+i) = 0$$

and

$$i \in [hc, hp-d-1],$$

where  $\mathbf{w}$  is the reference trajectory vector. Minimizing the cost function, yields the equation for the optimal control sequence:

$$\mathbf{u}_d^* = (\mathbf{G}^T \mathbf{G} + \lambda \mathbf{I}_{hc}) \mathbf{G}^T [\mathbf{w} - \mathbf{y}_0].$$

As  $y_j$  and  $j = \overline{hi, hp}$  are elements in the first row of the matrix  $(\mathbf{G}^T \mathbf{G} + \lambda \mathbf{I}_{hc})^{-1} \mathbf{G}^T$ , applying the receding horizon principle yields the control algorithm equation as

$$D(z^{-1})u(k) = \sum_{j=hi}^{hp} \gamma_j [w(k+j|k) - \hat{y}_0(k+j|k)].$$

$$\hat{y}_0(k+j|k) = \frac{H_{j-d}(z^{-1})D(z^{-1})}{C(z^{-1})}u(k-1) + \frac{F_{j-d}(z^{-1})}{C(z^{-1})}y(k)$$

Substitution using form of the control algorithm equation: yields this

$$C(z^{-1})D(z^{-1})u(k) = - \sum_{j=hi}^{hp} \gamma_j H_{j-d}(z^{-1})D(z^{-1})u(k-1) - \sum_{j=hi}^{hp} \gamma_j F_{j-d}(z^{-1})y(k) + \sum_{j=hi}^{hp} \gamma_j C(z^{-1})w(k+)$$

The polynomial form of the control algorithm follows as

$$R(z^{-1})u(k) + S(z^{-1})y(k) = T(z^{-1})w(k+hp),$$

where:

$$R(z^{-1}) = \left( C(z^{-1}) + \sum_{j=hi}^{hp} \gamma_j z^{-1} H_{j-d}(z^{-1}) \right) D(z^{-1}),$$

$$S(z^{-1}) = \sum_{j=hi}^{hp} \gamma_j F_{j-d}(z^{-1}),$$

and

$$T(z^{-1}) = C(z^{-1}) \sum_{j=hi}^{hp} \gamma_j z^{-hp+j}.$$

## Limitations

To obtain the  $R$ ,  $R$ , and  $T$  polynomials, use the discrete-time instead of the continuous-time transfer function.



## Ports

### Input

**r — Plant reference**

scalar

Plant system reference signal.

Data Types: `single` | `double`**y — Plant output**

scalar

Plant system output signal.

Data Types: `single` | `double`

### Output

**u — Controller output**

scalar

Control system output signal.

Data Types: `single` | `double`

## Parameters

**Controller parameterization — Parameterization method**

Controller polynomials (default) | Generate polynomials

Method for parameterizing the controller. If you know the discrete-time  $R$ ,  $S$ , and  $T$  polynomial values, select `Controller polynomials`. Otherwise, select `Generate polynomials`.

**Dependencies**

Selecting a parameterization method enables other parameters.

**R polynomial — R polynomial values**

1 (default) | positive, scalar or vector

Vector of the  $R$  polynomials for the RST control.

**Dependencies**

Selecting Controller polynomials for the **Controller parameterization** parameter enables this parameter.

**S polynomial — S polynomial values**

1 (default) | positive, scalar or vector

Vector of the  $S$  polynomials for the RST control.

**Dependencies**

Selecting Controller polynomials for the **Controller parameterization** parameter enables this parameter.

**T polynomial — T polynomial values**

1 (default) | positive, scalar or vector

Vector of the  $T$  polynomials for the RST control.

**Dependencies**

Selecting Controller polynomials for the **Controller parameterization** parameter enables this parameter.

**Model discrete transfer function numerator — Transfer function numerator**

1 (default) | scalar or vector

Numerator of the system discretized transfer function. To determine the discrete transfer function, if you have a license for Control System Toolbox™, use the c2d function.

**Dependencies**

Selecting Generate polynomials for the **Controller parameterization** parameter enables this parameter.

**Model discrete transfer function denominator — Transfer function denominator**

[1 0.5] (default) | vector

Denominator of the system discretized transfer function. To determine the discrete transfer function, if you have a license for Control System Toolbox, use the `c2d` function.

**Dependencies**

Selecting `Generate polynomials` for the **Controller parameterization** parameter enables this parameter.

**Control horizon (samples) — Number of control-horizon samples**

5 (default) | positive integer

Number of samples in the control horizon.

**Dependencies**

Selecting `Generate polynomials` for the **Controller parameterization** parameter enables this parameter.

**Control weighting factor — Weighting factor**

0.5 (default) | positive number

Weighting factor for the RST controller.

**Dependencies**

Selecting `Generate polynomials` for the **Controller parameterization** parameter enables this parameter.

**System dead time (samples) — Number of dead-time samples**

2 (default) | 0 or a positive integer

Number of samples of the dead time.

**Dependencies**

Selecting `Generate polynomials` for the **Controller parameterization** parameter enables this parameter.

**Sample time (-1 for inherited) — Sampling interval**

-1 (default) | default value or a positive number

Time interval between samples. If the block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

## References

- [1] Camacho, E. F. and C. Bordons. *Model Predictive Control*. Second Edition, London: Springer, 2007.

## See Also

### Blocks

Smith Predictor Controller | State-Feedback Controller

**Introduced in R2017b**

## S-R Latch

Model an S-R Latch behaviorally



## Library

Logic

## Description

The S-R Latch block is an abstracted behavioral model of a set-reset latch. It does not model the internal individual MOSFET devices (see “Basic Assumptions and Limitations” on page 1-1352 for details). Therefore, the block runs quickly during simulation but retains the correct I/O behavior.

If the gate voltage is greater than the threshold voltage  $V_{TH}$ , then the input taken is 1 (HIGH). Otherwise, the input is zero (LOW). The gate threshold voltage  $V_{TH}$  is halfway between the **Low level input voltage** ( $V_{IL}$ ) and **High level input voltage** ( $V_{IH}$ ) parameters.

The block output logic level is either HIGH or LOW, according to the logic levels of the gate inputs and the S-R latch truth table.

S	R	Q
0	0	1
0	1	0
1	0	1
1	1	1

The block models the gate as follows:

- The gate inputs have infinite resistance and finite or zero capacitance.
- The gate output offers a selection of two models: **Linear** and **Quadratic**. For more information, see “Selecting the Output Model for Logic Blocks”. Use the **Output current-voltage relationship** parameter to specify the output model.
- You can specify propagation delay for both output models. For **Linear** output, the block sets the value of the gate output capacitor such that the resistor-capacitor time constant equals the **Propagation delay** parameter value. For **Quadratic** output, the gate input demand is lagged to approximate the **Propagation delay** parameter value.

The block output voltage depends on the output model selected:

- For **Linear** model, output high is the **High level output voltage** parameter value, and output low is the **Low level output voltage** parameter value.
- For **Quadratic** model, the output voltage for High and Low states is a function of the output current, as explained in “Quadratic Model Output and Parameters”. For zero load current, output high is  $V_{cc}$  (the **Supply voltage** parameter value), and output low is zero volts.

## Basic Assumptions and Limitations

The block does not model the internal individual MOSFET devices that make up the gate (except for the final MOSFET pair if you select the **Quadratic** option for the **Output current-voltage relationship** parameter). This limitation has the following implications:

- The behavior of this block is abstracted. In particular, response to input noise and inputs that are around the logic threshold voltage can be inaccurate. Also, dynamic response is approximate.
- The linear drop in output voltage as a function of output current is an approximation to the MOSFET or bipolar output behavior.
- Modeling of the output as a controlled voltage source is representative of a totem-pole or push-pull output stage. To model a device with an open-collector:
  - 1 Connect the output pin to the base of an NPN Bipolar Transistor or PNP Bipolar Transistor block.
  - 2 Set the **Output resistance** parameter to a suitable value.

## Parameters

- “Inputs Tab” on page 1-1353
- “Outputs Tab” on page 1-1353
- “Initial Conditions Tab” on page 1-1355

### Inputs Tab

#### Low level input voltage

Voltage value less than which the block interprets the input voltage as LOW. The default value is 2 V.

#### High level input voltage

Voltage value greater than which the block interprets the input voltage as HIGH. The default value is 3 V.

#### Average input capacitance

Fixed capacitance that approximates the input capacitance for a MOSFET gate. You can usually find this capacitance value on a manufacturer datasheet. The default value is 5 pF. Setting this value to zero can result in faster simulation times.

### Outputs Tab

#### Output current-voltage relationship

Select the output model, Linear or Quadratic. The default value is Linear.

#### Low level output voltage

Voltage value at the output when the output logic level is LOW. The default value is 0 V. This parameter is available when you select the Linear option for the **Output current-voltage relationship** parameter.

#### High level output voltage

Voltage value at the output when the output logic level is HIGH. The default value is 5 V. This parameter is available when you select the Linear option for the **Output current-voltage relationship** parameter.

#### Output resistance

Value of the series output resistor that is used to model the drop in output voltage resulting from the output current. The default value is 25  $\Omega$ . You can derive this value

from a datasheet by dividing the high-level output voltage by the maximum low-level output current. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter.

**Supply voltage**

Supply voltage value applied to the gate in your circuit. The default value is 5 V. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

**Measurement voltage**

The gate supply voltage for which mask data output resistances and currents are defined. The default value is 5 V. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

**Logic HIGH output resistance at zero current and at I<sub>OH</sub>**

A row vector [  $R_{OH1}$   $R_{OH2}$  ] of two resistance values. The first value  $R_{OH1}$  is the gradient of the output voltage-current relationship when the gate is logic HIGH and there is no output current. The second value  $R_{OH2}$  is the gradient of the output voltage-current relationship when the gate is logic HIGH and the output current is  $I_{OH}$ . The default value is [ 25 250 ]  $\Omega$ . This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

**Logic HIGH output current I<sub>OH</sub> when shorted to ground**

The resulting current when the gate is in the logic HIGH state, but the load forces the output voltage to zero. The default value is 63 mA. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

**Logic LOW output resistance at zero current and at I<sub>OL</sub>**

A row vector [  $R_{OL1}$   $R_{OL2}$  ] of two resistance values. The first value  $R_{OL1}$  is the gradient of the output voltage-current relationship when the gate is logic LOW and there is no output current. The second value  $R_{OL2}$  is the gradient of the output voltage-current relationship when the gate is logic LOW and the output current is  $I_{OL}$ . The default value is [ 30 800 ]  $\Omega$ . This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

**Logic LOW output current I<sub>OL</sub> when shorted to Vcc**

The resulting current when the gate is in the logic LOW state, but the load forces the output voltage to the supply voltage Vcc. The default value is -45 mA. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.



**Propagation delay**

Time it takes for the output to swing from LOW to HIGH or HIGH to LOW after the input logic levels change. The default value is 25 ns.

**Protection diode on resistance**

The gradient of the voltage-current relationship for the protection diodes when forward biased. The default value is 5  $\Omega$ . This parameter is available when you select the Quadratic option for the **Output current-voltage relationship** parameter.

**Protection diode forward voltage**

The voltage above which the protection diode is turned on. The default value is 0.6 V. This parameter is available when you select the Quadratic option for the **Output current-voltage relationship** parameter.

**Initial Conditions Tab****Output initial state**

Specify whether the initial output state of the block is High or Low. This parameter is used for both linear and quadratic output states, provided that the **Propagation delay** parameter is greater than zero and the Solver Configuration block does not have the **Start simulation from steady state** option selected. The default value is Low.

**Ports**

This block has the following ports:

S

Electrical input port corresponding to the set pin

R

Electrical input port corresponding to the reset pin

Q

Electrical output port corresponding to the output pin

# Schmitt Trigger

Behavioral model of Schmitt trigger



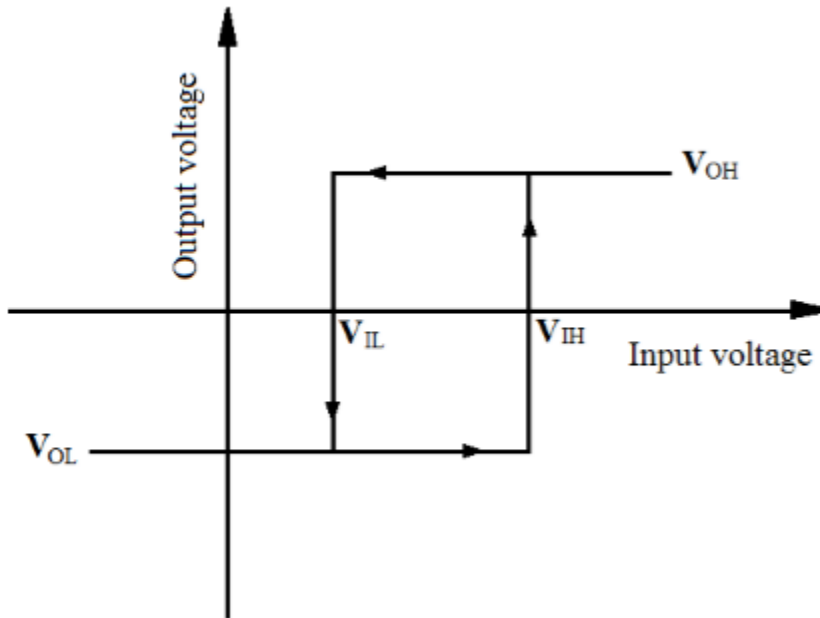
## Library

Logic

## Description

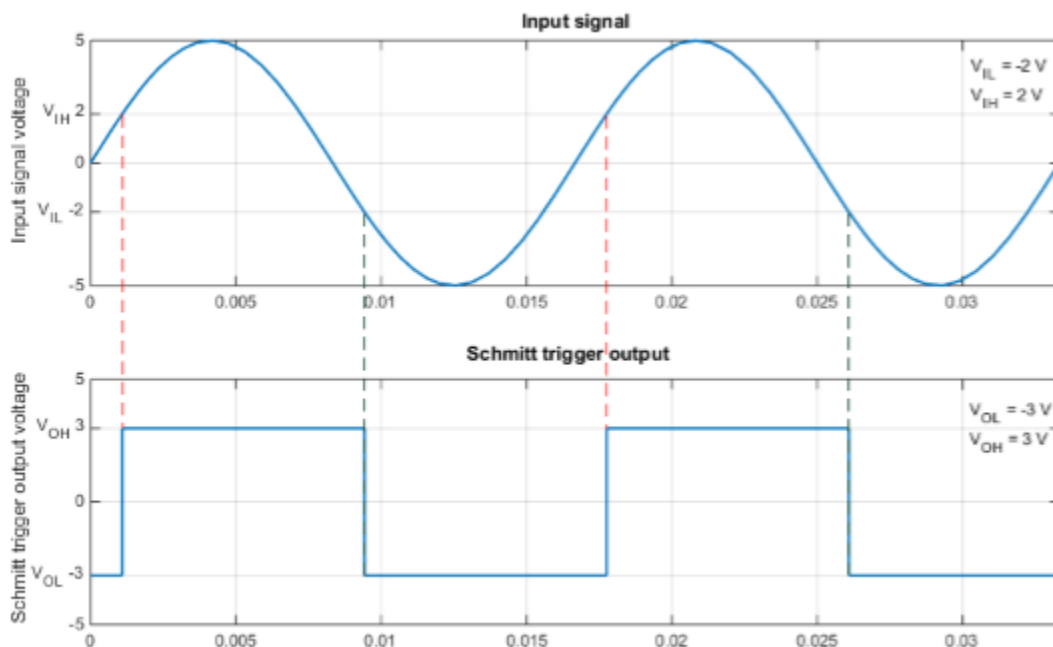
The Schmitt Trigger block implements a behavioral model of Schmitt trigger.

The block output logic level is HIGH when the input rises above the **High level input voltage** ( $V_{IH}$ ) value and does not go LOW until the input falls below the lower-valued **Low level input voltage** ( $V_{IL}$ ) value. This logic implements a hysteresis characteristic between input and output.



In the graphic,  $V_{OH}$  and  $V_{OL}$  correspond to the **High level output voltage** and **Low level output voltage** values, respectively.

The next figure shows a sample output of the block with parameters  $V_{IH} = 2V$ ,  $V_{IL} = -2V$ ,  $V_{OH} = 3V$ , and  $V_{OL} = -3V$ .



The block determines the logic levels of the gate inputs as follows:

- If the gate voltage is greater than the threshold voltage, the block interprets the input as logic 1.
- Otherwise, the block interprets the input as logic 0.

The threshold voltage is the voltage value at midpoint between the **High level input voltage** parameter value and the **Low level input voltage** parameter value.

---

**Note** To improve simulation speed, the block does not model all the internal individual MOSFET devices that make up the gate. See “Basic Assumptions and Limitations” on page 1-1359 for details.

---

The block models the gate as follows:

- The gate inputs have infinite resistance and finite or zero capacitance.

- The gate output offers a selection of two models: **Linear** and **Quadratic**. For more information, see “Selecting the Output Model for Logic Blocks”. Use the **Output current-voltage relationship** parameter to specify the output model.
- You can specify propagation delay for both output models. For **Linear** output, the block sets the value of the gate output capacitor such that the resistor-capacitor time constant equals the **Propagation delay** parameter value. For **Quadratic** output, the gate input demand is lagged to approximate the **Propagation delay** parameter value.

The block output voltage depends on the output model selected:

- For **Linear** model, output high is the **High level output voltage** parameter value, and output low is the **Low level output voltage** parameter value.
- For **Quadratic** model, the output voltage for High and Low states is a function of the output current, as explained in “Quadratic Model Output and Parameters”. For zero load current, output high is  $V_{cc}$  (the **Supply voltage** parameter value), and output low is zero volts.

## Basic Assumptions and Limitations

The block does not model the internal individual MOSFET devices that make up the gate (except for the final MOSFET pair if you select the **Quadratic** option for the **Output current-voltage relationship** parameter). This limitation has the following implications:

- The block does not accurately model the gate's response to input noise and inputs that are around the logic threshold voltage.
- The block does not accurately model dynamic response.

For circuits that involve a feedback path around a set of logic gates, you might need to set a nonzero propagation delay on one or more gates.

This block is implemented using event equations. This means that you must provide an initial output state that is consistent with the block input at time zero. For example, if you set initial output state HIGH, but the initial input voltage is below the **Low level input voltage**, then the initial output stays HIGH, the state only correcting itself when the input voltage rises above the **High level input voltage** value.

## Parameters

- “Inputs Tab” on page 1-1360
- “Outputs Tab” on page 1-1360
- “Initial Conditions Tab” on page 1-1362

### Inputs Tab

#### Low level input voltage

Voltage value below which the block interprets the input voltage as logic LOW. The default value is 2 V.

#### High level input voltage

Voltage value above which the block interprets the input voltage as logic HIGH. The default value is 3 V.

#### Average input capacitance

Fixed capacitance that approximates the input capacitance for a MOSFET gate. The MOSFET capacitance depends on the applied voltage. When you drive this block with another gate, the **Average input capacitance** produces a rise time similar to that of the MOSFET. You can usually find this capacitance value on a manufacturer datasheet. The default value is 5 pF. Setting this value to zero can result in faster simulation times.

### Outputs Tab

#### Output current-voltage relationship

Select the output model, Linear or Quadratic. The default value is Linear.

#### Low level output voltage

Voltage value at the output when the output logic level is LOW. The default value is 0 V. This parameter is available when you select the Linear option for the **Output current-voltage relationship** parameter.

#### High level output voltage

Voltage value at the output when the output logic level is HIGH. The default value is 5 V. This parameter is available when you select the Linear option for the **Output current-voltage relationship** parameter.

### Output resistance

Value of the series output resistor that is used to model the drop in output voltage resulting from the output current. The default value is 25  $\Omega$ . You can derive this value from a datasheet by dividing the high-level output voltage by the maximum low-level output current. This parameter is available when you select the **Linear** option for the **Output current-voltage relationship** parameter.

### Supply voltage

Supply voltage value applied to the gate in your circuit. The default value is 5 V. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Measurement voltage

The gate supply voltage for which mask data output resistances and currents are defined. The default value is 5 V. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Logic HIGH output resistance at zero current and at $I_{OH}$

A row vector [  $R_{OH1}$   $R_{OH2}$  ] of two resistance values. The first value  $R_{OH1}$  is the gradient of the output voltage-current relationship when the gate is logic HIGH and there is no output current. The second value  $R_{OH2}$  is the gradient of the output voltage-current relationship when the gate is logic HIGH and the output current is  $I_{OH}$ . The default value is [ 25 250 ]  $\Omega$ . This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Logic HIGH output current $I_{OH}$ when shorted to ground

The resulting current when the gate is in the logic HIGH state, but the load forces the output voltage to zero. The default value is 63 mA. This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

### Logic LOW output resistance at zero current and at $I_{OL}$

A row vector [  $R_{OL1}$   $R_{OL2}$  ] of two resistance values. The first value  $R_{OL1}$  is the gradient of the output voltage-current relationship when the gate is logic LOW and there is no output current. The second value  $R_{OL2}$  is the gradient of the output voltage-current relationship when the gate is logic LOW and the output current is  $I_{OL}$ . The default value is [ 30 800 ]  $\Omega$ . This parameter is available when you select the **Quadratic** option for the **Output current-voltage relationship** parameter.

## Logic LOW output current I\_OL when shorted to Vcc

The resulting current when the gate is in the logic LOW state, but the load forces the output voltage to the supply voltage Vcc. The default value is -45 mA. This parameter is available when you select the Quadratic option for the **Output current-voltage relationship** parameter.

## Propagation delay

Time it takes for the output to swing from LOW to HIGH or HIGH to LOW after the input logic levels change. The default value is 25 ns.

## Protection diode on resistance

The gradient of the voltage-current relationship for the protection diodes when forward biased. The default value is 5  $\Omega$ . This parameter is available when you select the Quadratic option for the **Output current-voltage relationship** parameter.

## Protection diode forward voltage

The voltage above which the protection diode is turned on. The default value is 0.6 V. This parameter is available when you select the Quadratic option for the **Output current-voltage relationship** parameter.

## Initial Conditions Tab

### Output initial state

Specify whether the initial output state of the block is High or Low. This parameter is used for both linear and quadratic output states, provided that the **Propagation delay** parameter is greater than zero and the Solver Configuration block does not have the **Start simulation from steady state** option selected. The default value is Low.

## Ports

The block has the following ports:

A

Electrical input port

J

Electrical output port

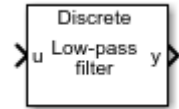


**Introduced in R2015a**

## Second-Order Filter

Low-pass, high-pass, band-pass, or band-stop second-order filter

**Library:** Simscape / Electrical / Power Systems / Control / General Control

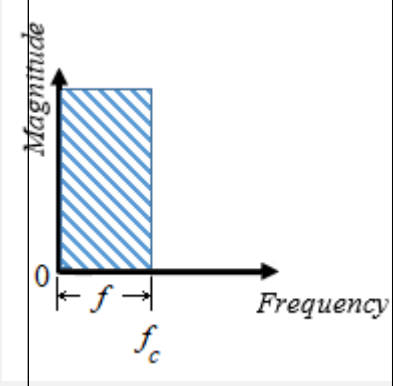
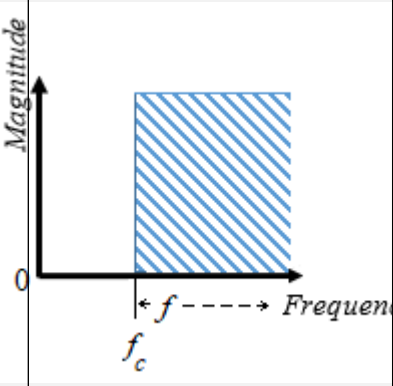


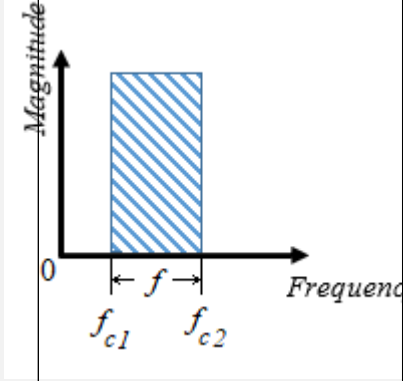
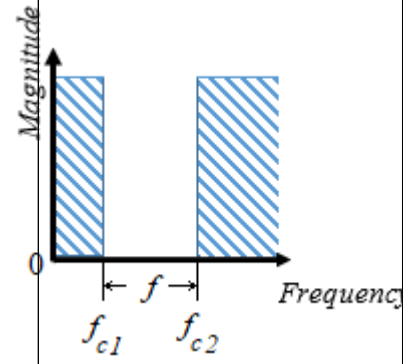
### Description

The Second-Order Filter block implements different types of second-order filters. Filters are useful for attenuating noise in measurement signals.

The block provides these filter types:

- Low pass — Allows signals,  $f$ , only in the range of frequencies below the cutoff frequency,  $f_c$ , to pass.
- High pass — Allows signals,  $f$ , only in the range of frequencies above the cutoff frequency,  $f_c$ , to pass.
- Band pass — Allows signals,  $f$ , only in the range of frequencies between two cutoff frequencies,  $f_{c1}$  and  $f_{c2}$ , to pass.
- Band stop — Prevents signals,  $f$ , only in the range of frequencies between two cutoff frequencies,  $f_{c1}$  and  $f_{c2}$ , from passing.

Filter Type	Frequency Range, $f$	
Low-Pass		$f < f_c$
High-Pass		$f > f_c$

Filter Type	Frequency Range, $f$	
<p><b>Band-Pass</b></p>		$f_{c1} < f < f_{c2}$
<p><b>Band-Stop</b></p>		$f_{c1} < f < f_{c2}$

## Equations

The second order derivative state equation for the filter is:

$$\frac{d^2x}{dt^2} = u - 2\zeta\omega_n \frac{dx}{dt} - \omega_n^2 x$$

Where:

- $x$  is the filter internal state.

- $u$  is the filter input.
- $\omega_n$  is the filter natural frequency.
- $\zeta$  is the filter damping factor.

For each filter type, the table maps the block output,  $y(x)$ , as a function of the internal state of the filter, to the s-domain transfer function,  $G(s)$ .

Filter Type	Output, $y(x)$	Transfer Function, $G(s)$
<b>Low-Pass</b>	$\omega_n^2 x$	$\frac{\omega_n^2}{s^2 + 2\zeta\omega_n s + \omega_n^2}$
<b>High-Pass</b>	$\frac{d^2 x}{dt^2}$	$\frac{s^2}{s^2 + 2\zeta\omega_n s + \omega_n^2}$
<b>Band-Pass</b>	$2\zeta\omega_n \frac{dx}{dt}$	$\frac{2\zeta\omega_n s}{s^2 + 2\zeta\omega_n s + \omega_n^2}$
<b>Band-Stop</b>	$\frac{d^2 x}{dt^2} + x$	$\frac{s^2 + \omega_n^2}{s^2 + 2\zeta\omega_n s + \omega_n^2}$

For Initialization:

$$\dot{x}(0) = \left. \frac{dx}{dt} \right|_{t=0}$$

$$u(0) = u_1(0) + u_2(0)$$

$$u_1(0) = A_0 e^{j\varphi_0}$$

$$u_2(0) = b_0 e^{j\frac{\pi}{2}}$$

Where:

- $x(0)$  is the initial state of the filter.
- $u(0)$  is the initial input to the filter.
- $u_1(0)$  is the AC component of the steady-state initial input.
- $A_0$  is the initial amplitude.
- $\varphi_0$  is the initial phase.
- $u_2(0)$  is the DC component of the steady-state initial input.
- $b_0$  is the initial bias.

In the  $s$ -domain  $s = j\omega_0$ . Therefore, for the initial frequency,  $\omega_0$ :

$$\dot{x}(0) = \text{Im} \left( \frac{j\omega_0 u_1(0)}{-\omega_0^2 + j\omega_0 2\zeta\omega_n + \omega_n^2} \right).$$

$$x(0) = \text{Im} \left( \frac{\dot{x}(0)\omega_n^2}{j\omega_0} + u_2(0) \right)$$

## Ports

### Input

#### **u** — Filter input

scalar

Filter input.

Data Types: single | double

## Output

### **y** — Filtered output

scalar

Filtered output.

Data Types: `single` | `double`

## Parameters

### Main

#### **Filter type** — Filter type

`Low-pass (default)` | `High-pass` | `Band-pass` | `Band-stop`

Type of second-order filter.

#### **Natural frequency (Hz)** — Natural frequency

`60 (default)` | positive scalar

Natural frequency, in Hz.

### Initial Conditions

#### **Damping factor** — Damping factor

`0.707 (default)` | nonnegative scalar

Damping factor of the filter.

#### **Sample time (-1 for inherited)** — Block sample time

`-1 (default)` | `0` | positive scalar

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For inherited discrete-time operation, specify `-1`. For discrete-time operation, specify a positive integer. For continuous-time operation, specify `0`.

If this block is in a masked subsystem, or other variant subsystem that allows you to switch between continuous operation and discrete operation, promote the sample time parameter. Promoting the sample time parameter ensures correct switching between the continuous and discrete implementations of the block. For more information, see “Promote Parameter to Mask” (Simulink).

### **Initial amplitude — Initial amplitude**

0 (default) | nonnegative scalar

Amplitude at the start of simulation.

### **Initial phase (rad) — Initial phase**

0 (default) | nonnegative scalar

Phase, in rad, at the start of simulation.

### **Initial frequency (Hz) — Initial frequency**

0 (default) | scalar

Frequency, in Hz, at the start of simulation.

### **Initial bias — Initial bias**

0 (default) | nonnegative scalar

Bias at the start of simulation.

## **References**

[1] Agarwal, A. and Lang, J. H. *Foundations of Analog and Digital Electronic Circuits*. New York: Elsevier, 2005.

## **See Also**

Low-Pass Filter (Discrete or Continuous) | SM PSS1A | Second-Order Low-Pass Filter (Discrete or Continuous) | Variable-Frequency Second-Order Filter | Washout (Discrete or Continuous)

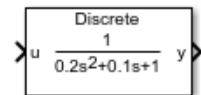
**Introduced in R2018b**



# Second-Order Low-Pass Filter (Discrete or Continuous)

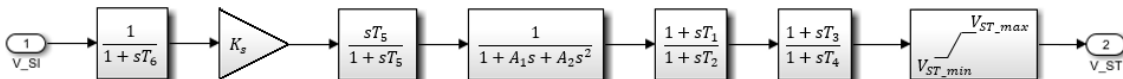
Discrete-time or continuous-time second-order low-pass filter

**Library:** Simscape / Electrical / Power Systems / Control / General Control



## Description

The Second-Order Low-Pass Filter (Discrete or Continuous) block implements a second-order low pass filter in conformance with IEEE Std 421.5-2016<sup>[1]</sup>. In the standard, the filter is a single input, single output signal conditioner that is used in the Power System Stabilizer PSS1A.



In the PSS1A, the filter accounts for some of the low-frequency effects of high-frequency torsional filters. It can also help shape the gain and phase characteristics of the stabilizer.

To switch between continuous and discrete implementations of the Second-Order Low-Pass Filter (Discrete or Continuous) block, adjust the **Sample time** parameter value.

## Ports

### Input

**u** — Filter input

scalar or vector

Filter input.

Data Types: single | double

## Output

### **y — Filtered output**

scalar or vector

Filtered output. The output has the same number of elements as the input.

Data Types: single | double

## Parameters

### **Coefficient of $s$ , $A1$ — $A1$ coefficient**

0.1 (default) | positive scalar

$A1$ , the signal conditioning frequency filter coefficient of  $s$ .

### **Coefficient of $s^2$ , $A2$ — $A2$ coefficient**

0.2 (default) | positive scalar

$A2$ , the signal conditioning frequency filter coefficient of  $s^2$ .

### **Sample time (-1 for inherited) — Block sample time**

-1 (default) | 0 | positive scalar

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For inherited discrete-time operation, specify -1. For discrete-time operation, specify a positive integer. For continuous-time operation, specify 0.

If this block is in a masked subsystem, or other variant subsystem that allows you to switch between continuous operation and discrete operation, promote the sample time parameter. Promoting the sample time parameter ensures correct switching between the continuous and discrete implementations of the block. For more information, see “Promote Parameter to Mask” (Simulink).

## References

- [1] *IEEE Recommended Practice for Excitation System Models for Power System Stability Studies*. IEEE Std 421.5-2016. Piscataway, NJ: IEEE-SA, 2016.

## See Also

Low-Pass Filter (Discrete or Continuous) | SM PSS1A | Second-Order Filter | Variable-Frequency Second-Order Filter | Washout (Discrete or Continuous)

**Introduced in R2018b**

## Secondary Winding

(To be removed) Linear nonideal transformer winding

---

**Note** The Secondary Winding block will be removed in a future release. Use the Winding block instead. To learn how to create a custom transformer without this block, refer to “Push-Pull Buck Converter in Continuous Conduction Mode”.

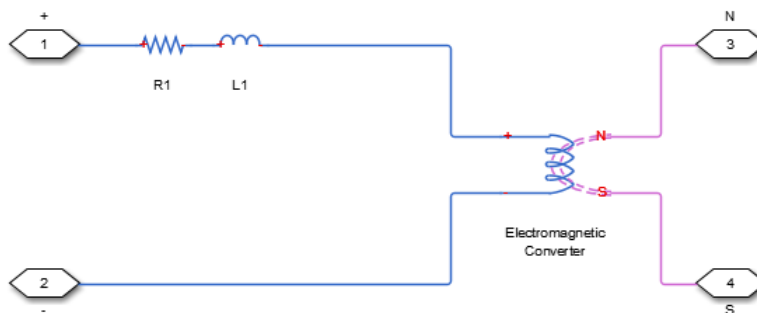
---

## Library

Simscape / Electrical / Power Systems / Passive Devices / Transformers / Fundamental Components

## Description

The Secondary Winding block models linear nonideal winding of a transformer with linear winding leakage effects. The figure shows the equivalent circuit diagram for the secondary winding.



- $R_l$  is the leakage resistance.
- $L_l$  is the leakage inductance.

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Parameters

### Main Tab

#### Number of winding turns

Number of wire turns on the transformer winding. The default value is 10.

#### Leakage resistance

Power loss in the winding. The default value is  $1e-3$  Ohm.

#### Leakage inductance

Magnetic flux loss in the winding. The default value is  $1e-3$  H.

## Ports

The block has the following ports:

+

Positive electrical conserving port

-

Negative electrical conserving port

N

North magnetic conserving port

S

South magnetic conserving port

## **See Also**

Electromagnetic Converter | Winding

## **Topics**

“Custom Zigzag Transformer”

“Push-Pull Buck Converter in Continuous Conduction Mode”

“Push-Pull Buck Converter in Discontinuous Conduction Mode”

**Introduced in R2013b**

# Serial-In Parallel-Out Shift Register

Discrete serial-in, parallel-out shift register

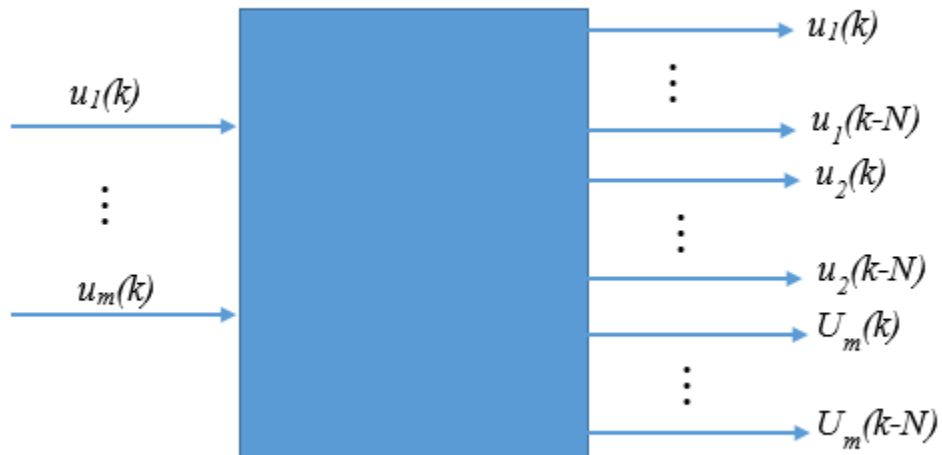
**Library:** Simscape / Electrical / Power Systems / Control / General Control



## Description

The Serial-In Parallel-Out Shift Register block implements a serial-in parallel-out shift register in discrete time. You can use a shift register to convert between serial and parallel interfaces or to implement a circuit delay or hardware stack.

This block outputs a vector of last  $N$  samples of the input signal. If the input signal is a vector, the block outputs the last  $N$  samples of each input signal.



## Ports

### Input

**u — Serial input**

scalar | vector

Serial input signal.

Data Types: single | double

### Output

**y — Parallel output**

vector

Parallel output signal.

Data Types: single | double

## Parameters

**Number of samples — Number of input samples**

4 (default) | positive integer

Number of register stages or samples.

**Initial condition — Initial value of the input sample**

0 (default) | real number

Initial value of the  $N-1$  samples preceding simulation start time. The value must be a scalar or a vector of the same size as the input signal.

**Sample time (-1 for inherited) — Block sample time**

-1 (default) | 0 | positive scalar

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).



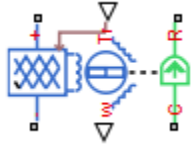
For inherited discrete-time, specify -1. For discrete-time, specify a positive integer. For continuous-time, specify 0.

## **See Also**

**Introduced in R2018b**

## Servomotor

Brushless motor model with closed-loop torque control



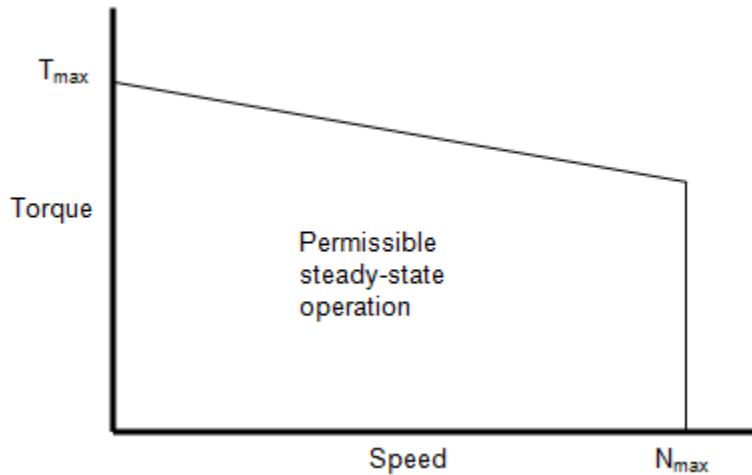
## Library

Rotational Actuators

## Description

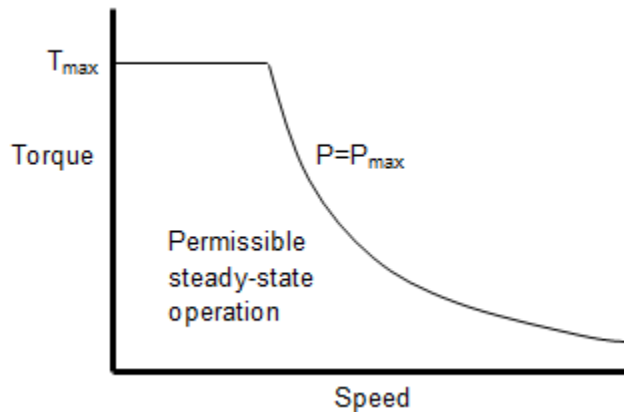
The Servomotor block represents a brushless motor model with closed-loop torque control. This block abstracts the torque-speed behavior of the combined motor and motor driver in order to support system-level simulation where simulation speed is important.

The block permits only the range of torques and speeds that the torque-speed envelope defines. In the default block configuration, you specify this data in the block dialog box as a set of speed data points and corresponding maximum torque values. The following figure shows a typical torque-speed envelope for a servomotor.



Specify the torque-speed envelope for the positive torque region only, that is, quadrants 1 and 4. If you specify only for positive speeds (quadrant 1 or, equivalently, the motoring region), then the quadrant 4 torque envelope is defined by the block as the mirror image of quadrant 1. The servomotor torque-speed envelope has the same profile when the motor is operating in a reverse direction (quadrants 2 and 3).

Instead of providing tabulated torque-speed data, you can specify a maximum torque and a maximum power. This results in the torque-speed envelope profile shown below. The other three operating quadrants are constrained by this same profile.



The block produces a positive torque acting from the mechanical C to R ports.

## Modeling Electrical Losses

The block allows both simplified and tabulated definition of electrical losses. The default, simplified, behavior is to model the losses as the sum of the following four terms:

- A series resistance between the DC power supply and the motor drive.
- Fixed losses independent of torque and speed,  $P_0$ . Use this to account for fixed converter losses.
- A torque-dependent electrical loss  $k\tau^2$ , where  $\tau$  is the torque and  $k$  is a constant. This represents ohmic losses in the copper windings.
- A speed-dependent electrical loss  $k_w\omega^2$ , where  $\omega$  is the speed and  $k_w$  is a constant. This represents iron losses due to eddy currents.

Alternatively, you can provide tabulated loss values as a function of motor speed and load torque. When using this option, provide data for all of the operating quadrants that your simulation will run in. If you provide partial data (for example, just for the quadrant 1 forward motoring region), then the other quadrants are assumed to repeat the same pattern of losses. This will normally be correct for the reverse motoring region, but may be an approximation for the braking/generating quadrants. The block does no extrapolation of loss values for speed and torque magnitudes that exceed the range of the table.

Finally, you can specify electrical losses by using tabulated efficiency data, instead of a single efficiency measurement or tabulated loss data. When using this option, also provide data for all of the operating quadrants that your simulation will run in. If you provide partial data (for example, just for the quadrant 1 forward motoring region), then the other quadrants are assumed to repeat the same pattern of losses.

The best practice is to provide tabulated loss data as a function of speed and torque, rather than tabulated efficiency data, because:

- Efficiency becomes ill-defined for zero speed or zero torque.
- Using losses, you can also account for fixed losses that are still present for zero speed or torque.

If you use the tabulated efficiencies option:

- The block converts the efficiency values you provide into losses and uses the tabulated losses for simulation.
- Efficiency values you provide for zero speed or zero torque are ignored, and losses are assumed zero when either torque or speed is zero.
- The block uses linear interpolation to determine losses. Provide tabulated data for low speeds and low torques, as required, to get the desired level of accuracy for lower power conditions.
- The block does no extrapolation of loss values for speed and torque magnitudes that exceed the range of the table.

When you provide tabulated loss or efficiency data, you can also specify it as a function of speed, load torque, and DC supply voltage. This option is useful when the supply voltage is not regulated and can vary during the simulation. One example is an electric vehicle drivetrain that does not have a DC-DC regulator upstream of the motor drive. Use the Servomotor block to model the motor drive and provide tabulated loss or efficiency values as a function of motor speed, load torque, and DC supply voltage.

## Block Variants

The block provides four modeling variants, accessible by right-clicking the block in your block diagram and then selecting the appropriate option from the context menu, under **Simscape > Block choices**:

- **No thermal port** — Basic model that does not simulate faults or thermal effects. This is the default.

- **Show thermal port** — Model with exposed thermal port. This model does not simulate faults.
- **Faultable| No thermal port** — Model with exposed fault control port. This model does not simulate thermal effects.
- **Faultable | Show thermal port** — Model that lets you simulate both faults and thermal effects. Both the thermal port and the fault input port are exposed.

## Thermal Ports

The block has an optional thermal port, hidden by default. To expose the thermal port, select one of the block variants that model thermal effects, as described in “Block Variants” on page 1-1383. This action displays the thermal port H on the block icon, and adds the **Temperature Dependence** and **Thermal Port** tabs to the block dialog box. These tabs are described further on this reference page.

Use the thermal port to simulate the effects of copper resistance losses that convert electrical power to heat. For more information on using thermal ports in actuator blocks, see “Simulating Thermal Effects in Rotational and Translational Actuators”.

## Simulating Faults

You can use the physical signal input port F to simulate servomotor failure, as well as connecting and disconnecting the DC supply. You cannot simulate disconnecting the DC supply by simply opening a switch, because there must be a finite voltage on the servomotor terminals, producing the current that balances the electrical and mechanical power.

To expose the fault control port, select one of the faultable block variants, as described in “Block Variants” on page 1-1383. This action displays the physical signal input port F on the block icon, and adds the **Faults** tab to the block dialog box. This tabs are is described further on this reference page.

If a signal is connected to port F, then the block operates according to the parameter settings on the **Faults** tab. For example, if **Fault condition** is Faulted if  $F \geq \text{Fault threshold}$ , then when the signal at port F rises above the **Fault threshold** value, the servomotor stops operating, zero current is taken from the supply side, and zero current is supplied to the load side.

## Basic Assumptions and Limitations

This model is based on the following assumptions:

- The motor driver tracks a torque demand with a time constant  $T_c$ .
- Motor speed fluctuations due to mechanical load do not affect the motor torque tracking.

## Parameters

- “Electrical Torque Tab” on page 1-1385
- “Electrical Losses Tab” on page 1-1386
- “Faults Tab” on page 1-1389
- “Mechanical Tab” on page 1-1390
- “Temperature Dependence Tab” on page 1-1390
- “Thermal Port Tab” on page 1-1392

## Electrical Torque Tab

### Parameterize by

Select one of the following methods for block parameterization:

- **Tabulated torque-speed envelope** — Provide the vectors of rotational speeds and corresponding maximum torque values. This is the default option.
- **Maximum torque and power** — Define the torque-speed envelope by providing values for maximum permissible torque and motor power.

### Vector of rotational speeds

Rotational speeds for permissible steady-state operation. This parameter is visible only if you select **Tabulated torque-speed envelope** for the **Parameterize by** parameter. The default value is [0 3.75e+03 7.5e+03 8e+03] rpm. To avoid poor performance due to an infinite slope in the torque-speed curve, specify a vector of rotational speeds that does not contain duplicate consecutive values.

### Vector of maximum torque values

Maximum torque values for permissible steady-state operation. This parameter is visible only if you select **Tabulated torque-speed envelope** for the

**Parameterize by** parameter. These values correspond to the speeds in the **Vector of rotational speeds** parameter and define the torque-speed envelope for the motor. The default value is [0.09 0.08 0.07 0] Nm.

### **Maximum torque**

The maximum permissible motor torque. This parameter is visible only if you select **Maximum torque and power** for the **Parameterize by** parameter. The default value is 0.1 Nm.

### **Maximum power**

The maximum permissible motor power. This parameter is visible only if you select **Maximum torque and power** for the **Parameterize by** parameter. The default value is 30 W.

### **Torque Control time constant, Tc**

Time constant with which the motor driver tracks a torque demand. The default value is 0.02 s.

## **Electrical Losses Tab**

### **Parameterize losses by**

Select one of the following methods for electrical loss parameterization:

- **Single efficiency measurement** — Model the losses as the sum of the four terms, listed in the block description, at a single measurement point. This is the default option.
- **Tabulated loss data as a function of speed and torque** — Determine the losses by two-dimensional table lookup based on the provided tabulated data for motor speeds, load torques, and corresponding losses.
- **Tabulated efficiency data as a function of speed and torque** — Determine the losses by two-dimensional table lookup based on the provided tabulated data for motor speeds, load torques, and corresponding efficiencies.
- **Tabulated loss data as a function of speed, torque, and DC supply voltage** — Determine the losses by three-dimensional table lookup based on the provided tabulated data for motor speeds, load torques, DC supply voltages, and corresponding losses.
- **Tabulated efficiency data as a function of speed, torque, and DC supply voltage** — Determine the losses by three-dimensional table lookup based on the provided tabulated data for motor speeds, load torques, DC supply voltages, and corresponding efficiencies.



See “Modeling Electrical Losses” on page 1-1382 for details.

### Motor and driver overall efficiency (percent)

The block defines overall efficiency as

$$\eta = 100 \frac{\tau_0 \omega_0}{\tau_0 \omega_0 + P_0 + k \tau_0^2 + k_w \omega_0^2}$$

where:

- $\tau_0$  represents the **Torque at which efficiency is measured**.
- $\omega_0$  represents the **Speed at which efficiency is measured**.
- $P_0$  represents the **Fixed losses independent of torque or speed**.
- $k \tau_0^2$  represents the torque-dependent electrical losses.
- $k_w \omega_0^2$  represents the speed-dependent iron losses.

At initialization, the block solves the efficiency equation for  $k$ . The block neglects losses associated with the rotor damping. This parameter is visible only if **Parameterize losses by** is set to **Single efficiency measurement**. The default value is 100 %.

### Speed at which efficiency is measured

Speed that the block uses to calculate torque-dependent electrical losses. This parameter is visible only if **Parameterize losses by** is set to **Single efficiency measurement**. The default value is 3.75e+03 rpm.

### Torque at which efficiency is measured

Torque that the block uses to calculate torque-dependent electrical losses. This parameter is visible only if **Parameterize losses by** is set to **Single efficiency measurement**. The default value is 0.08 Nm.

### Iron losses

Iron losses at the speed and torque at which efficiency is defined. This parameter is visible only if **Parameterize losses by** is set to **Single efficiency measurement**. The default value is 0 W.

### Fixed losses independent of torque and speed

Fixed electrical loss associated with the driver when the motor current and torque are zero. This parameter is visible only if **Parameterize losses by** is set to **Single efficiency measurement**. The default value is 0 W.

### **Vector of speeds (w) for tabulated losses**

The vector of speed values, to be used for table lookup when calculating losses. This parameter is visible only if **Parameterize losses by** is set to Tabulated loss data as a function of speed and torque, Tabulated loss data as a function of speed, torque, and DC supply voltage, Tabulated efficiency data as a function of speed and torque, or Tabulated efficiency data as a function of speed, torque, and DC supply voltage. The default value is [-8000 -4000 0 4000 8000] rpm.

### **Vector of torques (T) for tabulated losses**

The vector of speed values, to be used for table lookup when calculating losses. This parameter is visible only if **Parameterize losses by** is set to Tabulated loss data as a function of speed and torque, Tabulated loss data as a function of speed, torque, and DC supply voltage, Tabulated efficiency data as a function of speed and torque, or Tabulated efficiency data as a function of speed, torque, and DC supply voltage. The default value is [0 0.03 0.06 0.09] Nm.

### **Corresponding losses, P(w,T)**

Tabulated values for electrical losses as a function of speed and torque, to be used for 2D table lookup. Each value in the matrix specifies the losses for a specific combination of speed and torque. The matrix size must match the dimensions defined by the speed and torque vectors. This parameter is visible only if **Parameterize losses by** is set to Tabulated loss data as a function of speed and torque. The default value is [1.49 1.67 2.21 3.10; 0.42 0.69 1.14 2.03; 0.06 0.24 0.78 1.68; 0.42 0.69 1.14 2.03; 1.49 1.67 2.21 3.10] W.

### **Corresponding efficiency (percent), E(w,T)**

Tabulated efficiency values, in percent, as a function of speed and torque, to be used for 2D table lookup. Each value in the matrix specifies the efficiency for a specific combination of speed and torque. The matrix size must match the dimensions defined by the speed and torque vectors. Efficiency values you provide for zero speed or zero torque are ignored, and losses are assumed zero when either torque or speed is zero. The block uses linear interpolation to determine losses. Provide tabulated data for low speeds and low torques, as required, to get the desired level of accuracy for lower power conditions. This parameter is visible only if **Parameterize losses by** is set to Tabulated efficiency data as a function of speed and torque. The default value is [95 95 95 95; 95 95 95 95; 95 95 95 95; 95 95 95 95; 95 95 95 95].

### Vector of DC supply voltages (**v**) for tabulated losses

The vector of DC supply voltages, to be used for table lookup when calculating losses. This parameter is visible only if **Parameterize losses by** is set to Tabulated loss data as a function of speed, torque, and DC supply voltage or Tabulated efficiency data as a function of speed, torque, and DC supply voltage. The default value is [100 200 400] V.

### Corresponding losses, **P(w,T,v)**

Tabulated values for electrical losses as a function of speed, torque, and DC supply voltage, to be used for 3D table lookup. Each value in the matrix specifies the losses for a specific combination of speed, torque, and DC supply voltage. The matrix size must match the dimensions defined by the three vectors. This parameter is visible only if **Parameterize losses by** is set to Tabulated loss data as a function of speed, torque, and DC supply voltage. The default value is ones(5,4,3) W.

### Corresponding efficiency (percent), **E(w,T,v)**

Tabulated efficiency values, in percent, as a function of speed, torque, and DC supply voltage, to be used for 3D table lookup. Each value in the matrix specifies the efficiency for a specific combination of speed, torque, and DC supply voltage. The matrix size must match the dimensions defined by the three vectors. Efficiency values you provide for zero speed or zero torque are ignored, and losses are assumed zero when either torque or speed is zero. The block uses linear interpolation to determine losses. Provide tabulated data for low speeds and low torques, as required, to get the desired level of accuracy for lower power conditions. This parameter is visible only if **Parameterize losses by** is set to Tabulated efficiency data as a function of speed, torque, and DC supply voltage. The default value is 95\*ones(5,4,3).

### Supply series resistance

The equivalent resistance used in series with the DC supply to model electrical losses that are proportional to the driver supply current. The block assumes that the DC supply current is approximately constant under constant load conditions. The default value is 0  $\Omega$ .

## Faults Tab

This tab appears only for blocks with an exposed fault control port. For more information, see “Simulating Faults” on page 1-1384.

### **Fault condition**

Selects whether the fault is triggered by a signal that is high or low:

- **Faulted if  $F \geq \text{Fault threshold}$**  — Servomotor is disabled if the signal at port F rises above the threshold value. This is the default option.
- **Faulted if  $F \leq \text{Fault threshold}$**  — Servomotor is disabled if the signal at port F falls below the threshold value.

### **Fault threshold**

The threshold value used to detect a fault. The default value is 0.5.

## **Mechanical Tab**

### **Rotor inertia**

Rotor resistance to change in motor motion. The default value is  $5e-06 \text{ kg}\cdot\text{m}^2$ . The value can be zero.

### **Rotor damping**

Rotor damping. The default value is  $1e-05 \text{ N}\cdot\text{m}/(\text{rad/s})$ . The value can be zero.

### **Initial rotor speed**

Rotor speed at the start of the simulation. The default value is 0 rpm.

## **Temperature Dependence Tab**

This tab appears only for blocks with an exposed thermal port. For more information, see “Thermal Ports” on page 1-1384.

### **Resistance temperature coefficient**

Parameter  $\alpha$  in the equation defining resistance as a function of temperature, as described in “Thermal Model for Actuator Blocks”. This parameter is visible only if the **Parameterize losses by** parameter on the **Electrical Losses** tab is set to **Single efficiency measurement**. The default value is for copper, and is 0.00393 1/K.

### **Measurement temperature**

The temperature for which motor parameters are defined. If you parameterize electrical losses by tabulated loss data, then this is the temperature for which the **Corresponding losses,  $P(w,T)$**  are given on the **Electrical Losses** tab. The default value is 25 °C.

### Second measurement temperature

The temperature for which the **Corresponding losses,  $P(w,T)$ , at second measurement temperature** are given. This parameter is visible only if the **Parameterize losses by** parameter on the **Electrical Losses** tab is set to Tabulated loss data. The default value is 125 °C.

### Corresponding losses, $P(w,T)$ , at second measurement temperature

Iron losses at the second measurement temperature, corresponding to the speed and torque tabulated values on the **Electrical Losses** tab. This parameter is visible only if the **Parameterize losses by** parameter on the **Electrical Losses** tab is set to Tabulated loss data. The default value is [1.49 1.74 2.49 3.74;0.42 0.67 1.42 2.67;0.06 0.31 1.06 2.31;0.42 0.67 1.42 2.67;1.49 1.74 2.49 3.74] W.

### Corresponding efficiency (percent), $E(w,T)$ , at second measurement temperature

Tabulated efficiency values, in percent, at the second measurement temperature, corresponding to the speed and torque tabulated values on the **Electrical Losses** tab. This parameter is visible only if the **Parameterize losses by** parameter on the **Electrical Losses** tab is set to Tabulated efficiency data as a function of speed and torque. The default value is [95 95 95 95; 95 95 95 95; 95 95 95; 95 95 95 95; 95 95 95 95].

### Corresponding losses, $P(w,T,v)$ , at second measurement temperature

Iron losses at the second measurement temperature, corresponding to the speed, torque, and DC supply voltage tabulated values on the **Electrical Losses** tab. This parameter is visible only if the **Parameterize losses by** parameter on the **Electrical Losses** tab is set to Tabulated loss data as a function of speed, torque, and DC supply voltage. The default value is ones(5,4,3) W.

### Corresponding efficiency (percent), $E(w,T,v)$ , at second measurement temperature

Tabulated efficiency values, in percent, at the second measurement temperature, corresponding to the speed, torque, and DC supply voltage tabulated values on the **Electrical Losses** tab. This parameter is visible only if the **Parameterize losses by** parameter on the **Electrical Losses** tab is set to Tabulated efficiency data as a function of speed, torque, and DC supply voltage. The default value is 95\*ones(5,4,3).

## **Thermal Port Tab**

This tab appears only for blocks with an exposed thermal port. For more information, see “Thermal Ports” on page 1-1384.

### **Thermal mass**

Thermal mass of the electrical winding, defined as the energy required to raise the temperature by one degree. The default value is 100 J/K.

### **Initial temperature**

The temperature of the thermal port at the start of simulation. The default value is 25 °C.

## **Ports**

This block has the following ports:

+

Positive electrical DC supply

-

Negative electrical DC supply

Tr

Reference torque demand

w

Mechanical speed output

C

Mechanical rotational conserving port

R

Mechanical rotational conserving port

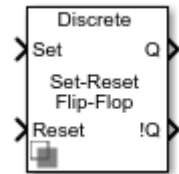
## **See Also**

DC Motor | Induction Motor | Shunt Motor | Universal Motor

# Set-Reset Flip-Flop

Implement a set-reset flip-flop or bistable multivibrator

**Library:** Simscape / Electrical / Power Systems / Control / General Control



## Description

The Set-Reset Flip-Flop block implements a set-reset flip-flop or bistable multivibrator.

The block maintains the output signals, Q and !Q, unless an external trigger is applied. An external trigger (*Set*) produces a change of state, which is maintained until a second external trigger (*Reset*) is applied.

The table shows the relationship between the block input and output signals.

Set	Reset	Q	!Q
0	0	Last Q	Last !Q
0	1	0	1
1	0	1	0
1	1	Undefined	Undefined

When the state is undefined, the priority is provided as an external parameter.

## Ports

### Input

**Set** — Set input signal

0 | 1

Input signal that triggers a state change.

Data Types: Boolean

**Reset — Reset input signal**

0 | 1

Input signal that resets a state change.

Data Types: Boolean

## Output

**Q — Output signal**

0 | 1

Output signal **Q**, with the same dimensions and data type as the input **signal**.

Data Types: Boolean

**!Q — Complement output signal**

0 | 1

Output signal **!Q**, with the same dimensions and data type as the input **signal**.

Data Types: Boolean

## Parameters

**Priority when undefined state — State priority**

Set (default) | Reset

Priority for the undefined state, that is, when both *Set* and *Reset* are true.

**Initial condition for Q state — Initial value of the output Q**

0 (default) | 1

Initial condition for *Q* state.

**Sample time (-1 for inherited) — Block sample time**

-1 (default) | 0 | positive scalar



Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For inherited discrete-time operation, specify  $-1$ . For discrete-time operation, specify a positive integer. For continuous-time operation, specify  $0$ .

If this block is in a masked subsystem, or other variant subsystem that allows you to switch between continuous operation and discrete operation, promote the sample time parameter. Promoting the sample time parameter ensures correct switching between the continuous and discrete implementations of the block. For more information, see “Promote Parameter to Mask” (Simulink).

## See Also

**Introduced in R2018b**

## SFFM Current Source

Model single-frequency FM current source



## Library

SPICE-Compatible Components/Sources

## Description

The SFFM Current Source block represents a single-frequency current source whose frequency-modulated output current value is independent of the voltage across its terminals. The following equation describes the current through the source as a function of time:

$$I_{out} = IO + IA * \sin((2\pi * FC * Time) + MI * \sin(2\pi * FS * Time))$$

where:

- $IO$  is the **Current offset**, **IO** parameter value.
- $IA$  is the **Current amplitude**, **IA** parameter value.
- $FC$  is the **Carrier frequency**, **FC** parameter value.
- $MI$  is the **Modulation index**, **MI** parameter value.
- $FS$  is the **Signal frequency**, **FS** parameter value.

The block uses a small conductance internally to prevent numerical simulation issues. The conductance connects the + and - ports of the device and has a conductance  $GMIN$ :

- By default,  $GMIN$  matches the **Minimum conductance GMIN** parameter of the SPICE Environment Parameters block, whose default value is 1e-12.

- To change *GMIN*, add a SPICE Environment Parameters block to your model and set the **Minimum conductance GMIN** parameter to the desired value.

## Parameters

### Current offset, IO

The magnitude of the time-independent part of the output current. The default value is 0 A.

### Current amplitude, IA

The magnitude of the sinusoidal part of the output current. The default value is 0 A.

### Carrier frequency, FC

Frequency of the carrier wave. The default value is 0 Hz. The value must be greater than or equal to 0.

### Modulation index, MI

The amount by which the modulated signal varies around its unmodulated level. The default value is 0. The value must be greater than or equal to 0.

### Signal frequency, FS

Frequency of the modulated signal. The default value is 0 Hz. The value must be greater than or equal to 0.

## Ports

The block has the following ports:

+

Positive electrical voltage.

-

Negative electrical voltage.

## See Also

SFFM Voltage Source

## SFFM Voltage Source

Model single-frequency FM voltage source



## Library

SPICE-Compatible Components/Sources

## Description

The SFFM Voltage Source block represents a single-frequency voltage source whose frequency-modulated output voltage value is independent of the current through the source. The following equation describes the output voltage as a function of time:

$$V_{out} = VO + VA * \sin((2\pi * FC * Time) + MI * \sin(2\pi * FS * Time))$$

where:

- $VO$  is the **Voltage offset**, **VO** parameter value.
- $VA$  is the **Voltage amplitude**, **VA** parameter value.
- $FC$  is the **Carrier frequency**, **FC** parameter value.
- $MI$  is the **Modulation index**, **MI** parameter value.
- $FS$  is the **Signal frequency**, **FS** parameter value.

## Parameters

### Voltage offset, $VO$

The magnitude of the time-independent part of the output voltage. The default value is 0 V.

**Voltage amplitude, VA**

The magnitude of the sinusoidal part of the output voltage. The default value is 0 V.

**Carrier frequency, FC**

Frequency of the carrier wave. The default value is 0 Hz. The value must be greater than or equal to 0.

**Modulation index, MI**

The amount by which the modulated signal varies around its unmodulated level. The default value is 0. The value must be greater than or equal to 0.

**Signal frequency, FS**

Frequency of the modulated signal. The default value is 0 Hz. The value must be greater than or equal to 0.

**Ports**

The block has the following ports:

+

Positive electrical voltage.

-

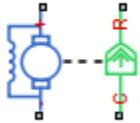
Negative electrical voltage.

**See Also**

SFFM Current Source

## Shunt Motor

Model electrical and torque characteristics of shunt motor

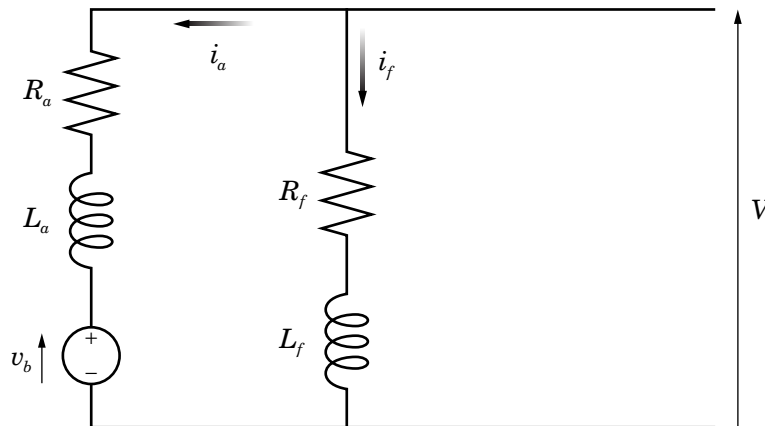


## Library

Rotational Actuators

## Description

The Shunt Motor block represents the electrical and torque characteristics of a shunt motor using the following equivalent circuit model.



When you set the **Model parameterization** parameter to **By equivalent circuit parameters**, you specify the equivalent circuit parameters for this model:

- $R_a$  — **Armature resistance**
- $L_a$  — **Armature inductance**
- $R_f$  — **Field winding resistance**
- $L_f$  — **Field winding inductance**

The Shunt Motor block computes the motor torque as follows:

- 1 The magnetic field in the motor induces the following back emf  $v_b$  in the armature:

$$v_b = L_{af} i_f \omega$$

where  $L_{af}$  is a constant of proportionality and  $\omega$  is the angular velocity.

- 2 The mechanical power is equal to the power reacted by the back emf:

$$P = v_b i_a = L_{af} i_f i_a \omega$$

- 3 The motor torque is:

$$T = P / \omega = L_{af} i_f i_a$$

The torque-speed characteristic for the Shunt Motor block model is related to the parameters in the preceding figure. When you set the **Model parameterization** parameter to **By rated power, rated speed & no-load speed**, the block solves for the equivalent circuit parameters as follows:

- 1 For the steady-state torque-speed relationship,  $L$  has no effect.
- 2 Sum the voltages around the loop:

$$V = i_a R_a + L_{af} i_f \omega$$

$$V = i_f R_f$$

- 3 Solve the preceding equations for  $i_a$  and  $i_f$ :

$$i_f = \frac{V}{R_f}$$

$$i_a = \frac{V}{R_a} \left( 1 - \frac{L_{af} \omega}{R_f} \right)$$

- 4 Substitute these values of  $i_a$  and  $i_f$  into the equation for torque:

$$T = \frac{L_{af}}{R_a R_f} \left( 1 - \frac{L_{af} \omega}{R_f} \right) V^2$$

The block uses the rated speed and power to calculate the rated torque. The block uses the rated torque and no-load speed values to get one equation that relates  $R_a$  and  $L_{af}/R_f$ . It uses the no-load speed at zero torque to get a second equation that relates these two quantities. Then, it solves for  $R_a$  and  $L_{af}/R_f$ .

The block models motor inertia  $J$  and damping  $B$  for all values of the **Model parameterization** parameter. The output torque is:

$$T_{load} = \frac{L_{af}}{R_a R_f} \left( 1 - \frac{L_{af} \omega}{R_f} \right) V^2 - J \dot{\omega} - B \omega$$

The block produces a positive torque acting from the mechanical C to R ports.

## Thermal Ports

The block has two optional thermal ports, one per winding, hidden by default. To expose the thermal ports, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal ports on the block icon, and adds the **Temperature Dependence** and **Thermal Port** tabs to the block dialog box. These tabs are described further on this reference page.

Use the thermal ports to simulate the effects of copper resistance losses that convert electrical power to heat. For more information on using thermal ports in actuator blocks, see “Simulating Thermal Effects in Rotational and Translational Actuators”.

## Parameters

- “Electrical Torque Tab” on page 1-1403
- “Mechanical Tab” on page 1-1404



- “Temperature Dependence Tab” on page 1-1404
- “Thermal Port Tab” on page 1-1405

## Electrical Torque Tab

### Model parameterization

Select one of the following methods for block parameterization:

- By equivalent circuit parameters — Provide electrical parameters for an equivalent circuit model of the motor. This is the default method.
- By rated power, rated speed & no-load speed — Provide power and speed parameters that the block converts to an equivalent circuit model of the motor.

### Armature resistance

Resistance of the armature. This parameter is only visible when you select By equivalent circuit parameters for the **Model parameterization** parameter. The default value is 110  $\Omega$ .

### Field winding resistance

Resistance of the field winding. This parameter is only visible when you select By equivalent circuit parameters for the **Model parameterization** parameter. The default value is 2.5e+03  $\Omega$ .

### Back-emf constant

The ratio of the voltage generated by the motor to the motor speed. The default value is 5.11 s\*V/rad/A.

### Armature inductance

Inductance of the armature. If you do not have information about this inductance, set the value of this parameter to a small, nonzero number. The default value is 0.1 H. The value can be zero.

### Field winding inductance

Inductance of the field winding. If you do not have information about this inductance, set the value of this parameter to a small, nonzero number. The default value is 0.1 H. The value can be zero.

### **No-load speed**

Speed of the motor when no load is applied. This parameter is only visible when you select **By rated power, rated speed & no-load speed** for the **Model parameterization** parameter. The default value is  $4.6e+03$  rpm.

### **Rated speed (at rated load)**

Motor speed at the rated load. This parameter is only visible when you select **By rated power, rated speed & no-load speed** for the **Model parameterization** parameter. The default value is  $4e+03$  rpm.

### **Rated load (mechanical power)**

The mechanical load for which the motor is rated to operate. This parameter is only visible when you select **By rated power, rated speed & no-load speed** for the **Model parameterization** parameter. The default value is 50 W.

### **Rated DC supply voltage**

The voltage at which the motor is rated to operate. This parameter is only visible when you select **By rated power, rated speed & no-load speed** for the **Model parameterization** parameter. The default value is 220 V.

### **Starting current at rated DC supply voltage**

The initial current when starting the motor with the rated DC supply voltage. This parameter is only visible when you select **By rated power, rated speed & no-load speed** for the **Model parameterization** parameter. The default value is 2.09 A.

## **Mechanical Tab**

### **Rotor inertia**

Rotor inertia. The default value is  $2e-04$  kg\*m<sup>2</sup>. The value can be zero.

### **Rotor damping**

Rotor damping. The default value is  $1e-06$  N\*m/(rad/s). The value can be zero.

### **Initial rotor speed**

Speed of the rotor at the start of the simulation. The default value is 0 rpm.

## **Temperature Dependence Tab**

This tab appears only for blocks with exposed thermal ports. For more information, see “Thermal Ports” on page 1-1402.

**Resistance temperature coefficients, [ $\alpha_f$   $\alpha_a$ ]**

A 1 by 2 row vector defining the coefficient  $\alpha$  in the equation relating resistance to temperature, as described in “Thermal Model for Actuator Blocks”. The first element corresponds to the field winding, and the second to the armature. The default value is for copper, and is [ 0.00393 0.00393 ] 1/K.

**Measurement temperature**

The temperature for which motor parameters are defined. The default value is 25 °C.

**Thermal Port Tab**

This tab appears only for blocks with exposed thermal ports. For more information, see “Thermal Ports” on page 1-1402.

**Thermal masses, [Mf Ma]**

A 1 by 2 row vector defining the thermal mass for the field and armature windings. The thermal mass is the energy required to raise the temperature by one degree. The default value is [ 100 100 ] J/K.

**Initial temperatures, [Tf Ta]**

A 1 by 2 row vector defining the temperature of the field and armature thermal ports at the start of simulation. The default value is [ 25 25 ] °C.

**Ports**

The block has the following ports:

- +  
Positive electrical input.
- Negative electrical input.
- C  
Mechanical rotational conserving port.
- R  
Mechanical rotational conserving port.

Hf

Field winding thermal port. For more information, see “Thermal Ports” on page 1-1402.

Ha

Armature winding thermal port. For more information, see “Thermal Ports” on page 1-1402.

## References

[1] Bolton, W. *Mechatronics: Electronic Control Systems in Mechanical and Electrical Engineering*, 3rd edition Pearson Education, 2004.

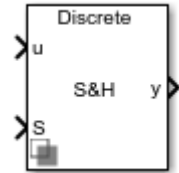
## See Also

DC Motor | Induction Motor | Servomotor | Universal Motor

# Signal Sample and Hold

Sample and hold the input signal

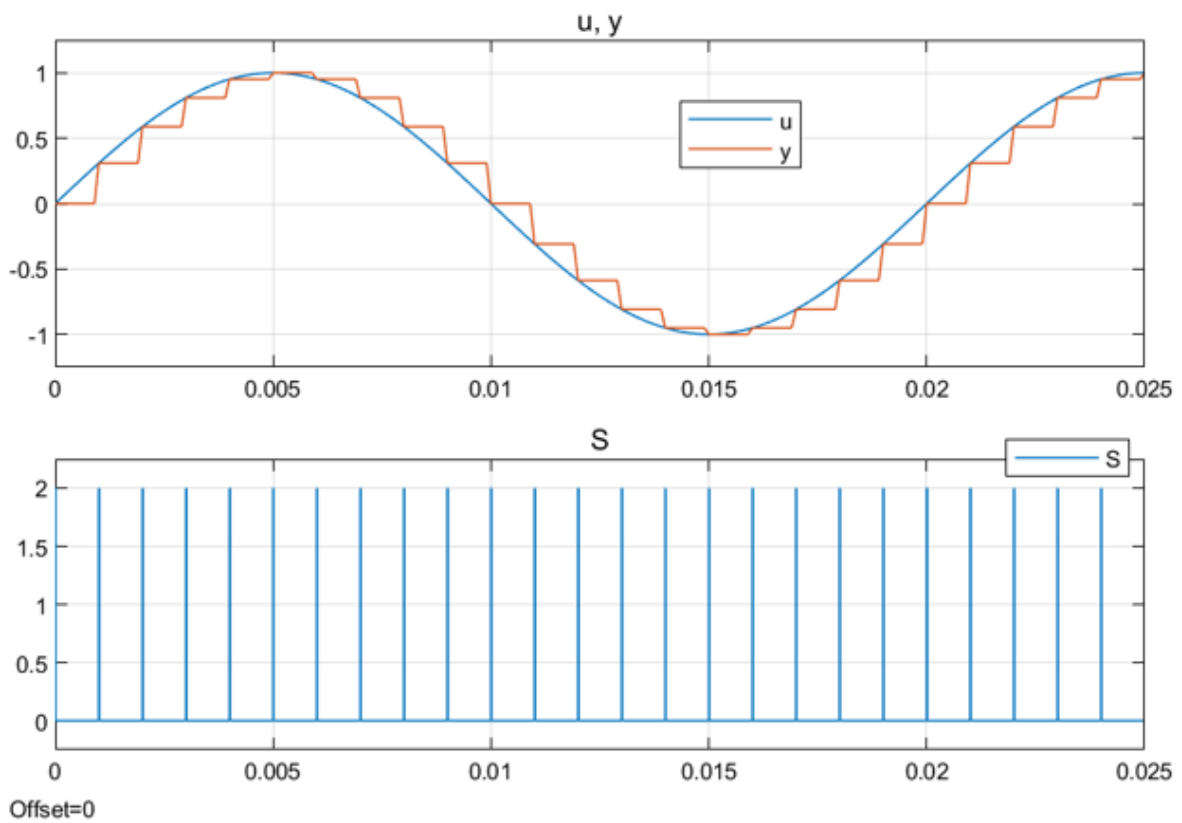
**Library:** Simscape / Electrical / Power Systems / Control /  
General Control



## Description

The Signal Sample and Hold block implements a signal sample and hold in either discrete or continuous time.

When input  $S$  is `true`, output  $y$  is equal to input  $u$ . When input  $S$  is `false`, the block holds the output until  $S$  becomes `true` again.



## Ports

### Input

**u** — Input signal

scalar | vector

Input signal.

Data Types: single | double

**S — Control signal**

0 | 1

Sample pulse of 0 for false or 1 for true.

Data Types: Boolean

**Output****y — Output signal**

scalar | vector

Output signal.

Data Types: single | double

**Parameters****Initial condition — Initial output value**

scalar | vector

Specify initial condition. The value must be a scalar or a vector of the same size as the input signal.

**Sample time (-1 for inherited) — Block sample time**

-1 (default) | 0 | positive scalar

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For inherited discrete-time operation, specify -1. For discrete-time operation, specify a positive integer. For continuous-time operation, specify 0.

If this block is in a masked subsystem, or other variant subsystem that allows you to switch between continuous operation and discrete operation, promote the sample time parameter. Promoting the sample time parameter ensures correct switching between the continuous and discrete implementations of the block. For more information, see “Promote Parameter to Mask” (Simulink).

## **See Also**

**Introduced in R2018b**



# Sinusoidal Current Source

Model damped sinusoidal current source



## Library

SPICE-Compatible Components/Sources

## Description

The Sinusoidal Current Source block represents a damped sinusoidal current source whose output current is independent of the voltage across the terminals of the source. The following equations describe the current through the source as a function of time:

$$I_{out}(Time \leq TD) = IO$$

$$I_{out}(Time > TD) = IO + IA * e^{-(Time-TD)*DF} * \sin(2\pi * FREQ * (Time - TD))$$

where:

- $IO$  is the **Current offset**, **IO** parameter value.
- $IA$  is the **Sinusoidal amplitude**, **IA** parameter value.
- $FREQ$  is the **Sinusoidal frequency**, **FREQ** parameter value.
- $TD$  is the **Time delay**, **TD** parameter value.
- $DF$  is the **Damping factor**, **DF** parameter value.

The block uses a small conductance internally to prevent numerical simulation issues. The conductance connects the + and - ports of the device and has a conductance  $GMIN$ :

- By default,  $GMIN$  matches the **Minimum conductance GMIN** parameter of the SPICE Environment Parameters block, whose default value is 1e-12.

- To change *GMIN*, add a SPICE Environment Parameters block to your model and set the **Minimum conductance GMIN** parameter to the desired value.

## Parameters

### Current offset, **I0**

The magnitude of the time-independent part of the output current. The default value is 0 A.

### Sinusoidal amplitude, **IA**

The magnitude of the sinusoidal part of the output current. The default value is 0 A.

### Sinusoidal frequency, **FREQ**

The frequency of the output sine wave. The default value is 1e+06 Hz. The value can be less than 0.

### Time delay, **TD**

The time at which the sine wave first starts. The default value is 0 s. The value can be less than 0.

### Damping factor, **DF**

The amount by which to amplify or reduce the exponential damping term that multiplies the sine wave to produce the output current. The default value is 0 1/s. The value must be greater than or equal to 0.

## Ports

The block has the following ports:

+

Positive electrical voltage.

-

Negative electrical voltage.

## **See Also**

Sinusoidal Voltage Source

## Sinusoidal Voltage Source

Model damped sinusoidal voltage source



### Library

SPICE-Compatible Components/Sources

### Description

The Sinusoidal Voltage Source block represents a damped sinusoidal voltage source whose output voltage is independent of the current through the source. The following equations describe the output as a function of time:

$$V_{out}(Time \leq TD) = VO$$

$$V_{out}(Time > TD) = VO + VA * e^{-(Time-TD)*DF} * \sin(2\pi * FREQ * (Time - TD))$$

where:

- $VO$  is the **Voltage offset**, **VO** parameter value.
- $VA$  is the **Sinusoidal amplitude**, **VA** parameter value.
- $FREQ$  is the **Sinusoidal frequency**, **FREQ** parameter value.
- $TD$  is the **Time delay**, **TD** parameter value.
- $DF$  is the **Damping factor**, **DF** parameter value.

## Parameters

### Voltage offset, $V_0$

The magnitude of the time-independent part of the output voltage. The default value is 0 V.

### Sinusoidal amplitude, $V_A$

The magnitude of the sinusoidal part of the output voltage. The default value is 0 V.

### Sinusoidal frequency, $FREQ$

The frequency of the output sine wave. The default value is 1e+06 Hz. The value can be less than 0.

### Time delay, $TD$

The time at which the sine wave first starts. The default value is 0 s. The value can be less than 0.

### Damping factor, $DF$

The amount by which to amplify or reduce the exponential damping term that multiplies the sine wave to produce the output voltage. The default value is 0 1/s. The value must be greater than or equal to 0.

## Ports

The block has the following ports:

+

Positive electrical voltage.

-

Negative electrical voltage.

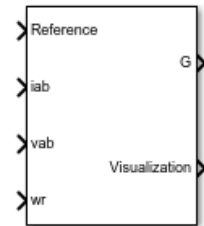
## See Also

Sinusoidal Current Source

# Single-Phase ASM Direct Torque Control

Single-phase ASM Direct Torque Control

**Library:** Simscape / Electrical / Power Systems / Control / ASM Control

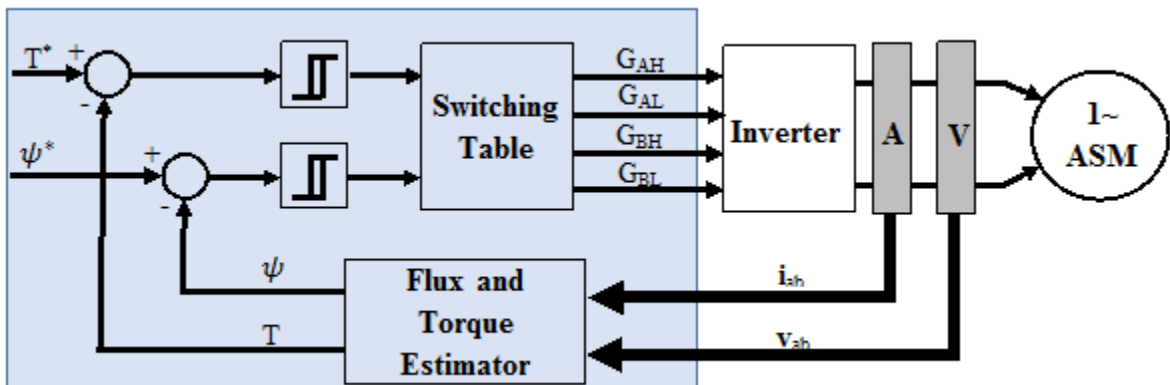


## Description

The Single-Phase ASM Direct Torque Control block implements a single-phase asynchronous machine (ASM) direct torque control structure.

## Equations

This diagram shows the direct torque control architecture for single-phase machines



The torque and flux estimation is based on machine voltage equations. The discrete-time voltage equations using the backward Euler discretization method, are:

$$\psi_a = (v_a - i_a R_{as}) \frac{T_s z}{z-1}$$

$$\psi_b = (v_b - i_b R_{bs}) \frac{T_s z}{z-1}$$

where:

- $R_{as}$  and  $R_{bs}$  are the main winding resistance and the auxiliary winding resistance, respectively.
- $i_a$  and  $i_b$  are the main winding current and the auxiliary winding current, respectively.
- $v_a$  and  $v_b$  are the main and auxiliary winding voltage, respectively.
- $\psi_a$  and  $\psi_b$  are the main and auxiliary winding flux, respectively.

The torque and flux are obtained from:

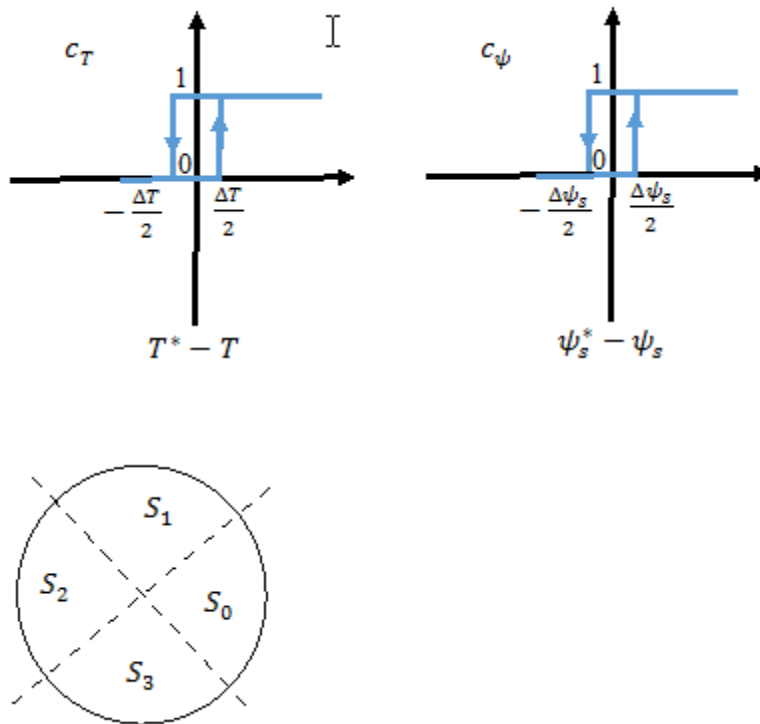
$$T = p(a\psi_a i_b - \frac{1}{a}\psi_b i_a)$$

$$\psi_s = \sqrt{\psi_a^2 + \psi_b^2}$$

where:

- $p$  is the number of pole pairs.
- $a$  is the auxiliary to main windings turn ratio.

Employing simple hysteresis comparators detects the status of flux and torque errors. The following figures illustrate the hysteresis comparators and the switching sectors.



The table shows the optimum switching table (inverter high side).

$c_\psi, c_T, S(\theta)$		$S_0$	$S_1$	$S_2$	$S_3$
$c_\psi = 1$	$c_T = 1$	1, 1	0, 1	0, 0	1, 0
	$c_T = 0$	1, 0	1, 1	0, 1	0, 0
$c_\psi = 0$	$c_T = 1$	0, 1	0, 0	1, 0	1, 1
	$c_T = 0$	0, 0	1, 0	1, 1	0, 1

The torque reference can be provided as an input, or, in the case of speed control, be generated internally using a PI speed controller.

The flux reference is generated internally using:



$$\psi_s^* = \frac{2\pi f_n \psi_n}{p \min(|\omega_r|, \frac{2\pi f_n}{p})}$$

where,

- $\omega_r$  is the rotor angular mechanical speed in rad/s.
- $f_n$  is the rated frequency.
- $\psi_n$  is the rated flux.

## Limitations

- The power inverter dead-times are not considered in this block. For hardware implementation, add the dead-time externally.

## Ports

### Input

#### Reference — Torque or angular velocity reference

scalar

Specify the angular velocity reference in rad/s or the torque reference in Nm.

Data Types: single | double

#### **iab** — Stator currents

scalar

Stator currents for main and auxiliary windings.

Data Types: single | double

#### **vab** — Stator voltages

scalar

Stator voltages for main and auxiliary windings.

Data Types: single | double

## **wr — Rotor angular speed**

scalar

Measure rotor angular speed in rad/s.

Data Types: `single` | `double`

## **Output**

### **G — Inverter gate pulses**

0 | 1

Inverter gate pulses, specified as a Boolean. The dead-time is not considered.

Data Types: `single` | `double`

### **Visualization — Internal signals for visualization**

Reference | Rotor velocity | TqRef | FluxRef | Torque | Flux

Bus containing internal signals for visualization. The signals are:

- Reference torque or speed
- Rotor velocity, in rad/s
- Torque Reference, in Nm
- Flux Reference, in Wb
- Torque, in Nm
- Flux, in Wb

Data Types: `single` | `double` | `bus`

## **Parameters**

### **General**

#### **Control mode — Specify control mode**

Speed control (default) | Torque control

Specify the control mode. The reference input is taken as angular speed, in rad/s, for Speed control and torque, in Nm, for Torque control.

**Rated electrical frequency (Hz) – Specify rated electrical frequency**

60 (default) | positive scalar

Rated electrical frequency, in Hz.

**Number of pole pairs – Specify pole pairs**

2 (default) | positive scalar

Number of pole pairs.

**Main winding stator resistance, Ras (Ohm) – Specify main winding stator resistance**

1.8180 (default) | positive scalar

Resistance of the main winding stator, in Ohms.

**Auxiliary winding stator resistance, Rbs (Ohm) – Specify auxiliary winding stator resistance**

6.4260 (default) | positive scalar

Resistance of the auxiliary winding stator, in Ohms.

**Auxiliary/main windings turn ratio – Specify the auxiliary-to-main windings turn ratio**

1.1 (default) | positive scalar

Auxiliary-to-main windings turns ratio.

**Sample time (-1 for inherited) – Block sample time**

-1 (default) | positive scalar

Time, in s, between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

If this block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly using a positive scalar.

## Outer Loop

This table shows how the visibility of some parameters in the **Outer Loop** tab depends on the option that you choose for the **Control mode** parameter in the **General** tab.

Control mode	Available Parameters
Speed control	Speed controller proportional gain
	Speed controller integral gain
	Speed controller integral anti-windup gain
	Maximum torque (Nm)
	Minimum torque (Nm)
	Rated flux (Wb)
Torque control	Rated flux (Wb)

### Speed controller proportional gain — Specify the speed controller proportional gain

1 (default) | positive scalar

Proportional gain for the PI speed controller.

### Speed controller integral gain — Specify the speed controller integral gain

10 (default) | positive scalar

Integral gain for the PI speed controller.

### Speed controller integral anti-windup gain — Specify the speed controller integral anti-windup gain

1000 (default) | positive scalar

Integral anti-windup gain.

### Maximum torque (Nm) — Specify the maximum torque

5 (default) | positive scalar

Maximum torque used in speed controller saturation, in Nm.

### Minimum torque (Nm) — Specify the minimum torque

-5 (default) | scalar

Minimum torque used in speed controller saturation, in Nm.

**Rated flux (Wb) — Specify the rated flux**

0.5 (default) | positive scalar

Rated flux, in Wb. The rated flux is used to compute the flux reference.

## Inner Loop

**Flux hysteresis band (Wb) — Specify the flux hysteresis band**

0.02 (default) | positive scalar

Flux hysteresis band for the controller, in Wb.

**Torque hysteresis band (Nm) — Specify the torque hysteresis band**

0.1 (default) | positive scalar

Torque hysteresis band for the controller, in Nm.

## References

- [1] Takahashi, I., and T. Noguchi. "A New Quick-Response and High-Efficiency Control Strategy of an Induction Motor." *IEEE Transactions on Industry Applications*. Vol. IA-22, Number 5, 1986, pp. 820 - 827.

## See Also

### Blocks

ASM Direct Torque Control | Single-Phase ASM Field-Oriented Control

### Topics

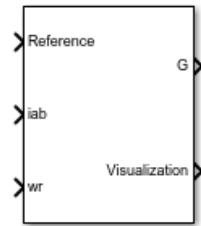
"Single-Phase Asynchronous Machine Direct Torque Control"

**Introduced in R2018b**

# Single-Phase ASM Field-Oriented Control

Single-Phase ASM Field-Oriented Control

**Library:** Simscape / Electrical / Power Systems / Control / ASM Control

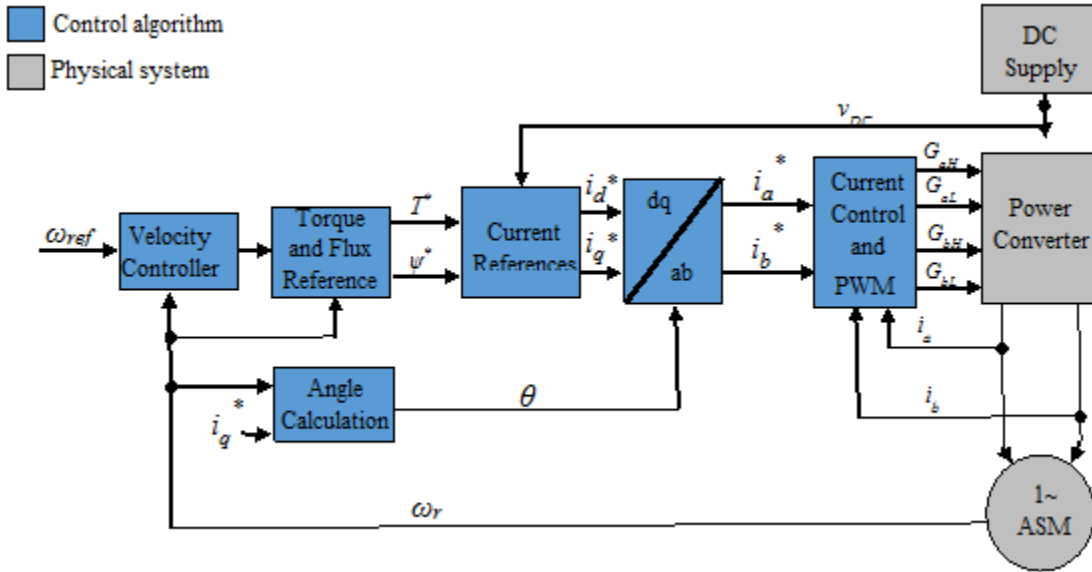


## Description

The Single-Phase ASM Field-Oriented Control block implements a single-phase asynchronous machine (ASM) field-oriented control structure.

## Equations

The field-oriented control architecture for single-phase machines is given as:



The torque reference can be provided as an input, or, in the case of speed control, be generated internally using a PI speed controller. If a speed controller is employed, the torque reference is obtained from:

$$T^* = \left( K_{p_\omega} + K_{i_\omega} \frac{T_s z}{z-1} \right) (\omega_{ref} - \omega_r)$$

where  $\omega_r$  is the rotor angular mechanical speed in rad/s.

The flux reference is generated internally by using:

$$\psi^* = \frac{2\pi f_n \psi_n}{p \min(|\omega_r|, \frac{2\pi f_n}{p})}$$

where:

- $p$  is the number of pole pairs.
- $f_n$  is the rated frequency.

- $\psi_n$  is the rated flux.

Current references are obtained from machine parameters:

$$i_q^* = \frac{(L_{ms} + L_{lar})T^*}{paL_{ms}\psi^*}$$

$$i_d^* = \frac{\psi^*}{a^2 L_{ms}}$$

where:

- $L_{ms}$  is the main winding mutual inductance.
- $L_{lar}$  is the main winding rotor leakage inductance.
- $a$  is the auxiliary-to-main windings turn ratio.

The angle is calculated by solving:

$$\frac{d\theta}{dt} = p\omega_r + \frac{a^3 L_{ms} R_{ar}}{L_{ms} + L_{lar}}$$

The transformation to the stationary reference frame is done by using:

$$\begin{pmatrix} i_a \\ i_b \end{pmatrix} = \begin{pmatrix} a^2 \sin(\theta) & a^2 \cos(\theta) \\ -\cos(\theta) & \sin(\theta) \end{pmatrix} \begin{pmatrix} i_d \\ i_q \end{pmatrix}$$

## Limitations

- The control structure is implemented in a single sample rate.



## Ports

### Input

#### Reference — Torque or angular velocity reference

scalar

Specify the angular velocity reference, in rad/s, or the torque reference, in Nm.

Data Types: `single` | `double`

#### **iab** — Stator currents

scalar

Stator currents for main and auxiliary windings.

Data Types: `single` | `double`

#### **wr** — Rotor angular speed

scalar

Measured rotor angular speed, in rad/s.

Data Types: `single` | `double`

### Output

#### **G** — Inverter gate pulses

0 | 1

Inverter gate pulses, specified as a Boolean. The dead-time is not considered.

Data Types: `single` | `double`

#### **Visualization** — Internal signals for visualization

Reference | Rotor velocity |  $\theta$  | `iab` | `iabRef` | `TqRef` | `FluxRef`

Bus containing internal signals for visualization. The signals are:

- Reference torque or speed
- Rotor velocity, in rad/s

- $\theta$ , in radians
- Stator currents ( $iab$ ), in A
- Stator current reference ( $iabRef$ ), in A
- Torque Reference ( $TqRef$ ), in Nm
- Flux Reference ( $FluxRef$ ), in Wb

Data Types: `single` | `double` | `bus`

## Parameters

### General

#### **Control mode — Specify control mode**

`Speed control` (default) | `Torque control`

Specify the control mode. The reference input is taken as angular speed, in rad/s, for `Speed control` and torque, in Nm for `Torque control`.

#### **Rated electrical frequency (Hz) — Specify rated electrical frequency**

`60` (default) | positive scalar

Rated electrical frequency, in Hz.

#### **Number of pole pairs — Specify pole pairs**

`2` (default) | positive scalar

Number of pole pairs.

#### **Main winding rotor resistance, $R_{ar}$ ' (Ohm) — Specify main winding rotor resistance**

`3.7080` (default) | positive scalar

Main winding rotor resistance, in Ohms.

#### **Main winding rotor leakage inductance, $L_{lar}$ ' (H) — Specify main winding rotor leakage inductance**

`0.0050` (default) | positive scalar

Main winding rotor leakage inductance, in H.

**Main winding mutual inductance,  $L_{ms}$  (H) – Specify main winding mutual inductance**

0.1595 (default) | positive scalar

Main winding mutual inductance, in H.

**Auxiliary/main windings turn ratio – Specify the auxiliary-to-main windings turn ratio**

1.1 (default) | positive scalar

Auxiliary-to-main windings turns ratio.

**Fundamental sample time (s) – Specify the block sample time**

5e-6 (default) | positive scalar

Sample time for the block, in s.

**Control sample time (s) – Specify the control sample time**

5e-5 (default) | positive scalar

Control sample time, in s.

**Outer Loop Tab**

This table shows how the visibility of some parameters in the **Outer Loop** tab depend on the option that you choose for the **Control mode** parameter in the **General** tab.

<b>Control mode</b>	<b>Available Parameters</b>
Speed control	Speed controller proportional gain
	Speed controller integral gain
	Speed controller integral anti-windup gain
	Maximum torque (Nm)
	Minimum torque (Nm)
	Rated flux (Wb)
Torque control	Rated flux (Wb)

**Speed controller proportional gain – Specify the speed controller proportional gain**

1 (default) | positive scalar

Proportional gain for the PI speed controller.

**Speed controller integral gain — Specify the speed controller integral gain**

10 (default) | positive scalar

Integral gain for the PI speed controller.

**Speed controller integral anti-windup gain — Specify the speed controller integral anti-windup gain**

1000 (default) | positive scalar

Integral anti-windup gain.

**Maximum torque (Nm) — Specify the maximum torque**

5 (default) | positive scalar

Maximum torque used in speed controller saturation, in Nm.

**Minimum torque (Nm) — Specify the minimum torque**

-5 (default) | scalar

Minimum torque used in speed controller saturation, in Nm.

**Rated flux (Wb) — Specify the rated flux**

0.5 (default) | positive scalar

Rated flux, in Wb. The rated flux is used to compute the flux reference.

## Inner Loop

This table shows how the visibility of some parameters in the **Inner Loop** tab depend on the option that you choose for the **Control type** parameter.

Control type	Available Parameters
Hysteresis control	Current hysteresis band (A)
PI control	A-phase current proportional gain
	A-phase current integral gain
	A-phase current anti-windup gain
	B-phase current proportional gain

Control type	Available Parameters
	B-phase current integral gain
	B-phase current anti-windup gain

**Control type — Specify the control type**

Hysteresis control (default) | PI control

Control type for current control loop.

**Current hysteresis band (A) — Specify the current hysteresis band**

0.5 (default) | positive scalar

Current hysteresis band for the controller, in A.

**A-phase current proportional gain — Main winding current proportional gain**

1 (default) | positive scalar

Proportional gain for the A-phase (main winding) controller current.

**A-phase current integral gain — Main winding current integral gain**

500 (default)

Integral gain for the A-phase (main winding) controller current.

**A-phase current anti-windup gain — Main winding current anti-windup gain**

1000 (default)

Anti-windup gain for the A-phase (main winding) controller current.

**B-phase current proportional gain — Auxiliary winding current proportional gain**

1 (default)

Proportional gain for the B-phase (auxiliary winding) controller current.

**B-phase current integral gain — Auxiliary winding current integral gain**

500 (default)

Integral gain for the B-phase (auxiliary winding) controller current.

**B-phase current anti-windup gain — Auxiliary winding current anti-windup gain**

1000 (default)

Anti-windup gain for the B-phase (auxiliary winding) controller current.

**References**

- [1] Correa, M. B. R., Jacobina, C. B., Lima, A. M. N., Da Silva, E. R. C. "Field Oriented Control of a Single-Phase Induction Motor Drive." *PESC 98 Record. 29th Annual IEEE Power Electronics Specialists Conference*. Vol. 2, 1998, pp. 990 - 996.

**See Also**

**Blocks**

ASM Field-Oriented Control | Single-Phase ASM Direct Torque Control

**Topics**

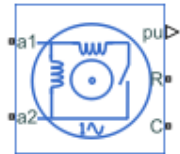
"Single-Phase Asynchronous Machine Field-Oriented Control"

**Introduced in R2018b**

# Single-Phase Asynchronous Machine (fundamental)

Single-phase asynchronous machine with fundamental parameterization

**Library:** Simscape / Electrical / Power Systems / Machines / Asynchronous Machine (Single Phase)



## Description

The Single-Phase Asynchronous Machine (fundamental) block represents a single-phase asynchronous machine with a squirrel cage rotor that you parameterize using fundamental parameters.

Each of four model variants for the Single-Phase Asynchronous Machine (fundamental) block corresponds to a **Block choice** option. To access the block choices, in the model window, right-click the block, and then use either of these methods:

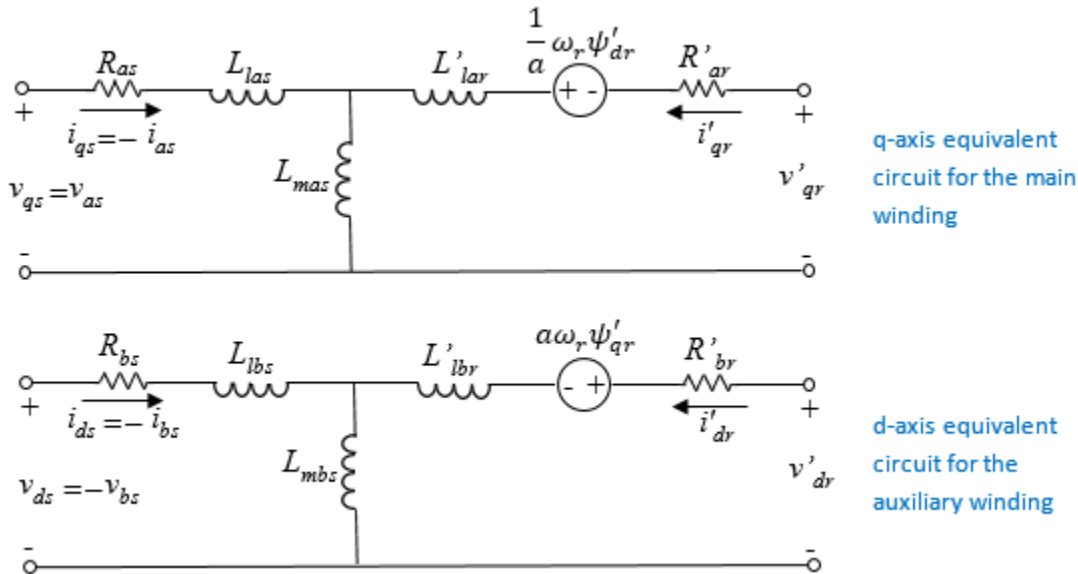
- From the context menu, select **Simscape > Block choices**.
- On the Simulink Editor menu bar, select **View > Property Inspector**. In the **Property Inspector** window, click the value of the **Block choice**.

The model variants are:

- Split-phase
- Capacitor-start
- Capacitor-start-capacitor-run
- Main and auxiliary windings

The single phase asynchronous machine consists of a squirrel cage rotor and two stator windings, the main and the auxiliary windings. The auxiliary winding is typically only active during startup. However, to improve performance, the auxiliary winding can be

active during running for low-power applications. The figure shows the equivalent  $d$ - and  $q$ -axis circuits for the main and auxiliary windings.



The table defines the variables.

Variable	Definition
$v_{qs}v_{ds}$	Stator voltages in the $d$ - $q$ representation
$i_{qs}i_{ds}$	Stator currents in the $d$ - $q$ representation
$\Psi_{qr}\Psi_{dr}$	Rotor fluxes in the $d$ - $q$ representation
$\omega$	Rotor electrical speed
$a$	Auxiliary/main windings turn ratio
$R_{as}$	Main winding stator resistance
$L_{as}$	Main winding stator leakage inductance
$R'_{ar}$	Main winding rotor resistance
$L'_{lar}$	Main winding rotor leakage inductance
$L_{mas}$	Main winding mutual inductance



Variable	Definition
$R_{bs}$	Auxiliary winding stator resistance
$R_{lbs}$	Auxiliary winding stator leakage inductance
$R'_{br}$	Auxiliary winding rotor resistance
$L'_{lbr}$	Auxiliary winding rotor leakage inductance
$L_{mbs}$	Auxiliary winding mutual inductance

## Equations

These transformations reduce the rotor resistance, rotor leakage inductance, and stator mutual inductance to a single set of values by defining the equivalent circuits with respect to the main winding:

$$L_{ms} = L_{mas} = \frac{1}{a^2} L_{mbs}$$

$$R'_{ar} = \frac{1}{a^2} R'_{br}$$

$$L'_{lar} = \frac{1}{a^2} L'_{lbr}$$

The voltage equations for the stator and rotor are:

$$v_{qs} = R_{as} i_{qs} + \frac{d\psi_{qs}}{dt}$$

$$v_{ds} = R_{bs} i_{ds} + \frac{d\psi_{ds}}{dt}$$

$$0 = R'_{ar} i'_{qr} + \frac{d\psi'_{qr}}{dt} - \frac{1}{a} \omega_r \psi'_{dr}$$

$$0 = a^2 R'_{ar} i'_{dr} + \frac{d\Psi'_{dr}}{dt} + a\omega_r \Psi'_{qr}$$

$$\Psi_{qs} = (L_{las} + L_{ms}) i_{qs} + L_{ms} i'_{qr}$$

$$\Psi_{ds} = (L_{lbs} + a^2 L_{ms}) i_{ds} + a^2 L_{ms} i'_{dr}$$

$$\Psi'_{qr} = (L'_{lar} + L_{ms}) i'_{qr} + L_{ms} i_{qs}$$

$$\Psi'_{dr} = a^2 (L'_{lar} + L_{ms}) i'_{dr} + a^2 L_{ms} i_{ds}$$

The expression for the electromagnetic torque,  $T$ , is obtained by applying the principle of virtual displacement [1].

$$T = p \left( a \Psi'_{qr} i'_{dr} - \frac{1}{a} \Psi'_{dr} i'_{qr} \right)$$

The mechanical equation is

$$J \frac{d\omega_m}{dt} = T - T_L - B_m \omega_m,$$

where  $\omega_m$  is the rotor angular velocity.

In split-phase machines, the auxiliary winding is displaced at 90 electrical degrees from the main winding and operates only until the speed reaches the disconnection speed, which is typically 70 to 80 percent of rated speed. In this configuration, the auxiliary winding has high resistance and small reactance compared to the main winding. The resulting phase difference makes the machine behave like a two-phase machine.

The capacitor-start machine is a type of split-phase machine that uses a capacitor in series with the auxiliary winding to start the machine. In this configuration, auxiliary and main windings have the same number of turns. The value of the capacitor ensures that the current in the auxiliary coil leads the current in the main winding by approximately 80 electrical degrees.

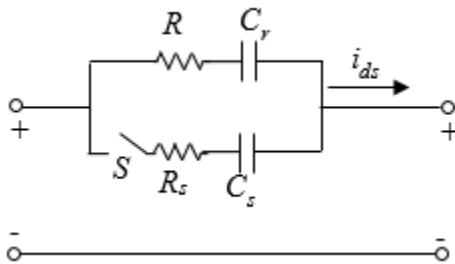
The capacitor-start voltage is:

$$v_c = R_s i_{ds} + \int \frac{1}{C_s} \omega_r i_{ds}.$$

When a capacitor is connected in series with the auxiliary winding, the voltage equation for the  $d$ -axis is

$$v_{ds} = R_{bs} i_{ds} + \frac{d\psi_{ds}}{dt} - R_s i_{ds} - \int \frac{1}{C_s} \omega_r i_{ds}.$$

The extension is obtained immediately for capacitor-start-capacitor-run machines. In this configuration, two capacitors are connected in parallel.



The  $d$ -axis voltage after disconnecting the capacitor-start is

$$v_{ds} = R_{bs} i_{ds} + \frac{d\psi_{ds}}{dt} - R_r i_{ds} - \int \frac{1}{C_r} \omega_r i_{ds}.$$

## Display Option

To display the machine per-unit base values in the MATLAB Command Window, right-click the block and, from the **Electrical** menu, select **Display Base Values**.

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Ports

### Output

#### **pu — Machine per-unit measurements**

physical signal | vector

Physical signal vector port associated with the machine per-unit measurements. The vector elements are:

- pu\_torque
- pu\_velocity
- pu\_vds
- pu\_vqs
- pu\_v0s = 0
- pu\_ids
- pu\_iqs
- pu\_i0s = 0

### Conserving

#### **a1 — Main winding positive terminal**

electrical | scalar

Electrical conserving port associated with the main winding positive terminal.

**a2 — Main winding negative terminal**

electrical | scalar

Electrical conserving port associated with the main winding negative terminal.

**b1 — Auxiliary winding positive terminal**

electrical | scalar

Electrical conserving port associated with the auxiliary winding positive terminal.

**Dependencies**

This port is only visible if you set the **Block choice** to `Main` and `auxiliary` windings.

**b2 — Auxiliary winding negative terminal**

electrical | scalar

Electrical conserving port associated with the auxiliary winding negative terminal.

**Dependencies**

This port is only visible if you set the **Block choice** to `Main` and `auxiliary` windings.

**R — Machine rotor**

mechanical rotational

Mechanical rotational conserving port associated with the machine rotor.

**C — Machine case**

mechanical rotational

Mechanical rotational conserving port associated with the machine case.

## Parameters

### Main

**Rated apparent power — Rated apparent power**

180 V\*A (default) | positive scalar

Rated apparent power of the asynchronous machine.

**Rated voltage — Rated voltage**

220 V (default) | positive scalar

RMS line-line voltage.

**Rated electrical frequency — Rated electrical frequency**

60 Hz (default) | positive scalar

Nominal electrical frequency corresponding to the rated apparent power.

**Number of pole pairs — Number of pole pairs**

2 (default) | positive integer

Number of machine pole pairs. The value is used as a back-electromotive force constant.

## Electrical

**Main winding stator resistance, Ras (pu) — Main winding stator resistance**

0.0135 (default) | positive scalar

Per-unit stator resistance for the main winding.

**Main winding stator leakage inductance, Llas (pu) — Main winding stator leakage inductance**

0.0188 (default) | positive scalar

Per-unit leakage inductance for the main winding.

**Main winding rotor resistance, Rar' (pu) — Main winding rotor resistance**

0.0276 (default) | positive scalar

Per-unit rotor resistance for the main winding.

**Main winding rotor leakage inductance, Llar' (pu) — Main winding rotor leakage inductance**

0.0140 (default) | positive scalar

Per-unit rotor leakage inductance for the main winding.

**Main winding mutual inductance,  $L_{ms}$  (pu) – Main winding mutual inductance**

0.4472 (default) | positive scalar

Per-unit mutual inductance for the main winding.

**Auxiliary winding stator resistance,  $R_{bs}$  (pu) – Auxiliary winding stator resistance**

0.0478 (default) | positive scalar

Per-unit stator resistance for the auxiliary winding.

**Auxiliary winding stator leakage inductance,  $L_{lbs}$  (pu) – Auxiliary winding stator leakage inductance**

0.0216 (default) | positive scalar

Per-unit stator leakage inductance for the auxiliary winding.

**Auxiliary/main windings turn ratio – Turn ratio**

1.1 (default) | positive scalar

Winding ratio between the main and auxiliary winding.

**Capacitor-start resistance,  $R_s$  (pu) – Capacitor-start resistance**

0.0223 (default) | positive scalar

Per-unit capacitor-start resistance.

**Dependencies**This parameter is exposed when **Block choice** is set to Capacitor\_start or Capacitor-start-capacitor-run.**Capacitor-start capacitance,  $C_s$  (pu) – Capacitor-start capacitance**

9.1232 (default) | positive scalar

Per-unit capacitor-start capacitance.

**Dependencies**This parameter is exposed when **Block choice** is set to Capacitor\_start or Capacitor-start-capacitor-run.

## **Capacitor-run resistance, Rr (pu) — Capacitor-run resistance**

0.0669 (default) | positive scalar

Per-unit capacitor-run resistance.

### **Dependencies**

This parameter is exposed when **Block choice** is set to Capacitor\_start or Capacitor-start-capacitor-run.

## **Capacitor-run resistance, Cr (pu) — Capacitor-run capacitance**

0.000015 (default) | positive scalar

Per-unit capacitor-run capacitance.

### **Dependencies**

This parameter is exposed when **Block choice** is set to Capacitor-start-capacitor-run.

## **Disconnection speed (% of rated speed) — Disconnection speed**

80 (default) | scalar in range [0,100]

Speed when capacitor is disconnected, expressed as a percentage of rated speed. The value must be between 0 and 100, inclusive.

### **Dependencies**

This parameter is hidden when **Block choice** is set to Main and auxiliary windings.

## **Mechanical**

### **Rotor inertia — Rotor inertia**

0.01 kg\*m<sup>2</sup> (default) | positive scalar

Inertia of the rotor.

### **Rotor damping — Rotor damping**

0 N\*m/(rad/s) (default) | scalar

Damping of the rotor.



## References

- [1] Krause, P. C. "Simulation of Unsymmetrical 2-Phase Induction Machines." *IEEE Transactions on Power Apparatus and Systems*. Vol 84, Number 11, 1965, pp. 1025-1037.

## See Also

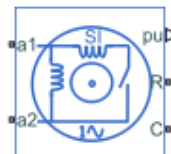
Single-Phase Asynchronous Machine (fundamental, SI) | Single-Phase ASM Field-Oriented Control | Single-Phase ASM Direct Torque Control

**Introduced in R2018b**

## Single-Phase Asynchronous Machine (fundamental, SI)

Single-phase asynchronous machine with fundamental parameterization in SI units

**Library:** Simscape / Electrical / Power Systems / Machines / Asynchronous Machine (Single Phase)



### Description

The Single-Phase Asynchronous Machine (fundamental, SI) block represents a single-phase asynchronous machine with a squirrel cage rotor that you parameterize using SI fundamental parameters.

Each of four model variants for the Single-Phase Asynchronous Machine (fundamental,SI) block corresponds to a **Block choice** option. To access the block choices, in the model window, right-click the block, and then use either of these methods:

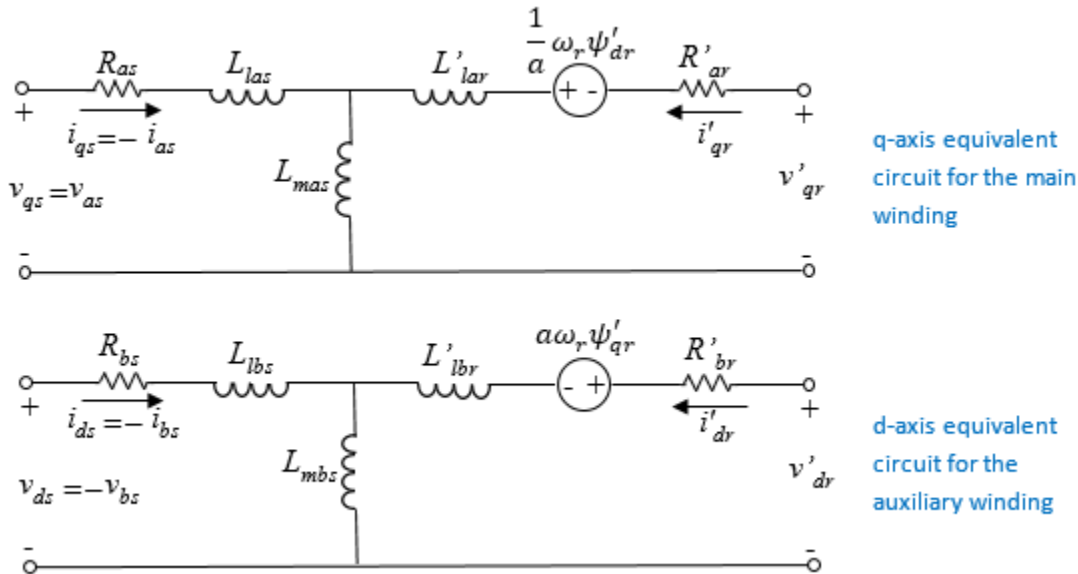
- From the context menu, select **Simscape > Block choices**.
- On the Simulink Editor menu bar, select **View > Property Inspector**. In the **Property Inspector** window, click the value of the **Block choice**.

The model variants are:

- Split-phase
- Capacitor-start
- Capacitor-start-capacitor-run
- Main and auxiliary windings

The single phase asynchronous machine consists of a squirrel cage rotor and two stator windings, the main and the auxiliary windings. The auxiliary winding is typically only active during startup. However, to improve performance, the auxiliary winding can be

active during running for low-power applications. The figure shows the equivalent  $d$ - and  $q$ -axis circuits for the main and auxiliary windings.



The table defines the variables.

Variable	Definition
$v_{qs} v_{ds}$	Stator voltages in the $d$ - $q$ representation
$i_{qs} i_{ds}$	Stator currents in the $d$ - $q$ representation
$\Psi_{qr} \Psi_{dr}$	Rotor fluxes in the $d$ - $q$ representation
$\omega$	Rotor electrical speed
$a$	Auxiliary/main windings turn ratio
$R_{as}$	Main winding stator resistance
$L_{as}$	Main winding stator leakage inductance
$R'_{ar}$	Main winding rotor resistance
$L'_{lar}$	Main winding rotor leakage inductance
$L_{mas}$	Main winding mutual inductance

Variable	Definition
$R_{bs}$	Auxiliary winding stator resistance
$R_{lbs}$	Auxiliary winding stator leakage inductance
$R'_{br}$	Auxiliary winding rotor resistance
$L'_{lbr}$	Auxiliary winding rotor leakage inductance
$L_{mbs}$	Auxiliary winding mutual inductance

## Equations

These transformations reduce the rotor resistance, rotor leakage inductance, and stator mutual inductance to a single set of values by defining the equivalent circuits with respect to the main winding:

$$L_{ms} = L_{mas} = \frac{1}{a^2} L_{mbs}$$

$$R'_{ar} = \frac{1}{a^2} R'_{br}$$

$$L'_{lar} = \frac{1}{a^2} L'_{lbr}$$

The voltage equations for the stator and rotor are:

$$v_{qs} = R_{as} i_{qs} + \frac{d\psi_{qs}}{dt}$$

$$v_{ds} = R_{bs} i_{ds} + \frac{d\psi_{ds}}{dt}$$

$$0 = R'_{ar} i'_{qr} + \frac{d\psi'_{qr}}{dt} - \frac{1}{a} \omega_r \psi'_{dr}$$

$$0 = \alpha^2 R_{ar}' i_{dr}' + \frac{d\Psi_{dr}'}{dt} + \alpha \omega_r \Psi_{qr}'$$

$$\Psi_{qs} = (L_{las} + L_{ms}) i_{qs} + L_{ms} i_{qr}'$$

$$\Psi_{ds} = (L_{lbs} + \alpha^2 L_{ms}) i_{ds} + \alpha^2 L_{ms} i_{dr}'$$

$$\Psi_{qr}' = (L'_{lar} + L_{ms}) i_{qr}' + L_{ms} i_{qs}$$

$$\Psi_{dr}' = \alpha^2 (L'_{lar} + L_{ms}) i_{dr}' + \alpha^2 L_{ms} i_{ds}$$

The expression for the electromagnetic torque,  $T$ , is obtained by applying the principle of virtual displacement [1].

$$T = p \left( \alpha \Psi_{qr}' i_{dr}' - \frac{1}{\alpha} \Psi_{dr}' i_{qr}' \right)$$

The mechanical equation is

$$J \frac{d\omega_m}{dt} = T - T_L - B_m \omega_m,$$

where  $\omega_m$  is the rotor angular velocity.

In split-phase machines, the auxiliary winding is displaced at 90 electrical degrees from the main winding and operates only until the speed reaches the disconnection speed, which is typically 70 to 80 percent of rated speed. In this configuration, the auxiliary winding has high resistance and small reactance compared to the main winding. The resulting phase difference makes the machine behave like a two-phase machine.

The capacitor-start machine is a type of split-phase machine that uses a capacitor in series with the auxiliary winding to start the machine. In this configuration, auxiliary and main windings have the same number of turns. The value of the capacitor ensures that the current in the auxiliary coil leads the current in the main winding by approximately 80 electrical degrees.

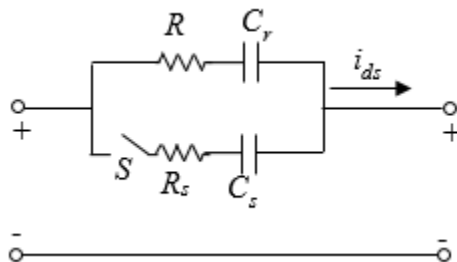
The capacitor-start voltage is:

$$v_c = R_s i_{ds} + \int \frac{1}{C_s} \omega_r i_{ds}.$$

When a capacitor is connected in series with the auxiliary winding, the voltage equation for the  $d$ -axis is

$$v_{ds} = R_{bs} i_{ds} + \frac{d\psi_{ds}}{dt} - R_s i_{ds} - \int \frac{1}{C_s} \omega_r i_{ds}.$$

The extension is obtained immediately for capacitor-start-capacitor-run machines. In this configuration, two capacitors are connected in parallel.



The  $d$ -axis voltage after disconnecting the capacitor-start is

$$v_{ds} = R_{bs} i_{ds} + \frac{d\psi_{ds}}{dt} - R_r i_{ds} - \int \frac{1}{C_r} \omega_r i_{ds}.$$

## Display Option

To display the machine per-unit base values in the MATLAB Command Window, right-click the block and, from the **Electrical** menu, select **Display Base Values**.

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Ports

### Output

#### **pu — Machine per-unit measurements**

physical signal | vector

Physical signal vector port associated with the machine per-unit measurements. The vector elements are:

- pu\_torque
- pu\_velocity
- pu\_vds
- pu\_vqs
- pu\_v0s = 0
- pu\_ids
- pu\_iqs
- pu\_i0s = 0

### Conserving

#### **a1 — Main winding positive terminal**

electrical | scalar

Electrical conserving port associated with the main winding positive terminal.

## **a2 — Main winding negative terminal**

electrical | scalar

Electrical conserving port associated with the main winding negative terminal.

## **b1 — Auxiliary winding positive terminal**

electrical | scalar

Electrical conserving port associated with the auxiliary winding positive terminal.

### **Dependencies**

This port is only visible if you set the **Block choice** to `Main` and `auxiliary windings`.

## **b2 — Auxiliary winding negative terminal**

electrical | scalar

Electrical conserving port associated with the auxiliary winding negative terminal.

### **Dependencies**

This port is only visible if you set the **Block choice** to `Main` and `auxiliary windings`.

## **R — Machine rotor**

mechanical rotational

Mechanical rotational conserving port associated with the machine rotor.

## **C — Machine case**

mechanical rotational

Mechanical rotational conserving port associated with the machine case.

# Parameters

## **Main**

### **Rated apparent power — Rated apparent power**

180 V\*A (default) | positive scalar



Rated apparent power of the asynchronous machine.

**Rated voltage — Rated voltage**

220 V (default) | positive scalar

RMS line-line voltage.

**Rated electrical frequency — Rated electrical frequency**

60 Hz (default) | positive scalar

Nominal electrical frequency corresponding to the rated apparent power.

**Number of pole pairs — Number of pole pairs**

2 (default) | positive integer

Number of machine pole pairs. The value is used as a back-electromotive force constant.

## Electrical

**Main winding stator resistance,  $R_{as}$  — Main winding stator resistance**

1.8180 0hm (default) | positive scalar

Stator resistance for the main winding.

**Main winding stator leakage inductance,  $L_{las}$  — Main winding stator leakage inductance**

0.0067 H (default) | positive scalar

Leakage inductance for the main winding.

**Main winding rotor resistance,  $R_{ar'}$  — Main winding rotor resistance**

3.7080 0hm (default) | positive scalar

Rotor resistance for the main winding.

**Main winding rotor leakage inductance,  $L_{lar'}$  — Main winding rotor leakage inductance**

0.0050 H (default) | positive scalar

Rotor leakage inductance for the main winding.

**Main winding mutual inductance,  $L_{ms}$  — Main winding mutual inductance**

0.1595 H (default) | positive scalar

Mutual inductance for the main winding.

**Auxiliary winding stator resistance,  $R_{bs}$  — Auxiliary winding stator resistance**

6.4260 Ohm (default) | positive scalar

Stator resistance for the auxiliary winding.

**Auxiliary winding stator leakage inductance,  $L_{lbs}$  — Auxiliary winding stator leakage inductance**

0.0077 H (default) | positive scalar

Stator leakage inductance for the auxiliary winding.

**Auxiliary/main windings turn ratio — Windings turn ratio**

1.1 (default) | positive scalar

Winding ratio between the main and auxiliary winding.

**Capacitor-start resistance,  $R_s$  — Capacitor-start resistance**

3 Ohm (default) | positive scalar

Capacitor-start resistance.

**Dependencies**

This parameter is exposed when **Block choice** is set to Capacitor\_start or Capacitor-start-capacitor-run.

**Capacitor-start capacitance,  $C_s$  — Capacitor-start capacitance**

0.00018 F (default) | positive scalar

Capacitor-start capacitance.

**Dependencies**

This parameter is exposed when **Block choice** is set to Capacitor\_start or Capacitor-start-capacitor-run.

**Capacitor-run resistance,  $R_r$  — Capacitor-run resistance**

9 Ohm (default) | positive scalar

Per-unit capacitor-run resistance.

**Dependencies**

This parameter is exposed when **Block choice** is set to Capacitor\_start or Capacitor-start-capacitor-run.

**Capacitor-run capacitance, Cr — Capacitor-run capacitance**

0.000015 F (default) | positive scalar

Capacitor-run capacitance.

**Dependencies**

This parameter is exposed when **Block choice** is set to Capacitor\_start or Capacitor-start-capacitor-run.

**Disconnection speed (% of rated speed) — Disconnection speed**

80 (default) | scalar in range [0,100]

Speed when capacitor is disconnected, expressed as a percentage of rated speed. The value must be between 0 and 100, inclusive.

**Dependencies**

This parameter is hidden when **Block choice** is set to Main and auxiliary windings.

**Mechanical****Rotor inertia — Rotor inertia**

0.01 kg\*m<sup>2</sup> (default) | positive scalar

Rotor inertia.

**Rotor damping — Rotor damping**

0 N\*m/(rad/s) (default) | positive scalar

Rotor damping.

**References**

- [1] Krause, P. C. "Simulation of Unsymmetrical 2-Phase Induction Machines." *IEEE Transactions on Power Apparatus and Systems*. Vol 84, Number 11, 1965, pp. 1025-1037.

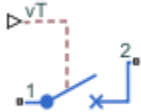
## **See Also**

Single-Phase Asynchronous Machine (fundamental) | Single-Phase ASM Field-Oriented Control | Single-Phase ASM Direct Torque Control

**Introduced in R2018b**

# Single-Phase Circuit Breaker

Single-pole single-throw circuit breaker



## Library

Simscape / Electrical / Power Systems / Switches & Breakers / Fundamental Components

## Description

The Single-Phase Circuit Breaker block models a single-phase circuit breaker that uses an external signal and phase current information to break an electrical circuit.

The table shows how the external signal  $vT$  controls the block behavior.

Condition	Block Behavior	Resistance Parameter Used
$vT < \text{Threshold}$	The breaker is closed. Port 1 connects to port 2.	<b>Closed Resistance</b>
$vT \geq \text{Threshold}$	When the current in port 1 goes through zero, the phase disconnects from port 2. The breaker is open.	<b>Open Conductance</b>

## Parameters

### Closed resistance

Resistance between ports 1 and 2 when the breaker is closed. The default value is 0.001 Ohm.

### Open conductance

Conductance between ports 1 and 2 when the breaker is open. The default value is 1e-6 1/Ohm.

### Threshold

Threshold voltage for the control port vT. The block uses the threshold voltage and the value of vT at the start of the simulation to determine whether the breaker is initially open or closed. When the voltage rises above the threshold, the breaker opens. When the control port voltage falls below the threshold, the breaker closes. The default value is 0.5 V.

## Ports

The block has the following ports:

1

Electrical conserving port

2

Electrical conserving port

vT

Scalar control port, which is either a physical signal or an electrical port.

## See Also

Circuit Breaker | Single-Phase Circuit Breaker (with arc)

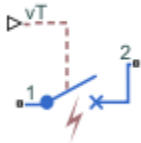
## Topics

“Switch Between Physical Signal and Electrical Ports”

**Introduced in R2013b**

# Single-Phase Circuit Breaker (with arc)

Single-pole single-throw circuit breaker with Mayr arc representation



## Library

Simscape / Electrical / Power Systems / Switches & Breakers / Fundamental Components

## Description

The Single-Phase Circuit Breaker (with arc) block represents a single-phase circuit breaker with Mayr arc representation controlled by an external control signal  $vT$ . If  $vT$  is less than the threshold, then the breaker is closed. If  $vT$  is greater than or equal to the threshold, then the breaker opens with an arc during the current interruption. The external signal can open and close the breaker repeatedly.

The table shows how the external signal  $vT$  controls the block behavior.

Condition	Block Behavior
$vT < \text{Threshold}$	The circuit breaker is closed. Port 1 is connected to port 2.
$vT \geq \text{Threshold}$	The circuit breaker is either opening or open. Port 1 is connected to port 2 via a nonlinear conductance.

The Single-Phase Circuit Breaker (with arc) block has a higher computational overhead than the Single-Phase Circuit Breaker block. If the fidelity of the representation of arc current or voltage is your overriding requirement, use the Single-Phase Circuit Breaker (with arc) block and use a global Simulink variable-step solver. Otherwise, use the Single-Phase Circuit Breaker block.

## Mayr Arc Model Equations

The defining equations for the breaker are

$$x = \ln(g)$$

and

$$i = gv,$$

where:

- $g$  is the arc conductance.
- $x$  is an internal state variable.
- $v$  is the voltage across the breaker.
- $i$  is the current through the breaker.

When the breaker is closed,

$$\frac{dx}{dt} = 0.$$

When the breaker is opening or open,

$$\frac{dx}{dt} = \frac{1}{\tau} \left( \frac{gv^2}{P} - 1 \right),$$

where:

- $\tau$  is the arc time constant.
- $P$  is the cooling power.

## Parameters

### Arc time constant, tau

The default value is  $0.3e-6$  s.



**Cooling power, P**

The default value is 30900 W.

**Initial arc conductance, g0**

Conductance between ports 1 and 2 when the breaker is closed. The default value is  $1e4$  S.

**Threshold**

Threshold voltage for the control port vT. The block uses the threshold voltage and the value of vT at the start of the simulation to determine whether the breaker is initially open or closed. When the voltage rises above the threshold, the breaker opens. When the control port voltage falls below the threshold, the breaker closes. The default value is 0.5 V.

**Ports**

The block has the following ports:

1

Electrical conserving port

2

Electrical conserving port

vT

Scalar control port, which is either a physical signal or an electrical port

**References**

- [1] Schavemaker, P. H., and L. Van der Sluis. "The Arc Model Blockset." *Proceedings of the Second IASTED International Conference POWER AND ENERGY SYSTEMS (EuroPES)*. Crete, Greece, June 25-28, 2002, pp. 644-648.

**See Also**

Circuit Breaker | Single-Phase Circuit Breaker

**Topics**

"Switch Between Physical Signal and Electrical Ports"

**Introduced in R2015b**

# Single-Phase Permanent Magnet Synchronous Motor

Single-phase PMSM

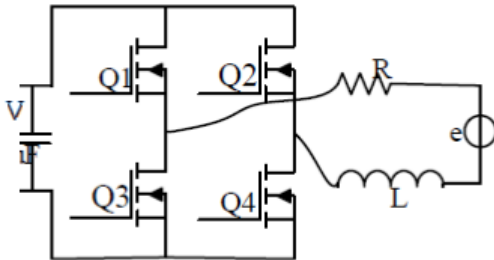
**Library:** Simscape / Electrical / Power Systems / Machines / Permanent Magnet Rotor



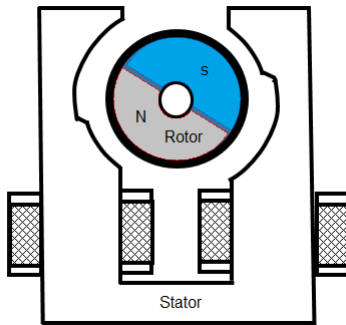
## Description

The Single-Phase Permanent Magnet Synchronous Motor represents a single-phase permanent magnet synchronous motor (PMSM), a type of DC motor that is useful for automation applications.

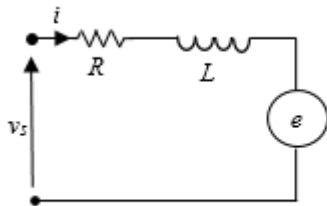
The figure shows the topology of the single-phase PMSM drive.



The figure shows the motor construction with a single pole-pair on the rotor. Single-phase PMSMs are not self-starting unless the air gap is asymmetrical.



The figure shows the equivalent circuit for the Single-Phase Permanent Magnet Synchronous Motor block.



## Equations

The motor voltage equations are

$$v_s = Ri + L \frac{di}{dt} + e$$

and

$$v_s = V_m \sin(\omega_s t + \varepsilon),$$

where:

- $v_s$  is the supply voltage.

- $i$  is the instantaneous motor current.
- $R$  is the resistance of the windings.
- $L$  is the self-inductance of the windings.
- $e$  is the back-electromotive force (BEMF).
- $\omega_s$  is the angular frequency of the supply voltage.
- $\varepsilon$  is the angle of the supply voltage.

The BEMF is

$$e = k_e \omega_r \sin(\theta_r),$$

where:

- $\omega_r$  is the rotor electrical angular velocity.
- $\theta_r$  is the rotor position.
- $k_e$  is the BEMF constant.

Due to the large low-permeability gaps between the stator and rotor, the saturation can be neglected. Therefore, the electric torque equations are

$$T_e = i\psi_m \sin(\theta_r)$$

and

$$\psi_m = \frac{k_e}{p},$$

where:

- $T_e$  is the electric torque.
- $\psi_m$  is the permanent magnet flux linkage.
- $p$  is the number of pole pairs.

The mechanical equation is

$$J_m \frac{d\omega_r}{dt} = T_e - T_L - B_m \omega_r,$$

where:

- $J_m$  is the rotor inertia.
- $T_L$  is the torque load.
- $B_m$  is the friction coefficient.

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Limitations and Assumptions

- The machine air gap is free of saliency effects.
- The stator current has negligible effect on the flux distribution under normal operating conditions.
- The hysteresis, saturation effects, and eddy currents are neglected.

## Ports

### Conserving

#### **R — Machine rotor**

mechanical rotational

Mechanical rotational conserving port associated with the machine rotor.

#### **C — Machine case**

mechanical rotational

Mechanical rotational conserving port associated with the machine case.

**+ – Negative**

electrical

Electrical conserving port associated with the supply positive terminal.

**- – Positive**

electrical

Electrical conserving port associated with the supply negative terminal.

## Parameters

### Main

**Number of pole pairs – Rotor pole pairs**

2 (default) | positive integer

Number of permanent magnet pole pairs on the rotor.

**Permanent magnet flux linkage parameterization – Parameterization method**

Specify flux linkage (default) | Specify back EMF constant

Method for parameterizing the stator.

**Dependencies**

Selecting Specify flux linkage exposes the **Permanent magnet flux linkage** parameter.

Selecting Specify back EMF constant exposes the **Back EMF constant** parameter.

**Permanent magnet flux linkage – Flux linkage**

0.2 Wb (default) | positive scalar

Peak permanent magnet flux linkage.

**Dependencies**

Selecting Specify flux linkage for the **Permanent magnet flux linkage parameterization** parameter exposes the **Permanent magnet flux linkage** parameter.

**Back EMF constant — Back-electromotive force constant**

0.4 V/(rad/s) (default) | positive scalar

Back-electromotive force constant.

**Dependencies**

Selecting Specify back EMF constant for the **Permanent magnet flux linkage parameterization** parameter exposes the **Back EMF constant** parameter.

**Inductance of stator coil — Inductance**

1.15 H (default) | positive scalar

The direct-axis inductance.

**Resistance of stator coil — Resistance**

150 Ohm (default) | positive scalar

Resistance of each of the stator windings.

**Rotor position at standstill due to asymmetric airgap — Rotor angular position at standstill**

45 deg (default) | positive scalar in the interval [0, 360] deg

Rotor angular position at standstill due to asymmetric air gap.

**Mechanical**

**Rotor inertia — Rotor moment of inertia**

1e-6 kg\*m<sup>2</sup> (default) | positive scalar

Inertia of the rotor attached to mechanical translational port **R**.

**Rotor damping — Rotor damping**

0 N\*m/(rad/s) (default) | positive scalar

Damping of the rotor.

**References**

- [1] Ertugrul, N. and C. Doudle. "Dynamic analysis of a single-phase line-starting permanent magnet synchronous motor." *Proceedings of International Conference*



*on Power Electronics, Drives and Energy Systems for Industrial Growth. Vol. 1, 1996, pp. 603–609.*

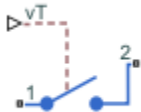
## **See Also**

Brushless DC Motor | Permanent Magnet Synchronous Motor

**Introduced in R2018b**

# Single-Phase Switch

Single-pole single-throw switch



## Library

Simscape / Electrical / Power Systems / Switches & Breakers / Fundamental Components

## Description

The Single-Phase Switch block models a single-pole single-throw switch that uses an external signal to connect port 1 to port 2 via internal resistance.

The table shows how the external signal  $vT$  controls the block behavior.

Condition	Block Behavior	Resistance Parameter Used
$vT \leq \text{Threshold}$	The switch is open. Port 1 is connected to port 2 via large internal resistance.	<b>Open conductance</b>
$vT > \text{Threshold}$	The switch is closed. Port 1 is connected to port 2 via small internal resistance.	<b>Closed resistance</b>

## Parameters

### Closed resistance

Resistance between ports 1 and 2 when the switch is closed. The default value is 0.001 Ohm.

**Open conductance**

Conductance between ports 1 and 2 when the switch is open. The default value is  $1e-6 \text{ 1/}\Omega$ .

**r Threshold**

Threshold voltage for the control port  $v_T$ . When the voltage is above the threshold, the switch is closed. The default value is  $0.5 \text{ V}$ .

**Ports**

The block has the following ports:

1

Electrical conserving port

2

Electrical conserving port

$v_T$

Scalar control port, which is either a physical signal or an electrical port.

**See Also**

[Single-Phase Two-Way Switch](#) | [Switch](#) | [Two-Way Switch](#)

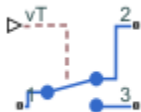
**Topics**

[“Switch Between Physical Signal and Electrical Ports”](#)

**Introduced in R2013b**

# Single-Phase Two-Way Switch

Single-pole double-throw switch



## Library

Simscape / Electrical / Power Systems / Switches & Breakers / Fundamental Components

## Description

The Single-Phase Two-Way Switch block models a single-pole double-throw switch that uses an external signal to connect the port 1 to either of two ports 2 or 3 via internal resistance.

The table shows how the external signal  $vT$  controls the block behavior.

Condition	Block Behavior	Resistance Parameter Used
$vT \leq \text{Threshold}$	Port 1 is connected to port 2 via internal resistance. Port 3 is unconnected.	<b>Open conductance</b> (port 1 to port 3). <b>Closed resistance</b> (port 1 to port 2)
$vT > \text{Threshold}$	Port 1 is connected to port 3 via internal resistance. Port 2 is unconnected.	<b>Open conductance</b> (port 1 to port 2). <b>Closed resistance</b> (port 1 to port 3)

## Parameters

### Closed resistance

Resistance between ports 1 and 3 when the switch is closed. The default value is 0.001 Ohm.

### Open conductance

Conductance between ports 1 and 2 when the switch is open. The default value is 1e-6 1/Ohm.

### Threshold

Threshold voltage for the control port vT. When the voltage is above the threshold, the switch is closed. The default value is 0.5 V.

## Ports

The block has the following ports:

1

Electrical conserving port

2

Electrical conserving port

3

Electrical conserving port

vT

Scalar control port, which is either a physical signal or an electrical port.

## See Also

[Single-Phase Switch](#) | [Switch](#) | [Two-Way Switch](#)

## Topics

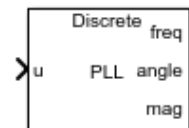
[“Switch Between Physical Signal and Electrical Ports”](#)

**Introduced in R2013b**

## Sinusoidal Measurement (PLL)

Estimate sinusoidal characteristics using a phase-locked loop

**Library:** Simscape / Electrical / Power Systems / Control / Measurements



### Description

The Sinusoidal Measurement (PLL) block estimates the frequency, phase angle, and magnitude of a single-phase sinusoidal signal or individual phases of a multiphase sinusoidal signal. The block uses an enhanced phase-locked loop (PLL) strategy to estimate these sinusoidal characteristics of the input signal.

Use this block in control applications when the frequency, phase angle, or magnitude is required and cannot be measured directly. To provide faster phase locking for balanced three-phase input signals, use the Three-Phase Sinusoidal Measurement (PLL) block.

### Equations

The phase-locked loop generates a sinusoid that approximates the input signal  $u(t)$  with the form:

$$y(t) = A(t) \sin\left(\phi_0 + \int 2\pi f(t) dt\right),$$

where:

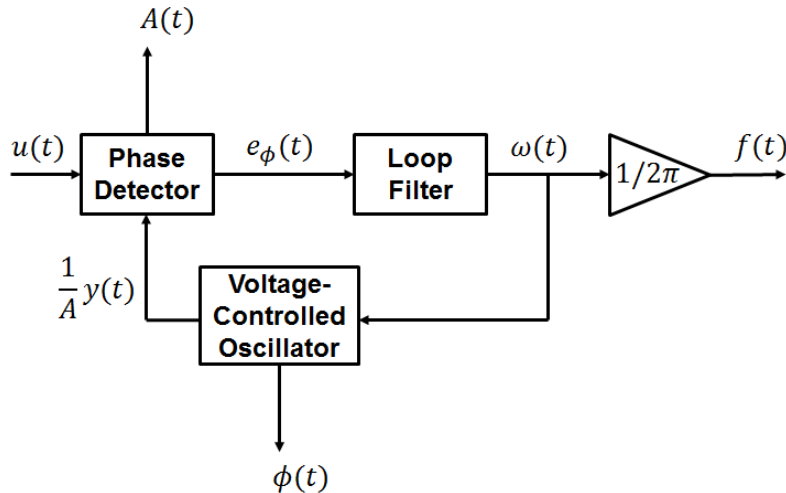
- $y$  is the estimate of the input signal.
- $A$  is the estimate of the amplitude of the input signal.
- $\phi_0$  is the initial phase angle of the input signal.

The estimated phase angle  $\phi$  is the angle of this generated sinusoid:

$$\phi(t) = \phi_0 + \int 2\pi f(t) dt,$$

where  $f$  is the frequency of the sinusoid, and  $\phi_0$  is the initial phase angle.

This diagram shows the overall structure of the phase-locked loop.



In the diagram:

- The phase detector produces an error signal relative to the phase difference  $e_\phi$  between the input sinusoid  $u$  and the synthesized sinusoid  $y$ . It also outputs an estimate of the amplitude  $A$ .
- The loop filter provides an estimate of the input angular frequency  $\omega$  by filtering out the high-frequency components of the phase difference. The block also outputs the converted frequency  $f$  in Hz.
- The voltage-controlled oscillator integrates the angular speed to produce the phase estimate  $\phi$ . The oscillator also generates the normalized synthesized sinusoid  $(1/A)y$  which it sends to the Phase Detector for comparison.

## Ports

### Input

**u** – Input signal

scalar or vector

Periodic input signal.

Data Types: `single` | `double`

## Output

### **freq — Frequency**

scalar or vector

Estimated frequency of the input signal, in Hz.

Data Types: `single` | `double`

### **angle — Phase angle**

scalar or vector

Estimated phase angle of the input signal, in rad.

Data Types: `single` | `double`

### **mag — Magnitude**

scalar or vector

Estimated magnitude of the input signal.

Data Types: `single` | `double`

## Parameters

### **Phase detector integral gain — PD integral gain**

1000 (default) | positive scalar or vector

Integral gain for the phase detector. This determines the aggressiveness of the PLL in tracking and locking to the magnitude.

If the input signal is a vector, use scalar parameters or use vector parameters that are the same size as the input signal.

### **Loop filter proportional gain — LF proportional gain**

400 (default) | positive scalar or vector



Proportional gain for the loop filter. This determines the aggressiveness of the PLL in tracking and locking to the phase angle. Increase this value to improve reaction time of the tracking to step changes in the phase angle.

If the input signal is a vector, use scalar parameters or use vector parameters that are the same size as the input signal.

**Loop filter integral gain – LF integral gain**

20000 (default) | positive scalar or vector

Integral gain for the loop filter. Increase this value to increase the rate at which steady-state error is eliminated in the phase angle. This value also determines the aggressiveness of the PLL in tracking and locking to the phase.

If the input signal is a vector, use scalar parameters or use vector parameters that are the same size as the input signal.

**Initial frequency (Hz) – Initial frequency**

60 Hz (default) | scalar or vector

Initial estimate of the input frequency. If the input signal is a vector, use scalar parameters or use vector parameters that are the same size as the input signal.

**Initial phase angle (rad) – Initial phase**

0 rad (default) | scalar or vector

Initial estimate of the phase angle. If the input signal is a vector, use scalar parameters or use vector parameters that are the same size as the input signal.

**Initial magnitude – Initial magnitude**

1 (default) | scalar or vector

Initial estimate of the magnitude. If the input signal is a vector, use scalar parameters or use vector parameters that are the same size as the input signal.

**Sample time (-1 for inherited) – Block sample time**

-1 (default) | 0 | positive scalar

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For inherited discrete-time operation, specify -1. For discrete-time operation, specify a positive integer. For continuous-time operation, specify 0.

If this block is in a masked subsystem, or other variant subsystem that allows you to switch between continuous operation and discrete operation, promote the sample time parameter. Promoting the sample time parameter ensures correct switching between the continuous and discrete implementations of the block. For more information, see “Promote Parameter to Mask” (Simulink).

## References

- [1] Karimi-Ghartemani, M., and M. R. Iravani. "A New Phase-Locked Loop (PLL) System." *IEEE Transactions on Industrial Electronics*. Proceedings of the 44th IEEE Symposium on Circuits and Systems, vol. 1, pp. 421-424. IEEE, 2001..

## See Also

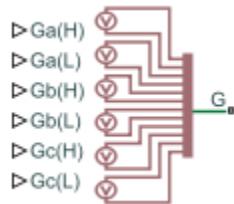
### Blocks

RMS Measurement | Three-Phase Sinusoidal Measurement (PLL)

**Introduced in R2017b**

# Six-Pulse Gate Multiplexer

Multiplex gate input signals to Converter block



## Library

Simscape / Electrical / Power Systems / Semiconductors

## Description

The Six-Pulse Gate Multiplexer block routes gate voltage signals to the six switching devices in a Converter block. The block multiplexes the six separate gate signals into a single vector.

If you want to use Simscape Electrical Electronics and Mechatronics blocks to model the electronics that drive the Converter block, you can switch the input ports of the Six-Pulse Gate Multiplexer block from physical signal ports to electrical ports.

When you switch the block inputs to electrical ports, the block shows additional electrical reference input ports. The additional electrical reference ports are associated with the individual phase voltages that connect to the high-side switching devices in the Converter block and the negative DC voltage common to each low-side switching device in the Converter block.

## Ports

The block has the following ports:

### Ga(H) , Gb(H) , Gc(H)

Ports associated with the gate terminals of the Converter block high-side switching devices. You can set the ports to either physical signal or electrical ports.

### Ga(L) , Gb(L) , Gc(L)

Ports associated with the gate terminals of the Converter block low-side switching devices. You can set the ports to either physical signal or electrical ports.

### G

Vector output port associated with the multiplexed gate signals. Connect this port to the G port of the Converter block.

### a, b, c

Electrical conserving ports associated with the individual phase voltages that connect to the high-side switching devices of the Converter block. These ports are visible only if you set the input ports of the Six-Pulse Gate Multiplexer block to electrical ports.

### L

Electrical conserving port associated with the negative DC voltage common to each low-side switching device in the Converter block. These ports are visible only if you set the input ports of the Six-Pulse Gate Multiplexer block to electrical ports.

## See Also

Converter

## Topics

*“SM Torque Control”*

*“SM Velocity Control”*

*“Switch Between Physical Signal and Electrical Ports”*

*“Synchronous Reluctance Machine Torque Control”*

*“Three-Phase Two-Level PWM Generator”*

*“Three-Phase Voltage-Sourced Converter (FLB)”*

## Introduced in R2013b

# Sliding Mode Controller

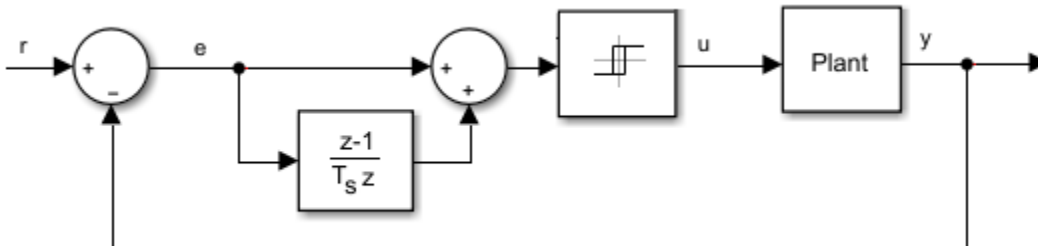
Hysteresis-based sliding mode control

**Library:** Simscape / Electrical / Power Systems / Control / General Control



## Description

The Sliding Mode Controller block implements hysteresis-based sliding mode control (SMC).



## Ports

### Input

**r** — Plant reference

scalar

Plant system reference signal.

Data Types: single | double

**y — Plant output**

scalar

Plant system output signal.

Data Types: `single` | `double`

## Output

**u — Controller output**

scalar

Control system output signal.

Data Types: `single` | `double`

## Parameters

**Hysteresis band — Hysteresis bandwidth**

0.2 (default)

Total hysteresis bandwidth, distributed symmetrically about the set point.

**Control action upper limit — Control signal upper limit,  $u_{max}$**

10 (default) | scalar greater than the value of the **Control action lower limit** parameter

Upper limit for the control output signal.

**Control action lower limit — Control signal lower limit,  $u_{min}$**

-10 (default) | scalar

Lower limit for the control output signal.

**Sample time (-1 for inherited) — Block sample time**

-1 (default) | positive scalar

Time, in s, between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

If this block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly using a positive scalar.

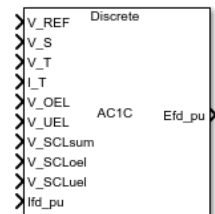
## **See Also**

**Introduced in R2018a**

## SM AC1C

Synchronous machine AC1C excitation system including an Automatic Voltage Regulator (AVR) and an exciter

**Library:** Simscape / Electrical / Power Systems / Control / SM Control



## Description

The SM AC1C block implements a synchronous machine type AC1C excitation system model in conformance with IEEE 421.5-2016<sup>[1]</sup>.

Use this block to model the control and regulation of the field voltage of a synchronous machine operating as a generator using an AC rotating exciter.

You can switch between continuous and discrete implementations of the block by using the Sample time parameter. To configure the integrator for continuous time, set the **Sample time** property to 0. To configure the integrator for discrete time, set the **Sample time** property to a positive, nonzero value, or to -1 to inherit the sample time from an upstream block.

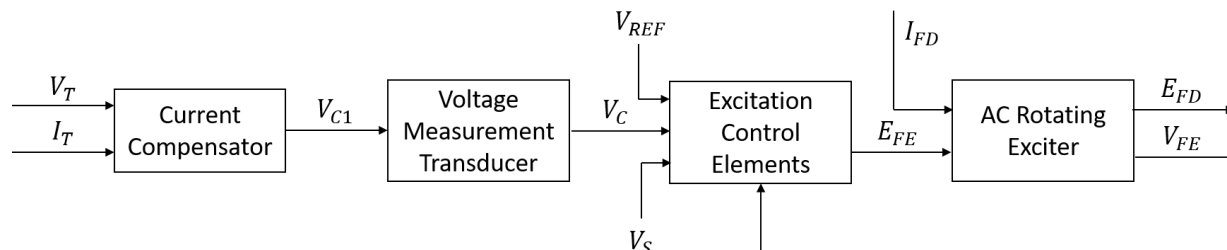
The SM AC1C block is made up of four major components:

- The Current Compensator modifies the measured terminal voltage as a function of terminal current.
- The Voltage Measurement Transducer simulates the dynamics of a terminal voltage transducer using a low-pass filter.
- The Excitation Control Elements component compares the voltage transducer output with a terminal voltage reference to produce a voltage error. This voltage error is then passed through a voltage regulator to produce the exciter field voltage.



- The AC Rotating Exciter models the AC rotating exciter, producing a field voltage to be applied to the controlled synchronous machine. The block also feeds the exciter field current (given the standard symbol  $V_{FE}$ ) back to the excitation system.

This diagram shows the overall structure of the AC1C excitation system model:



In the diagram:

- $V_T$  and  $I_T$  are the measured terminal voltage and current of the synchronous machine.
- $V_{C1}$  is the current-compensated terminal voltage.
- $V_C$  is the filtered, current-compensated terminal voltage.
- $V_{REF}$  is the reference terminal voltage.
- $V_S$  is the power system stabilizer voltage.
- $E_{FE}$  and  $V_{FE}$  are the exciter field voltage and current, respectively.
- $E_{FD}$  and  $I_{FD}$  are the field voltage and current, respectively.

The following sections describe each of the major parts of the block in detail.

## Current Compensator and Voltage Measurement Transducer

The current compensator is modeled as:

$$V_{C1} = V_T + I_T \sqrt{R_C^2 + X_C^2},$$

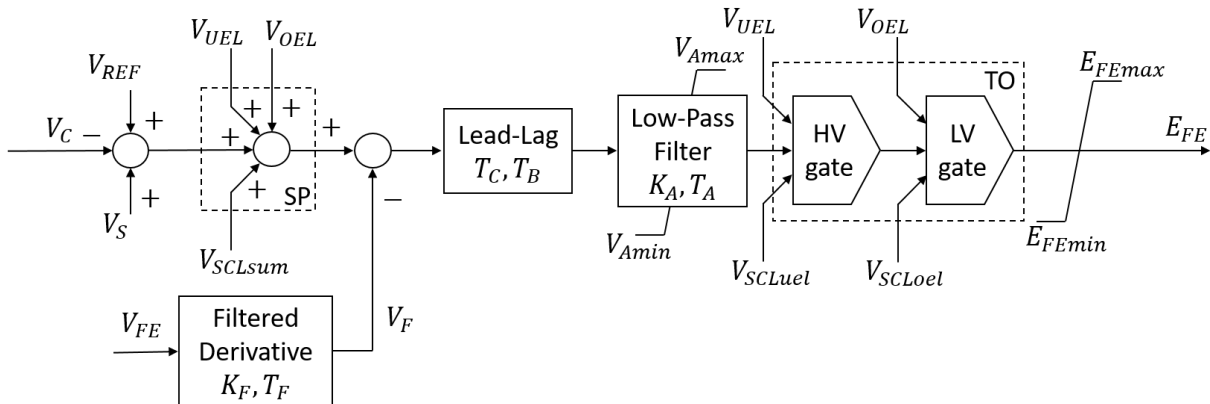
where:

- $R_C$  is the load compensation resistance.
- $X_C$  is the load compensation reactance.

The voltage measurement transducer is implemented as a Low-Pass Filter block with time constant  $T_R$ . Refer to the documentation for this block for the exact discrete and continuous implementations.

## Excitation Control Elements

This diagram illustrates the overall structure of the excitation control elements:



In the diagram:

- SP is the summation point input location for the overexcitation limiter (OEL), underexcitation limiter (UEL), and stator current limiter (SCL) voltages. For more information about using limiters with this block, see “Field Current Limiters” on page 1-1485.
- The Lead-Lag block models additional dynamics associated with the voltage regulator. Here,  $T_C$  is the lead time constant and  $T_B$  is the lag time constant. Refer to the documentation for this block for the exact discrete and continuous implementations.
- The Low-Pass Filter block models the major dynamics of the voltage regulator. Here,  $K_A$  is the regulator gain and  $T_A$  is the major time constant of the regulator. The minimum and maximum anti-windup saturation limits for the block are  $V_{Amin}$  and  $V_{Amax}$ , respectively.
- TO is the take-over point input location for the OEL, UEL, and SCL voltages. For more information about using limiters with this block, see “Field Current Limiters” on page 1-1485.
- The Filtered Derivative block models the rate feedback path for stabilization of the excitation system. Here,  $K_F$  and  $T_F$  are the gain and time constant of this system,

respectively. Refer to the documentation for the Filtered Derivative block for the exact discrete and continuous implementations.

- $E_{FEmin}$  and  $E_{FEmax}$  are the minimum and maximum saturation limits for the output exciter field voltage  $E_{FE}$ .

## Field Current Limiters

You can use various field current limiters to modify the output of the voltage regulator under unsafe operating conditions:

- Use an overexcitation limiter to prevent overheating of the field winding due to excessive field current demand.
- Use an underexcitation limiter to boost field excitation when it is too low, risking desynchronization.
- Use a stator current limiter to prevent overheating of the stator windings due to excessive current.

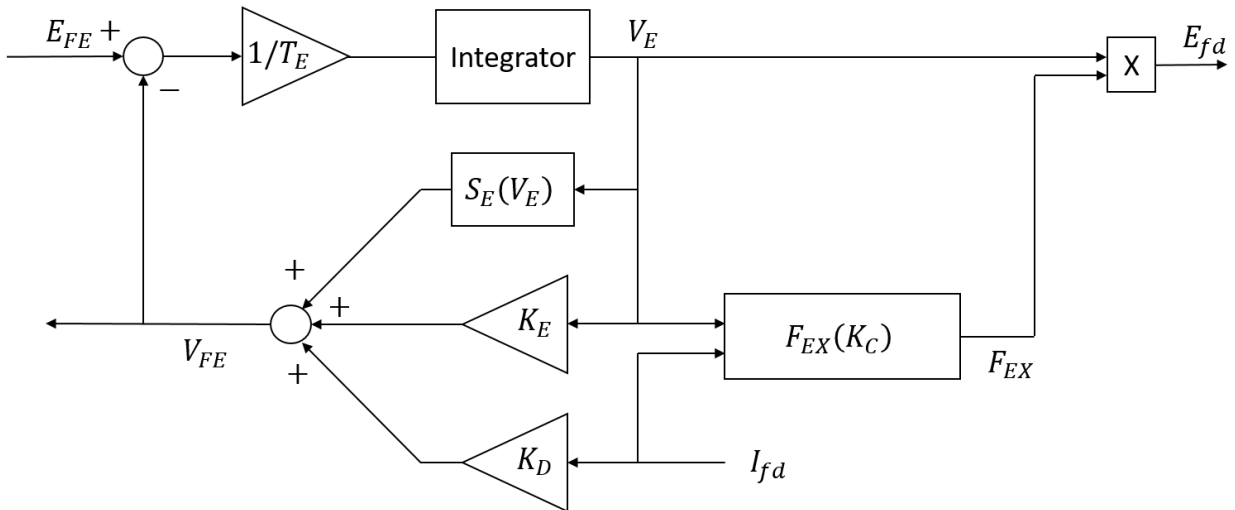
Attach the output of any of these limiters at one of these points:

- The summation point as part of the AVR feedback loop
- The take-over point to override the usual behavior of the AVR

If you are using the stator current limiter at the summation point, use the single input  $V_{SCLsum}$ . If you are using the stator current limiter at the take-over point, use both an overexcitation input  $V_{SCLoel}$  and an underexcitation input  $V_{SCLuel}$ .

## AC Rotating Exciter

This diagram illustrates the overall structure of the AC rotating exciter:



In the diagram:

- The exciter field current  $V_{FE}$  is modeled as the summation of three signals:
  - The nonlinear function  $S_E(V_E)$  models the saturation of the exciter output voltage.
  - The proportional term  $K_E$  models the linear relationship between exciter output voltage and the exciter field current.
  - The demagnetizing effect of the load current on the exciter output voltage is modelled using the demagnetization constant  $K_D$  in the feedback loop.
- The Integrator block integrates the difference between  $E_{FE}$  and  $V_{FE}$  to generate the exciter alternator output voltage  $V_E$ .  $T_E$  is the time constant for this process.
- The nonlinear function  $F_{EX}$  models the exciter output voltage drop from rectifier regulation. This function depends on the constant  $K_C$  which itself is a function of commutating reactance.

## Ports

### Input

**V\_REF — Voltage reference**

scalar

Voltage regulator reference set point, in per-unit representation.

Data Types: single | double

### **V\_S — Input from stabilizer**

scalar

Input from the power system stabilizer, in per-unit representation.

Data Types: single | double

### **V\_T — Terminal voltage**

scalar

Terminal voltage magnitude in per-unit representation.

Data Types: single | double

### **I\_T — Terminal current**

scalar

Terminal current magnitude in per-unit representation.

Data Types: single | double

### **V\_OEL — Overexcitation limit signal**

scalar

Input from the overexcitation limiter, in per-unit representation.

#### **Dependencies**

- To ignore the input from the overexcitation limiter, set **Alternate OEL input locations** to Unused.
- To use the input from the overexcitation limiter at the summation point, set **Alternate OEL input locations** to Summation point.
- To use the input from the overexcitation limiter at the take-over point, set **Alternate OEL input locations** to Take-over.

Data Types: single | double

### **V\_UEL — Underexcitation limit signal**

scalar

Input from the underexcitation limiter, in per-unit representation.

**Dependencies**

- To ignore the input from the underexcitation limiter, set **Alternate UEL input locations** to Unused.
- To use the input from the underexcitation limiter at the summation point, set **Alternate UEL input locations** to Summation point.
- To use the input from the underexcitation limiter at the take-over point, set **Alternate UEL input locations** to Take-over.

Data Types: single | double

**V\_SCLsum — Summation point stator current limit signal**

scalar

Input from the stator current limiter when using the summation point, in per-unit representation.

**Dependencies**

- To ignore the input from the stator current limiter, set **Alternate SCL input locations** to Unused.
- To use the input from the stator current limiter at the summation point, set **Alternate SCL input locations** to Summation point.

Data Types: single | double

**V\_SCLoel — Take-over stator current limit (OEL)**

scalar

Input from the stator current limiter to prevent field overexcitation when using the take-over point, in per-unit representation.

**Dependencies**

- To ignore the input from the stator current limiter, set **Alternate SCL input locations** to Unused.
- To use the input from the stator current limiter at the take-over point, set **Alternate SCL input locations** to Take-over.

Data Types: single | double

**V\_SCLuel — Take-over stator current limit (UEL)**

scalar

Input from the stator current limiter to prevent field underexcitation when using the take-over point, in per-unit representation.

### Dependencies

- To ignore the input from the stator current limiter, set **Alternate SCL input locations** to Unused.
- To use the input from the stator current limiter at the take-over point, set **Alternate SCL input locations** to Take-over.

Data Types: single | double

### I<sub>fd</sub>\_pu — Measured field current

scalar

Measured per-unit field current of the synchronous machine.

Data Types: single | double

## Output

### E<sub>fd</sub>\_pu — Field voltage

scalar

Per-unit field voltage to be applied to the field circuit of the synchronous machine.

Data Types: single | double

## Parameters

### General

#### Initial field voltage, E<sub>fd0</sub> (pu) — Initial output voltage

1 (default) | real number

Initial per unit voltage to be applied to the field circuit of the synchronous machine.

#### Sample time (-1 for inherited) — Block sample time

-1 (default) | 0 | positive scalar

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For inherited discrete-time operation, specify -1. For discrete-time operation, specify a positive integer. For continuous-time operation, specify 0.

If this block is in a masked subsystem, or other variant subsystem that allows you to switch between continuous operation and discrete operation, promote the sample time parameter. Promoting the sample time parameter ensures correct switching between the continuous and discrete implementations of the block. For more information, see “Promote Parameter to Mask” (Simulink).

## Pre-control

### **Resistive component of load compensation, $R_C$ (pu) — Compensation resistance**

0 (default) | positive number

Resistance used in the current compensation system. Set this and  $X_C$  to 0 to disable current compensation.

### **Reactance component of load compensation, $X_C$ (pu) — Compensation reactance**

0 (default) | positive number

Reactance used in the current compensation system. Set this and  $R_C$  to 0 to disable current compensation.

### **Regulator input filter time constant, $T_R$ (s) — Regulator time constant**

0 (default) | positive number

Equivalent time constant for the voltage transducer filtering.

## Control

### **Regulator output gain, $K_A$ (pu) — Regulator gain**

400 (default) | positive number

Gain associated with the voltage regulator.



**Regulator output time constant,  $T_A$  (s) – Regulator time constant**

0.02 (default) | positive number

Major time constant of the voltage regulator.

**Regulator denominator (lag) time constant,  $T_B$  (s) – Regulator lag time constant**

0 (default) | positive number

Equivalent lag time constant in the voltage regulator. Set this to 0 when the additional lag dynamics are negligible.

**Regulator numerator (lead) time constant,  $T_C$  (s) – Regulator lead time constant**

0 (default) | positive number

Equivalent lead time constant in the voltage regulator. Set this to 0 when the additional lead dynamics are negligible.

**Rate feedback excitation system stabilizer gain,  $K_F$  (pu) – Rate feedback gain**

0.03 (default) | positive number

Rate feedback block gain for stabilization of excitation system.

**Rate feedback time constant,  $T_F$  (s) – Rate feedback time constant**

1 (default) | positive number

Rate feedback block time constant for stabilization of excitation system.

**Maximum regulator output,  $V_{Amax}$  (pu) – Regulator output upper limit**

14.5 (default) | real number

Maximum per-unit output voltage of the regulator.

**Minimum regulator output,  $V_{Amin}$  (pu) – Regulator output lower limit**

-14.5 (default) | real number

Minimum per-unit output voltage of the regulator.

**Maximum exciter field voltage,  $E_{FEmax}$  (pu) – Exciter voltage upper limit**

6.03 (default) | real number

Maximum per unit field voltage to be applied to exciter.

**Minimum exciter field voltage, E\_FEmin (pu) — Exciter voltage lower limit**  
-5.43 (default) | real number

Minimum per unit field voltage to be applied to exciter.

**Alternate OEL input locations (V\_OEL) — OEL input location**  
Unused (default) | Summation point | Take-over

Select overexcitation limiter input location.

**Alternate UEL input locations (V\_UEL) — UEL input location**  
Unused (default) | Summation point | Take-over

Select underexcitation limiter input location.

**Alternate SCL input locations (V\_SCL) — SCL input location**  
Unused (default) | Summation point | Take-over

Select stator current limiter input location. To specify the SCL input:

- If you select `Summation point`, use the `V_SCLsum` inport port.
- If you select `Take-over`, use the `V_SCLoel` and `V_SCLuel` inport ports.

## Exciter

**Exciter field proportional constant, K\_E (pu) — Exciter field gain**  
1 (default) | positive number

Proportional constant for exciter field.

**Exciter field time constant, T\_E (s) — Exciter field time constant**  
0.8 (default) | positive number

Time constant for exciter field.

**Rectifier loading factor proportional to commutating reactance, K\_C (pu) — Rectifier loading factor**  
0.2 (default) | positive number

Rectifier loading factor proportional to commutating reactance.

**Demagnetizing factor, function of exciter alternator reactances,  $K_D$  (pu) – Demagnetization factor**

0.38 (default) | positive number

Demagnetization factor related to exciter alternator reactances.

**Exciter output voltage for saturation factor  $S_E(E_1)$ ,  $E_1$  (pu) – First saturation output voltage**

4.18 (default) | positive number

Exciter output voltage for first saturation factor.

**Exciter saturation factor at exciter output voltage  $E_1$ ,  $S_E(E_1)$  (1) – First saturation lookup voltage**

0.1 (default) | positive number

First exciter saturation factor.

**Exciter output voltage for saturation factor  $S_E(E_2)$ ,  $E_2$  (pu) – Second saturation output voltage**

3.14 (default) | positive number

Exciter output voltage for second saturation factor.

**Exciter saturation factor at exciter output voltage  $E_2$ ,  $S_E(E_2)$  (1) – Second saturation lookup voltage**

0.03 (default) | positive number

Second exciter saturation factor.

## References

- [1] *IEEE Recommended Practice for Excitation System Models for Power System Stability Studies*. IEEE Std 421.5-2016. Piscataway, NJ: IEEE-SA, 2016.

## See Also

### Blocks

Lead-Lag (Discrete or Continuous) | PMSM Current Controller with Pre-Control | PMSM Current Reference Generator

**Introduced in R2017b**

# SM Current Controller

Synchronous machine current controller

**Library:** Simscape / Electrical / Power Systems / Control / SM Control



## Description

The SM Current Controller block implements a discrete time PI-based SM current controller in the rotor d-q reference frame.

## Defining Equations

The block is discretized using the backward Euler method due to its first-order simplicity and its stability.

Three PI current controllers implemented in the rotor reference frame produce the reference voltage vector:

$$v_d^{ref} = \left( K_{p\_id} + K_{i\_id} \frac{T_s z}{z-1} \right) (i_d^{ref} - i_d) + v_{d\_FF},$$

$$v_q^{ref} = \left( K_{p\_iq} + K_{i\_iq} \frac{T_s z}{z-1} \right) (i_q^{ref} - i_q) + v_{q\_FF},$$

and

$$v_f^{ref} = \left( K_{p\_if} + K_{i\_if} \frac{T_s z}{z-1} \right) (i_f^{ref} - i_f),$$

where:

- $v_d^{ref}$ ,  $v_q^{ref}$ , and  $v_f^{ref}$  are the  $d$ -axis,  $q$ -axis, and field reference voltages, respectively.
- $i_d^{ref}$ ,  $i_q^{ref}$ , and  $i_f^{ref}$  are the  $d$ -axis,  $q$ -axis, and field reference currents, respectively.
- $i_d$ ,  $i_q$ , and  $i_f$  are the  $d$ -axis,  $q$ -axis, and field currents, respectively.
- $K_{p\_id}$ ,  $K_{p\_iq}$ , and  $K_{p\_if}$  are the proportional gains for the  $d$ -axis,  $q$ -axis and field controllers, respectively.
- $K_{i\_id}$ ,  $K_{i\_iq}$ , and  $K_{i\_if}$  are the integral gains for the  $d$ -axis,  $q$ -axis and field controllers, respectively.
- $v_{d\_FF}$ , and  $v_{q\_FF}$  are the feedforward voltages for the  $d$ -axis and  $q$ -axis, respectively, obtained from the machine mathematical equations and provided as inputs.
- $T_s$  is the sample time of the discrete controller.

Using PI control results in a zero in the closed-loop transfer function which can be canceled by introducing a zero-cancellation block in the feedforward path. The zero cancellation transfer functions in discrete time are:

$$G_{ZC\_id}(z) = \frac{\frac{T_s K_{i\_id}}{K_{p\_id}}}{z + \left( \frac{T_s - \frac{K_{p\_id}}{K_{i\_id}}}{\frac{K_{p\_id}}{K_{i\_id}}} \right)},$$

$$G_{ZC\_iq}(z) = \frac{\frac{T_s K_{i\_iq}}{K_{p\_iq}}}{z + \left( \frac{T_s - \frac{K_{p\_iq}}{K_{i\_iq}}}{\frac{K_{p\_iq}}{K_{i\_iq}}} \right)},$$

and

$$G_{ZC\_if}(z) = \frac{\frac{T_s K_{i\_if}}{K_{p\_if}}}{z + \left( \frac{T_s - \frac{K_{p\_if}}{K_{i\_if}}}{\frac{K_{p\_if}}{K_{i\_if}}} \right)}$$

Saturation must be imposed when the stator voltage vector exceeds the voltage phase limit  $V_{ph\_max}$ :

$$\sqrt{v_d^2 + v_q^2} \leq V_{ph\_max},$$

where  $v_d$ , and  $v_q$  are the  $d$ -axis and  $q$ -axis voltages, respectively.

In the case of axis prioritization, the voltages  $v_1$  and  $v_2$  are introduced, where:

- $v_1 = v_d$  and  $v_2 = v_q$  for  $d$ -axis prioritization.
- $v_1 = v_q$  and  $v_2 = v_d$  for  $q$ -axis prioritization.

The constrained (saturated) voltages  $v_1^{sat}$  and  $v_2^{sat}$  are obtained as follows:

$$v_1^{sat} = \min \left( \max \left( v_1^{unsat}, -V_{ph\_max} \right), V_{ph\_max} \right),$$

and

$$v_2^{sat} = \min \left( \max \left( v_2^{unsat}, -V_{2\_max} \right), V_{2\_max} \right),$$

where:

- $v_1^{unsat}$  and  $v_2^{unsat}$  are the unconstrained (unsaturated) voltages.
- $v_{2\_max}$  is the maximum value of  $v_2$  that does not exceed the voltage phase limit, given

$$\text{by } v_{2\_max} = \sqrt{\left( V_{ph\_max} \right)^2 - \left( v_1^{sat} \right)^2}.$$

In the case that the direct and quadrature axes have the same priority (d-q equivalence) the constrained voltages are obtained as follows:

$$v_d^{sat} = \min \left( \max \left( v_d^{unsat}, -V_{d\_max} \right), V_{d\_max} \right),$$

and

$$v_q^{sat} = \min \left( \max \left( v_q^{unsat}, -V_{q\_max} \right), V_{q\_max} \right),$$

where

$$V_{d\_max} = \frac{V_{ph\_max} |v_d^{unsat}|}{\sqrt{(v_d^{unsat})^2 + (v_q^{unsat})^2}},$$

and

$$V_{q\_max} = \frac{V_{ph\_max} |v_q^{unsat}|}{\sqrt{(v_d^{unsat})^2 + (v_q^{unsat})^2}}.$$

The constrained (saturated) field voltage  $v_f^{sat}$  is limited according to the maximum admissible value:

$$v_f^{sat} = \min \left( \max \left( v_f^{unsat}, -V_{f\_max} \right), V_{f\_max} \right),$$

where:

- $v_f^{unsat}$  is the unconstrained (unsaturated) field voltage.
- $V_{f\_max}$  is the maximum allowable field voltage.

An anti-windup mechanism is employed to avoid saturation of integrator output. In such a situation, the integrator gains become:



$$K_{i\_id} + K_{aw\_id} \left( v_d^{sat} - v_d^{unsat} \right),$$

$$K_{i\_iq} + K_{aw\_iq} \left( v_q^{sat} - v_q^{unsat} \right),$$

and

$$K_{i\_if} + K_{aw\_if} \left( v_f^{sat} - v_f^{unsat} \right),$$

where  $K_{aw\_id}$ ,  $K_{aw\_iq}$ , and  $K_{aw\_if}$  are the anti-windup gains for the  $d$ -axis,  $q$ -axis and field controllers, respectively.

## Assumptions

- The plant model for direct and quadrature axis can be approximated with a first order system.
- This control solution is used only for synchronous motors with sinusoidal flux distribution and field windings.

## Ports

### Input

#### **idqfRef** — Reference currents, A

vector

Reference d-q and field currents for control of synchronous motor.

Data Types: single | double

#### **idqf** — Measured currents, A

vector

Actual d-q and field axis currents of controlled synchronous motor.

Data Types: single | double

## **vdqFF — Purpose, V**

vector

Feedforward pre-control voltages.

Data Types: `single` | `double`

## **VphMax — Maximum phase voltage, V**

scalar

Maximum allowable voltage in each phase.

Data Types: `single` | `double`

## **VfMax — Maximum field voltage, V**

scalar

Maximum allowable field voltage.

Data Types: `single` | `double`

## **Reset — External reset**

scalar

External reset signal (rising edge) for integrators.

Data Types: `single` | `double`

## **Output**

### **vdqfRef — Reference voltages, V**

vector

Reference d-q and field voltages for control of synchronous motor.

Data Types: `single` | `double`

## **Parameters**

### **General**

#### **Sample time (-1 for inherited) — Block sample time**

-1 (default) | -1 or positive number

Sample time for the block (-1 for inherited). If this block is used inside a triggered subsystem, the sample time should be -1. If this block is used in a continuous variable-step model, then the sample time can be explicitly specified.

**Discretization sample time — Block discretization sample time**

0.001 (default) | -1 or positive number

Specify the discretization sample time when zero-cancellation is active and sample time is set to -1 (e.g., when the block is used inside a triggered subsystem).

**Axis prioritization — Axis prioritization for voltage limiter**

q-axis (default) | d-axis | d-q equivalence

Prioritize or maintain ratio between d and q axes when block limits voltage.

**Enable zero cancellation — Feedforward zero-cancellation**

off (default) | on

Enable or disable zero-cancellation on the feedforward path.

**Enable pre-control voltage — Pre-control voltage**

on (default) | off

Enable or disable pre-control voltage.

**d-q control****D-axis current proportional gain — D-axis proportional gain**

1 (default) | positive number

Proportional gain of PI controller used for direct-axis current control.

**D-axis current integral gain — D-axis integral gain**

100 (default) | positive number

Integrator gain of PI controller used for direct-axis current control.

**D-axis current anti-windup gain — D-axis anti-windup gain**

1 (default) | positive number

Anti-windup gain of PI controller used for direct-axis current control.

**Q-axis current proportional gain — Q-axis proportional gain**

1 (default) | positive number

Proportional gain of PI controller used for quadrature-axis current control.

### **Q-axis current integral gain — Q-axis integral gain**

100 (default) | positive number

Integrator gain of PI controller used for quadrature-axis current control.

### **Q-axis current anti-windup gain — Q-axis anti-windup gain**

1 (default) | positive number

Anti-windup gain of PI controller used for quadrature-axis current control.

## **Field control**

### **Field current proportional gain — Field current proportional gain**

9 (default) | positive number

Proportional gain of PI controller used for field current control.

### **Field current integral gain — Field current integral gain**

350 (default) | positive number

Integrator gain of PI controller used for field current control.

### **Field current anti-windup gain — Field current anti-windup gain**

1 (default) | positive number

Anti-windup gain of PI controller used for field current control.

## **References**

- [1] Märgner, M., and W. Hackmann. "Control challenges of an externally excited synchronous machine in an automotive traction drive application." *Emobility-Electrical Power Train, 2006*, pp. 1-6.

## **See Also**

### **Blocks**

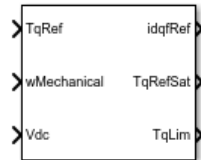
SM AC1C | SM Current Reference Generator

**Introduced in R2017b**

# SM Current Reference Generator

Synchronous machine current reference generator

**Library:** Simscape / Electrical / Power Systems / Control / SM Control



## Description

The SM Current Reference Generator block implements a current reference generator for SM current control in the rotor d-q reference frame.

## Defining Equations

The SM Current Reference Generator block can obtain the current reference using one of these methods:

- Zero  $d$ -axis control (ZDAC).
- Lookup tables.

For the ZDAC method, the block sets:

- The  $d$ -axis current reference  $i_d^{ref}$  to zero:

$$i_d^{ref} = 0,$$

- The field current reference  $i_f^{ref}$  using the torque reference:

$$i_f^{ref} = \frac{|T_{ref} i_{f,max}|}{T_{max}},$$

where  $i_{f,max}$  is the maximum field current and  $T_{max}$  is the maximum torque.

- The  $q$ -axis current reference  $i_q^{ref}$  using the torque equation:

$$i_q^{ref} = \frac{T_{ref}}{K_t i_f^{ref}},$$

where  $T_{ref}$  is the reference torque input and  $K_t$  is the torque constant of the synchronous machine expressed by the simplified torque equation  $T = K_t i_f i_q$ .

For operation below the base speed of the synchronous machine, ZDAC is a suitable method. Above base speed, a field weakening controller is required to adjust the  $d$ -axis reference.

To pregenerate the current references for several operating points, define three lookup tables using the lookup tables approach:

$$i_d^{ref} = f(n_m, T_{ref}, v_{dc}),$$

$$i_q^{ref} = g(n_m, T_{ref}, v_{dc}),$$

and

$$i_f^{ref} = h(n_m, T_{ref}, v_{dc}).$$

## Ports

### Input

#### **TqRef — Reference torque, N\*m**

scalar

Desired mechanical torque produced by the synchronous machine.

Data Types: single | double

**wMechanical — Rotor mechanical speed, rad/s**

scalar

Mechanical angular velocity of the synchronous machine rotor, obtained via direct measurement from the synchronous machine.

Data Types: `single` | `double`

**Vdc — DC-link voltage, V**

scalar

DC-link voltage of the converter. For the ZDAC method, this value is used to limit the output reference torque and torque limit. For the lookup table method, this value is used as an input to the lookup tables.

Data Types: `single` | `double`

## Output

**idqfRef — Reference currents, A**

vector

Reference d-q and field currents to be given as inputs to a current controller.

Data Types: `single` | `double`

**TqRefSat — Reference torque, N\*m**

scalar

Reference torque saturated by the calculated torque limit **TqLim**.

Data Types: `single` | `double`

**TqLim — Torque limit, N\*m**

scalar

Torque limit imposed by both the electrical and mechanical constraints of the system.

Data Types: `single` | `double`



## Parameters

### General Parameters

#### **Nominal dc-link voltage (V) — Rated DC voltage**

300V (default) | positive number

Nominal DC-link voltage of the electrical source.

#### **Maximum power (W) — Maximum power**

30000W (default) | positive number

Maximum synchronous machine power.

#### **Maximum torque (N\*m) — Maximum torque**

250N\*m (default) | positive number

Maximum synchronous machine torque.

#### **Maximum field current (A) — Maximum field current**

25A (default) | positive number

Maximum field current of the synchronous machine.

#### **Sample time (-1 for inherited) — Block sample time**

-1 (default) | -1 or positive number

Sample time for the block (-1 for inherited). If this block is used inside a triggered subsystem, the sample time should be -1. If this block is used in a continuous variable-step model, then the sample time can be explicitly specified.

### Reference Generation Strategy

#### **Current references — Current reference strategy**

Zero d-axis control (default) | Lookup-table based

Select the strategy for determining current references.

#### **Torque constant (N\*m/A) — Torque constant**

0.0375 (default) | positive number

Torque constant of the synchronous machine.

**Mechanical speed vector,  $w_{\text{Mechanical}}$  (rpm) — Rotor speed lookup vector**  
[0, 3000] (default) | positive monotonically increasing vector

Speed vector used in the lookup-tables for determining current references.

**Torque reference vector,  $T_{\text{qRef}}$  (N\*m) — Torque reference lookup vector**  
[-100, 0, 100] (default) | positive monotonically increasing vector

Torque vector used in the lookup-tables for determining current references.

**DC-link voltage vector,  $V_{\text{dc}}$  (V) — DC-link voltage lookup vector**  
[300, 350] (default) | positive monotonically increasing vector

DC-link voltage vector used in the lookup-tables for determining current references.

**D-axis current reference matrix,  $i_{\text{d}}(w_{\text{Mechanical}}, T_{\text{qRef}}, V_{\text{dc}})$ , (A) — Reference d-axis current values**  
 $\text{zeros}(2, 3, 2)$  (default) | real matrix

Direct-axis current reference lookup data.

**Q-axis current reference matrix,  $i_{\text{q}}(w_{\text{Mechanical}}, T_{\text{qRef}}, V_{\text{dc}})$ , (A) — Reference q-axis current values**  
 $\text{zeros}(2, 3, 2)$  (default) | real matrix

Quadrature-axis current reference lookup data.

**Field current reference matrix,  $i_{\text{f}}(w_{\text{Mechanical}}, T_{\text{qRef}}, V_{\text{dc}})$ , (A) — Reference field current values**  
 $\text{zeros}(2, 3, 2)$  (default) | real matrix

Field current reference lookup data.

## References

- [1] Girardin, A., and G. Friedrich. "Optimal control for a wound rotor synchronous starter generator." *Industry Applications Conference, 2006*, pp. 14-19.
- [2] Carpiuc, S., C. Lazar, and D. I. Patrascu. "Optimal Torque Control of the Externally Excited Synchronous Machine." *Control Engineering and Applied Informatics*, 14(2), 2012, pp. 80-88.

## See Also

### Blocks

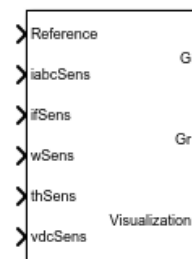
SM AC1C | SM Current Controller

**Introduced in R2017b**

## SM Field-Oriented Control

Synchronous machine field-oriented control

**Library:** Simscape / Electrical / Power Systems / Control / SM Control



### Description

The SM Field-Oriented Control block implements a synchronous machine (SM) field-oriented control structure. Field Oriented Control (FOC) is a performant AC motor control strategy that decouples torque and flux by transforming the stationary phase currents to a rotating frame. Use FOC when rotor speed and position are known and your application requires:

- High torque and low current at startup
- High efficiency

### Equations

The SM FOC is made up of several control blocks from the Control library. To see and modify these blocks and the FOC's internal structure, right-click the block in Simulink and select **Mask > Look Under Mask**. The overall control structure is made up of several parts:

- The outer loop controller converts the reference signal you supply to the reference  $d$ -axis,  $q$ -axis, and field currents.

You can choose the type of reference signal you provide using the `Control mode` parameter:

- **Velocity control** — Control or regulate the rotation speed of the synchronous machine. An internal Velocity Controller block generates a reference torque from the rotor speed error.
- **Torque control** — Control or regulate the mechanical torque of the SM.

An internal SM Current Reference Generator block generates the reference currents using a proportional-integral (PI) controller, minimizing the torque error.

- The inner loop controller converts the current references into voltage references. An internal SM Current Controller generates the voltage references using a PI controller minimizing the current error, and the feedforward terms:

$$v_{d\_FF} = -\omega_e L_{qi} i_q$$

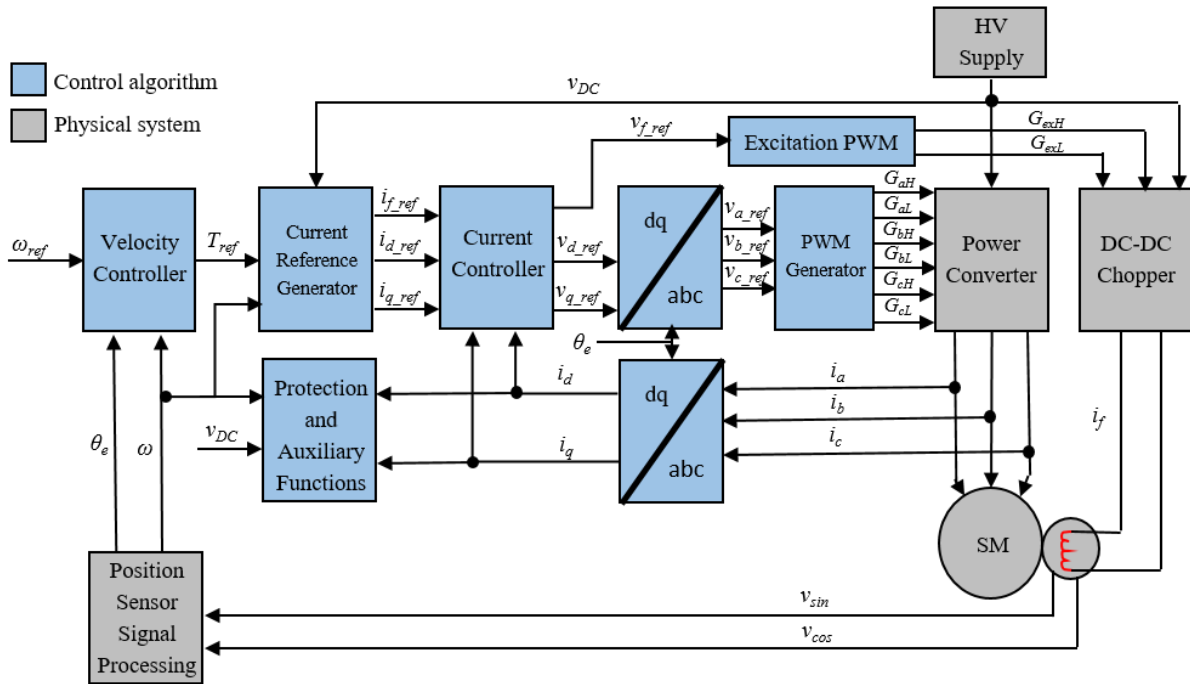
$$v_{q\_FF} = \omega_e (L_d i_d + L_{mf} i_f)$$

$$v_{f\_FF} = 0$$

where:

- $\omega_e$  is the rotor electrical angular velocity.
- $L_d$  and  $L_q$  are the  $d$ - and  $q$ -axis stator inductances.
- $L_{mf}$  is the mutual field armature inductance.
- $i_d$ ,  $i_q$ , and  $i_f$  are the stator  $d$ - $q$  and field excitation currents, respectively.
- The PWM Generator converts the reference stator voltages into gate pulses to be passed to a Power Converter that is powering the stator windings of the synchronous machine.
- The Excitation PWM Generator converts the reference field voltage into gate pulses to be passed to a DC-DC Chopper powering the SM field winding.

This diagram shows the overall architecture of the block.



In the diagram:

- $\omega$  and  $\omega_{ref}$  are the measured and reference angular velocities, respectively.
- $T_{ref}$  is the reference electromagnetic torque. If you configure the block for speed control, a Velocity Controller generates this reference torque.
- $i$  and  $v$  are stator currents and voltages. Subscripts  $d$ ,  $q$ , and  $f$ , represent the  $d$ -axis,  $q$ -axis, and field winding. Subscripts  $a$ ,  $b$ , and  $c$ , represent the three stator windings.
- $\theta_e$  is the rotor electrical angle.
- $G$  is a gate pulse, subscripts  $H$  and  $L$  represent high and low, and subscripts  $a$ ,  $b$ , and  $c$ , represent the three stator windings. Subscript  $ex$  represents the field excitation pulses.

You can choose to implement either velocity or torque control with the `Control mode` parameter. The block implements velocity control exactly as shown in the diagram. The block implements torque control by removing the Velocity Controller block and accepting the reference torque directly.

## Assumptions

The machine parameters are known.

## Limitations

The control structure is implemented with a single sample rate.

## Ports

### Input

#### **Reference — System reference**

scalar

System reference specified as torque reference in N\*m or velocity reference in rad/s, depending on the control mode selected.

Data Types: `single` | `double`

#### **iabcSens — Measured phase currents**

vector

Measured stator phase currents, in A.

Data Types: `single` | `double`

#### **ifSens — Measured field current**

scalar

Measured rotor field current, in A.

Data Types: `single` | `double`

#### **wSens — Rotor speed**

scalar

Measured mechanical angular velocity of rotor, in rad/s.

Data Types: `single` | `double`

## **thSens — Rotor angle**

scalar

Measured mechanical angle of rotor, in rad.

Data Types: `single` | `double`

## **vdcSens — DC-link voltage**

scalar

Measured DC-link voltage, in V.

Data Types: `single` | `double`

## **Output**

### **G — Stator converter gate pulses**

vector

Six pulse waveforms that determine switching behavior in the attached power converter.

Data Types: `single` | `double`

### **Gr — Excitation chopper gate pulses**

vector

Waveforms that determine switching behavior in the attached excitation chopper. The size of the waveform depends on the selected chopper type. To specify the chopper type, use the **Chopper type** parameter:

- First and fourth quadrant chopper — The output waveform has two pulses.
- Four-quadrant chopper — The output waveform has four pulses.

Data Types: `single` | `double`

### **Visualization — Visualization signals**

bus

Bus containing signals for visualization, including:

- Reference
- wElectrical



- `iabc`
- `theta`
- `Vdc`
- `PwmEnable`
- `TqRef`
- `TqLim`
- `idqRef`
- `idqf`
- `vdqRef`
- `modWave`
- `DCexcit`

Data Types: `single` | `double`

## Parameters

### General

#### **Control mode — Control mode strategy**

`Velocity control (default)` | `Torque control`

Specify either a torque control or velocity control strategy.

#### **Nominal dc-link voltage (V) — Rated DC voltage**

`300 V (default)` | positive number

Nominal DC-link voltage of the electrical source.

#### **Maximum power (W) — Maximum power**

`60000 W (default)` | positive number

Maximum machine power.

#### **Maximum torque (N\*m) — Maximum torque**

`250 N*m (default)` | positive number

Maximum machine torque.

**Maximum field current (A) — Maximum field current**

25 A (default) | positive number

Maximum current in the field winding.

**Inverter dc-link voltage threshold (V) — DC-link voltage threshold**

100 V (default) | positive number

Voltage threshold to activate the power inverter.

**Number of rotor pole pairs — Pole pairs**

4 (default) | positive integer

Number of pole pairs on the rotor.

**Fundamental sample time (s) — Block sample time**

5e-6 (default) | positive number

Fundamental sample time for the block.

## **Outer Loop**

**Control Type — Control type strategy**

PI control (default) | P control | P-PI control

Specify the type of the control strategy.

**Controller proportional gain — Proportional gain of PI controller**

1 (default) | positive number

Proportional gain of the PI controller.

**Controller integral gain — Integral gain of PI controller**

1 (default) | positive number

Integral gain of the PI controller.

**P controller proportional gain — Proportional gain of P controller**

1 (default) | positive number

Proportional gain of P controller.

**Integral anti-windup gain — Anti-windup gain**

1 (default) | positive number

Anti-windup gain of the PI controller.

**Current references — Current reference strategy**

Zero d-axis control (default) | Lookup-table based

Select the current reference strategy.

**Mechanical speed vector,  $w_{\text{Mechanical}}$  (rpm) — Rotor speed lookup vector**

[0, 3000] rpm (default) | positive monotonically increasing vector

Speed vector used in the lookup tables for determining current references.

**Torque reference vector,  $T_q\text{Ref}$  (N\*m) — Torque reference lookup vector**

[-100, 0, 100] N\*m (default) | positive monotonically increasing vector

Torque vector used in the lookup tables for determining current references.

**DC-link voltage vector,  $V_{dc}$  (V) — DC-link voltage lookup vector**

[300, 350] V (default) | positive monotonically increasing vector

DC-link voltage vector used in the lookup tables for determining current references.

**D-axis current reference matrix,  $i_d(w_{\text{Mechanical}}, T_q\text{Ref}, V_{dc})$  (A) — Reference d-axis current values**

zeros(2,3,2) A (default) | real matrix

Direct-axis current reference lookup data.

**Q-axis current reference matrix,  $i_q(w_{\text{Mechanical}}, T_q\text{Ref}, V_{dc})$  (A) — Reference q-axis current values**

zeros(2,3,2) A (default) | real matrix

Quadrature-axis current reference lookup data.

**Field current reference matrix,  $i_f(w_{\text{Mechanical}}, T_q\text{Ref}, V_{dc})$ , (A) — Reference field current values**

zeros(2,3,2) A (default) | real matrix

Field current reference lookup data.

**Torque constant — Motor torque constant**

0.04 N\*m/A (default) | positive scalar

Synchronous machine torque constant. This value is numerically equivalent to the back EMF constant expressed in  $V / (\text{rad/s})$ .

## Inner Loop

**D-axis current proportional gain — D-axis proportional gain**

1 (default) | positive number

Proportional gain of the PI controller used for direct-axis current control.

**D-axis current integral gain — D-axis integral gain**

100 (default) | positive number

Integrator gain of the PI controller used for direct-axis current control.

**D-axis current anti-windup gain — D-axis anti-windup gain**

1 (default) | positive number

Anti-windup gain of the PI controller used for direct-axis current control.

**Q-axis current proportional gain — Q-axis proportional gain**

1 (default) | positive number

Proportional gain of the PI controller used for quadrature-axis current control.

**Q-axis current integral gain — Q-axis integral gain**

100 (default) | positive number

Integrator gain of the PI controller used for quadrature-axis current control.

**Q-axis current anti-windup gain — Q-axis anti-windup gain**

1 (default) | positive number

Anti-windup gain of the PI controller used for quadrature-axis current control.

**Field current proportional gain — Field winding proportional gain**

10 (default) | positive number

Proportional gain of the PI controller used for field current control.

**Field current integral gain — Field winding integral gain**

1000 (default) | positive number

Integral gain of the PI controller used for field current control.

**Field current anti-windup gain — Field winding anti-windup gain**

1 (default) | positive number

Anti-windup gain of the PI controller used for field current control.

**Axis prioritization — Axis prioritization for voltage limiter**

q-axis (default) | d-axis | d-q equivalence

Prioritize or maintain ratio between  $d$ - and  $q$ -axis when the block limits voltage.

**Enable zero cancellation — Feedforward zero cancellation**

off (default) | on

Enable or disable zero cancellation on the feedforward path.

**Enable pre-control voltage — pre-control voltage**

on (default) | off

Enable or disable pre-control voltage.

**Machine parameters — Machine parameterization**

Constant parameters (default) | Lookup table based parameters

Specify how to parameterize the machine.

- Constant parameters — Specify machine parameters that are constant throughout the simulation.
- Lookup table based parameters — Specify machine parameters as lookup tables that depend on current.

**Dependencies**

Enabled when the **Enable pre-control voltage** parameter is selected.

**D-axis inductance (H) — Feedforward d-axis inductance**

0.00024 (default) | positive scalar

Direct-axis inductance for feedforward pre-control.

**Dependencies**

Enabled when the **Machine parameters** parameter is set to Constant parameters.

**Q-axis inductance (H) — Feedforward q-axis inductance**

0.00029 (default) | positive scalar

Quadrature-axis inductance for feedforward pre-control.

**Dependencies**

Enabled when the **Machine parameters** parameter is set to Constant parameters.

**Mutual field armature inductance (H) — Mutual inductance between field and armature**

0.007 (default)

Mutual inductance between the field and armature windings.

**Dependencies**

Enabled when the **Machine parameters** parameter is set to Constant parameters.

**D-axis current vector, id (A) — D-axis current breakpoint vector**

[-200, 0, 200] A (default) | monotonically increasing vector

Direct-axis current vector used in the lookup tables for parameters determination.

**Dependencies**

Enabled when the **Machine parameters** parameter is set to Lookup table based parameters.

**Q-axis current vector, iq (A) — Q-axis current breakpoint vector**

[-200, 0, 200] A (default) | monotonically increasing vector

Quadrature-axis current vector used in the lookup tables for parameters determination.

**Dependencies**

Enabled when the **Machine parameters** parameter is set to Lookup table based parameters.

**Field current vector, if (A) — Field winding current vector**

[0, 20] A (default) | monotonically increasing vector

Field current vector used in the lookup tables for parameters determination.

#### **Dependencies**

Enabled when the **Machine parameters** parameter is set to Lookup table based parameters.

#### **Ld matrix, Ld(id,iq) (H) – D-axis inductance lookup data**

0.0002 \* ones(3, 3) H (default) | positive matrix

$L_d$  matrix used as lookup table data.

#### **Dependencies**

Enabled when the **Machine parameters** parameter is set to Lookup table based parameters.

#### **Lq matrix, Lq(id,iq) (H) – Q-axis inductance lookup data**

0.0002 \* ones(3, 3) H (default) | positive matrix

$L_q$  matrix used as lookup table data.

#### **Dependencies**

Enabled when the **Machine parameters** parameter is set to Lookup table based parameters.

#### **Lmf matrix, Lmf(id,iq,if), (H) – Field-armature mutual inductance lookup**

0.007 \* ones(2, 2, 2) H (default) | positive matrix

$L_{mf}$  matrix used as lookup table data.

#### **Dependencies**

Enabled when the **Machine parameters** parameter is set to Lookup table based parameters.

## **PWM**

#### **PWM method – Pulse width modulation method**

SVM: space vector modulation (default) | SPWM: sinusoidal PWM

Specify the waveform technique.

## **Sampling mode — Wave-sampling method**

Natural (default) | Asymmetric | Symmetric

Specify whether the block samples the modulation waveform when the waves intersect or when the carrier wave is at one or both of its boundary conditions.

## **Switching frequency (Hz) — Switching rate**

1000 Hz (default) | positive integer

Specify the rate at which you want the switches in the power converter to switch.

## **Chopper type — DC-DC chopper type**

First and fourth quadrant chopper (default) | Four-quadrant chopper

Specify DC-DC chopper type.

## **Switching frequency excitation (Hz) — Excitation system switching frequency**

1000 Hz (default) | positive integer

Specify PWM switching frequency for excitation system.

## **References**

- [1] Märgner, M., and W. Hackmann. "Control challenges of an externally excited synchronous machine in an automotive traction drive application." *In Emobility-Electrical Power Train*. (2010): 1-6.
- [2] Carpiuc, S., C. Lazar, and D. Patrascu. "Optimal Torque Control of the Externally Excited Synchronous Machine." *Journal of Control Engineering and Applied Informatics*. 14, no 2 (2012): 80-88.

## **See Also**

SM Current Controller | SM Current Reference Generator

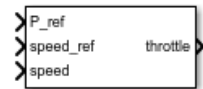
**Introduced in R2018a**



# SM Governor with Droop

Synchronous machine governor with droop

**Library:** Simscape / Electrical / Power Systems / Control / SM Control



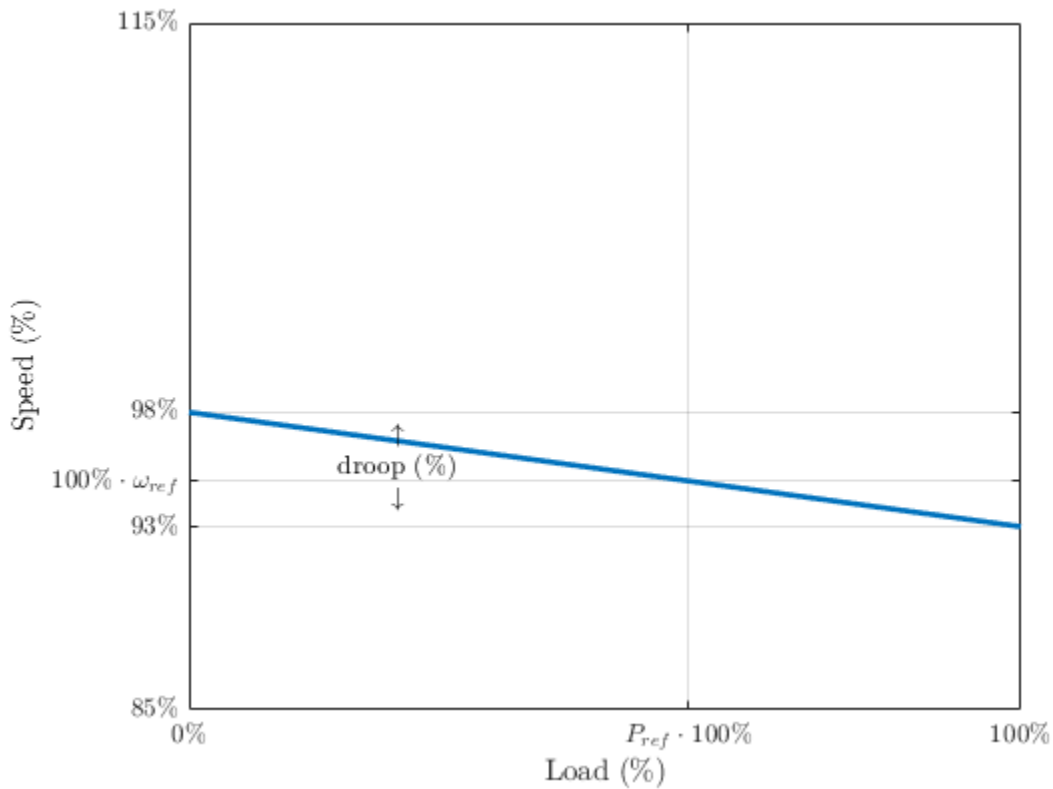
## Description

The SM Governor with Droop block implements a synchronous machine (SM) governor with a droop characteristic. Use this block to regulate or control the throttle input to a prime mover driving a synchronous generator.

The block uses the error between the measured and desired generator speeds to set the prime mover throttle position. For example, when the prime mover is rotating too slowly, the throttle is opened to increase the energy input to the generator and increase its speed.

## Operation

When multiple governor-driven generators are connected in parallel, droop characteristics ensure overall stability of the grid. Droop is defined as percent change in speed from no load to full load of the generator. This figure shows the speed-load relationship for a governor with 5% droop.



Here,

- $\omega_{ref}$  is the reference speed of the governor. Set this value as a per-unit quantity using the **speed\_ref** port.
- $P_{ref}$  is the reference load of the governor. Set this value as a per-unit quantity using the **P\_ref** port.
- **droop** is the droop percentage of the governor. Set this value as a percentage using the **Percentage droop, (%)** parameter.

The block calculates the reference throttle position, expressed as a per-unit quantity, as:

$$u_{throttle,ref} = P_{ref} - \frac{100}{droop} (\omega - \omega_{ref})$$

where  $\omega$  is the actual, per-unit generator speed.

The inertia of the valve introduces a delay between this reference throttle position and the actual throttle position, which is modeled as a first-order lag:

$$u_{throttle} = \left( \frac{1}{T_s s + 1} \right) u_{throttle,ref}$$

Here,  $T_s$  is the time constant. Set this value using the **Time constant of governor, (s)** parameter.

## Ports

### Input

#### **P\_ref — Desired per-unit generator load**

scalar

Reference generator load expressed as a per-unit quantity. The steady-state output of the governor running at its reference speed is equivalent to this value.

Data Types: `single` | `double`

#### **speed\_ref — Desired per-unit generator speed**

scalar

Reference generator speed expressed as a per-unit quantity.

Data Types: `single` | `double`

#### **speed — Actual per-unit generator speed**

scalar

Measured generator speed expressed as a per-unit quantity.

Data Types: `single` | `double`

## Output

### **throttle — Throttle position**

scalar

Throttle position of the governor, expressed as a per-unit quantity.

Data Types: `single` | `double`

## Parameters

### **Percentage droop, (%) — Droop**

5 % (default)

Percent change in governor speed from 0% to 100% load. If multiple governor-driven generators are connected in parallel, those with lower droop percentages are more sensitive to load changes.

### **Time constant of governor, (s) — First-order lag time constant**

0.2 (default)

Time constant of the governor first-order lag representing the throttle inertia dynamics.

### **Initial throttle position, (pu) — Initial throttle output**

0.5 (default)

Initial state of the throttle.

## See Also

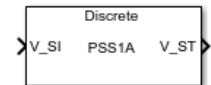
Synchronous Machine Round Rotor (fundamental)

**Introduced in R2018a**

# SM PSS1A

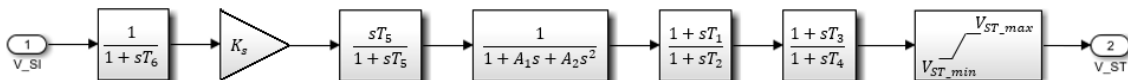
Single input power system stabilizer type PSS1A

**Library:** Simscape / Electrical / Power Systems / Control / SM Control



## Description

The SM PSS1A block implements a single-input PSS1A power system stabilizer (PSS) to maintain rotor angle stability in a synchronous machine (SM). Typically, you use a PSS to enhance damping of power system oscillations through excitation control.



## Ports

### Input

**V\_SI — Speed, frequency, or power**

scalar

Per-unit speed, frequency of the terminal bus voltage, compensated frequency, or electrical power.

Data Types: single | double

### Output

**V\_ST — Stabilization speed, frequency, or power**

scalar

Automatic voltage regulator (AVR) input stabilization signal, as limited by  $V_{ST\_min}$  and  $V_{ST\_max}$

Data Types: single | double

## Parameters

**Power system stabilizer (PSS) gain,  $K_S$  (pu) — PSS gain**

3.15 (default)

Power system stabilizer (PSS) forward path gain.

**PSS denominator constant for second-order block,  $A_1$  — Filter constant 1**

0 (default)

PSS signal conditioning filter coefficient 1.

**PSS denominator constant for second-order block,  $A_2$  — Filter constant 2**

0 (default)

PSS signal conditioning filter coefficient 2.

**PSS numerator (lead) compensating time constant,  $T_1$  (s) — Lead time constant 1**

0.76 (default)

Lead time constant 1.

**PSS denominator (lag) compensating time constant,  $T_2$  (s) — Lag time constant 2**

0.01 (default)

Lag time constant 2.

**PSS numerator (lead) compensating time constant,  $T_3$  (s) — Lead time constant 3**

0.76 (default)

Lead time constant 3.

**PSS denominator (lag) compensating time constant, T\_4 (s) – Lag time constant 4**

0.01 (default)

Lag time constant 4.

**PSS washout time constant, T\_5 (s) – Washout time constant**

10 (default)

Washout time constant.

**PSS transducer time constant, T\_6 (s) – Transducer time constant**

0 (default)

Transducer time constant.

**Maximum PSS output, V\_STmax (pu) – Maximum stabilization**

0.09 (default)

Maximum power system stabilizer (PSS) output to the automatic voltage regulator (AVR).

**Minimum PSS output, V\_STmin (pu) – Minimum stabilization**

-0.09 (default)

Minimum power system stabilizer (PSS) output to the automatic voltage regulator (AVR).

**Sample time (-1 for inherited) – Block sample time**

-1 (default) | 0 | positive scalar

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For inherited discrete-time operation, set the sample time to -1. For discrete-time operation, set the sample time to a positive scalar. For continuous-time operation, set the sample time to 0.

**References**

- [1] *IEEE Recommended Practice for Excitation System Models for Power System Stability Studies*. IEEE Std 421.5-2016. Piscataway, NJ: IEEE-SA, 2016.

[2] Kundur, P. *Power System Stability and Control*. New York, NY: McGraw Hill, 1993.

## **See Also**

Synchronous Machine Model 1.0 (SI parameters) | Synchronous Machine Field Circuit (pu) | Synchronous Machine Measurement | Synchronous Machine Model 2.1 (fundamental) | Synchronous Machine Model 2.1 (standard) | Synchronous Machine Round Rotor (fundamental) | Synchronous Machine Round Rotor (standard) | Synchronous Machine Salient Pole (fundamental) | Synchronous Machine Salient Pole (standard)

**Introduced in R2018b**



# Smith Predictor Controller

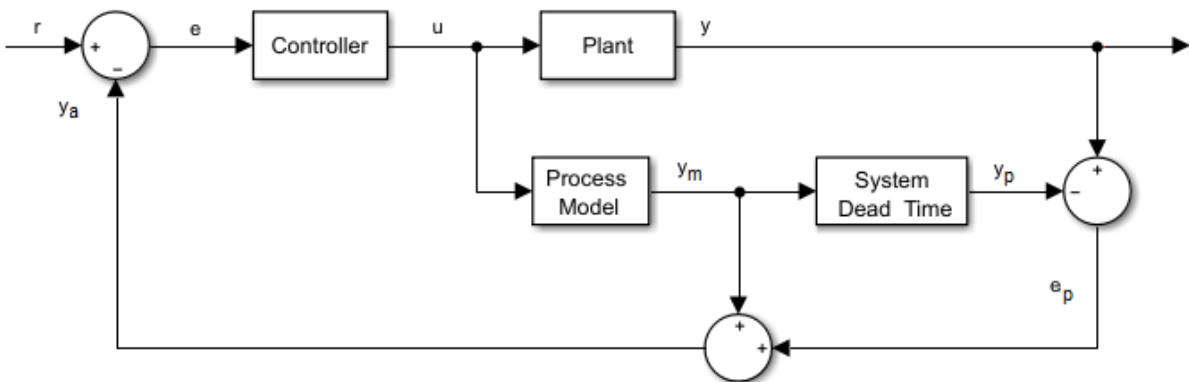
Discrete-time Smith dead-time compensator

**Library:** Simscape / Electrical / Power Systems / Control / General Control



## Description

The Smith Predictor Controller block compensates for dead time by implementing a Smith dead-time PI control structure in discrete time. This diagram shows the equivalent circuit for the block.



## Equations

The transfer function for a system with dead-time is

$$G_f(s) = G_p(s)e^{-\tau s},$$

where:

- $\tau$  is the system dead time.
- $G_p(s)$  is the process model.
- $G_f(s)$  is prediction error filter.

## Ports

### Input

#### **r — Plant reference**

scalar

Plant system reference signal.

Data Types: `single` | `double`

#### **Reset — Integrator reset**

scalar

External reset signal (rising edge) for the integrator.

Data Types: `Boolean`

#### **y — Plant output**

scalar

Plant system output signal.

Data Types: `single` | `double`

### Output

#### **u — Controller output**

scalar

Control system output signal.

Data Types: `single` | `double`

## Parameters

### Proportional gain — $K_p$

1 (default) | positive scalar

Proportional gain,  $K_p$ , of the PI controller.

### Integral gain — $K_i$

1 (default) | positive scalar

Integral gain,  $K_i$ , of the PI controller.

### Integrator initial condition — Initial integrator value

0 (default) | scalar

Value of the integrator at simulation start time.

### Control action upper limit — $u_{max}$

5 (default) | scalar greater than the value of the **Control action lower limit** parameter

Upper limit for the control output signal.

### Control action lower limit — $u_{min}$

0 (default) | scalar

Lower limit for the control output signal.

### Model discrete transfer function numerator — Transfer function numerator

1 (default) | scalar or vector

Numerator of the system discretized transfer function. To determine the discrete transfer function, if you have a license for Control System Toolbox, use the `c2d` function.

### Model discrete transfer function denominator — Transfer function denominator

[1 0.5] (default) | vector

Denominator of the system discretized transfer function. To determine the discrete transfer function, if you have a license for Control System Toolbox, use the `c2d` function.

### System dead time (samples) — Number of dead-time samples

2 (default) | 0 or a positive integer

Number of samples of the dead time.

**Sample time (-1 for inherited) — Sampling interval**

-1 (default) | default value or a positive number

Time interval between samples. If the block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

**References**

- [1] Velagic. J. "Design of Smith-like Predictive Controller with Communication with Communication Delay Adaptation." *International Journal of Electrical, Computer, Energetic, Electronic and Communication Engineering*. Vol 2, Number 11, 2008, pp. 2447-2481.

**See Also**

**Blocks**

RST Controller | State-Feedback Controller

**Introduced in R2017b**

# Solar Cell

Solar cell model



## Library

Sources

## Description

The Solar Cell block represents a solar cell current source.

The solar cell model includes the following components:

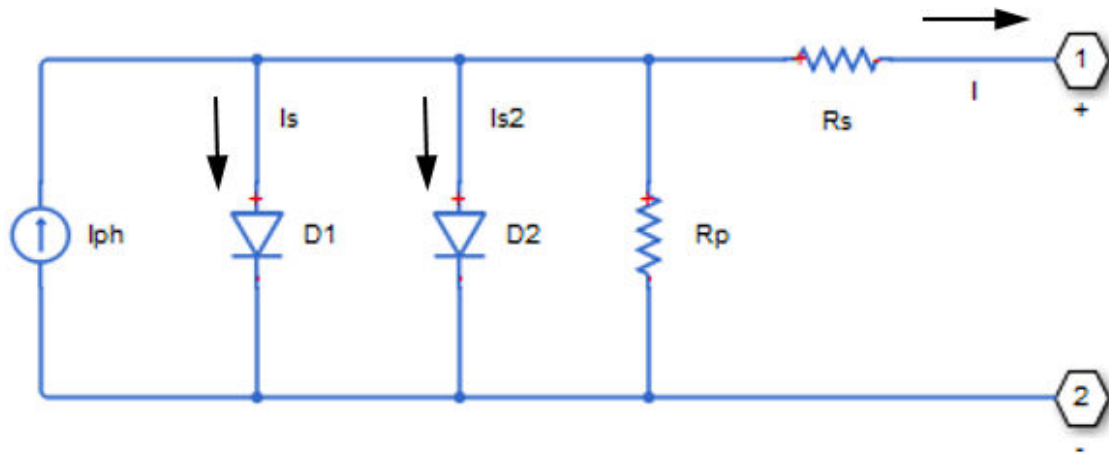
- “Solar-Induced Current” on page 1-1535
- “Temperature Dependence” on page 1-1537
- “Thermal Port” on page 1-1539

## Solar-Induced Current

The block represents a single solar cell as a resistance  $R_s$  that is connected in series with a parallel combination of the following elements:

- Current source
- Two exponential diodes
- Parallel resistor  $R_p$

The following illustration shows the equivalent circuit diagram:



The output current  $I$  is:

$$I = I_{ph} - I_s * \left( e^{(V + I * R_s) / (N_s * V_t)} - 1 \right) - I_{s2} * \left( e^{(V + I * R_s) / (N_2 * V_t)} - 1 \right) - (V + I * R_s) / R_p$$

where:

- $I_{ph}$  is the solar-induced current:

$$I_{ph} = I_{ph0} \times \frac{I_r}{I_{r0}}$$

where:

- $I_r$  is the irradiance (light intensity) in  $W/m^2$  falling on the cell.
- $I_{ph0}$  is the measured solar-generated current for the irradiance  $I_{r0}$ .
- $I_s$  is the saturation current of the first diode.
- $I_{s2}$  is the saturation current of the second diode.
- $V_t$  is the thermal voltage,  $kT/q$ , where:
  - $k$  is the Boltzmann constant.

- $T$  is the **Device simulation temperature** parameter value.
- $q$  is the elementary charge on an electron.
- $N$  is the quality factor (diode emission coefficient) of the first diode.
- $N_2$  is the quality factor (diode emission coefficient) of the second diode.
- $V$  is the voltage across the solar cell electrical ports.

The quality factor varies for amorphous cells, and is typically 2 for polycrystalline cells.

The block lets you choose between two models:

- An 8-parameter model where the preceding equation describes the output current
- A 5-parameter model that applies the following simplifying assumptions to the preceding equation:
  - The saturation current of the second diode is zero.
  - The impedance of the parallel resistor is infinite.

If you choose the 5-parameter model, you can parameterize this block in terms of the preceding equivalent circuit model parameters or in terms of the short-circuit current and open-circuit voltage the block uses to derive these parameters.

All models adjust the block resistance and current parameters as a function of temperature.

You can model any number of solar cells connected in series using a single Solar Cell block by setting the parameter **Number of series cells** to a value larger than 1. Internally the block still simulates only the equations for a single solar cell, but scales up the output voltage according to the number of cells. This results in a more efficient simulation than if equations for each cell were simulated individually.

If you want to model  $N$  cells in parallel, you can do so for single cells by scaling the parameter values accordingly. That is, multiply short-circuit current, diode saturation current, and solar-generated currents by  $N$ , and divide series resistance by  $N$ . To connect solar cell blocks in parallel, where each block contains multiple cells in series, make multiple copies of the block and connect accordingly.

## Temperature Dependence

Several solar cell parameters depend on temperature. The solar cell temperature is specified by the **Device simulation temperature** parameter value.

The block provides the following relationship between the solar-induced current  $I_{ph}$  and the solar cell temperature  $T$ :

$$I_{ph}(t) = I_{ph} * (1 + TIPH1 * (T - T_{meas}))$$

where:

- $TIPH1$  is the **First order temperature coefficient for Iph**, **TIPH1** parameter value.
- $T_{meas}$  is the **Measurement temperature** parameter value.

The block provides the following relationship between the saturation current of the first diode  $I_{s1}$  and the solar cell temperature  $T$ :

$$I_{s1}(T) = I_{s1} * \left( \frac{T}{T_{meas}} \right)^{(TXIS1/N)} * e^{\left( EG * \left( \frac{T}{T_{meas}} - 1 \right) / (N * V_t) \right)}$$

where  $TXIS1$  is the **Temperature exponent for Is**, **TXIS1** parameter value.

The block provides the following relationship between the saturation current of the second diode  $I_{s2}$  and the solar cell temperature  $T$ :

$$I_{s2}(T) = I_{s2} * \left( \frac{T}{T_{meas}} \right)^{(TXIS2/N_2)} * e^{\left( EG * \left( \frac{T}{T_{meas}} - 1 \right) / (N_2 * V_t) \right)}$$

where  $TXIS2$  is the **Temperature exponent for Is2**, **TXIS2** parameter value.

The block provides the following relationship between the series resistance  $R_s$  and the solar cell temperature  $T$ :

$$R_s(T) = R_s * \left( \frac{T}{T_{meas}} \right)^{TRS1}$$

where  $TRS1$  is the **Temperature exponent for Rs**, **TRS1** parameter value.

The block provides the following relationship between the parallel resistance  $R_p$  and the solar cell temperature  $T$ :



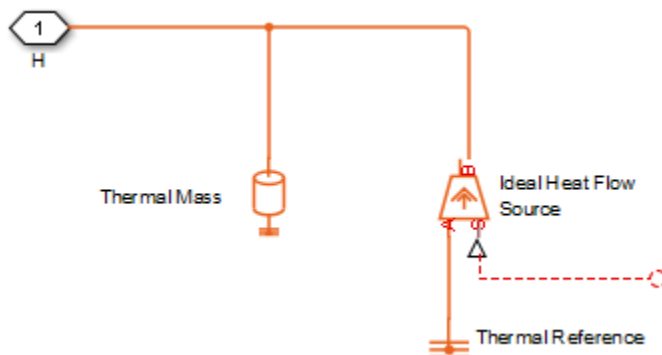
$$R_p(T) = R_p * \left( \frac{T}{T_{meas}} \right)^{TRP1}$$

where *TRP1* is the **Temperature exponent for Rp, TRP1** parameter value.

## Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal port H on the block icon, and adds the **Thermal Port** tab to the block dialog box.

The thermal port model, shown in the following illustration, represents just the thermal mass of the device. The thermal mass is directly connected to the component thermal port H. An internal Ideal Heat Flow Source supplies a heat flow to the port and thermal mass. This heat flow represents the internally generated heat.



The internally generated heat in the solar cell is calculated according to the equivalent circuit diagram, shown at the beginning of the reference page, in the “Solar-Induced Current” on page 1-1535 section. It is the sum of the  $i^2 \cdot R$  losses for each of the resistors plus the losses in each of the diodes.

The internally generated heat due to electrical losses is a separate heating effect to that of the solar irradiation. To model thermal heating due to solar irradiation, you must account for it separately in your model and add the heat flow to the physical node connected to the solar cell thermal port.

## Parameters

- “Cell Characteristics Tab” on page 1-1540
- “Configuration Tab” on page 1-1542
- “Temperature Dependence Tab” on page 1-1542
- “Thermal Port Tab” on page 1-1543

## Cell Characteristics Tab

### Parameterize by

Select one of the following methods for block parameterization:

- **By s/c current and o/c voltage, 5 parameter** — Provide short-circuit current and open-circuit voltage that the block converts to an equivalent circuit model of the solar cell. This is the default option.
- **By equivalent circuit parameters, 5 parameter** — Provide electrical parameters for an equivalent circuit model of the solar cell using the 5-parameter solar cell model that makes the following assumptions:
  - The saturation current of the second diode is zero.
  - The parallel resistor has infinite impedance.
- **By equivalent circuit parameters, 8 parameter** — Provide electrical parameters for an equivalent circuit model of the solar cell using the 8-parameter solar cell model.

### Short-circuit current, $I_{sc}$

The current that flows when you short-circuit the solar cell. This parameter is only visible when you select **By s/c current and o/c voltage, 5 parameter** for the **Parameterize by** parameter. The default value is 7.34 A.

### Open-circuit voltage, $V_{oc}$

The voltage across the solar cell when it is not connected. This parameter is only visible when you select **By s/c current and o/c voltage, 5 parameter** for the **Parameterize by** parameter. The default value is 0.6 V.

### Diode saturation current, $I_s$

The asymptotic reverse current of the first diode for increasing reverse bias in the absence of any incident light. This parameter is only visible when you select one of the following settings:

- By equivalent circuit parameters, 5 parameter for the **Parameterize by** parameter
- By equivalent circuit parameters, 8 parameter for the **Parameterize by** parameter

The default value is  $1e-06$  A.

### **Diode saturation current, Is2**

The asymptotic reverse current of the second diode for increasing reverse bias in the absence of any incident light. This parameter is only visible when you select By equivalent circuit parameters, 8 parameter for the **Parameterize by** parameter. The default value is 0 A.

### **Solar-generated current, Iph0**

The solar-induced current when the irradiance is  $I_{r0}$ . This parameter is only visible when you select one of the following settings:

- By equivalent circuit parameters, 5 parameter for the **Parameterize by** parameter
- By equivalent circuit parameters, 8 parameter for the **Parameterize by** parameter

The default value is 7.34 A.

### **Irradiance used for measurements, Ir0**

The irradiance that produces a current of  $I_{ph0}$  in the solar cell. The default value is  $1000$  W/m<sup>2</sup>.

### **Quality factor, N**

The emission coefficient of the first diode. The default value is 1.5.

### **Quality factor, N2**

The emission coefficient of the second diode. This parameter is only visible when you select By equivalent circuit parameters, 8 parameter for the **Parameterize by** parameter. The default value is 2.

### **Series resistance, Rs**

The internal series resistance. The default value is 0  $\Omega$ .

### **Parallel resistance, Rp**

The internal parallel resistance. This parameter is only visible when you select By equivalent circuit parameters, 8 parameter for the **Parameterize by** parameter. The default value is  $\text{inf}$   $\Omega$ .

## Configuration Tab

### Number of series cells

The number of series-connected solar cells modeled by the block. The default value is 1. The value must be greater than 0.

## Temperature Dependence Tab

### First order temperature coefficient for $I_{ph}$ , TIPH1

The order of the linear increase in the solar-generated current as temperature increases. The default value is 0 1/K. The value must be greater than or equal to 0.

### Energy gap, EG

The solar cell activation energy. The default value is 1.11 eV. The value must be greater than or equal to 0.1.

### Temperature exponent for $I_s$ , TXIS1

The order of the exponential increase in the current from the first diode as temperature increases. The default value is 3. The value must be greater than or equal to 0.

### Temperature exponent for $I_{s2}$ , TXIS2

The order of the exponential increase in the current from the second diode as temperature increases. This parameter is only visible when you select **By equivalent circuit parameters**, 8 parameter for the **Parameterize by** parameter. The default value is 3. The value must be greater than or equal to 0.

### Temperature exponent for $R_s$ , TRS1

The order of the exponential increase in the series resistance as temperature increases. The default value is 0. The value must be greater than or equal to 0.

### Temperature exponent for $R_p$ , TRP1

The order of the exponential increase in the parallel resistance as temperature increases. This parameter is only visible when you select **By equivalent circuit parameters**, 8 parameter for the **Parameterize by** parameter. The default value is 0. The value must be greater than or equal to 0.

### Measurement temperature

The temperature at which the solar cell parameters were measured. The default value is 25 C. The value must be greater than 0.

### **Device simulation temperature**

The temperature at which the solar cell is simulated. The default value is 25 C. The value must be greater than 0.

### **Thermal Port Tab**

This tab appears only for blocks with exposed thermal ports. For more information, see “Thermal Port” on page 1-1539.

#### **Thermal mass**

The heat energy required to raise the temperature of the solar cell by one degree. When modeling more than one cell in series, specify the thermal mass for a single cell. This value gets multiplied internally by the number of cells to determine the total thermal mass. The default value is 100 J/K.

#### **Initial temperature**

The temperature of the solar cell at the start of simulation. The default value is 25 C.

## **Ports**

The block has the following ports:

I<sub>r</sub>

Incident irradiance

+

Positive electrical voltage

-

Negative electrical voltage

## **References**

- [1] Gow, J.A. and C.D. Manning. “Development of a Photovoltaic Array Model for Use in Power-Electronics Simulation Studies.” *IEEE Proceedings of Electric Power Applications*, Vol. 146, No. 2, 1999, pp. 193-200.

## Solenoid

Model electrical characteristics and generated force of solenoid

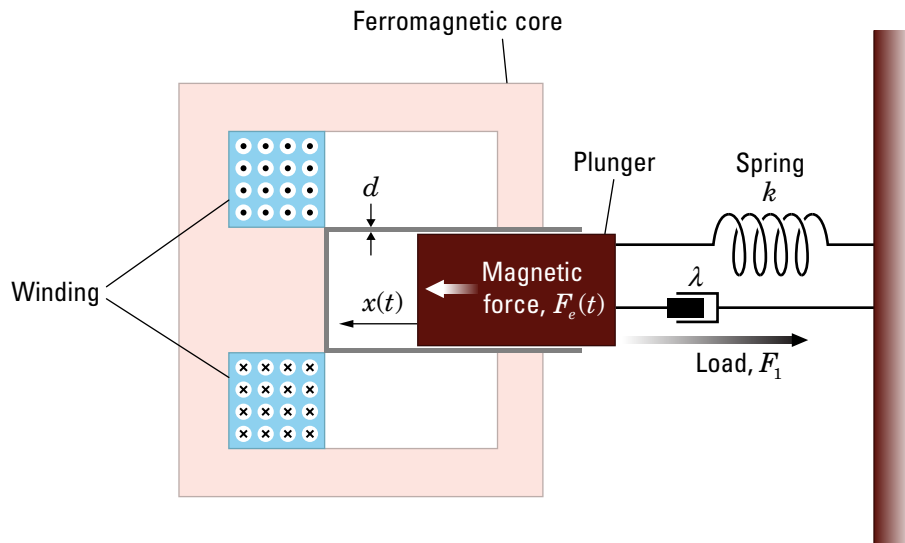


## Library

Translational Actuators

## Description

The Solenoid block represents the electrical characteristics and generated force for the solenoid in the following figure:



The return spring is optional. To remove the effects of this spring from the model, set the **Spring constant** parameter to 0.

The equation of motion for the plunger as a function of position,  $x$ , is:

$$F_l + m\ddot{x} + \lambda\dot{x} + kx = F_e$$

where  $F_e$  is the electromagnetic force,  $F_l$  is the load force,  $\lambda$  is the viscous damping term and  $m$  is the plunger mass. The electromagnetic force is related to the solenoid current and inductance by:

$$F_e = \frac{1}{2} i^2 \frac{\partial L(x)}{\partial x}$$

The inductance, which is derived in [1], can be written as:

$$\frac{\partial L(x)}{\partial x} = \frac{-\beta}{(\alpha + \beta x)^2}$$

where  $\alpha$  and  $\beta$  are constants. Plugging the preceding equation into the equation for electromagnetic force gives the force-stroke relationship of the solenoid for a current  $i_0$ :

$$F = \frac{1}{2} i_0^2 \frac{-\beta}{(\alpha + \beta x)^2}$$

The Solenoid block solves for  $\alpha$  and  $\beta$  by taking the two specified force and stroke measurements and substituting them into the preceding equation. It solves the resulting equations for  $\alpha$  and  $\beta$ .

A positive current from the electrical + to - ports creates a negative force (i.e., a pulling force) from the mechanical C to R ports.

## Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal port H on the

block icon, and adds the **Temperature Dependence** and **Thermal Port** tabs to the block dialog box.

Use the thermal port to simulate the effects of copper resistance losses that convert electrical power to heat. For more information on using thermal ports and on the **Temperature Dependence** and **Thermal Port** tab parameters, see “Simulating Thermal Effects in Rotational and Translational Actuators”.

## Variables

Use the **Variables** section of the block interface to set the priority and initial target values for the block variables prior to simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Use the **Position, X** variable to set the target for the initial plunger position at the start of simulation.

## Parameters

- “Magnetic Force Tab” on page 1-1546
- “Mechanical Tab” on page 1-1547

## Magnetic Force Tab

### Forces [F1 F2]

A vector of the force values at the two points on the force-stroke curve. The second measurement point must be at a stroke that is greater than that of the first measurement point. When the manufacturer does not provide a force-stroke curve, set F1 to the holding force (when  $X1 = 0$ ) and F2 to the pull-in force when running the solenoid at the **Rated voltage Vdc** and **Rated current Idc** values. The default value is [7.5 0.75] N.

### Stroke [X1 X2]

A vector of the stroke (plunger distance from the fully closed position) values at the two points on the force-stroke curve. The second measurement point must be at a stroke that is greater than that of the first measurement point. To ensure a finite force value, the points must meet the condition



$$\frac{X2}{X1} > \sqrt{\frac{F1}{F2}}$$

The default value is [1 5] mm.

### Rated voltage Vdc

The voltage at which the solenoid is rated to operate. This voltage value is used to measure the **Forces [F1 F2]** and **Stroke [X1 X2]** values. The default value is 50 V.

### Rated current Idc

The current that flows when the solenoid is supplied with the **Rated voltage Vdc** voltage. The default value is 0.05 A.

## Mechanical Tab

### Spring constant

Constant representing the stiffness of the spring that acts to retract the plunger when the solenoid is powered off. The force is zero when the plunger is displaced to the **Stroke for zero spring force** parameter value. The default value is 200 N/m. Set the spring constant to zero if there is no spring.

### Stroke for zero spring force

The stroke at which the spring provides no force. The default value is 5 mm.

### Damping

The term  $\lambda$  in the equation of motion for the plunger as a function of position that linearly damps the plunger motion. The default value is 1 N/(m/s). The value can be zero.

### Plunger mass

The weight of the solenoid plunger. The default value is 0.05 kg. The value can be zero.

### Maximum stroke

The maximum amount by which the plunger can be displaced. You can use this parameter to model a hard endstop that limits the stroke. The default value is Inf mm, which means no stroke limit.

### Contact stiffness

Stiffness of the plunger contact that models the hard stop at the minimum ( $x = 0$ ) and maximum ( $x =$  **Maximum stroke**) plunger positions. The default value is 1e+06 N/m.

### **Contact damping**

Damping of the plunger contact that models the hard stop at the minimum ( $x = 0$ ) and maximum ( $x =$  **Maximum stroke**) plunger positions. The default value is 500 N/(m/s).

## **Ports**

The block has the following ports:

+

Positive electrical input

-

Negative electrical input

C

Mechanical translational conserving port

R

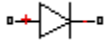
Mechanical translational conserving port

## **References**

[1] S.E. Lyshevski. *Electromechanical Systems, Electric Machines, and Applied Mechatronics*. CRC, 1999.

# SPICE Diode

Model SPICE-compatible diode



## Library

SPICE-Compatible Components/Semiconductor Devices

## Description

The SPICE Diode block represents a SPICE-compatible diode.

The SPICE Diode block model includes the following components:

- “Current-Voltage Model” on page 1-1549
- “Junction Charge Model” on page 1-1551
- “Temperature Dependence” on page 1-1551

## Current-Voltage Model

The block provides the following relationship between the diode current  $I_d$  and the diode voltage  $V_d$  after adjusting the applicable model parameters for temperature.

Applicable Range of $V_d$ Values	Corresponding $I_d$ Equation
$V_d > 80 * V_t$	$I_d = IS \left( \left( \frac{V_d}{V_t} - 79 \right) e^{80} - 1 \right) + V_d * G \text{ min}$
$80 * V_t \geq V_d \geq -3 * V_t$	$I_d = IS * (e^{V_d/V_t} - 1) + V_d * G \text{ min}$

Applicable Range of $V_d$ Values	Corresponding $I_d$ Equation
$-3 * V_t > V_d \geq -BV$	$I_d = -IS \left( 1 + \frac{27}{(V_d / V_t)^3 e^3} \right) + V_d * G \text{ min}$
$V_d < -BV$	$I_d = -IBV * \left( e^{-(BV+V_d)/V_t} - 1 \right) -$ $IS * \left( 1 - \left( \frac{3}{e * BV / V_t} \right)^3 \right) + V_d * G \text{ min}$

Where:

- $IS$  is the **Saturation current, IS** parameter value.
- $V_t = N * k * T / q$
- $N$  is the **Emission coefficient, N** parameter value.
- $q$  is the elementary charge on an electron.
- $k$  is the Boltzmann constant.
- $T$  is the diode temperature:
  - If you select Device temperature for the **Model temperature dependence using** parameter,  $T$  is the sum of the **Circuit temperature** value plus the **Offset local circuit temperature, TOFFSET** parameter value. The **Circuit temperature** value comes from the SPICE Environment Parameters block, if one exists in the circuit. Otherwise, it comes from the default value for this block.
  - If you select Fixed temperature for the **Model temperature dependence using** parameter,  $T$  is the **Fixed circuit temperature, TFIXED** parameter value.
- $GMIN$  is the diode minimum conductance. By default,  $GMIN$  matches the **Minimum conductance, GMIN** parameter of the SPICE Environment Parameters block, whose default value is 1e-12. To change  $GMIN$ , add a SPICE Environment Parameters block to your model and set the **Minimum conductance, GMIN** parameter to the desired value.
- $BV$  is the **Reverse breakdown voltage, BV** parameter value.

## Junction Charge Model

The block provides the following relationship between the diode charge  $Q_d$  and the diode voltage  $V_d$  after adjusting the applicable model parameters for temperature.

Applicable Range of $V_d$ Values	Corresponding $Q_d$ Equation
$V_d < FC * VJ$	$Q_d = TT * I_d + CJO * VJ * \frac{1 - \left(1 - \frac{V_d}{VJ}\right)^{1-MG}}{1 - MG}$
$V_d \geq FC * VJ$	$Q_d = TT * I_d + CJO * \left( F1 + \frac{F3 * (V_d - FC * VJ) + \left(\frac{MG}{2 * VJ}\right) * (V_d^2 - (FC * VJ)^2)}{F2} \right)$

Where:

- $FC$  is the **Capacitance coefficient**,  $FC$  parameter value.
- $VJ$  is the **Junction potential**,  $VJ$  parameter value.
- $TT$  is the **Transit time**,  $TT$  parameter value.
- $CJO$  is the **Zero-bias junction capacitance**,  $CJO$  parameter value.
- $MG$  is the **Grading coefficient**,  $M$  parameter value.
- $F1 = VJ * (1 - (1 - FC)^{(1-MG)}) / (1 - MG)$
- $F2 = (1 - FC)^{(1+MG)}$
- $F3 = 1 - FC * (1 + MG)$

## Temperature Dependence

Several diode parameters depend on temperature. There are two ways to specify the diode temperature:

- When you select Device temperature for the **Model temperature dependence using** parameter, the diode temperature is

$$T = T_C + T_O$$

where:

- $T_C$  is the **Circuit temperature** parameter value from the SPICE Environment Parameters block. If this block doesn't exist in the circuit,  $T_C$  is the default value of this parameter.
- $T_O$  is the **Offset local circuit temperature, TOFFSET** parameter value.
- When you select Fixed temperature for the **Model temperature dependence using** parameter, the diode temperature is the **Fixed circuit temperature, TFIXED** parameter value.

The block provides the following relationship between the saturation current  $IS$  and the diode temperature  $T$ :

$$IS(T) = IS * \left( T / T_{meas} \right)^{\frac{XTI}{ND}} * e^{\left( \frac{T}{T_{meas}} - 1 \right) * \frac{EG}{V_t}}$$

where:

- $IS$  is the **Transport saturation current, IS** parameter value.
- $T_{meas}$  is the **Parameter extraction temperature, TMEAS** parameter value.
- $XTI$  is the **Saturation current temperature exponent, XTI** parameter value.
- $ND$  is the **Emission coefficient, N** parameter value.
- $EG$  is the **Activation energy, EG** parameter value.
- $V_t = kT/q$ .

The block provides the following relationship between the junction potential  $VJ$  and the diode temperature  $T$ :

$$VJ(T) = VJ * \left( \frac{T}{T_{meas}} \right) - \frac{3 * k * T}{q} * \log \left( \frac{T}{T_{meas}} \right) - \left( \frac{T}{T_{meas}} \right) * EG_{T_{meas}} + EG_T$$

where:

- $VJ$  is the **Junction potential**,  $VJ$  parameter value.

- 

$$EG_{T_{meas}} = 1.16eV - (7.02e - 4 * T_{meas}^2) / (T_{meas} + 1108)$$

- 

$$EG_T = 1.16eV - (7.02e - 4 * T^2) / (T + 1108)$$

The block provides the following relationship between the junction capacitance  $CJO$  and the diode temperature  $T$ :

$$CJO(T) = CJO * \left[ 1 + MG * \left( 400e - 6 * (T - T_{meas}) - \frac{VJ(T) - VJ}{VJ} \right) \right]$$

where  $CJO$  is the **Zero-bias junction capacitance**,  $CJO$  parameter value.

## Basic Assumptions and Limitations

The model is based on the following assumptions:

- The SPICE Diode block does not support noise analysis.
- The SPICE Diode block applies initial conditions across junction capacitors and not across the block ports.

## Parameters

### Main Tab

#### Device area, AREA

The diode area. This value multiplies the **Saturation current**,  $IS$ , **Zero-bias junction capacitance**  $CJO$ , and **Reverse breakdown current**,  $IBV$  parameter values. It divides the **Ohmic resistance**,  $RS$  parameter value. The default value is 1  $m^2$ . The value must be greater than 0.

#### Number of parallel devices, SCALE

The number of parallel diodes the block represents. This value multiplies the output current and device charges. The default value is 1. The value must be greater than 0.

### **Saturation current, IS**

The magnitude of the current that the ideal diode equation approaches asymptotically for very large reverse bias levels. The default value is  $1e-14$  A/m<sup>2</sup>. The value must be greater than or equal to 0.

### **Ohmic resistance, RS**

The series diode connection resistance. The default value is  $0.01$  m<sup>2</sup>\*Ω. The value must be greater than or equal to 0.

### **Emission coefficient, N**

The diode emission coefficient or ideality factor. The default value is 1. The value must be greater than 0.

## **Junction Capacitance Tab**

### **Model junction capacitance**

Select one of the following options for modeling the junction capacitance:

- **No** — Do not include junction capacitance in the model. This is the default option.
- **Yes** — Specify zero-bias junction capacitance, junction potential, grading coefficient, forward-bias depletion capacitance coefficient, and transit time.

### **Zero-bias junction capacitance CJO**

The value of the capacitance placed in parallel with the exponential diode term. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is 0 F/m<sup>2</sup>. The value must be greater than or equal to 0.

### **Junction potential, VJ**

The junction potential. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is 1 V. The value must be greater than 0.01 V.

### **Grading coefficient, M**

The grading coefficient. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is 0.5. The value must be greater than 0 and less than 0.9.

### **Capacitance coefficient, FC**

The fitting coefficient that quantifies the decrease of the depletion capacitance with applied voltage. This parameter is only visible when you select **Yes** for the **Model**



**junction capacitance** parameter. The default value is 0.5. The value must be greater than or equal to 0 and less than 0.95.

### **Transit time, TT**

The transit time of the minority carriers that cause diffusion capacitance. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is 0 s. The value must be greater than or equal to 0.

### **Specify initial condition**

Select one of the following options for specifying an initial condition:

- **No** — Do not specify an initial condition for the model. This is the default option.
- **Yes** — Specify the initial diode voltage.

---

**Note** The SPICE Diode block applies the initial diode voltage across the junction capacitors and not across the ports.

---

### **Initial voltage, V0**

Diode voltage at the start of the simulation. This parameter is only visible when you select **Yes** for the **Model junction capacitance** and **Yes** for the **Specify initial condition** parameter. The default value is 0 V.

---

**Note** The block applies the initial condition across the diode junction, so the initial condition is only effective when charge storage is included, i.e. when one or both of the **Zero-bias junction capacitance, CJO** and **Transit time, TT** parameters are greater than zero.

---

## **Reverse Breakdown Tab**

### **Model reverse breakdown**

Select one of the following options for modeling the diode reverse breakdown:

- **No** — Don't model reverse breakdown. This is the default option.
- **Yes** — Introduce a second exponential term to the diode I-V relationship, thereby modeling a rapid increase in conductance as the breakdown voltage is exceeded.

### **Reverse breakdown current, IBV**

The diode current that corresponds to the **Reverse breakdown voltage, BV** value. This parameter is only visible when you select **Yes** for the **Model reverse**

**breakdown** parameter. The default value is  $0.001 \text{ A/m}_2$ . The value must be greater than  $0$ .

---

**Note** The Diode model does not use this parameter at this time.

---

### **Reverse breakdown voltage, BV**

The voltage below which to model the rapid increase in conductance that occurs at diode breakdown. This parameter is only visible when you select **Yes** for the **Model reverse breakdown** parameter. The default value is  $\text{Inf V}$ . The value must be greater than or equal to  $0$ .

## **Temperature Tab**

### **Model temperature dependence using**

Select one of the following options for modeling the diode temperature dependence:

- **Device temperature** — Use the device temperature, which is the **Circuit temperature** parameter value (from the SPICE Environment Parameters block, if one exists in the circuit, or the default value for this block otherwise) plus the **Offset local circuit temperature, TOFFSET** parameter value.
- **Fixed temperature** — Use a temperature that is independent of the circuit temperature to model temperature dependence.

### **Saturation current temperature exponent, XTI**

The order of the exponential increase in the saturation current as temperature increases. This parameter is only visible when you select **Device temperature** for the **Model temperature dependence using** parameter. The default value is  $3$ . The value must be greater than  $0$ .

### **Activation energy, EG**

The diode activation energy. This parameter is only visible when you select **Device temperature** for the **Model temperature dependence using** parameter. The default value is  $1.11 \text{ eV}$ . The value must be greater than or equal to  $0.1$ .

### **Offset local circuit temperature, TOFFSET**

The amount by which the diode temperature differs from the circuit temperature. This parameter is only visible when you select **Device temperature** for the **Model temperature dependence using** parameter. The default value is  $0 \text{ K}$ .

**Parameter extraction temperature, TMEAS**

The temperature at which the diode parameters were measured. The default value is 300.15 K. The value must be greater than 0.

**Fixed circuit temperature, TFIXED**

The temperature at which to simulate the diode. This parameter is only visible when you select `Fixed` temperature for the **Model temperature dependence using** parameter. The default value is 300.15 K. The value must be greater than 0.

## Ports

The block has the following ports:

+

Positive electrical voltage.

-

Negative electrical voltage.

## See Also

Diode

## SPICE Environment Parameters

Set parameters that apply to all connected SPICE-compatible blocks



### Library

SPICE-Compatible Components/Utilities

### Description

The SPICE Environment Parameters block lets you set parameters that apply to all SPICE-compatible blocks in an electrical network:

- Circuit temperature
- Minimum conductance

If your Simulink model does not contain a SPICE Environment Parameters block, all blocks use the default values of these parameters. You must connect every network in the system to a SPICE Environment Parameters block to override the default values.

---

**Note** The simple semiconductor models in the Semiconductors sublibrary are not temperature dependent, so the SPICE Environment Parameters block only changes the minimum conductance parameter used by the exponential diode and bipolar transistor models.

---

### Parameters

#### Circuit temperature

The temperature of the connected SPICE-compatible blocks. The default value is 300.15 K.

**Minimum conductance GMIN**

The minimum conductance used by some blocks. The default value is  $1e-12$   $1/\Omega$ .

**Ports**

The block has the following ports:

OUT

Electrical output.

## SPICE NJFET

Model SPICE-compatible N-Channel JFET



## Library

SPICE-Compatible Components/Semiconductor Devices

## Description

The NJFET block represents a SPICE-compatible N-channel JFET.

The NJFET block model includes the following components:

- “Gate-Source Current-Voltage Model” on page 1-1560
- “Gate-Drain Current-Voltage Model” on page 1-1561
- “Drain-Source Current-Voltage Model” on page 1-1562
- “Junction Charge Model” on page 1-1563
- “Temperature Dependence” on page 1-1564

## Gate-Source Current-Voltage Model

The block provides the following relationship between the gate-source current  $I_{gs}$  and the gate-source voltage  $V_{gs}$  after adjusting the applicable model parameters for temperature.

Applicable Range of $V_{gs}$ Values	Corresponding $I_{gs}$ Equation
$V_{gs} > 80 * V_t$	$I_{gs} = IS * \left( \left( \frac{V_{gs}}{V_t} - 79 \right) e^{80} - 1 \right) + V_{gs} * G \text{ min}$

Applicable Range of $V_{gs}$ Values	Corresponding $I_{gs}$ Equation
$80 * V_t \geq V_{gs}$	$I_{gs} = IS * (e^{V_{gs}/V_t} - 1) + V_{gs} * G \text{ min}$

Where:

- $IS$  is the **Saturation current, IS** parameter value.
- $V_t = ND * k * T / q$
- $ND$  is the **Emission coefficient, N** parameter value.
- $q$  is the elementary charge on an electron.
- $k$  is the Boltzmann constant.
- $T$  is the diode temperature:
  - If you select Device temperature for the **Model temperature dependence using** parameter,  $T$  is the sum of the **Circuit temperature** value plus the **Offset local circuit temperature, TOFFSET** parameter value. The **Circuit temperature** value comes from the SPICE Environment Parameters block, if one exists in the circuit. Otherwise, it comes from the default value for this block.
  - If you select Fixed temperature for the **Model temperature dependence using** parameter,  $T$  is the **Fixed circuit temperature, TFIXED** parameter value.
- $GMIN$  is the diode minimum conductance. By default,  $GMIN$  matches the **Minimum conductance GMIN** parameter of the SPICE Environment Parameters block, whose default value is 1e-12. To change  $GMIN$ , add a SPICE Environment Parameters block to your model and set the **Minimum conductance GMIN** parameter to the desired value.

## Gate-Drain Current-Voltage Model

The block provides the following relationship between the gate-drain current  $I_{gd}$  and the gate-drain voltage  $V_{gd}$  after adjusting the applicable model parameters for temperature.

Applicable Range of $V_{gd}$ Values	Corresponding $I_{gd}$ Equation
$V_{gd} > 80 * V_t$	$I_{gd} = IS * \left( \left( \frac{V_{gd}}{V_t} - 79 \right) e^{80} - 1 \right) + V_{gd} * G \text{ min}$
$80 * V_t \geq V_{gd}$	$I_{gd} = IS * \left( e^{V_{gd}/V_t} - 1 \right) + V_{gd} * G \text{ min}$

## Drain-Source Current-Voltage Model

The block provides the following relationship between the drain-source current  $I_{ds}$  and the drain-source voltage  $V_{ds}$  in normal mode ( $V_{ds} \geq 0$ ) after adjusting the applicable model parameters for temperature.

Applicable Range of $V_{gs}$ and $V_{gd}$ Values	Corresponding $I_{ds}$ Equation
$V_{gs} - V_{to} \leq 0$	$I_{ds} = 0$
$0 < V_{gs} - V_{to} \leq V_{ds}$	$I_{ds} = \beta (V_{gs} - V_{to})^2 (1 + \lambda V_{ds})$
$0 < V_{ds} < V_{gs} - V_{to}$	$I_{ds} = \beta V_{ds} (2(V_{gs} - V_{to}) - V_{ds}) (1 + \lambda V_{ds})$

Where:

- $V_{to}$  is the **Threshold voltage, VTO** parameter value.
- $\beta$  is the **Transconductance, BETA** parameter value.
- $\lambda$  is the **Channel modulation, LAMBDA** parameter value.

The block provides the following relationship between the drain-source current  $I_{ds}$  and the drain-source voltage  $V_{ds}$  in inverse mode ( $V_{ds} < 0$ ) after adjusting the applicable model parameters for temperature.



Applicable Range of $V_{gs}$ and $V_{gd}$ Values	Corresponding $I_{ds}$ Equation
$V_{gd} - V_{to} \leq 0$	$I_{ds} = 0$
$0 < V_{gd} - V_{to} \leq -V_{ds}$	$I_{ds} = -\beta(V_{gd} - V_{to})^2(1 - \lambda V_{ds})$
$0 < -V_{ds} < V_{gd} - V_{to}$	$I_{ds} = \beta V_{ds} (2(V_{gd} - V_{to}) + V_{ds})(1 - \lambda V_{ds})$

## Junction Charge Model

The block provides the following relationship between the gate-source charge  $Q_{gs}$  and the gate-source voltage  $V_{gs}$  after adjusting the applicable model parameters for temperature.

Applicable Range of $V_{gs}$ Values	Corresponding $Q_{gs}$ Equation
$V_{gs} < FC * VJ$	$Q_{gs} = \frac{CGS * VJ * \left(1 - \left(1 - \frac{V_{gs}}{VJ}\right)^{1-MG}\right)}{1 - MG}$
$V_{gs} \geq FC * VJ$	$Q_{gs} = CGS * \left( F1 + \frac{F3 * (V_{gs} - FC * VJ) + \frac{MG * (V_{gs}^2 - (FC * VJ)^2)}{2 * VJ}}{F2} \right)$

Where:

- $FC$  is the **Capacitance coefficient FC** parameter value.
- $VJ$  is the **Junction potential VJ** parameter value.
- $CGS$  is the **Zero-bias GS capacitance, CGS** parameter value.
- $MG$  is the **Grading coefficient, M** parameter value.

•

$$F1 = \frac{VJ * (1 - (1 - FC)^{1-MG})}{1 - MG}$$

•

$$F2 = (1 - FC)^{1+MG}$$

•

$$F3 = 1 - FC * (1 + MG)$$

The block provides the following relationship between the gate-drain charge  $Q_{gd}$  and the gate-drain voltage  $V_{gd}$  after adjusting the applicable model parameters for temperature.

Applicable Range of $V_{gd}$ Values	Corresponding $Q_{gd}$ Equation
$V_{gd} < FC * VJ$	$Q_{gd} = \frac{CGD * VJ * \left( 1 - \left( 1 - \frac{V_{gd}}{VJ} \right)^{1-MG} \right)}{1 - MG}$
$V_{gd} \geq FC * VJ$	$Q_{gd} = CGD * \left( F1 + \frac{F3 * (V_{gd} - FC * VJ) + \frac{MG * (V_{gd}^2 - (FC * VJ)^2)}{2 * VJ}}{F2} \right)$

Where:

- $CGD$  is the **Zero-bias GD capacitance, CGD** parameter value.

## Temperature Dependence

Several transistor parameters depend on temperature. There are two ways to specify the transistor temperature:

- When you select Device temperature for the **Model temperature dependence using** parameter, the transistor temperature is

$$T = T_C + T_O$$

where:

- $T_C$  is the **Circuit temperature** parameter value from the SPICE Environment Parameters block. If this block doesn't exist in the circuit,  $T_C$  is the default value of this parameter.
- $T_O$  is the **Offset local circuit temperature, TOFFSET** parameter value.
- When you select Fixed temperature for the **Model temperature dependence using** parameter, the transistor temperature is the **Fixed circuit temperature, TFIXED** parameter value.

The block provides the following relationship between the saturation current  $IS$  and the transistor temperature  $T$ :

$$IS(T) = IS * \left( T / T_{meas} \right)^{\frac{XTI}{ND}} * e^{\left( \frac{T}{T_{meas}} - 1 \right) * \frac{EG}{V_t}}$$

where:

- $IS$  is the **Saturation current, IS** parameter value.
- $T_{meas}$  is the **Parameter extraction temperature, TMEAS** parameter value.
- $XTI$  is the **Saturation current temperature exponent, XTI** parameter value.
- $EG$  is the **Energy gap, EG** parameter value.
- $V_t = ND * k * T / q$
- $ND$  is the **Emission coefficient, N** parameter value.

The block provides the following relationship between the junction potential  $VJ$  and the transistor temperature  $T$ :

$$VJ(T) = VJ * \left( \frac{T}{T_{meas}} \right) - \frac{3 * k * T}{q} * \log \left( \frac{T}{T_{meas}} \right) - \left( \frac{T}{T_{meas}} \right) * EG_{T_{meas}} + EG_T$$

where:

- $VJ$  is the **Junction potential VJ** parameter value.

- $EG_{T_{meas}} = 1.16eV - (7.02e - 4 * T_{meas}^2) / (T_{meas} + 1108)$
- $EG_T = 1.16eV - (7.02e - 4 * T^2) / (T + 1108)$

The block provides the following relationship between the gate-source junction capacitance  $CGS$  and the transistor temperature  $T$ :

$$CGS(T) = CGS * \left[ 1 + MG * \left( 400e - 6 * (T - T_{meas}) - \frac{VJ(T) - VJ}{VJ} \right) \right]$$

where:

- $CGS$  is the **Zero-bias GS capacitance, CGS** parameter value.

The block uses the  $CGS(T)$  equation to calculate the gate-drain junction capacitance by substituting  $CGD$  (the **Zero-bias GD capacitance, CGD** parameter value) for  $CGS$ .

The block provides the following relationship between the forward and reverse beta and the transistor temperature  $T$ :

$$\beta(T) = \beta * \left( \frac{T}{T_{meas}} \right)$$

where  $\beta$  is the **Transconductance, BETA** parameter value.

## Basic Assumptions and Limitations

The model is based on the following assumptions:

- The NJFET block does not support noise analysis.
- The NJFET block applies initial conditions across junction capacitors and not across the block ports.

# Parameters

## Main Tab

### Device area, AREA

The transistor area. This value multiplies the **Transconductance, BETA, Zero-bias GS capacitance, CGS, Zero-bias GD capacitance, CGD, and Saturation current, IS** parameter values. It divides the **Source resistance, RS** and **Drain resistance, RD** parameter values. The default value is  $1 \text{ m}^2$ . The value must be greater than 0.

### Number of parallel devices, SCALE

The number of parallel transistors the block represents. This value multiplies the output current and device charges. The default value is 1. The value must be greater than 0.

### Threshold voltage, VTO

The gate-source voltage above which the transistor produces a nonzero drain current. The default value is  $-2 \text{ V}$ .

### Transconductance, BETA

The derivative of drain current with respect to gate voltage. The default value is  $1\text{e-}04 \text{ A/m}^2/\text{V}^2$ . The value must be greater than or equal to 0.

### Channel modulation, LAMBDA

The channel-length modulation. The default value is  $0 \text{ 1/V}$ .

### Saturation current, IS

The magnitude of the current that the ideal diode equation approaches asymptotically for very large reverse bias levels. The default value is  $1\text{e-}14 \text{ A/m}^2$ . The value must be greater than or equal to 0.

### Emission coefficient, N

The transistor emission coefficient or ideality factor. The default value is 1. The value must be greater than 0.

### Source resistance, RS

The transistor source resistance. The default value is  $1\text{e-}4 \text{ m}^2*\Omega$ . The value must be greater than or equal to 0.

### Drain resistance, RD

The transistor drain resistance. The default value is  $0.01 \text{ m}^2*\Omega$ . The value must be greater than or equal to 0.

## Junction Capacitance Tab

### Model junction capacitance

Select one of the following options for modeling the junction capacitance:

- **No** — Do not include junction capacitance in the model. This is the default option.
- **Yes** — Specify zero-bias junction capacitance, junction potential, grading coefficient, forward-bias depletion capacitance coefficient, and transit time.

### Zero-bias GS capacitance, CGS

The value of the capacitance placed between the gate and the source. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is  $0 \text{ F/m}^2$ . The value must be greater than or equal to  $0$ .

### Zero-bias GD capacitance, CGD

The value of the capacitance placed between the gate and the drain. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is  $0 \text{ F/m}^2$ . The value must be greater than or equal to  $0$ .

### Junction potential VJ

The junction potential. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is  $1 \text{ V}$ . The value must be greater than  $0.01 \text{ V}$ .

### Grading coefficient, M

The transistor grading coefficient. The default value is  $0.5$ . The value must be greater than  $0$  and less than  $0.9$ .

### Capacitance coefficient FC

The fitting coefficient that quantifies the decrease of the depletion capacitance with applied voltage. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is  $0.5$ . The value must be greater than or equal to  $0$  and less than or equal to  $0.95$ .

### Specify initial condition

Select one of the following options for specifying an initial condition:

- **No** — Do not specify an initial condition for the model. This is the default option.
- **Yes** — Specify the initial diode voltage.

---

**Note** The NJFET block applies the initial diode voltage across the junction capacitors and not across the ports.

---

### Initial condition voltage ICVDS

Drain-source voltage at the start of the simulation. This parameter is only visible when you select **Yes** for the **Model junction capacitance** and **Yes** for the **Specify initial condition** parameter. The default value is 0 V.

### Initial condition voltage ICVGS

Gate-source voltage at the start of the simulation. This parameter is only visible when you select **Yes** for the **Model junction capacitance** and **Yes** for the **Specify initial condition** parameter. The default value is 0 V.

## Temperature Tab

### Model temperature dependence using

Select one of the following options for modeling the diode temperature dependence:

- **Device temperature** — Use the device temperature, which is the **Circuit temperature** value plus the **Offset local circuit temperature, TOFFSET** value. The **Circuit temperature** value comes from the SPICE Environment Parameters block, if one exists in the circuit. Otherwise, it comes from the default value for this block.
- **Fixed temperature** — Use a temperature that is independent of the circuit temperature to model temperature dependence.

### Saturation current temperature exponent, XTI

The order of the exponential increase in the saturation current as temperature increases. The default value is 0. The value must be greater than or equal to 0.

### Activation energy, EG

The energy gap that affects the increase in the saturation current as temperature increases. The default value is 1.11 eV. The value must be greater than 0.1 eV.

### Offset local circuit temperature, TOFFSET

The amount by which the transistor temperature differs from the circuit temperature. This parameter is only visible when you select **Device temperature** for the **Model temperature dependence using** parameter. The default value is 0 K.

### **Fixed circuit temperature, TFIXED**

The temperature at which to simulate the transistor. This parameter is only visible when you select `Fixed temperature` for the **Model temperature dependence using** parameter. The default value is `300.15` K. The value must be greater than `0`.

### **Parameter extraction temperature, TMEAS**

The temperature at which the transistor parameters were measured. The default value is `300.15` K. The value must be greater than `0`.

## **Ports**

The block has the following ports:

G

Electrical conserving port associated with the transistor gate terminal.

D

Electrical conserving port associated with the transistor drain terminal.

S

Electrical conserving port associated with the transistor source terminal.

## **References**

- [1] G. Massobrio and P. Antognetti. *Semiconductor Device Modeling with SPICE*. 2nd Edition, McGraw-Hill, 1993. Chapter 3.

## **See Also**

N-Channel JFET, SPICE PJFET



# SPICE NMOS

Model SPICE-compatible N-Channel MOSFET



## Library

SPICE-Compatible Components/Semiconductor Devices

## Description

The NMOS block represents a SPICE-compatible N-channel MOSFET.

The NMOS block model includes the following components:

- “Resistance Calculations” on page 1-1571
- “Bulk-Source Diode Model” on page 1-1572
- “Bulk-Drain Diode Model” on page 1-1573
- “Level 1 Drain Current Model” on page 1-1574
- “Level 3 Drain Current Model” on page 1-1577
- “Junction Charge Model” on page 1-1582
- “Temperature Dependence” on page 1-1586

## Resistance Calculations

The following table shows how the NMOS block calculates the transistor drain resistance. The abbreviations in the table represent the values of the following block parameters:

- **Drain resistance, RD**
- **Sheet resistance, RSH**
- **Number of drain squares, NRD**

Drain resistance, RD Parameter	Sheet resistance, RSH Parameter	Drain Resistance
NaN	NaN	0
RD	NaN or RSH	RD
NaN	RSH	RSH*NRD

The following table shows how the NMOS block calculates the transistor source resistance. The abbreviations in the table represent the values of the following block parameters:

- **Source resistance, RS**
- **Sheet resistance, RSH**
- **Number of source squares, NRS**

Source resistance, RS Parameter	Sheet resistance, RSH Parameter	Source Resistance
NaN	NaN	0
RS	NaN or RSH	RS
NaN	RSH	RSH*NRS

## Bulk-Source Diode Model

The block provides the following relationship between the bulk-source current  $I_{bs}$  and the bulk-source voltage  $V_{bs}$  after adjusting the applicable model parameters for temperature.

Applicable Range of $V_{bs}$ Values	Corresponding $I_{gs}$ Equation
$V_{bs} > 80 * V_{tn}$	$I_{bs} = IS_{bs} * \left( \left( \frac{V_{bs}}{V_{tn}} - 79 \right) e^{80} - 1 \right) + V_{bs} * G \text{ min}$
$80V_{tn} \geq V_{bs}$	$I_{bs} = IS_{bs} * \left( e^{V_{bs}/V_{tn}} - 1 \right) + V_{bs} * G \text{ min}$

Where:

- $IS_{bs}$  is
  - The product of the **Bulk jct sat current density, JS** parameter value and the **Area of source, AS** parameter value if both these parameter values and the **Area of drain, AD** parameter value are nonzero.
  - The **Bulk saturation current, IS** parameter value, otherwise.
- $V_{tn} = NkT/q$
- $q$  is the elementary charge on an electron, 1.6021918e-19 C.
- $N$  is the **Emission coefficient, N** parameter value.
- $k$  is the Boltzmann constant.
- $T$  is the diode temperature:
  - If you select Device temperature for the **Model temperature dependence using** parameter,  $T$  is the sum of the **Circuit temperature** value plus the **Offset local circuit temperature, TOFFSET** parameter value. The **Circuit temperature** value comes from the SPICE Environment Parameters block, if one exists in the circuit. Otherwise, it comes from the default value for this block.
  - If you select Fixed temperature for the **Model temperature dependence using** parameter,  $T$  is the **Fixed circuit temperature, TFIXED** parameter value.
- $GMIN$  is the diode minimum conductance. By default,  $GMIN$  matches the **Minimum conductance GMIN** parameter of the SPICE Environment Parameters block, whose default value is 1e-12. To change  $GMIN$ , add a SPICE Environment Parameters block to your model and set the **Minimum conductance GMIN** parameter to the desired value.

## Bulk-Drain Diode Model

The block provides the following relationship between the bulk-drain current  $I_{bd}$  and the bulk-drain voltage  $V_{bd}$  after adjusting the applicable model parameters for temperature.

Applicable Range of $V_{bs}$ Values	Corresponding $I_{gs}$ Equation
$V_{bd} > 80 * V_{tn}$	$I_{bd} = IS_{bd} * \left( \left( \frac{V_{bd}}{V_{tn}} - 79 \right) e^{80} - 1 \right) + V_{bd} * G \text{ min}$

Applicable Range of $V_{bs}$ Values	Corresponding $I_{gs}$ Equation
$80V_{tn} \geq V_{bd}$	$I_{bd} = IS_{bd} * (e^{V_{bd}/V_{in}} - 1) + V_{bd} * G \text{ min}$

Where:

- $IS_{bd}$  is
  - The product of the **Bulk jct sat current density, JS** parameter value and the **Area of drain, AD** parameter value if both these parameter values and the **Area of source, AS** parameter value are nonzero.
  - The **Bulk saturation current, IS** parameter value, otherwise.

## Level 1 Drain Current Model

The block provides the following relationship between the drain current  $I_d$  and the drain-source voltage  $V_{ds}$  in normal mode ( $V_{ds} \geq 0$ ) after adjusting the applicable model parameters for temperature.

### Normal Mode

Applicable Range of $V_{gs}$ and $V_{ds}$ Values	Corresponding $I_d$ Equation
$V_{gs} - V_{on} \leq 0$	$I_d = 0$
$0 < V_{gs} - V_{on} \leq V_{ds}$	$I_d = BETA * (V_{gs} - V_{on})^2 \frac{(1 + LAMBDA * V_{ds})}{2}$
$0 < V_{ds} < V_{gs} - V_{on}$	$I_d = BETA * V_{ds} \left( (V_{gs} - V_{on}) - \frac{V_{ds}}{2} \right) (1 + LAMBDA * V_{ds})$

Where:

- $V_{on}$  is:

- $MTYPE * VBI + GAMMA \sqrt{PHI - V_{bs}}$  if  $V_{bs} \leq 0$ .
- $MTYPE * VBI + GAMMA \left( \sqrt{PHI} - \frac{V_{bs}}{2\sqrt{PHI}} \right)$  if  $0 < V_{bs} \leq 2 * PHI$ .
- $MTYPE * VBI$  if  $V_{bs} > 2 * PHI$ .
- $MTYPE$  is 1.
- $BETA$  is  $KP * WIDTH / (LENGTH - 2 * LD)$
- $KP$  is:
  - The **Transconductance, KP** parameter value, if this parameter has a numerical value.
  - $U0 * 3.9 * \epsilon_0 / TOX$ , if **Transconductance, KP** is NaN and you specify values for both the **Oxide thickness, TOX** and **Substrate doping, NSUB** parameters.
- $WIDTH$  is the **Width of channel, WIDTH** parameter value.
- $LENGTH$  is the **Length of channel, LENGTH** parameter value.
- $LD$  is the **Lateral diffusion, LD** parameter value.
- $VBI$  is an built-in voltage value the block uses in calculations. The value is a function of temperature. For a detailed definition, see "Temperature Dependence" on page 1-1586.
- $PHI$  is:
  - The **Surface potential, PHI** parameter value, if this parameter has a numerical value.
  - $2 * kT_{meas} / q * \log(NSUB / n_i)$ , if **Surface potential, PHI** is NaN and you specify values for both the **Oxide thickness, TOX** and **Substrate doping, NSUB** parameters.
- $LAMBDA$  is the **Channel modulation, LAMBDA** parameter value.
- $GAMMA$  is:
  - The **Bulk threshold, GAMMA** parameter value, if this parameter has a numerical value.

- $TOX * \sqrt{2 * 11.7 * \epsilon_0 * q * NSUB} / (3.9 * \epsilon_0)$ , if **Bulk threshold**, **GAMMA** is NaN and you specify values for both the **Oxide thickness**, **TOX** and **Substrate doping**, **NSUB** parameters.
- $\epsilon_0$  is the permittivity of free space, 8.854214871e-12 F/m.
- $n_i$  is the carrier concentration of intrinsic silicon, 1.45e10 cm<sup>-3</sup>.

The block provides the following relationship between the drain current  $I_d$  and the drain-source voltage  $V_{ds}$  in inverse mode ( $V_{ds} < 0$ ) after adjusting the applicable model parameters for temperature.

### Inverse Mode

Applicable Range of $V_{gd}$ and $V_{ds}$ Values	Corresponding $I_d$ Equation
$V_{gd} - V_{on} \leq 0$	$I_d = 0$
$0 < V_{gd} - V_{on} \leq -V_{ds}$	$I_d = -BETA (V_{gd} - V_{on})^2 (1 - LAMBDA * V_{ds}) / 2$
$0 < V_{ds} < V_{gd} - V_{on}$	$I_d = BETA * V_{ds} \left( (V_{gd} - V_{on}) + V_{ds} / 2 \right) (1 - LAMBDA * V_{ds})$

Where:

- $V_{on}$  is:
  - $MTYPE * VBI + GAMMA \sqrt{PHI - V_{bd}}$  if  $V_{bd} \leq 0$ .
  - $MTYPE * VBI + GAMMA \left( \sqrt{PHI} - \frac{V_{bd}}{2\sqrt{PHI}} \right)$  if  $0 < V_{bd} \leq 2 * PHI$ .
  - $MTYPE * VBI$  if  $V_{bd} > 2 * PHI$ .

## Level 3 Drain Current Model

The block provides the following model for drain current  $I_{ds}$  in normal mode ( $V_{ds} \geq 0$ ) after adjusting the applicable model parameters for temperature.

$$I_{DS} = I_{DS0} * Scale_{VMAX} * Scale_{LChan} * Scale_{INV}$$

Where:

- $I_{DS0}$  is the Basic Drain Current Model on page 1-1577.
- $Scale_{VMAX}$  is the Velocity Saturation Scaling on page 1-1580.
- $Scale_{LChan}$  is the Channel Length Modulation Scaling on page 1-1580.
- $Scale_{INV}$  is the Weak Inversion Scaling on page 1-1582.

The blocks uses the same model for drain current in inverse mode ( $V_{ds} < 0$ ), with the following substitutions:

- $V_{bs} - V_{ds}$  for  $V_{bs}$
- $V_{gs} - V_{ds}$  for  $V_{ds}$
- $-V_{ds}$  for  $V_{ds}$

The block provides the following relationship between the drain current  $I_{ds}$  and the drain-source voltage  $V_{ds}$ :

$$I_{DS0} = BETA * F_{gate} * \left( V_{GSX} - V_{TH} - \frac{1 + F_B}{2} * V_{DSX} \right) * V_{DSX}$$

- The block calculates BETA as described in “Level 1 Drain Current Model” on page 1-1574.
- The block calculates  $F_{GATE}$  using the following equation:

$$F_{gate} = \frac{1}{1 + THETA * (V_{gsx} - V_{TH})}$$

- *THETA* is the **Vgs dependence on mobility, THETA** parameter value.

- 

$$V_{gsx} = \max(V_{GS}, V_{on})$$

- If you specify a nonzero value for the **Fast surface state density, NFS** parameter, the block calculates  $V_{on}$  using the following equation:

$$V_{on} = V_{TH} + x_n V_T$$

Otherwise,  $V_{on} = V_{TH}$ .

- The block calculates  $x_n$  using the following equation:

$$x_n = 1 + \frac{q * NFS}{COX} + \frac{\left( GAMMA * F_s * \sqrt{V_{bulk}} + \frac{F_n * V_{bulk}}{WIDTH} \right)}{2 * V_{bulk}}$$

- The block calculates  $V_{bulk}$  as follows:

- If  $V_{BS} \leq 0$ ,  $V_{bulk} = PHI - V_{BS}$ .

- Otherwise, the block calculates  $V_{bulk}$  using the following equation:

$$V_{bulk} = \frac{PHI}{\left( 1 + \frac{V_{BS}}{2 * PHI} \right)^2}$$

- $V_T = kT/q$

- The block calculates  $V_{TH}$  using the equation following equation:

$$V_{TH} = V_{BI} - \frac{8.15e^{-22} * ETA}{COX * (LENGTH - 2 * LD)^3} * V_{DS} + GAMMA * F_s * \sqrt{V_{bulk}} + F_n * V_{bulk}$$

- For information about how the block calculates  $V_{BI}$ , see “Temperature Dependence” on page 1-1586.

- *ETA* is the **Vds dependence threshold volt, ETA** parameter value.



- $COX = \epsilon_{ox}/TOX$ , where  $\epsilon_{ox}$  is the permittivity of the oxide and  $TOX$  is the **Oxide thickness, TOX** parameter value.
- If you specify a nonzero value for the **Junction depth, XJ** parameter and a value for the **Substrate doping, NSUB** parameter, the block calculates  $F_s$  using the following equations:

$$\alpha = \frac{2\epsilon_{si}}{qNSUB}$$

$$XD = \sqrt{\alpha}$$

$$wc = .0631353 + .8013292 * \frac{XD * \sqrt{V_{bulk}}}{XJ}$$

$$-.01110777 * \left( \frac{XD * \sqrt{V_{bulk}}}{XJ} \right)^2 + \frac{LD}{XJ}$$

$$F_s = 1 - \left( wc * \sqrt{1 - \left( \frac{XD * \sqrt{V_{bulk}}}{XJ + XD * \sqrt{V_{bulk}}} \right)^2} - \frac{LD}{XJ} \right)$$

where  $\epsilon_{si}$  is the permittivity of silicon.

Otherwise,  $F_s = 1$ .

- The block calculates  $F_B$  using the following equation:

$$F_B = \frac{GAMMA * F_s}{4 * \sqrt{V_{bulk}}} + F_n$$

- The block calculates  $F_n$  using the following equation:

$$F_n = \frac{DELTA * \pi * \epsilon_{si}}{2 * COX * WIDTH}$$

- $DELTA$  is the **Width effect on threshold, DELTA** parameter value.
- $V_{DSX}$  is the lesser of  $V_{DS}$  and the saturation voltage,  $V_{dsat}$ .

- If you specify a positive value for the **Max carrier drift velocity, VMAX** parameter, the block calculates  $V_{dsat}$  using the following equation:

$$V_{dsat} = \frac{V_{gsx} - V_{TH}}{1 + F_B} + \frac{(LENGTH - 2 * LD) * VMAX}{UO * F_{gate}}$$

$$- \sqrt{\left( \frac{V_{gsx} - V_{TH}}{1 + F_B} \right)^2 + \left( \frac{(LENGTH - 2 * LD) * VMAX}{UO * F_{gate}} \right)^2}$$

Otherwise, the block calculates  $V_{dsat}$  using the following equation:

$$V_{dsat} = \frac{V_{gsx} - V_{TH}}{1 + F_B}$$

If you specify a positive value for the **Max carrier drift velocity, VMAX** parameter, the block calculates  $Scale_{VMAX}$  using the following equation:

$$Scale_{VMAX} = \frac{1}{1 + \frac{UO * F_{gate}}{(LENGTH - 2 * LD) * VMAX} * V_{DSX}}$$

Otherwise,  $Scale_{VMAX} = 1$ .

The block scales the drain current to account for channel length modulation if the block meets all of the following criteria:

- $V_{DS} > V_{sat}$
- The **Max carrier drift velocity, VMAX** parameter value is zero or  $\alpha$  is nonzero.

The block scales the drain current using the following equation:

$$Scale_{LChan} = \frac{1}{1 - \frac{\Delta l}{(LENGTH - 2 * LD)}}$$

The block uses the following procedure to calculate  $\Delta l$ :

1 The block first calculates the intermediate value  $\Delta l_0$ .

- If you specify a positive value for the **Max carrier drift velocity, VMAX** parameter, the block computes the intermediate value  $g_{dsat}$  as the greater of  $1e-12$  and the result of the following equation:

$$I_{DS0} * \left( 1 - \frac{1}{1 + Scale_{g_{dsat}} * V_{DSX}} \right) * Scale_{g_{dsat}}$$

where:

$$Scale_{g_{dsat}} = \frac{UO * F_{gate}}{(LENGTH - 2 * LD) * VMAX}$$

Then, the block uses the following equation to calculate the intermediate value  $\Delta l_0$ :

$$\Delta l_0 = \sqrt{\left( \frac{KA * I_{DS}}{2 * (LENGTH - 2 * LD) * g_{dsat}} \right)^2 + KA * (V_{DS} - V_{dsat})} - \frac{KA * I_{DS}}{2 * (LENGTH - 2 * LD) * g_{dsat}}$$

where  $KA$  is the product of the **Mobility modulation, KAPPA** parameter value and  $\alpha$ .

- Otherwise, the block uses the following equation to calculate the intermediate value  $\Delta l_0$ :

$$\Delta l = \sqrt{KA * (V_{DS} - V_{dsat})}$$

2 The block checks for punch through and calculates  $\Delta l$ .

- If  $\Delta l_0$  is greater than  $(LENGTH - 2 * LD) / 2$ , the block calculates  $\Delta l$  using the following equation:

$$\Delta l = \left( 1 - \frac{(LENGTH - 2 * LD)}{4 * \Delta l_0} \right) * (LENGTH - 2 * LD)$$

- Otherwise,  $\Delta l = \Delta l_0$ .

If  $V_{GS}$  is less than  $V_{on}$ , the block calculates  $Scale_{INV}$  using the following equation:

$$Scale_{INV} = e^{\frac{V_{gs} - V_{on}}{x_n * V_T}}$$

Otherwise,  $Scale_{INV} = 1$ .

## Junction Charge Model

The block models the following junction charges:

- Junction Overlap Charges
- Bulk Junction Charges

The block calculates the following junction overlap charges:

- $Q_{GS} = CGSO * WIDTH * V_{gs}$

Where:

- $Q_{GS}$  is the gate-source overlap charge.
- $CGSO$  is the **G-S overlap capacitance**, **CGSO** parameter value.
- $WIDTH$  is the **Width of channel**, **WIDTH** parameter value.
- $Q_{GD} = CGDO * WIDTH * V_{gd}$

Where:

- $Q_{GD}$  is the gate-drain overlap charge.
- $CGDO$  is the **G-D overlap capacitance**, **CGDO** parameter value.
- $Q_{GB} = CGBO * (LENGTH - 2 * LD) * V_{gb}$

Where:

- $Q_{GB}$  is the gate-bulk overlap charge.
- $CGBO$  is the **G-B overlap capacitance**, **CGBO** parameter value.
- $LENGTH$  is the **Length of channel**, **LENGTH** parameter value.
- $LD$  is the **Lateral diffusion**, **LD** parameter value.

The block provides the following relationship between the bulk-drain bottom junction charge  $Q_{bottom}$  and the junction voltage  $V_{bd}$  after adjusting the applicable model parameters for temperature.

Applicable Range of $V_{bd}$ Values	Corresponding $Q_{bottom}$ Equation
$V_{bd} < FC * PB$	$Q_{bottom} = \frac{CBD * PB * \left( 1 - \left( 1 - \frac{V_{bd}}{PB} \right)^{1-MJ} \right)}{1-MJ} \quad \text{if } CBD > 0.$ $Q_{bottom} = \frac{CJ * AD * PB * \left( 1 - \left( 1 - \frac{V_{bd}}{PB} \right)^{1-MJ} \right)}{1-MJ} \quad \text{otherwise.}$

Applicable Range of $V_{bd}$ Values	Corresponding $Q_{bottom}$ Equation
$V_{bd} \geq FC * PB$	$Q_{bottom} = CBD *$ $\left( F1 + \frac{F3 * (V_{bd} - FC * PB) + \frac{MJ * (V_{bd}^2 - (FC * PB)^2)}{2 * PB}}{F2} \right) \text{ if } CBD > 0.$ $Q_{bottom} = CJ * AD *$ $\left( F1 + \frac{F3 * (V_{bd} - FC * PB) + \frac{MJ * (V_{bd}^2 - (FC * PB)^2)}{2 * PB}}{F2} \right)$ <p>otherwise.</p>

Where:

- $PB$  is the **Bulk junction potential**,  $PB$  parameter value.
- $FC$  is the **Capacitance coefficient**  $FC$  parameter value.
- $CBD$  is the **Zero-bias BD capacitance**,  $CBD$  parameter value.
- $CJ$  is the **Bottom junction cap per area**,  $CJ$  parameter value.
- $AD$  is the **Area of drain**,  $AD$  parameter value.
- $MJ$  is the **Bottom grading coefficient**,  $MJ$  parameter value.
- 

$$F1 = \frac{PB * (1 - (1 - FC)^{1 - MJ})}{1 - MJ}$$

$$F2 = (1 - FC)^{1 + MJ}$$

- $$F3 = 1 - FC * (1 + MJ)$$

The block uses the equations in the preceding table to calculate the bulk-source bottom junction charge, with the following substitutions:

- $V_{bs}$  replaces  $V_{bd}$ .
- AS (the **Area of source, AS** parameter value) replaces AD.
- CBS (the **Zero-bias BS capacitance, CBS** parameter value) replaces CBD.

The block provides the following relationship between the bulk-drain sidewall junction charge  $Q_{sidewall}$  and the junction voltage  $V_{bd}$  after adjusting the applicable model parameters for temperature.

Applicable Range of $V_{bd}$ Values	Corresponding $Q_{sidewall}$ Equation
$V_{bd} < FC * PB$	$Q_{sidewall} = \frac{CJSW * PD * PB * \left( 1 - \left( 1 - \frac{V_{bd}}{PB} \right)^{1-MGSW} \right)}{1 - MGSW}$
$V_{bd} \geq FC * PB$	$Q_{sidewall} = CJSW * PD * \left( F1 + \frac{F3 * (V_{bd} - FC * PB) + \frac{MGSW * (V_{bd}^2 - (FC * PB)^2)}{2 * PB}}{F2} \right)$

Where:

- CJSW is the **Side jct cap/area of jct perimeter, CJSW** parameter value.
- PD is the **Perimeter of drain, AD** parameter value.
- MGSW is the **Side grading coefficient, MJSW** parameter value.

- 

$$F1 = \frac{PB * (1 - (1 - FC)^{1 - MJSW})}{1 - MJSW}$$

- 

$$F2 = (1 - FC)^{1 + MJSW}$$

- 

$$F3 = 1 - FC * (1 + MJSW)$$

The block uses the equations in the preceding table to calculate the bulk-source sidewall junction charge and the sidewall junction voltage, with the following substitutions:

- $V_{bs}$  replaces  $V_{bd}$ .
- $PS$  (the **Perimeter of source, PS** parameter value) replaces  $PD$ .

## Temperature Dependence

Several transistor parameters depend on temperature. There are two ways to specify the transistor temperature:

- When you select `Device` temperature for the **Model temperature dependence using** parameter, the transistor temperature is

$$T = T_C + T_O$$

where:

- $T_C$  is the **Circuit temperature** parameter value from the SPICE Environment Parameters block. If this block doesn't exist in the circuit,  $T_C$  is the default value of this parameter.
- $T_O$  is the **Offset local circuit temperature, TOFFSET** parameter value.
- When you select `Fixed` temperature for the **Model temperature dependence using** parameter, the transistor temperature is the **Fixed circuit temperature, TFIXED** parameter value.

The block provides the following relationship between the transconductance  $KP$  and the transistor temperature  $T$ :



$$KP(T) = \frac{KP}{\left(\frac{T}{T_{meas}}\right)^{3/2}}$$

where:

- $KP$  is the **Transconductance, KP** parameter value.
- $T_{meas}$  is the **Parameter extraction temperature, TMEAS** parameter value.

The block provides the following relationship between the surface potential  $PHI$  and the transistor temperature  $T$ :

$$PHI(T) = \frac{T}{T_{meas}} \left( PHI + \frac{kT_{meas}}{q} \left( \log \left( \frac{T_{meas}}{300.15} \right)^3 + \frac{q}{k} \left( \frac{1.115}{300.15} - \frac{EG_{T_{meas}}}{T_{meas}} \right) \right) \right) \\ - \frac{kT}{q} \left( \log \left( \frac{T}{300.15} \right)^3 + \frac{q}{k} \left( \frac{1.115}{300.15} - \frac{EG_T}{T} \right) \right)$$

where:

- $EG_{T_{meas}} = 1.16eV - (7.02e - 4 * T_{meas}^2) / (T_{meas} + 1108)$
- $EG_T = 1.16eV - (7.02e - 4 * T^2) / (T + 1108)$

The block provides the following relationship between the built-in voltage  $VBI$  and the transistor temperature  $T$ :

$$VBI(T) = VTO + MTYPE * \left( \frac{PHI(T) - PHI}{2} - GAMMA \sqrt{PHI} \right) \\ + \frac{EG_{T_{meas}} - EG_T}{2}$$

where:

- $VTO$  is:

- The **Threshold voltage, VTO** parameter value, if this parameter has a numerical value.

- $$\Phi - 3.25 + EG_{T_{meas}} / 2 + MTYPE * PHI / 2 - NSS * q * TOX / (3.9 * \epsilon_0)$$

+  $MTYPE * (GAMMA * \sqrt{PHI} + PHI)$ , if **Threshold voltage, VTO** is NaN and you specify values for both the **Oxide thickness, TOX** and **Substrate doping, NSUB** parameters.

- $\Phi$  is:

- 3.2, if **TPG** (the **Gate type?**, **TPG** parameter value) is 0.

- $$3.25 + EG_{T_{meas}} / 2 - MTYPE * TPG * EG_{T_{meas}} / 2$$
, otherwise.

- **GAMMA** is:

- The **Bulk threshold, GAMMA** parameter value, if this parameter has a numerical value.

- $$TOX * \sqrt{2 * 11.7 * \epsilon_0 * q * NSUB} / (3.9 * \epsilon_0)$$
, if **Bulk threshold, GAMMA** is NaN and you specify values for both the **Oxide thickness, TOX** and **Substrate doping, NSUB** parameters.

The block provides the following relationship between the bulk saturation current  $IS$  and the transistor temperature  $T$ :

$$IS(T) = IS * e^{\frac{-qEG_T}{ND*kT} + \frac{qEG_{T_{meas}}}{ND*kT_{meas}}}$$

where:

- $ND$  is the **Emission coefficient, N** parameter value.
- $IS$  is the **Bulk saturation current, IS** parameter value.

The block provides the following relationship between the bulk junction saturation current density  $JS$  and the transistor temperature  $T$ :

$$JS(T) = JS * e^{\frac{-qEG_T}{ND*kT} + \frac{qEG_{T_{meas}}}{ND*kT_{meas}}}$$

where:

- $JS$  is the **Bulk jct sat current density, JS** parameter value.

The block provides the following relationship between the bulk junction potential  $PB$  and the transistor temperature  $T$ :

$$PB(T) = \frac{PB + \frac{kT_{meas}}{q} \left( \log \left( \frac{T_{meas}}{300.15} \right)^3 + \frac{q}{k} \left( \frac{1.115}{300.15} - \frac{EG_{T_{meas}}}{T} \right) \right)}{T_{meas}/T} - \frac{kT}{q} \left( \log \left( \frac{T}{300.15} \right)^3 + \frac{q}{k} \left( \frac{1.115}{300.15} - \frac{EG_T}{T} \right) \right)$$

where:

- $PB$  is the **Bulk junction potential, PB** parameter value.

The block provides the following relationship between the bulk-drain junction capacitance  $CBD$  and the transistor temperature  $T$ :

$$CBD(T) = CBD \frac{pbo + MJ * (4 * 10^4 * (T - 300.15) * pbo - (PB(T) - pbo))}{pbo + MJ * (4 * 10^4 * (T_{meas} - 300.15) * pbo - (PB - pbo))}$$

where:

- $CBD$  is the **Zero-bias BD capacitance, CBD** parameter value.
- $MJ$  is the **Bottom grading coefficient, MJ** parameter value.
- 

$$pbo = \frac{PB + \frac{kT_{meas}}{q} \left( \log \left( \frac{T_{meas}}{300.15} \right)^3 + \frac{q}{k} \left( \frac{1.115}{300.15} - \frac{EG_{T_{meas}}}{T} \right) \right)}{T_{meas}/300.15}$$

The block uses the  $CBD(T)$  equation to calculate:

- The bulk-source junction capacitance by substituting  $CBS$  (the **Zero-bias BS capacitance, CBS** parameter value) for  $CBD$ .

- The bottom junction capacitance by substituting  $CJ$  (the **Bottom junction cap per area, CJ** parameter value) for  $CBD$ .

The block provides the following relationship between the sidewall junction capacitance  $CJSW$  and the transistor temperature  $T$ :

$$CJSW(T) = CJ \frac{pbo + MJSW * (4 * 10^4 * (T - 300.15) * pbo - (PB(T) - pbo))}{pbo + MJSW * (4 * 10^4 * (T_{meas} - 300.15) * pbo - (PB - pbo))}$$

where:

- $MJSW$  is the **Side grading coefficient, MJSW** parameter value.

## Basic Assumptions and Limitations

The model is based on the following assumptions:

- The NMOS block does not support noise analysis.
- The NMOS block applies initial conditions across junction capacitors and not across the block ports.

## Parameters

### Model Selection Tab

#### MOS model

Select one of the following MOSFET model options:

- Level 1 MOS — Use the “Level 1 Drain Current Model” on page 1-1574. This is the default option.
- Level 3 MOS — Use the “Level 3 Drain Current Model” on page 1-1577.

## Dimensions Tab

### Device area factor, AREA

The transistor area. This value multiplies the following parameter values:

- **Transconductance, KP**
- **Bulk saturation current, IS**
- **Bulk jct sat current density, JS**
- **Zero-bias BD capacitance, CBD**
- **Zero-bias BS capacitance, CBS**
- **G-S overlap capacitance, CGSO**
- **G-D overlap capacitance, CGDO**
- **G-B overlap capacitance, CGBO**
- **Bottom junction cap per area CJ**
- **Side jct cap/area of jct perimeter CJSW**

It divides the following parameter values:

- **Drain resistance, RD**
- **Source resistance, RS**
- **Sheet resistance, RSH**

The default value is 1. The value must be greater than 0.

### Number of parallel devices, SCALE

The number of parallel MOS instances for this device. This parameter multiplies the output current and device charge. The default value is 1. The value must be greater than 0.

### Length of channel, LENGTH

Length of the channel between the source and drain. The default value is  $1e-04$  m.

### Width of channel, WIDTH

Width of the channel between the source and drain. The default value is  $1e-04$  m.

### Area of drain, AD

Area of the transistor drain diffusion. The default value is  $0$  m<sup>2</sup>. The value must be greater than or equal to 0.

## Area of source, AS

Area of the transistor source diffusion. The default value is  $0 \text{ m}^2$ . The value must be greater than or equal to 0.

## Perimeter of drain, PD

Perimeter of the transistor drain diffusion. The default value is 0 m.

## Perimeter of source, PS

Perimeter of the transistor source diffusion. The default value is 0 m.

## Resistors Tab

### Drain resistance, RD

The transistor drain ohmic resistance. The default value is  $0.01 \Omega$ . If you set this parameter to NaN  $\Omega$ , this value means the parameter is unspecified, so the block calculates the drain resistance as described in “Resistance Calculations” on page 1-1571. The value must be equal to 0 or greater than or equal to  $R_{min}$ .  $R_{min}$  is a built-in model constant whose value is  $1\text{e-}12$ .

### Source resistance, RS

The transistor source ohmic resistance. The default value is  $1\text{e-}4 \Omega$ . If you set this parameter to NaN  $\Omega$ , this value means the parameter is unspecified, so the block calculates the drain resistance as described in “Resistance Calculations” on page 1-1571. The value must be equal to 0 or greater than or equal to  $R_{min}$ .  $R_{min}$  is a built-in model constant whose value is  $1\text{e-}12$ .

### Sheet resistance, RSH

Resistance per square of the transistor source and drain. The default value is NaN  $\Omega$ . This value means the parameter is unspecified. The block only uses this parameter value if you do not specify one or both of the **Drain resistance, RD** and **Source resistance, RS** parameter values, as described in “Resistance Calculations” on page 1-1571. The value must be greater than or equal to 0.

### Number of drain squares, NRD

Number of squares of resistance that make up the transistor drain diffusion. The default value is 1. The value must be greater than or equal to 0. The block only uses this parameter value if you do not specify one or both of the **Drain resistance, RD** and **Source resistance, RS** parameter values, as described in “Resistance Calculations” on page 1-1571.

**Number of source squares, NRS**

Number of squares of resistance that make up the transistor source diffusion. The default value is 1. The value must be greater than or equal to 0. The block only uses this parameter value if you do not specify one or both of the **Drain resistance, RD** and **Source resistance, RS** parameter values, as described in “Resistance Calculations” on page 1-1571.

**DC Currents Tab****Threshold voltage, VTO**

The gate-source voltage above which the transistor produces a nonzero drain current. The default value is 0 V. If you assign this parameter a value of NaN, the block calculates the value from the specified values of the **Oxide thickness, TOX** and **Substrate doping, NSUB** parameters. For more information about this calculation, see “Temperature Dependence” on page 1-1586.

**Transconductance, KP**

The derivative of drain current with respect to gate voltage. The default value is  $2e-05 \text{ A/V}^2$ . The value must be greater than or equal to 0. If you assign this parameter a value of NaN, the block calculates the value from the specified values of the **Oxide thickness, TOX** and **Substrate doping, NSUB** parameters. For more information about this calculation, see “Level 1 Drain Current Model” on page 1-1574 or “Level 3 Drain Current Model” on page 1-1577 as appropriate for the selected value of the **MOS model** parameter.

**Bulk threshold, GAMMA**

Body effect parameter, which relates the threshold voltage, VTH, to the body bias, VBS, as described in “Level 1 Drain Current Model” on page 1-1574 and “Level 3

Drain Current Model” on page 1-1577. The default value is  $0 \sqrt{V}$ . The value must be greater than or equal to 0. If you assign this parameter a value of NaN, the block calculates the value from the specified values of the **Oxide thickness, TOX** and **Substrate doping, NSUB** parameters. For more information about this calculation, see “Level 1 Drain Current Model” on page 1-1574 or “Level 3 Drain Current Model” on page 1-1577 as appropriate for the selected value of the **MOS model** parameter.

**Surface potential, PHI**

Twice the voltage at which the surface electron concentration becomes equal to the intrinsic concentration and the device transitions between depletion and inversion conditions. The default value is 0.6 V. The value must be greater than or equal to 0. If you assign this parameter a value of NaN, the block calculates the value from the

specified values of the **Oxide thickness, TOX** and **Substrate doping, NSUB** parameters. For more information about this calculation, see “Level 1 Drain Current Model” on page 1-1574 or “Level 3 Drain Current Model” on page 1-1577 as appropriate for the selected value of the **MOS model** parameter.

**Channel modulation, LAMBDA**

The channel-length modulation. This parameter is only visible when you select Level 1 MOS for the **MOS model** parameter. The default value is  $0.1/V$ .

**Bulk saturation current, IS**

The magnitude of the current that the junction approaches asymptotically for very large reverse bias levels. The default value is  $1e-14$  A. The value must be greater than or equal to 0.

**Bulk jct sat current density, JS**

The magnitude of the current per unit area that the junction approaches asymptotically for very large reverse bias levels. The default value is  $0$  A/m<sup>2</sup>. The value must be greater than or equal to 0.

**Emission coefficient, N**

The transistor emission coefficient or ideality factor. The default value is 1. The value must be greater than 0.

**Width effect on threshold, DELTA**

The factor that controls the effect of transistor width on threshold voltage. This parameter is only visible when you select Level 3 MOS for the **MOS model** parameter. The default value is 0.

**Max carrier drift velocity, VMAX**

The maximum drift velocity of the carriers. This parameter is only visible when you select Level 3 MOS for the **MOS model** parameter. The default value is  $0$  m/s.

**Fast surface state density, NFS**

The fast surface state density adjusts the drain current for the mobility reduction caused by the gate voltage. This parameter is only visible when you select Level 3 MOS for the **MOS model** parameter. The default value is  $0$  1/cm<sup>2</sup>.

**Vds dependence threshold volt, ETA**

The coefficient that controls how the threshold voltage depends on the drain-source voltage in the drain current calculation. This parameter is only visible when you select Level 3 MOS for the **MOS model** parameter. The default value is 0.



**V<sub>gs</sub> dependence on mobility, THETA**

The coefficient that controls how the mobility affects the gate voltage in the drain current calculation. This parameter is only visible when you select Level 3 MOS for the **MOS model** parameter. The default value is  $\theta 1/V$ .

**Mobility modulation, KAPPA**

The coefficient that controls how the mobility affects the channel length in the drain current calculation. This parameter is only visible when you select Level 3 MOS for the **MOS model** parameter. The default value is  $\theta .2$ .

**C-V Tab****Model junction capacitance**

Select one of the following options for modeling the junction capacitance:

- **No** — Do not include junction capacitance in the model. This is the default option.
- **Yes** — Specify zero-bias junction capacitance, junction potential, grading coefficient, forward-bias depletion and capacitance coefficient.

**G-S overlap capacitance, CGSO**

Gate-source capacitance due to the diffusion that occurs when the device operates in depletion mode. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is  $\theta$  F/m. The value must be equal to  $\theta$  or greater than or equal to *Cmin*. *Cmin* is a built-in model constant whose value is  $1e-18$ .

**G-D overlap capacitance, CGDO**

Gate-drain capacitance due to the diffusion that occurs when the device operates in depletion mode. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is  $\theta$  F/m. The value must be equal to  $\theta$  or greater than or equal to *Cmin*. *Cmin* is a built-in model constant whose value is  $1e-18$ .

**G-B overlap capacitance, CGBO**

Gate-base capacitance due to the diffusion that occurs when the device operates in depletion mode. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is  $\theta$  F/m. The value must be equal to  $\theta$  or greater than or equal to *Cmin*. *Cmin* is a built-in model constant whose value is  $1e-18$ .

**Zero-bias BD capacitance, CBD**

The value of the capacitance placed between the base and the drain. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is 0 F. The value must be equal to 0 or greater than or equal to *Cmin*. *Cmin* is a built-in model constant whose value is 1e-18.

**Zero-bias BS capacitance, CBS**

The value of the capacitance placed between the base and the source. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is 0 F. The value must be equal to 0 or greater than or equal to *Cmin*. *Cmin* is a built-in model constant whose value is 1e-18.

**Bottom junction cap per area CJ**

Zero-bias bulk junction bottom capacitance per junction area. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is 0 F/m<sup>2</sup>. The value must be equal to 0 or greater than or equal to *Cmin*. *Cmin* is a built-in model constant whose value is 1e-18.

**Bottom grading coefficient, MJ**

The transistor bottom grading coefficient. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is 0.5. The value must be equal to 0 or less than *MGmax*. *MGmax* is a built-in model constant whose value is 0.9.

**Side jct cap/area of jct perimeter CJSW**

Zero-bias bulk junction sidewall capacitance per junction perimeter. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is 0 F/m. The value must be equal to 0 or greater than or equal to *Cmin*. *Cmin* is a built-in model constant whose value is 1e-18.

**Side grading coefficient, MJSW**

The transistor sidewall grading coefficient. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is 0.5. The value must be equal to 0 or less than *MGmax*. *MGmax* is a built-in model constant whose value is 0.9.

**Bulk junction potential, PB**

The potential across the bulk junction. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is 0.8 V. The value must be equal to 0 or greater than or equal to *VJmin*. *VJmin* is a built-in model constant whose value is 0.01.

### Capacitance coefficient FC

The fitting coefficient that quantifies the decrease of the depletion capacitance with applied voltage. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is 0.5. The value must be equal to 0 or less than or equal to  $FCmax$ .  $FCmax$  is a built-in model constant whose value is 0.95.

### Specify initial condition

Select one of the following options for specifying an initial condition:

- No — Do not specify an initial condition for the model. This is the default option.
- Yes — Specify the initial diode voltage.

---

**Note** The NMOS block applies the initial diode voltage across the junction capacitors and not across the ports.

---

### Initial condition voltage ICVDS

Drain-source voltage at the start of the simulation. This parameter is only visible when you select **Yes** for the **Model junction capacitance** and **Yes** for the **Specify initial condition** parameter. The default value is 0 V.

### Initial condition voltage ICVGS

Gate-source voltage at the start of the simulation. This parameter is only visible when you select **Yes** for the **Model junction capacitance** and **Yes** for the **Specify initial condition** parameter. The default value is 0 V.

### Initial condition voltage ICVBS

Bulk-source voltage at the start of the simulation. This parameter is only visible when you select **Yes** for the **Model junction capacitance** and **Yes** for the **Specify initial condition** parameter. The default value is 0 V.

## Process Tab

### Oxide thickness, TOX

Thickness of the gate oxide. The default value is NaN m. The value must be greater than or equal to 0.

---

**Note** When you select Level 3 MOS for the **MOS model** parameter, the block uses a value of  $1e-7$  rather than NaN by default.

---

### **Lateral diffusion, LD**

Length of lateral diffusion. The default value is 0 m.

### **Substrate doping, NSUB**

Substrate doping. The default value is NaN 1/cm<sup>3</sup>. The value must be greater than or equal to 1.45e10 (the carrier concentration of intrinsic silicon).

### **Surface state density, NSS**

Substrate doping. The default value is 0 1/cm<sup>2</sup>.

### **Surface mobility, U0**

Zero-bias surface mobility coefficient. The default value is 600 cm<sup>2</sup>/V/s.

### **Junction depth, XJ**

Junction depth. This parameter is only visible when you select Level 3 MOS for the **MOS model** parameter. The default value is 0 m.

### **Gate type?, TPG**

Select one of the following MOSFET gate materials (as compared to the substrate):

- **Opposite of substrate** — The gate material is the opposite of the substrate. This means that TPG = 1 in the device equations. This is the default option.
- **Same as substrate** — The gate material is the same as the substrate. This means that TPG = -1 in the device equations.
- **Aluminum** — The gate material is aluminum. This means that TPG = 0 in the device equations.

## **Temperature Tab**

### **Model temperature dependence using**

Select one of the following options for modeling the diode temperature dependence:

- **Device temperature** — Use the device temperature, which is the **Circuit temperature** value plus the **Offset local circuit temperature, TOFFSET** value. The **Circuit temperature** value comes from the SPICE Environment Parameters block, if one exists in the circuit. Otherwise, it comes from the default value for this block.
- **Fixed temperature** — Use a temperature that is independent of the circuit temperature to model temperature dependence.

**Offset local circuit temperature, TOFFSET**

The amount by which the transistor temperature differs from the circuit temperature. This parameter is only visible when you select `Device temperature` for the **Model temperature dependence using** parameter. The default value is 0 K.

**Fixed circuit temperature, TFIXED**

The temperature at which to simulate the transistor. This parameter is only visible when you select `Fixed temperature` for the **Model temperature dependence using** parameter. The default value is 300.15 K. The value must be greater than 0.

**Parameter extraction temperature, TMEAS**

The temperature at which the transistor parameters were measured. The default value is 300.15 K. The value must be greater than 0.

## Ports

The block has the following ports:

G

Electrical conserving port associated with the transistor gate terminal.

D

Electrical conserving port associated with the transistor drain terminal.

S

Electrical conserving port associated with the transistor source terminal.

B

Electrical conserving port associated with the transistor bulk terminal.

## References

- [1] G. Massobrio and P. Antognetti. *Semiconductor Device Modeling with SPICE*. 2nd Edition, McGraw-Hill, 1993. Chapter 3.

## **See Also**

SPICE PMOS

# SPICE NPN

Model Gummel-Poon NPN Transistor



## Library

SPICE-Compatible Components/Semiconductor Devices

## Description

The NPN block represents a SPICE-compatible four-terminal Gummel-Poon NPN transistor. The substrate port is connected to the transistor body using a capacitor, so these devices are equivalent to a three-terminal transistor when you connect the substrate port to any other port and use the default value of zero for the **C-S junction capacitance, CJS** parameter.

The NPN block model includes the following components:

- “Current-Voltage and Base Charge Model” on page 1-1601
- “Base Resistance Model” on page 1-1604
- “Transit Charge Modulation Model” on page 1-1605
- “Junction Charge Model” on page 1-1606
- “Temperature Dependence” on page 1-1608

## Current-Voltage and Base Charge Model

The current-voltage relationships and base charge relationships for the transistor are calculated adjusting the applicable model parameters for temperature as described in the following sections:

- “Base-Emitter and Base-Collector Junction Currents” on page 1-1602

- “Terminal Currents” on page 1-1603
- “Base Charge Model” on page 1-1604

The base-emitter junction current is calculated using the following equations:

- When  $V_{BE} > 80 * V_{TF}$  :

$$I_{bef} = IS * \left( \left( \frac{V_{BE}}{V_{TF}} - 79 \right) * e^{80} - 1 \right) + G_{\min} * V_{BE}$$
$$I_{bee} = ISE * \left( (V_{BE} - 80 * V_{TF} + V_{TE}) * \frac{e^{(80 * V_{TF} / V_{TE})}}{V_{TE}} - 1 \right)$$

- When  $V_{BE} \leq 80 * V_{TF}$

$$I_{bef} = IS * \left( e^{(V_{BE} / V_{TF})} - 1 \right) + G_{\min} * V_{BE}$$
$$I_{bee} = ISE * \left( e^{(V_{BE} / V_{TE})} - 1 \right)$$

The base-collector junction current is calculated using the following equations:

- When  $V_{BC} > 80 * V_{TR}$  :

$$I_{bcr} = IS * \left( \left( \frac{V_{BC}}{V_{TR}} - 79 \right) * e^{80} - 1 \right) + G_{\min} * V_{BC}$$
$$I_{bcc} = ISC * \left( (V_{BC} - 80 * V_{TR} + V_{TC}) * \frac{e^{(80 * V_{TR} / V_{TC})}}{V_{TC}} - 1 \right)$$

- When  $V_{BC} \leq 80 * V_{TR}$



$$I_{bcf} = IS * (e^{(V_{BC}/V_{TR})} - 1) + G_{min} * V_{BC}$$

$$I_{bcc} = ISC * (e^{(V_{BC}/V_{TC})} - 1)$$

In the preceding equations:

- $V_{BE}$  is the base-emitter voltage and  $V_{BC}$  is the base-collector voltage.
- 
- $V_{TE} = NE * k * T / q$ ,  $V_{TC} = NC * k * T / q$ ,  $V_{TF} = NF * k * T / q$ , and  $V_{TR} = NR * k * T / q$ .
- $ISC$  and  $ISE$  are the **B-C leakage current, ISC** and **B-E leakage current, ISE** parameter values, respectively.
- $NE$ ,  $NC$ ,  $NF$ , and  $NR$  are the **B-E emission coefficient, NE**, **B-C emission coefficient, NC**, **Forward emission coefficient, NF** and **Reverse emission coefficient, NR** parameter values, respectively.
- $q$  is the elementary charge on an electron.
- $k$  is the Boltzmann constant.
- $T$  is the transistor temperature:
  - If you select Device temperature for the **Model temperature dependence using** parameter,  $T$  is the sum of the **Circuit temperature** value plus the **Offset local circuit temperature, TOFFSET** parameter value. The **Circuit temperature** value comes from the SPICE Environment Parameters block, if one exists in the circuit. Otherwise, it comes from the default value for this block.
  - If you select Fixed temperature for the **Model temperature dependence using** parameter,  $T$  is the **Fixed circuit temperature, TFIXED** parameter value.
- $G_{min}$  is the minimum conductance. By default,  $G_{min}$  matches the **Minimum conductance GMIN** parameter of the SPICE Environment Parameters block, whose default value is 1e-12. To change  $G_{min}$ , add a SPICE Environment Parameters block to your model and set the **Minimum conductance GMIN** parameter to the desired value.

The terminal currents,  $I_B$  and  $I_C$  are the base and collector currents, defined as positive into the device. They are calculated as:

$$I_B = - \left( \frac{I_{ebf}}{BF} + I_{ebe} + \frac{I_{cbr}}{BR} + I_{cbc} \right)$$

$$I_C = - \left( \frac{I_{ebf} - I_{cbr}}{q_b} - \frac{I_{cbr}}{BR} - I_{cbc} \right)$$

where *BF* and *BR* are the **Forward beta, BF** and **Reverse beta, BR** parameter values, respectively.

The base charge, *q<sub>b</sub>*, is calculated using the following equations:

$$q_b = \frac{q_1}{2} \left( 1 + \sqrt{0.5 * \left( \sqrt{(1 + 4 * q_2 - eps)^2 + eps^2} + 1 + 4 * q_2 - eps \right) + eps} \right)$$

$$q_1 = \left( 1 - \frac{V_{BC}}{VAF} - \frac{V_{BE}}{VAR} \right)^{-1}$$

$$q_2 = \frac{I_{bef}}{IKF} + \frac{I_{bcr}}{IKR}$$

where

- *VAF* and *VAR* are the **Forward Early voltage, VAF** and **Reverse Early voltage, VAR** parameters, respectively.
- *IKF* and *IKR* are the **Forward knee current, IKF** and **Reverse knee current, IKR** parameter values, respectively.
- *eps* is 1e-4.

## Base Resistance Model

The block models base resistance in one of two ways:

- If you use the default value of infinity for the **Half base resistance cur, IRB** parameter, the NPN block calculates the base resistance *r<sub>bb</sub>* as

$$r_{bb} = RBM + \frac{RB - RBM}{q_b}$$

where:

- $RBM$  is the **Minimum base resistance**,  $RBM$  parameter value.
- $RB$  is the **Zero-bias base resistance**,  $RB$  parameter value.
- If you specify a finite value for the **Half base resistance cur**,  $IRB$  parameter, the NPN block calculates the base resistance  $r_{bb}$  as

$$r_{bb} = RBM + 3 * (RB - RBM) * \left( \frac{\tan z - z}{z * \tan^2 z} \right)$$

where

$$z = \frac{\sqrt{1 + 144I_B / (\pi^2 IRB)} - 1}{(24 / \pi^2) \sqrt{(I_B / IRB)}}$$

## Transit Charge Modulation Model

If you specify nonzero values for the **Coefficient of TF**,  $XTF$  parameter, the block models transit charge modulation by scaling the **Forward transit time**,  $TF$  parameter value as follows:

$$TF_{\text{mod}} = \frac{TF * \left[ 1 + XTF * e^{V_{BC} / (1.44V_{TF})} \left( \frac{I_{BE}}{I_{BE} + ITF} \right)^2 \right]}{q_b}$$

where  $ITF$  is the **Coefficient of TF**,  $ITF$  parameter value.

## Junction Charge Model

The block lets you model junction charge. The base-collector charge  $Q_{bc}$  and the base-emitter charge  $Q_{be}$  depend on an intermediate value,  $Q_{dep}$  as follows, after adjusting the applicable model parameters for temperature:

- For the internal base-emitter junctions:

$$Q_{be} = TF_{mod} * I_{be} + Q_{dep}$$

- For the internal base-collector junctions:

$$Q_{bc} = TR * I_{bc} + XCJC * Q_{dep}$$

- For the external base-collector junctions:

$$Q_{bc,c} = (1 - XCJC) * Q_{dep}$$

$Q_{dep}$  depends on the junction voltage,  $V_{jct}$  ( $V_{BE}$  for the base-emitter junction and  $V_{BC}$  for the base-collector junction) as follows.

Applicable Range of $V_{jct}$ Values	Corresponding $Q_{dep}$ Equation
$V_{jct} < FC * VJ$	$Q_{dep} = C_{jct} * VJ * \frac{1 - (1 - V_{jct} / VJ)^{(1-MJ)}}{1 - MJ}$
$V_{jct} \geq FC * VJ$	$Q_{dep} = C_{jct} * \left[ F1 + \frac{F3 * (V_{jct} - FC * VJ) + \frac{MJ * [V_{jct}^2 - (FC * VJ)^2]}{2 * VJ}}{F2} \right]$

Where:

- $FC$  is the **Capacitance coefficient FC** parameter value.

- $VJ$  is:
  - The **B-E built-in potential**,  $VJE$  parameter value for the base-emitter junction.
  - The **B-C built-in potential**,  $VJC$  parameter value for the base-collector junction.
- $MJ$  is:
  - The **B-E exponential factor**,  $MJE$  parameter value for the base-emitter junction.
  - The **B-C exponential factor**,  $MJC$  parameter value for the base-collector junction.
- $C_{jct}$  is:
  - The **B-E depletion capacitance**,  $CJE$  parameter value for the base-emitter junction.
  - The **B-C depletion capacitance**,  $CJC$  parameter value for the base-collector junction.

$$F1 = VJ * (1 - (1 - FC)^{(1 - MJ)}) / (1 - MJ)$$

$$F2 = (1 - FC)^{(1 + MJ)}$$

$$F3 = 1 - FC * (1 + MJ)$$

The collector-substrate charge  $Q_{cs}$  depends on the collector-substrate voltage  $V_{cs}$  as follows, after adjusting the applicable model parameters for temperature.

Applicable Range of $V_{cs}$ Values	Corresponding $Q_{cs}$ Equation
$V_{cs} < 0$	$Q_{cs} = CJS * VJS * \left( \frac{1 - (1 - V_{cs} / VJS)^{(1 - MJS)}}{1 - MJS} \right)$
$V_{cs} \geq 0$	$Q_{cs} = CJS * (1 + MJS * V_{cs} / (2 * VJS)) * V_{cs}$

where:

- $CJS$  is the **C-S junction capacitance**,  $CJS$  parameter value.

- $V_{JS}$  is the **Substrate built-in potential**, **VJS** parameter value.
- $M_{JS}$  is the **Substrate exponential factor**, **MJS** parameter value.

## Temperature Dependence

Several transistor parameters depend on temperature. There are two ways to specify the transistor temperature:

- When you select `Device` temperature for the **Model temperature dependence using** parameter, the transistor temperature is

$$T = T_C + T_O$$

where:

- $T_C$  is the **Circuit temperature** parameter value from the SPICE Environment Parameters block. If this block doesn't exist in the circuit,  $T_C$  is the default value of this parameter.
- $T_O$  is the **Offset local circuit temperature**, **TOFFSET** parameter value.
- When you select `Fixed` temperature for the **Model temperature dependence using** parameter, the transistor temperature is the **Fixed circuit temperature**, **TFIXED** parameter value.

The block provides the following relationship between the saturation current  $IS$  and the transistor temperature  $T$ :

$$IS(T) = IS * (T/T_{meas})^{XTI} * e^{\left(\frac{T}{T_{meas}} - 1\right) \frac{EG}{V_t}}$$

where:

- $IS$  is the **Transport saturation current**, **IS** parameter value.
- $T_{meas}$  is the **Parameter extraction temperature**, **TMEAS** parameter value.
- $XTI$  is the **Temperature exponent for IS**, **XTI** parameter value.
- $EG$  is the **Energy gap**, **EG** parameter value.
- $V_t = kT/q$ .

The block provides the following relationship between the base-emitter junction potential  $VJE$  and the transistor temperature  $T$ :

$$VJE(T) = VJE * \left( \frac{T}{T_{meas}} \right) - \frac{3 * k * T}{q} * \log \left( \frac{T}{T_{meas}} \right) - \left( \frac{T}{T_{meas}} \right) * EG_{T_{meas}} + EG_T$$

where:

- $VJE$  is the **B-E built-in potential**, **VJE** parameter value.

$$EG_{T_{meas}} = 1.16eV - (7.02e - 4 * T_{meas}^2) / (T_{meas} + 1108)$$

$$EG_T = 1.16eV - (7.02e - 4 * T^2) / (T + 1108)$$

The block uses the  $VJE(T)$  equation to calculate the base-collector junction potential by substituting  $VJC$  (the **B-C built-in potential**, **VJC** parameter value) for  $VJE$ .

The block provides the following relationship between the base-emitter junction capacitance  $CJE$  and the transistor temperature  $T$ :

$$CJE(T) = CJE * \left[ 1 + MJE * \left( 400e - 6 * (T - T_{meas}) - \frac{VJE(T) - VJE}{VJE} \right) \right]$$

where:

- $CJE$  is the **B-E depletion capacitance**, **CJE** parameter value.
- $MJE$  is the **B-E exponential factor**, **MJE** parameter value.

The block uses the  $CJE(T)$  equation to calculate the base-collector junction capacitance by substituting  $CJC$  (the **B-C depletion capacitance**, **CJC** parameter value) for  $CJE$  and  $MJC$  (the **B-C exponential factor**, **MJC** parameter value) for  $MJE$ .

The block provides the following relationship between the forward and reverse beta and the transistor temperature  $T$ :

$$\beta(T) = \beta * \left( \frac{T}{T_{meas}} \right)^{XTB}$$

where:

- $\beta$  is the **Forward beta, BF** or **Reverse beta, BR** parameter value.
- $XTB$  is the **Beta temperature exponent, XTB** parameter value.

The block provides the following relationship between the base-emitter leakage current  $ISE$  and the transistor temperature  $T$ :

$$ISE(T) = ISE * \left( \frac{T}{T_{meas}} \right)^{-XTB} * \left( \frac{IS(T)}{IS} \right)^{1/NE}$$

where:

- $ISE$  is the **B-E leakage current, ISE** parameter value.
- $NE$  is the **B-E emission coefficient, NE** parameter value.

The block uses this equation to calculate the base-collector leakage current by substituting  $ISC$  (the **B-C leakage current, ISC** parameter value) for  $ISE$  and  $NC$  (the **B-C emission coefficient, NC** parameter value) for  $NE$ .

## Basic Assumptions and Limitations

The model is based on the following assumptions:

- The NPN block does not support noise analysis.
- The NPN block applies initial conditions across junction capacitors and not across the block ports.

## Parameters

### Main Tab

#### Device area, AREA

The transistor area. This value multiplies the following parameter values:



- **Transport saturation current, IS**
- **Forward knee current, IKF**
- **B-E leakage current, ISE**
- **Reverse knee current, IKR**
- **B-C leakage current, ISC**
- **Half base resistance cur, IRB**
- **B-E depletion capacitance, CJE**
- **Coefficient of TF, ITF**
- **B-C depletion capacitance, CJC**
- **C-S junction capacitance, CJS**

It divides the following parameter values:

- **Zero-bias base resistance, RB**
- **Minimum base resistance, RBM**
- **Emitter resistance, RE**
- **Collector resistance, RC**

The default value is 1 m<sup>2</sup>. The value must be greater than 0.

### **Number of parallel devices, SCALE**

The number of parallel transistors the block represents. This value multiplies the output current and device charges. The default value is 1. The value must be greater than 0.

## **Forward Gain Tab**

### **Transport saturation current, IS**

The magnitude of the current at which the transistor saturates. The default value is 1e-16 A/m<sup>2</sup>. The value must be greater than or equal to 0.

### **Forward beta, BF**

The ideal maximum reverse beta. The default value is 100. The value must be greater than 0.

### **Forward emission coefficient, NF**

The reverse emission coefficient or ideality factor. The default value is 1. The value must be greater than 0.

**B-E leakage current, ISE**

The base-emitter leakage current. The default value is  $\theta$  A/m<sup>2</sup>. The value must be greater than or equal to  $\theta$ .

**B-E emission coefficient, NE**

The base-collector emission coefficient or ideality factor. The default value is 1.5. The value must be greater than  $\theta$ .

**Forward knee current, IKF**

The current value at which forward-beta high-current roll-off occurs. The default value is Inf A/m<sup>2</sup>. The value must be greater than or equal to  $\theta$ .

**Forward Early voltage, VAF**

The forward Early voltage. The default value is Inf V. The value must be greater than or equal to  $\theta$ .

**Reverse Gain Tab**

**Reverse beta, BR**

The ideal maximum reverse beta. The default value is 1. The value must be greater than  $\theta$ .

**Reverse emission coefficient, NR**

The reverse emission coefficient or ideality factor. The default value is 1. The value must be greater than  $\theta$ .

**B-C leakage current, ISC**

The base-collector leakage current. The default value is  $\theta$  A/m<sup>2</sup>. The value must be greater than or equal to  $\theta$ .

**B-C emission coefficient, NC**

The base-collector emission coefficient or ideality factor. The default value is 2. The value must be greater than  $\theta$ .

**Reverse knee current, IKR**

The current value at which reverse-beta high-current roll-off occurs. The default value is Inf A/m<sup>2</sup>. The value must be greater than or equal to  $\theta$ .

**Reverse Early voltage, VAR**

The reverse Early voltage. The default value is Inf V. The value must be greater than or equal to  $\theta$ .

---

## Resistors Tab

### Emitter resistance, RE

The resistance of the emitter. The default value is  $1e-4 \text{ m}^2 \cdot \Omega$ . The value must be greater than or equal to 0.

### Collector resistance, RC

The resistance of the collector. The default value is  $0.01 \text{ m}^2 \cdot \Omega$ . The value must be greater than or equal to 0.

### Zero-bias base resistance, RB

The resistance of the base. The default value is  $1 \text{ m}^2 \cdot \Omega$ . The value must be greater than or equal to 0.

### Minimum base resistance, RBM

The minimum resistance of the base. The default value is  $0 \text{ m}^2 \cdot \Omega$ . The value must be less than or equal to the **Zero-bias base resistance, RB** parameter value.

### Half base resistance cur, IRB

The base current at which the base resistance has dropped to half of its zero-bias value. The default value is  $\text{Inf A/m}^2$ . The value must be greater than or equal to 0. Use the default value of  $\text{Inf}$  if you do not want to model the change in base resistance as a function of base current.

## Capacitance Tab

### Model junction capacitance

Select one of the following options for modeling the junction capacitance:

- **No** — Do not include junction capacitance in the model. This is the default option.
- **B-E Capacitance** — Model the junction capacitance across the base-emitter junction.
- **B-C Capacitance** — Model the junction capacitance across the base-collector junction.
- **C-S Capacitance** — Model the junction capacitance across the collector-substrate junction.

---

**Note** To include junction capacitance in the model:

- 1 Select **B-E Capacitance** and specify the base-emitter junction capacitance parameters.
- 2 Select **B-C Capacitance** and specify the base-collector junction capacitance parameters.
- 3 Select **C-S Capacitance** and specify the collector-substrate junction capacitance parameters.

You can specify or change any of the common parameters when you select any of the preceding options for the **Model junction capacitance** parameter.

---

### **B-E depletion capacitance, CJE**

The depletion capacitance across the base-emitter junction. This parameter is only visible when you select **B-E Capacitance** for the **Model junction capacitance** parameter. The default value is  $0 \text{ F/m}^2$ . The value must be greater than or equal to  $0$ .

### **B-E built-in potential, VJE**

The base-emitter junction potential. This parameter is only visible when you select **B-E Capacitance** for the **Model junction capacitance** parameter. The default value is  $0.75 \text{ V}$ . The value must be greater than or equal to  $0.01 \text{ V}$ .

### **B-E exponential factor, MJE**

The grading coefficient for the base-emitter junction. This parameter is only visible when you select **B-E Capacitance** for the **Model junction capacitance** parameter. The default value is  $0.33$ . The value must be greater than or equal to  $0$  and less than or equal to  $0.9$ .

### **Forward transit time, TF**

The transit time of the minority carriers that cause diffusion capacitance when the base-emitter junction is forward-biased. This parameter is only visible when you select **B-E Capacitance** for the **Model junction capacitance** parameter. The default value is  $0$ . The value must be greater than or equal to  $0$ .

### **Coefficient of TF, XTF**

The coefficient for the base-emitter and base-collector bias dependence of the transit time, which produces a charge across the base-emitter junction. This parameter is only visible when you select **B-E Capacitance** for the **Model junction capacitance** parameter. The default value is  $0$ . The value must be greater than or equal to  $0$ . Use the default value of  $0$  if you do not want to model the effect of base-emitter bias on transit time.

**VBC dependence of TF, VTF**

The coefficient for the base-emitter bias dependence of the transit time. This parameter is only visible when you select B-E Capacitance for the **Model junction capacitance** parameter. The default value is Inf V. The value must be greater than or equal to 0.

**Coefficient of TF, ITF**

The coefficient for the dependence of the transit time on collector current. This parameter is only visible when you select B-E Capacitance for the **Model junction capacitance** parameter. The default value is 0 A/m<sup>2</sup>. The value must be greater than or equal to 0. Use the default value of 0 if you do not want to model the effect of collector current on transit time.

**B-C depletion capacitance, CJC**

The depletion capacitance across the base-collector junction. This parameter is only visible when you select B-C Capacitance for the **Model junction capacitance** parameter. The default value is 0 F/m<sup>2</sup>. The value must be greater than 0.

**B-C built-in potential, VJC**

The base-collector junction potential. This parameter is only visible when you select B-C Capacitance for the **Model junction capacitance** parameter. The default value is 0.75 V. The value must be greater than or equal to 0.01 V.

**B-C exponential factor, MJC**

The grading coefficient for the base-collector junction. This parameter is only visible when you select B-C Capacitance for the **Model junction capacitance** parameter. The default value is 0.33. The value must be greater than or equal to 0 and less than or equal to 0.9.

**B-C capacitance fraction, XCJC**

The fraction of the base-collector depletion capacitance that is connected between the internal base and the internal collector. The rest of the base-collector depletion capacitance is connected between the external base and the internal collector. This parameter is only visible when you select B-C Capacitance for the **Model junction capacitance** parameter. The default value is 0. The value must be greater than or equal to 0 and less than or equal to 1.

**Reverse transit time, TR**

The transit time of the minority carriers that cause diffusion capacitance when the base-collector junction is reverse-biased. This parameter is only visible when you select B-C Capacitance for the **Model junction capacitance** parameter. The default value is 0 s. The value must be greater than or equal to 0.

### **Capacitance coefficient FC**

The fitting coefficient that quantifies the decrease of the depletion capacitance with applied voltage. This parameter is only visible when you select **B-E Capacitance** or **B-C Capacitance** for the **Model junction capacitance** parameter. The default value is 0.5. The value must be greater than or equal to 0 and less than or equal to 0.95.

### **Specify initial condition**

Select one of the following options for specifying an initial condition:

- **No** — Do not specify an initial condition for the model. This is the default option.
- **Yes** — Specify the initial transistor conditions.

---

**Note** The NPN block applies the initial transistor voltages across the junction capacitors and not across the ports.

---

This parameter is only visible when you select **B-E Capacitance** or **B-C Capacitance** for the **Model junction capacitance** parameter.

### **Initial condition voltage ICVBE**

Base-emitter voltage at the start of the simulation. This parameter is only visible when you select **B-E Capacitance** or **B-C Capacitance** for the **Model junction capacitance** and **Yes** for the **Specify initial condition** parameter. The default value is 0 V.

### **Initial condition voltage ICVCE**

Base-collector voltage at the start of the simulation. This parameter is only visible when you select **B-E Capacitance** or **B-C Capacitance** for the **Model junction capacitance** and **Yes** for the **Specify initial condition** parameter. The default value is 0 V.

### **C-S junction capacitance, CJS**

The collector-substrate junction capacitance. This parameter is only visible when you select **C-S Capacitance** for the **Model junction capacitance** parameter. The default value is 0 F/m<sup>2</sup>. The value must be greater than or equal to 0.

### **Substrate built-in potential, VJS**

The potential of the substrate. This parameter is only visible when you select **C-S Capacitance** for the **Model junction capacitance** parameter. The default value is 0.75 V.

### Substrate exponential factor, MJS

The grading coefficient for the collector-substrate junction. This parameter is only visible when you select C-S Capacitance for the **Model junction capacitance** parameter. The default value is 0. The value must be greater than or equal to 0 and less than or equal to 0.9.

## Temperature Tab

### Model temperature dependence using

Select one of the following options for modeling the transistor temperature dependence:

- **Device temperature** — Use the device temperature, which is the **Circuit temperature** value plus the **Offset local circuit temperature, TOFFSET** value. The **Circuit temperature** value comes from the SPICE Environment Parameters block, if one exists in the circuit. Otherwise, it comes from the default value for this block.
- **Fixed temperature** — Use a temperature that is independent of the circuit temperature to model temperature dependence.

### Beta temperature exponent, XTB

The forward and reverse beta temperature exponent that models base current temperature dependence. This parameter is only visible when you select **Device temperature** for the **Model temperature dependence using** parameter. The default value is 0. The value must be greater than or equal to 0.

### Energy gap, EG

The energy gap that affects the increase in the saturation current as temperature increases. This parameter is only visible when you select **Device temperature** for the **Model temperature dependence using** parameter. The default value is 1.11 eV. The value must be greater than or equal to 0.1.

### Temperature exponent for IS, XTI

The order of the exponential increase in the saturation current as temperature increases. This parameter is only visible when you select **Device temperature** for the **Model temperature dependence using** parameter. The default value is 3. The value must be greater than or equal to 0.

## Offset local circuit temperature, **TOFFSET**

The amount by which the transistor temperature differs from the circuit temperature. This parameter is only visible when you select `Device temperature` for the **Model temperature dependence using** parameter. The default value is 0 K.

## Parameter extraction temperature, **TMEAS**

The temperature at which the transistor parameters were measured. The default value is 300.15 K. The value must be greater than 0.

## Fixed circuit temperature, **TFIXED**

The temperature at which to simulate the transistor. This parameter is only visible when you select `Fixed temperature` for the **Model temperature dependence using** parameter. The default value is 300.15 K. The value must be greater than 0.

## Ports

The block has the following ports:

**B**

Electrical conserving port associated with the transistor base terminal.

**C**

Electrical conserving port associated with the transistor collector terminal.

**E**

Electrical conserving port associated with the transistor emitter terminal.

**S**

Electrical conserving port associated with the transistor substrate terminal.

## References

- [1] G. Massobrio and P. Antognetti. *Semiconductor Device Modeling with SPICE*. 2nd Edition, McGraw-Hill, 1993. Chapter 2.



## **See Also**

NPN Bipolar Transistor

## SPICE PJFET

Model SPICE-compatible P-Channel JFET



## Library

SPICE-Compatible Components/Semiconductor Devices

## Description

The PJFET block represents a SPICE-compatible P-channel JFET.

The PJFET block model includes the following components:

- “Source-Gate Current-Voltage Model” on page 1-1620
- “Drain-Gate Current-Voltage Model” on page 1-1621
- “Source-Drain Current-Voltage Model” on page 1-1622
- “Junction Charge Model” on page 1-1623
- “Temperature Dependence” on page 1-1624

## Source-Gate Current-Voltage Model

The block provides the following relationship between the source-gate current  $I_{sg}$  and the source-gate voltage  $V_{sg}$  after adjusting the applicable model parameters for temperature.

Applicable Range of $V_{sg}$ Values	Corresponding $I_{sg}$ Equation
$V_{sg} > 80 * V_t$	$I_{sg} = IS * \left( \left( \frac{V_{sg}}{V_t} - 79 \right) e^{80} - 1 \right) + V_{sg} * G \text{ min}$

Applicable Range of $V_{sg}$ Values	Corresponding $I_{sg}$ Equation
$80 * V_t \geq V_{sg}$	$I_{sg} = IS * \left( e^{V_{sg}/V_t} - 1 \right) + V_{sg} * G_{min}$

Where:

- $IS$  is the **Saturation current, IS** parameter value.
- $V_t = ND * k * T / q$
- $ND$  is the **Emission coefficient, N** parameter value.
- $q$  is the elementary charge on an electron.
- $k$  is the Boltzmann constant.
- $T$  is the diode temperature:
  - If you select Device temperature for the **Model temperature dependence using** parameter,  $T$  is the sum of the **Circuit temperature** value plus the **Offset local circuit temperature, TOFFSET** parameter value. The **Circuit temperature** value comes from the SPICE Environment Parameters block, if one exists in the circuit. Otherwise, it comes from the default value for this block.
  - If you select Fixed temperature for the **Model temperature dependence using** parameter,  $T$  is the **Fixed circuit temperature, TFIXED** parameter value.
- $GMIN$  is the diode minimum conductance. By default,  $GMIN$  matches the **Minimum conductance GMIN** parameter of the SPICE Environment Parameters block, whose default value is 1e-12. To change  $GMIN$ , add a SPICE Environment Parameters block to your model and set the **Minimum conductance GMIN** parameter to the desired value.

## Drain-Gate Current-Voltage Model

The block provides the following relationship between the drain-gate current  $I_{dg}$  and the drain-gate voltage  $V_{dg}$  after adjusting the applicable model parameters for temperature.

Applicable Range of $V_{dg}$ Values	Corresponding $I_{dg}$ Equation
$V_{dg} > 80 * V_t$	$I_{dg} = IS * \left( \left( \frac{V_{dg}}{V_t} - 79 \right) e^{80} - 1 \right) + V_{dg} * G \text{ min}$
$80 * V_t \geq V_{dg}$	$I_{dg} = IS * \left( e^{V_{dg}/V_t} - 1 \right) + V_{dg} * G \text{ min}$

## Source-Drain Current-Voltage Model

The block provides the following relationship between the source-drain current  $I_{sd}$  and the source-drain voltage  $V_{sd}$  in normal mode ( $V_{sd} \geq 0$ ) after adjusting the applicable model parameters for temperature.

Applicable Range of $V_{sg}$ and $V_{dg}$ Values	Corresponding $I_{sd}$ Equation
$V_{sg} - V_{to} \leq 0$	$I_{sd} = 0$
$0 < V_{sg} - V_{to} \leq V_{sd}$	$I_{sd} = -\beta * (V_{sg} - V_{to})^2 * (1 + \lambda * V_{sd})$
$0 < V_{sd} < V_{sg} - V_{to}$	$I_{sd} = \beta * V_{sd} * (2 * (V_{sg} - V_{to}) - V_{sd}) * (1 + \lambda * V_{sd})$

Where:

- $V_{to}$  is the **Threshold voltage, VTO** parameter value.
- $\beta$  is the **Transconductance, BETA** parameter value.
- $\lambda$  is the **Channel modulation, LAMBDA** parameter value.

The block provides the following relationship between the source-drain current  $I_{sd}$  and the source-drain voltage  $V_{sd}$  in inverse mode ( $V_{sd} < 0$ ) after adjusting the applicable model parameters for temperature.

Applicable Range of $V_{sg}$ and $V_{dg}$ Values	Corresponding $I_{sd}$ Equation
$V_{dg} - V_{to} \leq 0$	$I_{sd} = 0$
$0 < V_{dg} - V_{to} \leq -V_{sd}$	$I_{sd} = \beta * (V_{dg} - V_{to})^2 * (1 - \lambda * V_{sd})$
$0 < -V_{sd} < V_{dg} - V_{to}$	$I_{sd} = \beta * V_{sd} * (2 * (V_{dg} - V_{to}) + V_{sd}) * (1 - \lambda * V_{sd})$

## Junction Charge Model

The block provides the following relationship between the source-gate charge  $Q_{sg}$  and the source-gate voltage  $V_{sg}$  after adjusting the applicable model parameters for temperature.

Applicable Range of $V_{sg}$ Values	Corresponding $Q_{sg}$ Equation
$V_{sg} < FC * VJ$	$Q_{sg} = \frac{CGS * VJ * \left( 1 - \left( 1 - \frac{V_{sg}}{VJ} \right)^{1-MG} \right)}{1 - MG}$
$V_{sg} \geq FC * VJ$	$Q_{sg} = CGS * \left( F1 + \frac{F3 * (V_{sg} - FC * VJ) + \frac{MG * (V_{sg}^2 - (FC * VJ)^2)}{2 * VJ}}{F2} \right)$

Where:

- $FC$  is the **Capacitance coefficient FC** parameter value.
- $VJ$  is the **Junction potential VJ** parameter value.
- $CGS$  is the **Zero-bias GS capacitance, CGS** parameter value.
- $MG$  is the **Grading coefficient, M** parameter value.

•

$$F1 = \frac{VJ * (1 - (1 - FC)^{1-MG})}{1 - MG}$$

•

$$F2 = (1 - FC)^{1+MG}$$

•

$$F3 = 1 - FC * (1 + MG)$$

The block provides the following relationship between the drain-gate charge  $Q_{dg}$  and the drain-gate voltage  $V_{dg}$  after adjusting the applicable model parameters for temperature.

Applicable Range of $V_{dg}$ Values	Corresponding $Q_{dg}$ Equation
$V_{dg} < FC * VJ$	$Q_{dg} = \frac{CGD * VJ * \left( 1 - \left( 1 - \frac{V_{dg}}{VJ} \right)^{1-MG} \right)}{1 - MG}$
$V_{dg} \geq FC * VJ$	$Q_{dg} = CGD * \left( F1 + \frac{F3 * (V_{dg} - FC * VJ) + \frac{MG * (V_{dg}^2 - (FC * VJ)^2)}{2 * VJ}}{F2} \right)$

Where:

- $CGD$  is the **Zero-bias GD capacitance**, **CGD** parameter value.

## Temperature Dependence

Several transistor parameters depend on temperature. There are two ways to specify the transistor temperature:

- When you select `Device` temperature for the **Model temperature dependence using** parameter, the transistor temperature is

$$T = T_C + T_O$$

where:

- $T_C$  is the **Circuit temperature** parameter value from the SPICE Environment Parameters block. If this block doesn't exist in the circuit,  $T_C$  is the default value of this parameter.
- $T_O$  is the **Offset local circuit temperature, TOFFSET** parameter value.
- When you select Fixed temperature for the **Model temperature dependence using** parameter, the transistor temperature is the **Fixed circuit temperature, TFIXED** parameter value.

The block provides the following relationship between the saturation current  $IS$  and the transistor temperature  $T$ :

$$IS(T) = IS * \left( T / T_{meas} \right)^{\frac{XTI}{ND}} * e^{\left( \frac{T}{T_{meas}} - 1 \right) * \frac{EG}{V_t}}$$

where:

- $IS$  is the **Saturation current, IS** parameter value.
- $T_{meas}$  is the **Parameter extraction temperature, TMEAS** parameter value.
- $XTI$  is the **Saturation current temperature exponent, XTI** parameter value.
- $EG$  is the **Energy gap, EG** parameter value.
- $V_t = ND * k * T / q$
- $ND$  is the **Emission coefficient, N** parameter value.

The block provides the following relationship between the junction potential  $VJ$  and the transistor temperature  $T$ :

$$VJ(T) = VJ * \left( \frac{T}{T_{meas}} \right) - \frac{3 * k * T}{q} * \log \left( \frac{T}{T_{meas}} \right) - \left( \frac{T}{T_{meas}} \right) * EG_{T_{meas}} + EG_T$$

where:

- $VJ$  is the **Junction potential VJ** parameter value.

- $EG_{T_{meas}} = 1.16eV - (7.02e - 4 * T_{meas}^2) / (T_{meas} + 1108)$
- $EG_T = 1.16eV - (7.02e - 4 * T^2) / (T + 1108)$

The block provides the following relationship between the gate-source junction capacitance  $CGS$  and the transistor temperature  $T$ :

$$CGS(T) = CGS * \left[ 1 + MG * \left( 400e - 6 * (T - T_{meas}) - \frac{VJ(T) - VJ}{VJ} \right) \right]$$

where:

- $CGS$  is the **Zero-bias GS capacitance, CGS** parameter value.

The block uses the  $CGS(T)$  equation to calculate the gate-drain junction capacitance by substituting  $CGD$  (the **Zero-bias GD capacitance, CGD** parameter value) for  $CGS$ .

The block provides the following relationship between the forward and reverse beta and the transistor temperature  $T$ :

$$\beta(T) = \beta * \left( \frac{T}{T_{meas}} \right)$$

where  $\beta$  is the **Transconductance, BETA** parameter value.

## Basic Assumptions and Limitations

The model is based on the following assumptions:

- The PJFET block does not support noise analysis.
- The PJFET block applies initial conditions across junction capacitors and not across the block ports.



# Parameters

## Main Tab

### Device area, AREA

The transistor area. This value multiplies the **Transconductance, BETA, Zero-bias GS capacitance, CGS, Zero-bias GD capacitance, CGD, and Saturation current, IS** parameter values. It divides the **Source resistance, RS** and **Drain resistance, RD** parameter values. The default value is  $1 \text{ m}^2$ . The value must be greater than 0.

### Number of parallel devices, SCALE

The number of parallel transistors the block represents. This value multiplies the output current and device charges. The default value is 1. The value must be greater than 0.

### Threshold voltage, VTO

The gate-source voltage above which the transistor produces a nonzero drain current. The default value is  $-2 \text{ V}$ .

### Transconductance, BETA

The derivative of drain current with respect to gate voltage. The default value is  $1\text{e-}04 \text{ A/m}^2/\text{V}^2$ . The value must be greater than or equal to 0.

### Channel modulation, LAMBDA

The channel-length modulation. The default value is  $0 \text{ 1/V}$ .

### Saturation current, IS

The magnitude of the current that the ideal diode equation approaches asymptotically for very large reverse bias levels. The default value is  $1\text{e-}14 \text{ A/m}^2$ . The value must be greater than or equal to 0.

### Emission coefficient, N

The transistor emission coefficient or ideality factor. The default value is 1. The value must be greater than 0.

### Source resistance, RS

The transistor source resistance. The default value is  $1\text{e-}4 \text{ m}^2*\Omega$ . The value must be greater than or equal to 0.

### Drain resistance, RD

The transistor drain resistance. The default value is  $0.01 \text{ m}^2*\Omega$ . The value must be greater than or equal to 0.

## Junction Capacitance Tab

### Model junction capacitance

Select one of the following options for modeling the junction capacitance:

- **No** — Do not include junction capacitance in the model. This is the default option.
- **Yes** — Specify zero-bias junction capacitance, junction potential, grading coefficient, forward-bias depletion capacitance coefficient, and transit time.

### Zero-bias GS capacitance, CGS

The value of the capacitance placed between the gate and the source. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is  $0 \text{ F/m}^2$ . The value must be greater than or equal to  $0$ .

### Zero-bias GD capacitance, CGD

The value of the capacitance placed between the gate and the drain. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is  $0 \text{ F/m}^2$ . The value must be greater than or equal to  $0$ .

### Junction potential VJ

The junction potential. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is  $1 \text{ V}$ . The value must be greater than  $0.01 \text{ V}$ .

### Grading coefficient, M

The transistor grading coefficient. The default value is  $0.5$ . The value must be greater than  $0$  and less than  $0.9$ .

### Capacitance coefficient FC

The fitting coefficient that quantifies the decrease of the depletion capacitance with applied voltage. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is  $0.5$ . The value must be greater than or equal to  $0$  and less than or equal to  $0.95$ .

### Specify initial condition

Select one of the following options for specifying an initial condition:

- **No** — Do not specify an initial condition for the model. This is the default option.
- **Yes** — Specify the initial diode voltage.

---

**Note** The PJFET block applies the initial diode voltage across the junction capacitors and not across the ports.

---

### Initial condition voltage ICVDS

Drain-source voltage at the start of the simulation. This parameter is only visible when you select **Yes** for the **Model junction capacitance** and **Yes** for the **Specify initial condition** parameter. The default value is 0 V.

### Initial condition voltage ICVGS

Gate-source voltage at the start of the simulation. This parameter is only visible when you select **Yes** for the **Model junction capacitance** and **Yes** for the **Specify initial condition** parameter. The default value is 0 V.

## Temperature Tab

### Model temperature dependence using

Select one of the following options for modeling the diode temperature dependence:

- **Device temperature** — Use the device temperature, which is the **Circuit temperature** value plus the **Offset local circuit temperature, TOFFSET** value. The **Circuit temperature** value comes from the SPICE Environment Parameters block, if one exists in the circuit. Otherwise, it comes from the default value for this block.
- **Fixed temperature** — Use a temperature that is independent of the circuit temperature to model temperature dependence.

### Saturation current temperature exponent, XTI

The order of the exponential increase in the saturation current as temperature increases. The default value is 0. The value must be greater than or equal to 0.

### Activation energy, EG

The energy gap that affects the increase in the saturation current as temperature increases. The default value is 1.11 eV. The value must be greater than 0.1 eV.

### Offset local circuit temperature, TOFFSET

The amount by which the transistor temperature differs from the circuit temperature. This parameter is only visible when you select **Device temperature** for the **Model temperature dependence using** parameter. The default value is 0 K.

### **Fixed circuit temperature, TFIXED**

The temperature at which to simulate the transistor. This parameter is only visible when you select `Fixed temperature` for the **Model temperature dependence using** parameter. The default value is 300.15 K. The value must be greater than 0.

### **Parameter extraction temperature, TMEAS**

The temperature at which the transistor parameters were measured. The default value is 300.15 K. The value must be greater than 0.

## **Ports**

The block has the following ports:

G

Electrical conserving port associated with the transistor gate terminal.

D

Electrical conserving port associated with the transistor drain terminal.

S

Electrical conserving port associated with the transistor source terminal.

## **References**

- [1] G. Massobrio and P. Antognetti. *Semiconductor Device Modeling with SPICE*. 2nd Edition, McGraw-Hill, 1993. Chapter 3.

## **See Also**

P-Channel JFET, SPICE NJFET

# SPICE PMOS

Model SPICE-compatible P-Channel MOSFET



## Library

SPICE-Compatible Components/Semiconductor Devices

## Description

The PMOS block represents a SPICE-compatible P-channel MOSFET.

The PMOS block model includes the following components:

- “Resistance Calculations” on page 1-1631
- “Bulk-Source Diode Model” on page 1-1632
- “Bulk-Drain Diode Model” on page 1-1633
- “Level 1 Drain Current Model” on page 1-1634
- “Level 3 Drain Current Model” on page 1-1637
- “Junction Charge Model” on page 1-1642
- “Temperature Dependence” on page 1-1646

## Resistance Calculations

The following table shows how the PMOS block calculates the transistor drain resistance. The abbreviations in the table represent the values of the following block parameters:

- **Drain resistance, RD**
- **Sheet resistance, RSH**
- **Number of drain squares, NRD**

Drain resistance, RD Parameter	Sheet resistance, RSH Parameter	Drain Resistance
NaN	NaN	0
RD	NaN or RSH	RD
NaN	RSH	RSH*NRD

The following table shows how the PMOS block calculates the transistor source resistance. The abbreviations in the table represent the values of the following block parameters:

- **Source resistance, RS**
- **Sheet resistance, RSH**
- **Number of source squares, NRS**

Source resistance, RS Parameter	Sheet resistance, RSH Parameter	Source Resistance
NaN	NaN	0
RS	NaN or RSH	RS
NaN	RSH	RSH*NRS

## Bulk-Source Diode Model

The block provides the following relationship between the bulk-source current  $I_{sb}$  and the bulk-source voltage  $V_{sb}$  after adjusting the applicable model parameters for temperature.

Applicable Range of $V_{sb}$ Values	Corresponding $I_{gs}$ Equation
$V_{sb} > 80 * V_{tn}$	$I_{sb} = IS_{sb} * \left( \left( \frac{V_{sb}}{V_m} - 79 \right) e^{80} - 1 \right) + V_{sb} * G \text{ min}$
$80V_{tn} \geq V_{sb}$	$I_{sb} = IS_{sb} * \left( e^{V_{sb}/V_m} - 1 \right) + V_{sb} * G \text{ min}$

Where:

- $IS_{sb}$  is
  - The product of the **Bulk jct sat current density, JS** parameter value and the **Area of source, AS** parameter value if both these parameter values and the **Area of drain, AD** parameter value are nonzero.
  - The **Bulk saturation current, IS** parameter value, otherwise.
- $V_{tn} = NkT/q$
- $q$  is the elementary charge on an electron, 1.6021918e-19 C.
- $N$  is the **Emission coefficient, N** parameter value.
- $k$  is the Boltzmann constant.
- $T$  is the diode temperature:
  - If you select Device temperature for the **Model temperature dependence using** parameter,  $T$  is the sum of the **Circuit temperature** value plus the **Offset local circuit temperature, TOFFSET** parameter value. The **Circuit temperature** value comes from the SPICE Environment Parameters block, if one exists in the circuit. Otherwise, it comes from the default value for this block.
  - If you select Fixed temperature for the **Model temperature dependence using** parameter,  $T$  is the **Fixed circuit temperature, TFIXED** parameter value.
- $GMIN$  is the diode minimum conductance. By default,  $GMIN$  matches the **Minimum conductance GMIN** parameter of the SPICE Environment Parameters block, whose default value is 1e-12. To change  $GMIN$ , add a SPICE Environment Parameters block to your model and set the **Minimum conductance GMIN** parameter to the desired value.

## Bulk-Drain Diode Model

The block provides the following relationship between the bulk-drain current  $I_{db}$  and the bulk-drain voltage  $V_{db}$  after adjusting the applicable model parameters for temperature.

Applicable Range of $V_{db}$ Values	Corresponding $I_{gs}$ Equation
$V_{db} > 80 * V_{tn}$	$I_{db} = IS_{db} * \left( \left( \frac{V_{db}}{V_{tn}} - 79 \right) e^{80} - 1 \right) + V_{db} * G \text{ min}$

Applicable Range of $V_{db}$ Values	Corresponding $I_{gs}$ Equation
$80V_{tn} \geq V_{db}$	$I_{db} = IS_{db} * (e^{V_{db}/V_{tn}} - 1) + V_{db} * G \text{ min}$

Where:

- $IS_{db}$  is
  - The product of the **Bulk jct sat current density, JS** parameter value and the **Area of drain, AD** parameter value if both these parameter values and the **Area of source, AS** parameter value are nonzero.
  - The **Bulk saturation current, IS** parameter value, otherwise.

## Level 1 Drain Current Model

The block provides the following relationship between the drain current  $I_{sd}$  and the drain-source voltage  $V_{sd}$  in normal mode ( $V_{sd} \geq 0$ ) after adjusting the applicable model parameters for temperature.

### Normal Mode

Applicable Range of $V_{sg}$ and $V_{sd}$ Values	Corresponding $I_{sd}$ Equation
$V_{sg} - V_{on} \leq 0$	$I_{sd} = 0$
$0 < V_{sg} - V_{on} \leq V_{sd}$	$I_{sd} = BETA * (V_{sg} - V_{on})^2 \frac{(1 + LAMBDA * V_{sd})}{2}$
$0 < V_{sd} < V_{sg} - V_{on}$	$I_{sd} = BETA * V_{sd} \left( (V_{sg} - V_{on}) - \frac{V_{sd}}{2} \right) (1 + LAMBDA * V_{sd})$

Where:

- $V_{on}$  is:



- $MTYPE * VBI + GAMMA \sqrt{PHI - V_{sb}}$  if  $V_{sb} \leq 0$ .
- $MTYPE * VBI + GAMMA \left( \sqrt{PHI} - \frac{V_{sb}}{2\sqrt{PHI}} \right)$  if  $0 < V_{sb} \leq 2 * PHI$ .
- $MTYPE * VBI$  if  $V_{sb} > 2 * PHI$ .
- $MTYPE$  is -1.
- $BETA$  is  $KP * WIDTH / (LENGTH - 2 * LD)$
- $KP$  is:
  - The **Transconductance, KP** parameter value, if this parameter has a numerical value.
  - $U0 * 3.9 * \epsilon_0 / TOX$ , if **Transconductance, KP** is NaN and you specify values for both the **Oxide thickness, TOX** and **Substrate doping, NSUB** parameters.
- $WIDTH$  is the **Width of channel, WIDTH** parameter value.
- $LENGTH$  is the **Length of channel, LENGTH** parameter value.
- $LD$  is the **Lateral diffusion, LD** parameter value.
- $VBI$  is an built-in voltage value the block uses in calculations. The value is a function of temperature. For a detailed definition, see "Temperature Dependence" on page 1-1586.
- $PHI$  is:
  - The **Surface potential, PHI** parameter value, if this parameter has a numerical value.
  - $2 * kT_{meas} / q * \log(NSUB / n_i)$ , if **Surface potential, PHI** is NaN and you specify values for both the **Oxide thickness, TOX** and **Substrate doping, NSUB** parameters.
- $LAMBDA$  is the **Channel modulation, LAMBDA** parameter value.
- $GAMMA$  is:
  - The **Bulk threshold, GAMMA** parameter value, if this parameter has a numerical value.

- $TOX * \sqrt{2 * 11.7 * \epsilon_0 * q * NSUB} / (3.9 * \epsilon_0)$ , if **Bulk threshold**, **GAMMA** is NaN and you specify values for both the **Oxide thickness**, **TOX** and **Substrate doping**, **NSUB** parameters.
- $\epsilon_0$  is the permittivity of free space, 8.854214871e-12 F/m.
- $n_i$  is the carrier concentration of intrinsic silicon, 1.45e10 cm<sup>-3</sup>.

The block provides the following relationship between the drain current  $I_{sd}$  and the drain-source voltage  $V_{sd}$  in inverse mode ( $V_{sd} < 0$ ) after adjusting the applicable model parameters for temperature.

### Inverse Mode

Applicable Range of $V_{dg}$ and $V_{sd}$ Values	Corresponding $I_{sd}$ Equation
$V_{dg} - V_{on} \leq 0$	$I_{sd} = 0$
$0 < V_{dg} - V_{on} \leq -V_{sd}$	$I_{sd} = -BETA (V_{dg} - V_{on})^2 (1 - LAMBDA * V_{sd}) / 2$
$0 < V_{sd} < V_{dg} - V_{on}$	$I_{sd} = BETA * V_{sd} \left( (V_{dg} - V_{on}) + V_{sd} / 2 \right) (1 - LAMBDA * V_{sd})$

Where:

- $V_{on}$  is:
  - $MTYPE * VBI + GAMMA \sqrt{PHI - V_{db}}$  if  $V_{db} \leq 0$
  - $MTYPE * VBI + GAMMA \left( \sqrt{PHI} - \frac{V_{db}}{2\sqrt{PHI}} \right)$  if  $0 < V_{db} \leq 2 * PHI$

- $MTYPE * VBI$  if  $V_{db} > 2 * PHI$ .

### Level 3 Drain Current Model

The block provides the following model for drain current  $I_{sd}$  in normal mode ( $V_{sd} \geq 0$ ) after adjusting the applicable model parameters for temperature.

$$I_{SD} = I_{SD0} * Scale_{VMAX} * Scale_{LChan} * Scale_{INV}$$

Where:

- $I_{SD0}$  is the Basic Drain Current Model on page 1-1637.
- $Scale_{VMAX}$  is the Velocity Saturation Scaling on page 1-1640.
- $Scale_{LChan}$  is the Channel Length Modulation Scaling on page 1-1640.
- $Scale_{INV}$  is the Weak Inversion Scaling on page 1-1642.

The blocks uses the same model for drain current in inverse mode ( $V_{sd} < 0$ ), with the following substitutions:

- $V_{sb} - V_{sd}$  for  $V_{sb}$
- $V_{sg} - V_{sd}$  for  $V_{sd}$
- $-V_{sd}$  for  $V_{sd}$

The block provides the following relationship between the drain current  $I_{sd}$  and the drain-source voltage  $V_{ds}$ :

$$I_{SD0} = BETA * F_{gate} * \left( V_{SGX} - V_{TH} - \frac{1 + F_B}{2} * V_{SDX} \right) * V_{SDX}$$

- The block calculates BETA as described in “Level 1 Drain Current Model” on page 1-1634.
- The block calculates  $F_{GATE}$  using the following equation:

$$F_{gate} = \frac{1}{1 + THETA * (V_{sgv} - V_{TH})}$$

- **THETA** is the **Vgs dependence on mobility**, **THETA** parameter value.
- 
- $V_{sgv} = \max(V_{SG}, V_{on})$
- If you specify a nonzero value for the **Fast surface state density**, **NFS** parameter, the block calculates  $V_{on}$  using the following equation:

$$V_{on} = V_{TH} + x_n V_T$$

Otherwise,  $V_{on} = V_{TH}$ .

- The block calculates  $x_n$  using the following equation:

$$x_n = 1 + \frac{q * NFS}{COX} + \frac{\left( GAMMA * F_s * \sqrt{V_{bulk}} + \frac{F_n * V_{bulk}}{WIDTH} \right)}{2 * V_{bulk}}$$

- The block calculates  $V_{bulk}$  as follows:
  - If  $V_{SB} \leq 0$ ,  $V_{bulk} = PHI - V_{BS}$ .
  - Otherwise, the block calculates  $V_{bulk}$  using the following equation:

$$V_{bulk} = \frac{PHI}{\left( 1 + \frac{V_{SB}}{2 * PHI} \right)^2}$$

- $V_T = kT/q$
- The block calculates  $V_{TH}$  using the equation following equation:

$$V_{TH} = V_{BI} - \frac{8.15e^{-22} * ETA}{COX * (LENGTH - 2 * LD)^3} * V_{SD} + GAMMA * F_s * \sqrt{V_{bulk}} + F_n * V_{bulk}$$

- For information about how the block calculates  $V_{BI}$ , see “Temperature Dependence” on page 1-1646.
- $ETA$  is the **Vds dependence threshold volt, ETA** parameter value.
- $COX = \epsilon_{ox}/TOX$ , where  $\epsilon_{ox}$  is the permittivity of the oxide and  $TOX$  is the **Oxide thickness, TOX** parameter value.
- If you specify a nonzero value for the **Junction depth, XJ** parameter and a value for the **Substrate doping, NSUB** parameter, the block calculates  $F_s$  using the following equations:

$$\alpha = \frac{2\epsilon_{si}}{qNSUB}$$

$$XD = \sqrt{\alpha}$$

$$wc = .0631353 + .8013292 * \frac{XD * \sqrt{V_{bulk}}}{XJ}$$

$$-.01110777 * \left( \frac{XD * \sqrt{V_{bulk}}}{XJ} \right)^2 + \frac{LD}{XJ}$$

$$F_s = 1 - \left( wc * \sqrt{1 - \left( \frac{XD * \sqrt{V_{bulk}}}{XJ + XD * \sqrt{V_{bulk}}} \right)^2} - \frac{LD}{XJ} \right)$$

where  $\epsilon_{si}$  is the permittivity of silicon.

Otherwise,  $F_s = 1$ .

- The block calculates  $F_B$  using the following equation:

$$F_B = \frac{GAMMA * F_s}{4 * \sqrt{V_{bulk}}} + F_n$$

- The block calculates  $F_n$  using the following equation:

$$F_n = \frac{DELTA * \pi * \epsilon_{si}}{2 * COX * WIDTH}$$

- *DELTA* is the **Width effect on threshold, DELTA** parameter value.
- $V_{SDX}$  is the lesser of  $V_{SD}$  and the saturation voltage,  $V_{dsat}$ .
- If you specify a positive value for the **Max carrier drift velocity, VMAX** parameter, the block calculates  $V_{dsat}$  using the following equation:

$$V_{dsat} = \frac{V_{sgx} - V_{TH}}{1 + F_B} + \frac{(LENGTH - 2 * LD) * VMAX}{UO * F_{gate}}$$

$$- \sqrt{\left(\frac{V_{sgx} - V_{TH}}{1 + F_B}\right)^2 + \left(\frac{(LENGTH - 2 * LD) * VMAX}{UO * F_{gate}}\right)^2}$$

Otherwise, the block calculates  $V_{dsat}$  using the following equation:

$$V_{dsat} = \frac{V_{sgx} - V_{TH}}{1 + F_B}$$

If you specify a positive value for the **Max carrier drift velocity, VMAX** parameter, the block calculates  $Scale_{VMAX}$  using the following equation:

$$Scale_{VMAX} = \frac{1}{1 + \frac{UO * F_{gate}}{(LENGTH - 2 * LD) * VMAX} * V_{SDX}}$$

Otherwise,  $Scale_{VMAX} = 1$ .

The block scales the drain current to account for channel length modulation if the block meets all of the following criteria:

- $V_{SD} > V_{sat}$
- The **Max carrier drift velocity, VMAX** parameter value is zero or  $\alpha$  is nonzero.

The block scales the drain current using the following equation:

$$Scale_{LChan} = \frac{1}{1 - \frac{\Delta l}{(LENGTH - 2 * LD)}}$$

The block uses the following procedure to calculate  $\Delta l$ :

- 1 The block first calculates the intermediate value  $\Delta l_0$ .
  - If you specify a positive value for the **Max carrier drift velocity, VMAX** parameter, the block computes the intermediate value  $g_{dsat}$  as the greater of 1e-12 and the result of the following equation:

$$I_{SD0} * \left( 1 - \frac{1}{1 + Scale_{g_{dsat}} * V_{SDX}} \right) * Scale_{g_{dsat}}$$

where:

$$Scale_{g_{dsat}} = \frac{UO * F_{gate}}{(LENGTH - 2 * LD) * VMAX}$$

Then, the block uses the following equation to calculate the intermediate value  $\Delta l_0$ :

$$\Delta l_0 = \sqrt{\left( \frac{KA * I_{SD}}{2 * (LENGTH - 2 * LD) * g_{dsat}} \right)^2 + KA * (V_{SD} - V_{dsat})} - \frac{KA * I_{SD}}{2 * (LENGTH - 2 * LD) * g_{dsat}}$$

where  $KA$  is the product of the **Mobility modulation, KAPPA** parameter value and  $\alpha$ .

- Otherwise, the block uses the following equation to calculate the intermediate value  $\Delta l_0$ :

$$\Delta l = \sqrt{KA * (V_{SD} - V_{dsat})}$$

**2** The block checks for punch through and calculates  $\Delta l$ .

- If  $\Delta l_0$  is greater than  $(LENGTH - 2 * LD) / 2$ , the block calculates  $\Delta l$  using the following equation:

$$\Delta l = \left( 1 - \frac{(LENGTH - 2 * LD)}{4 * \Delta l_0} \right) * (LENGTH - 2 * LD)$$

- Otherwise,  $\Delta l = \Delta l_0$ .

If  $V_{SG}$  is less than  $V_{on}$ , the block calculates  $Scale_{INV}$  using the following equation:

$$Scale_{INV} = e^{\frac{V_{sg} - V_{on}}{x_n * V_T}}$$

Otherwise,  $Scale_{INV} = 1$ .

## Junction Charge Model

The block models the following junction charges:

- Junction Overlap Charges
- Bulk Junction Charges

The block calculates the following junction overlap charges:

- $Q_{SG} = CGSO * WIDTH * V_{sg}$

Where:

- $Q_{SG}$  is the gate-source overlap charge.
- $CGSO$  is the **G-S overlap capacitance**, **CGSO** parameter value.
- $WIDTH$  is the **Width of channel**, **WIDTH** parameter value.
- $Q_{DG} = CGDO * WIDTH * V_{dg}$



Where:

- $Q_{DG}$  is the gate-drain overlap charge.
- $CGDO$  is the **G-D overlap capacitance**,  $CGDO$  parameter value.
- $Q_{BG} = CGBO * (LENGTH - 2 * LD) * V_{bg}$

Where:

- $Q_{BG}$  is the gate-bulk overlap charge.
- $CGBO$  is the **G-B overlap capacitance**,  $CGBO$  parameter value.
- $LENGTH$  is the **Length of channel**,  $LENGTH$  parameter value.
- $LD$  is the **Lateral diffusion**,  $LD$  parameter value.

The block provides the following relationship between the bulk-drain bottom junction charge  $Q_{bottom}$  and the junction voltage  $V_{db}$  after adjusting the applicable model parameters for temperature.

Applicable Range of $V_{db}$ Values	Corresponding $Q_{bottom}$ Equation
$V_{db} < FC * PB$	$Q_{bottom} = \frac{CBD * PB * \left( 1 - \left( 1 - \frac{V_{db}}{PB} \right)^{1-MJ} \right)}{1-MJ} \quad \text{if } CBD > 0.$ $Q_{bottom} = \frac{CJ * AD * PB * \left( 1 - \left( 1 - \frac{V_{db}}{PB} \right)^{1-MJ} \right)}{1-MJ} \quad \text{otherwise.}$

Applicable Range of $V_{db}$ Values	Corresponding $Q_{bottom}$ Equation
$V_{db} \geq FC * PB$	$Q_{bottom} = CBD *$ $\left( F1 + \frac{F3 * (V_{db} - FC * PB) + \frac{MJ * (V_{db}^2 - (FC * PB)^2)}{2 * PB}}{F2} \right) \text{ if } CBD > 0.$ $Q_{bottom} = CJ * AD *$ $\left( F1 + \frac{F3 * (V_{db} - FC * PB) + \frac{MJ * (V_{db}^2 - (FC * PB)^2)}{2 * PB}}{F2} \right)$ <p>otherwise.</p>

Where:

- $PB$  is the **Bulk junction potential, PB** parameter value.
- $FC$  is the **Capacitance coefficient FC** parameter value.
- $CBD$  is the **Zero-bias BD capacitance, CBD** parameter value.
- $CJ$  is the **Bottom junction cap per area, CJ** parameter value.
- $AD$  is the **Area of drain, AD** parameter value.
- $MJ$  is the **Bottom grading coefficient, MJ** parameter value.
- 

$$F1 = \frac{PB * (1 - (1 - FC)^{1 - MJ})}{1 - MJ}$$

$$F2 = (1 - FC)^{1 + MJ}$$

- $$F3 = 1 - FC * (1 + MJ)$$

The block uses the equations in the preceding table to calculate the bulk-source bottom junction charge, with the following substitutions:

- $V_{sb}$  replaces  $V_{db}$ .
- AS** (the **Area of source, AS** parameter value) replaces  $AD$ .
- CBS** (the **Zero-bias BS capacitance, CBS** parameter value) replaces  $CBD$ .

The block provides the following relationship between the bulk-drain sidewall junction charge  $Q_{sidewall}$  and the junction voltage  $V_{db}$  after adjusting the applicable model parameters for temperature.

Applicable Range of $V_{db}$ Values	Corresponding $Q_{sidewall}$ Equation
$V_{db} < FC * PB$	$Q_{sidewall} = \frac{CJSW * PD * PB * \left( 1 - \left( 1 - \frac{V_{db}}{PB} \right)^{1-MJSW} \right)}{1 - MJSW}$
$V_{db} \geq FC * PB$	$Q_{sidewall} = CJSW * PD * \left( F1 + \frac{F3 * (V_{db} - FC * PB) + \frac{MJSW * (V_{db}^2 - (FC * PB)^2)}{2 * PB}}{F2} \right)$

Where:

- $CJSW$  is the **Side jct cap/area of jct perimeter, CJSW** parameter value.
- $PD$  is the **Perimeter of drain, AD** parameter value.
- $MJSW$  is the **Side grading coefficient, MJSW** parameter value.

- 

$$F1 = \frac{PB * (1 - (1 - FC)^{1 - MJSW})}{1 - MJSW}$$

- 

$$F2 = (1 - FC)^{1 + MJSW}$$

- 

$$F3 = 1 - FC * (1 + MJSW)$$

The block uses the equations in the preceding table to calculate the bulk-source sidewall junction charge and the sidewall junction voltage, with the following substitutions:

- $V_{sb}$  replaces  $V_{db}$ .
- $PS$  (the **Perimeter of source, PS** parameter value) replaces  $PD$ .

## Temperature Dependence

Several transistor parameters depend on temperature. There are two ways to specify the transistor temperature:

- When you select `Device` temperature for the **Model temperature dependence using** parameter, the transistor temperature is

$$T = T_C + T_O$$

where:

- $T_C$  is the **Circuit temperature** parameter value from the SPICE Environment Parameters block. If this block doesn't exist in the circuit,  $T_C$  is the default value of this parameter.
- $T_O$  is the **Offset local circuit temperature, TOFFSET** parameter value.
- When you select `Fixed` temperature for the **Model temperature dependence using** parameter, the transistor temperature is the **Fixed circuit temperature, TFIXED** parameter value.

The block provides the following relationship between the transconductance  $KP$  and the transistor temperature  $T$ :

$$KP(T) = \frac{KP}{\left(\frac{T}{T_{meas}}\right)^{3/2}}$$

where:

- $KP$  is the **Transconductance, KP** parameter value.
- $T_{meas}$  is the **Parameter extraction temperature, TMEAS** parameter value.

The block provides the following relationship between the surface potential  $PHI$  and the transistor temperature  $T$ :

$$PHI(T) = \frac{T}{T_{meas}} \left( PHI + \frac{kT_{meas}}{q} \left( \log \left( \frac{T_{meas}}{300.15} \right)^3 + \frac{q}{k} \left( \frac{1.115}{300.15} - \frac{EG_{T_{meas}}}{T_{meas}} \right) \right) \right) - \frac{kT}{q} \left( \log \left( \frac{T}{300.15} \right)^3 + \frac{q}{k} \left( \frac{1.115}{300.15} - \frac{EG_T}{T} \right) \right)$$

where:

- $EG_{T_{meas}} = 1.16eV - (7.02e - 4 * T_{meas}^2) / (T_{meas} + 1108)$
- $EG_T = 1.16eV - (7.02e - 4 * T^2) / (T + 1108)$

The block provides the following relationship between the built-in voltage  $VBI$  and the transistor temperature  $T$ :

$$VBI(T) = VTO + MTYPE * \left( \frac{PHI(T) - PHI}{2} - GAMMA \sqrt{PHI} \right) + \frac{EG_{T_{meas}} - EG_T}{2}$$

where:

- $VTO$  is:

- The **Threshold voltage, VTO** parameter value, if this parameter has a numerical value.

$$\Phi - 3.25 + EG_{T_{meas}} / 2 + MTYPE * PHI / 2 - NSS * q * TOX / (3.9 * \epsilon_0)$$

+  $MTYPE * (GAMMA * \sqrt{PHI} + PHI)$ , if **Threshold voltage, VTO** is NaN and you specify values for both the **Oxide thickness, TOX** and **Substrate doping, NSUB** parameters.

- $\Phi$  is:

- 3.2, if **TPG** (the **Gate type?**, **TPG** parameter value) is 0.

$$3.25 + EG_{T_{meas}} / 2 - MTYPE * TPG * EG_{T_{meas}} / 2, \text{ otherwise.}$$

- **GAMMA** is:

- The **Bulk threshold, GAMMA** parameter value, if this parameter has a numerical value.

$$TOX * \sqrt{2 * 11.7 * \epsilon_0 * q * NSUB} / (3.9 * \epsilon_0), \text{ if } \mathbf{Bulk\ threshold, GAMMA} \text{ is NaN and you specify values for both the } \mathbf{Oxide\ thickness, TOX} \text{ and } \mathbf{Substrate\ doping, NSUB} \text{ parameters.}$$

The block provides the following relationship between the bulk saturation current  $IS$  and the transistor temperature  $T$ :

$$IS(T) = IS * e^{\frac{-qEG_T}{ND*kT} + \frac{qEG_{T_{meas}}}{ND*kT_{meas}}}$$

where:

- $ND$  is the **Emission coefficient, N** parameter value.
- $IS$  is the **Bulk saturation current, IS** parameter value.

The block provides the following relationship between the bulk junction saturation current density  $JS$  and the transistor temperature  $T$ :

$$JS(T) = JS * e^{\frac{-qEG_T}{ND*kT} + \frac{qEG_{T_{meas}}}{ND*kT_{meas}}}$$

where:

- $JS$  is the **Bulk jct sat current density, JS** parameter value.

The block provides the following relationship between the bulk junction potential  $PB$  and the transistor temperature  $T$ :

$$PB(T) = \frac{PB + \frac{kT_{meas}}{q} \left( \log \left( \frac{T_{meas}}{300.15} \right)^3 + \frac{q}{k} \left( \frac{1.115}{300.15} - \frac{EG_{T_{meas}}}{T} \right) \right)}{T_{meas}/T} \\ - \frac{kT}{q} \left( \log \left( \frac{T}{300.15} \right)^3 + \frac{q}{k} \left( \frac{1.115}{300.15} - \frac{EG_T}{T} \right) \right)$$

where:

- $PB$  is the **Bulk junction potential, PB** parameter value.

The block provides the following relationship between the bulk-drain junction capacitance  $CBD$  and the transistor temperature  $T$ :

$$CBD(T) = CBD \frac{pbo + MJ * (4 * 10^4 * (T - 300.15) * pbo - (PB(T) - pbo))}{pbo + MJ * (4 * 10^4 * (T_{meas} - 300.15) * pbo - (PB - pbo))}$$

where:

- $CBD$  is the **Zero-bias BD capacitance, CBD** parameter value.
- $MJ$  is the **Bottom grading coefficient, MJ** parameter value.
- 

$$pbo = \frac{PB + \frac{kT_{meas}}{q} \left( \log \left( \frac{T_{meas}}{300.15} \right)^3 + \frac{q}{k} \left( \frac{1.115}{300.15} - \frac{EG_{T_{meas}}}{T} \right) \right)}{T_{meas}/300.15}$$

The block uses the  $CBD(T)$  equation to calculate:

- The bulk-source junction capacitance by substituting  $CBS$  (the **Zero-bias BS capacitance, CBS** parameter value) for  $CBD$ .

- The bottom junction capacitance by substituting  $CJ$  (the **Bottom junction cap per area, CJ** parameter value) for  $CBD$ .

The block provides the following relationship between the sidewall junction capacitance  $CJSW$  and the transistor temperature  $T$ :

$$CJSW(T) = CJ \frac{pbo + MJSW * (4 * 10^4 * (T - 300.15) * pbo - (PB(T) - pbo))}{pbo + MJSW * (4 * 10^4 * (T_{meas} - 300.15) * pbo - (PB - pbo))}$$

where:

- $MJSW$  is the **Side grading coefficient, MJSW** parameter value.

## Basic Assumptions and Limitations

The model is based on the following assumptions:

- The PMOS block does not support noise analysis.
- The PMOS block applies initial conditions across junction capacitors and not across the block ports.

## Parameters

### Model Selection Tab

#### MOS model

Select one of the following MOSFET model options:

- Level 1 MOS — Use the “Level 1 Drain Current Model” on page 1-1634. This is the default option.
- Level 3 MOS — Use the “Level 3 Drain Current Model” on page 1-1637.



## Dimensions Tab

### Device area factor, AREA

The transistor area. This value multiplies the following parameter values:

- **Transconductance, KP**
- **Bulk saturation current, IS**
- **Bulk jct sat current density, JS**
- **Zero-bias BD capacitance, CBD**
- **Zero-bias BS capacitance, CBS**
- **G-S overlap capacitance, CGSO**
- **G-D overlap capacitance, CGDO**
- **G-B overlap capacitance, CGBO**
- **Bottom junction cap per area CJ**
- **Side jct cap/area of jct perimeter CJSW**

It divides the following parameter values:

- **Drain resistance, RD**
- **Source resistance, RS**
- **Sheet resistance, RSH**

The default value is 1. The value must be greater than 0.

### Number of parallel devices, SCALE

The number of parallel MOS instances for this device. This parameter multiplies the output current and device charge. The default value is 1. The value must be greater than 0.

### Length of channel, LENGTH

Length of the channel between the source and drain. The default value is  $1e-04$  m.

### Width of channel, WIDTH

Width of the channel between the source and drain. The default value is  $1e-04$  m.

### Area of drain, AD

Area of the transistor drain diffusion. The default value is  $0$  m<sup>2</sup>. The value must be greater than or equal to 0.

## Area of source, AS

Area of the transistor source diffusion. The default value is  $0 \text{ m}^2$ . The value must be greater than or equal to 0.

## Perimeter of drain, PD

Perimeter of the transistor drain diffusion. The default value is 0 m.

## Perimeter of source, PS

Perimeter of the transistor source diffusion. The default value is 0 m.

## Resistors Tab

### Drain resistance, RD

The transistor drain ohmic resistance. The default value is  $0.01 \Omega$ . If you set this parameter to NaN  $\Omega$ , this value means the parameter is unspecified, so the block calculates the drain resistance as described in “Resistance Calculations” on page 1-1631. The value must be equal to 0 or greater than or equal to  $R_{min}$ .  $R_{min}$  is a built-in model constant whose value is  $1\text{e-}12$ .

### Source resistance, RS

The transistor source ohmic resistance. The default value is  $1\text{e-}4 \Omega$ . If you set this parameter to NaN  $\Omega$ , this value means the parameter is unspecified, so the block calculates the drain resistance as described in “Resistance Calculations” on page 1-1631. The value must be equal to 0 or greater than or equal to  $R_{min}$ .  $R_{min}$  is a built-in model constant whose value is  $1\text{e-}12$ .

### Sheet resistance, RSH

Resistance per square of the transistor source and drain. The default value is NaN  $\Omega$ . This value means the parameter is unspecified. The block only uses this parameter value if you do not specify one or both of the **Drain resistance, RD** and **Source resistance, RS** parameter values, as described in “Resistance Calculations” on page 1-1631. The value must be greater than or equal to 0.

### Number of drain squares, NRD

Number of squares of resistance that make up the transistor drain diffusion. The default value is 1. The value must be greater than or equal to 0. The block only uses this parameter value if you do not specify one or both of the **Drain resistance, RD** and **Source resistance, RS** parameter values, as described in “Resistance Calculations” on page 1-1631.

### Number of source squares, NRS

Number of squares of resistance that make up the transistor source diffusion. The default value is 1. The value must be greater than or equal to 0. The block only uses this parameter value if you do not specify one or both of the **Drain resistance, RD** and **Source resistance, RS** parameter values, as described in “Resistance Calculations” on page 1-1631.

## DC Currents Tab

### Threshold voltage, VTO

The gate-source voltage above which the transistor produces a nonzero drain current. The default value is 0 V. If you assign this parameter a value of NaN, the block calculates the value from the specified values of the **Oxide thickness, TOX** and **Substrate doping, NSUB** parameters. For more information about this calculation, see “Temperature Dependence” on page 1-1586.

### Transconductance, KP

The derivative of drain current with respect to gate voltage. The default value is  $2e-05 \text{ A/V}^2$ . The value must be greater than or equal to 0. If you assign this parameter a value of NaN, the block calculates the value from the specified values of the **Oxide thickness, TOX** and **Substrate doping, NSUB** parameters. For more information about this calculation, see “Level 1 Drain Current Model” on page 1-1574 or “Level 3 Drain Current Model” on page 1-1637 as appropriate for the selected value of the **MOS model** parameter.

### Bulk threshold, GAMMA

Body effect parameter, which relates the threshold voltage, VTH, to the body bias, VBS, as described in “Level 1 Drain Current Model” on page 1-1574. The default

value is  $0 \sqrt{V}$ . The value must be greater than or equal to 0. If you assign this parameter a value of NaN, the block calculates the value from the specified values of the **Oxide thickness, TOX** and **Substrate doping, NSUB** parameters. For more information about this calculation, see “Level 1 Drain Current Model” on page 1-1574 or “Level 3 Drain Current Model” on page 1-1637 as appropriate for the selected value of the **MOS model** parameter.

### Surface potential, PHI

Twice the voltage at which the surface electron concentration becomes equal to the intrinsic concentration and the device transitions between depletion and inversion conditions. The default value is 0.6 V. The value must be greater than or equal to 0. If you assign this parameter a value of NaN, the block calculates the value from the

specified values of the **Oxide thickness, TOX** and **Substrate doping, NSUB** parameters. For more information about this calculation, see “Level 1 Drain Current Model” on page 1-1574 or “Level 3 Drain Current Model” on page 1-1637 as appropriate for the selected value of the **MOS model** parameter.

**Channel modulation, LAMBDA**

The channel-length modulation. This parameter is only visible when you select Level 1 MOS for the **MOS model** parameter. The default value is  $\theta$  1/V.

**Bulk saturation current, IS**

The magnitude of the current that the junction approaches asymptotically for very large reverse bias levels. The default value is  $1e-14$  A. The value must be greater than or equal to  $\theta$ .

**Bulk jct sat current density, JS**

The magnitude of the current per unit area that the junction approaches asymptotically for very large reverse bias levels. The default value is  $\theta$  A/m<sup>2</sup>. The value must be greater than or equal to  $\theta$ .

**Emission coefficient, N**

The transistor emission coefficient or ideality factor. The default value is 1. The value must be greater than  $\theta$ .

**Width effect on threshold, DELTA**

The factor that controls the effect of transistor width on threshold voltage. This parameter is only visible when you select Level 3 MOS for the **MOS model** parameter. The default value is  $\theta$ .

**Max carrier drift velocity, VMAX**

The maximum drift velocity of the carriers. This parameter is only visible when you select Level 3 MOS for the **MOS model** parameter. The default value is  $\theta$  m/s.

**Fast surface state density, NFS**

The fast surface state density adjusts the drain current for the mobility reduction caused by the gate voltage. This parameter is only visible when you select Level 3 MOS for the **MOS model** parameter. The default value is  $\theta$  1/cm<sup>2</sup>.

**Vds dependence threshold volt, ETA**

The coefficient that controls how the threshold voltage depends on the drain-source voltage in the drain current calculation. This parameter is only visible when you select Level 3 MOS for the **MOS model** parameter. The default value is  $\theta$ .

### **V<sub>gs</sub> dependence on mobility, THETA**

The coefficient that controls how the mobility affects the gate voltage in the drain current calculation. This parameter is only visible when you select **Level 3 MOS** for the **MOS model** parameter. The default value is  $\theta / V$ .

### **Mobility modulation, KAPPA**

The coefficient that controls how the mobility affects the channel length in the drain current calculation. This parameter is only visible when you select **Level 3 MOS** for the **MOS model** parameter. The default value is  $\theta . 2$ .

## **C-V Tab**

### **Model junction capacitance**

Select one of the following options for modeling the junction capacitance:

- **No** — Do not include junction capacitance in the model. This is the default option.
- **Yes** — Specify zero-bias junction capacitance, junction potential, grading coefficient, forward-bias depletion and capacitance coefficient.

### **G-S overlap capacitance, CGSO**

Gate-source capacitance due to the diffusion that occurs when the device operates in depletion mode. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is  $\theta$  F/m. The value must be equal to  $\theta$  or greater than or equal to *Cmin*. *Cmin* is a built-in model constant whose value is  $1e-18$ .

### **G-D overlap capacitance, CGDO**

Gate-drain capacitance due to the diffusion that occurs when the device operates in depletion mode. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is  $\theta$  F/m. The value must be equal to  $\theta$  or greater than or equal to *Cmin*. *Cmin* is a built-in model constant whose value is  $1e-18$ .

### **G-B overlap capacitance, CGBO**

Gate-base capacitance due to the diffusion that occurs when the device operates in depletion mode. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is  $\theta$  F/m. The value must be equal to  $\theta$  or greater than or equal to *Cmin*. *Cmin* is a built-in model constant whose value is  $1e-18$ .

### **Zero-bias BD capacitance, CBD**

The value of the capacitance placed between the base and the drain. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is 0 F. The value must be equal to 0 or greater than or equal to *Cmin*. *Cmin* is a built-in model constant whose value is 1e-18.

### **Zero-bias BS capacitance, CBS**

The value of the capacitance placed between the base and the source. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is 0 F. The value must be equal to 0 or greater than or equal to *Cmin*. *Cmin* is a built-in model constant whose value is 1e-18.

### **Bottom junction cap per area CJ**

Zero-bias bulk junction bottom capacitance per junction area. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is 0 F/m<sup>2</sup>. The value must be equal to 0 or greater than or equal to *Cmin*. *Cmin* is a built-in model constant whose value is 1e-18.

### **Bottom grading coefficient, MJ**

The transistor bottom grading coefficient. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is 0.5. The value must be equal to 0 or less than *MGmax*. *MGmax* is a built-in model constant whose value is 0.9.

### **Side jct cap/area of jct perimeter CJSW**

Zero-bias bulk junction sidewall capacitance per junction perimeter. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is 0 F/m. The value must be equal to 0 or greater than or equal to *Cmin*. *Cmin* is a built-in model constant whose value is 1e-18.

### **Side grading coefficient, MJSW**

The transistor sidewall grading coefficient. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is 0.5. The value must be equal to 0 or less than *MGmax*. *MGmax* is a built-in model constant whose value is 0.9.

### **Bulk junction potential, PB**

The potential across the bulk junction. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is 0.8 V. The value must be equal to 0 or greater than or equal to *VJmin*. *VJmin* is a built-in model constant whose value is 0.01.

### Capacitance coefficient FC

The fitting coefficient that quantifies the decrease of the depletion capacitance with applied voltage. This parameter is only visible when you select **Yes** for the **Model junction capacitance** parameter. The default value is 0.5. The value must be equal to 0 or less than or equal to  $FCmax$ .  $FCmax$  is a built-in model constant whose value is 0.95.

### Specify initial condition

Select one of the following options for specifying an initial condition:

- No — Do not specify an initial condition for the model. This is the default option.
- Yes — Specify the initial diode voltage.

---

**Note** The PMOS block applies the initial diode voltage across the junction capacitors and not across the ports.

---

### Initial condition voltage ICVDS

Drain-source voltage at the start of the simulation. This parameter is only visible when you select **Yes** for the **Model junction capacitance** and **Yes** for the **Specify initial condition** parameter. The default value is 0 V.

### Initial condition voltage ICVGS

Gate-source voltage at the start of the simulation. This parameter is only visible when you select **Yes** for the **Model junction capacitance** and **Yes** for the **Specify initial condition** parameter. The default value is 0 V.

### Initial condition voltage ICVBS

Bulk-source voltage at the start of the simulation. This parameter is only visible when you select **Yes** for the **Model junction capacitance** and **Yes** for the **Specify initial condition** parameter. The default value is 0 V.

## Process Tab

### Oxide thickness, TOX

Thickness of the gate oxide. The default value is NaN m. The value must be greater than or equal to 0.

---

**Note** When you select Level 3 MOS for the **MOS model** parameter, the block uses a value of  $1e-7$  rather than NaN by default.

---

### **Lateral diffusion, LD**

Length of lateral diffusion. The default value is 0 m.

### **Substrate doping, NSUB**

Substrate doping. The default value is NaN 1/cm<sup>3</sup>. The value must be greater than or equal to 1.45e10 (the carrier concentration of intrinsic silicon).

### **Surface state density, NSS**

Substrate doping. The default value is 0 1/cm<sup>2</sup>.

### **Surface mobility, U0**

Zero-bias surface mobility coefficient. The default value is 600 cm<sup>2</sup>/V/s.

### **Junction depth, XJ**

Junction depth. This parameter is only visible when you select Level 3 MOS for the **MOS model** parameter. The default value is 0 m.

### **Gate type?, TPG**

Select one of the following MOSFET gate materials (as compared to the substrate):

- **Opposite of substrate** — The gate material is the opposite of the substrate. This means that TPG = 1 in the device equations. This is the default option.
- **Same as substrate** — The gate material is the same as the substrate. This means that TPG = -1 in the device equations.
- **Aluminum** — The gate material is aluminum. This means that TPG = 0 in the device equations.

## **Temperature Tab**

### **Model temperature dependence using**

Select one of the following options for modeling the diode temperature dependence:

- **Device temperature** — Use the device temperature, which is the **Circuit temperature** value plus the **Offset local circuit temperature, TOFFSET** value. The **Circuit temperature** value comes from the SPICE Environment Parameters block, if one exists in the circuit. Otherwise, it comes from the default value for this block.
- **Fixed temperature** — Use a temperature that is independent of the circuit temperature to model temperature dependence.



**Offset local circuit temperature, TOFFSET**

The amount by which the transistor temperature differs from the circuit temperature. This parameter is only visible when you select `Device temperature` for the **Model temperature dependence using** parameter. The default value is 0 K.

**Fixed circuit temperature, TFIXED**

The temperature at which to simulate the transistor. This parameter is only visible when you select `Fixed temperature` for the **Model temperature dependence using** parameter. The default value is 300.15 K. The value must be greater than 0.

**Parameter extraction temperature, TMEAS**

The temperature at which the transistor parameters were measured. The default value is 300.15 K. The value must be greater than 0.

## Ports

The block has the following ports:

G

Electrical conserving port associated with the transistor gate terminal.

D

Electrical conserving port associated with the transistor drain terminal.

S

Electrical conserving port associated with the transistor source terminal.

B

Electrical conserving port associated with the transistor bulk terminal.

## References

- [1] G. Massobrio and P. Antognetti. *Semiconductor Device Modeling with SPICE*. 2nd Edition, McGraw-Hill, 1993. Chapter 3.

## **See Also**

SPICE NMOS

# SPICE PNP

Model Gummel-Poon PNP Transistor



## Library

SPICE-Compatible Components/Semiconductor Devices

## Description

The PNP block represents a SPICE-compatible four-terminal Gummel-Poon PNP transistor. The substrate port is connected to the transistor body using a capacitor, so these devices are equivalent to a three-terminal transistor when you connect the substrate port to any other port and use the default value of zero for the **C-S junction capacitance, CJS** parameter.

The PNP block model includes the following components:

- “Current-Voltage and Base Charge Model” on page 1-1661
- “Base Resistance Model” on page 1-1664
- “Transit Charge Modulation Model” on page 1-1665
- “Junction Charge Model” on page 1-1666
- “Temperature Dependence” on page 1-1668

## Current-Voltage and Base Charge Model

The current-voltage relationships and base charge relationships for the transistor are calculated after adjusting the applicable model parameters for temperature as described in the following sections:

- “Emitter-Base and Collector-Base Junction Currents” on page 1-1662

- “Terminal Currents” on page 1-1663
- “Base Charge Model” on page 1-1664

The base-emitter junction current is calculated using the following equations:

- When  $V_{EB} > 80 * V_{TF}$  :

$$I_{ebf} = IS * \left( \left( \frac{V_{EB}}{V_{TF}} - 79 \right) * e^{80} - 1 \right) + G_{\min} * V_{EB}$$
$$I_{ebe} = ISE * \left( (V_{EB} - 80 * V_{TF} + V_{TE}) * \frac{e^{(80 * V_{TF} / V_{TE})}}{V_{TE}} - 1 \right)$$

- When  $V_{EB} \leq 80 * V_{TF}$

$$I_{ebf} = IS * \left( e^{(V_{EB} / V_{TF})} - 1 \right) + G_{\min} * V_{EB}$$
$$I_{ebe} = ISE * \left( e^{(V_{EB} / V_{TE})} - 1 \right)$$

The base-collector junction current is calculated using the following equations:

- When  $V_{CB} > 80 * V_{TR}$  :

$$I_{cbr} = IS * \left( \left( \frac{V_{CB}}{V_{TR}} - 79 \right) * e^{80} - 1 \right) + G_{\min} * V_{CB}$$
$$I_{cbc} = ISC * \left( (V_{CB} - 80 * V_{TR} + V_{TC}) * \frac{e^{(80 * V_{TR} / V_{TC})}}{V_{TC}} - 1 \right)$$

- When  $V_{CB} \leq 80 * V_{TR}$

$$I_{cbr} = IS * \left( e^{(V_{CB}/V_{TR})} - 1 \right) + G_{min} * V_{CB}$$

$$I_{cbc} = ISC * \left( e^{(V_{CB}/V_{TC})} - 1 \right)$$

In the preceding equations:

- $V_{EB}$  is the emitter-base voltage and  $V_{CB}$  is the collector-base voltage.
- 
- $V_{TE} = NE * k * T / q$ ,  $V_{TC} = NC * k * T / q$ ,  $V_{TF} = NF * k * T / q$ , and  $V_{TR} = NR * k * T / q$ .
- $ISC$  and  $ISE$  are the **B-C leakage current, ISC** and **B-E leakage current, ISE** parameter values, respectively.
- $NE$ ,  $NC$ ,  $NF$ , and  $NR$  are the **B-E emission coefficient, NE**, **B-C emission coefficient, NC**, **Forward emission coefficient, NF** and **Reverse emission coefficient, NR** parameter values, respectively.
- $q$  is the elementary charge on an electron.
- $k$  is the Boltzmann constant.
- $T$  is the transistor temperature:
  - If you select Device temperature for the **Model temperature dependence using** parameter,  $T$  is the sum of the **Circuit temperature** value plus the **Offset local circuit temperature, TOFFSET** parameter value. The **Circuit temperature** value comes from the SPICE Environment Parameters block, if one exists in the circuit. Otherwise, it comes from the default value for this block.
  - If you select Fixed temperature for the **Model temperature dependence using** parameter,  $T$  is the **Fixed circuit temperature, TFIXED** parameter value.
- $G_{min}$  is the minimum conductance. By default,  $G_{min}$  matches the **Minimum conductance GMIN** parameter of the SPICE Environment Parameters block, whose default value is 1e-12. To change  $G_{min}$ , add a SPICE Environment Parameters block to your model and set the **Minimum conductance GMIN** parameter to the desired value.

The terminal currents,  $I_B$  and  $I_C$  are the base and collector currents, defined as positive into the device. They are calculated as:

$$I_B = - \left( \frac{I_{ebf}}{BF} + I_{ebe} + \frac{I_{cbr}}{BR} + I_{cbc} \right)$$

$$I_C = - \left( \frac{I_{ebf} - I_{cbr}}{q_b} - \frac{I_{cbr}}{BR} - I_{cbc} \right)$$

where *BF* and *BR* are the **Forward beta, BF** and **Reverse beta, BR** parameter values, respectively.

The base charge, *q<sub>b</sub>*, is calculated using the following equations:

$$q_b = \frac{q_1}{2} \left( 1 + \sqrt{0.5 * \left( \sqrt{(1 + 4 * q_2 - eps)^2 + eps^2} + 1 + 4 * q_2 - eps \right) + eps} \right)$$

$$q_1 = \left( 1 - \frac{V_{CB}}{VAF} - \frac{V_{EB}}{VAR} \right)^{-1}$$

$$q_2 = \frac{I_{ebf}}{IKF} + \frac{I_{cbr}}{IKR}$$

where

- *VAF* and *VAR* are the **Forward Early voltage, VAF** and **Reverse Early voltage, VAR** parameters, respectively.
- *IKF* and *IKR* are the **Forward knee current, IKF** and **Reverse knee current, IKR** parameter values, respectively.
- *eps* is 1e-4.

## Base Resistance Model

The block models base resistance in one of two ways:

- If you use the default value of infinity for the **Half base resistance cur, IRB** parameter, the PNP block calculates the base resistance *r<sub>bb</sub>* as

$$r_{bb} = RBM + \frac{RB - RBM}{q_b}$$

where:

- $RBM$  is the **Minimum base resistance**,  $RBM$  parameter value.
- $RB$  is the **Zero-bias base resistance**,  $RB$  parameter value.
- If you specify a finite value for the **Half base resistance cur**,  $IRB$  parameter, the PNP block calculates the base resistance  $r_{bb}$  as

$$r_{bb} = RBM + 3 * (RB - RBM) * \left( \frac{\tan z - z}{z * \tan^2 z} \right)$$

where:

$$z = \frac{\sqrt{1 + 144I_B / (\pi^2 IRB)} - 1}{(24 / \pi^2) \sqrt{(I_B / IRB)}}$$

## Transit Charge Modulation Model

If you specify nonzero values for the **Coefficient of TF**,  $XTF$  parameter, the block models transit charge modulation by scaling the **Forward transit time**,  $TF$  parameter value as follows:

$$TF_{\text{mod}} = \frac{TF * \left[ 1 + XTF * e^{V_{CB} / (1.44V_{TF})} \left( \frac{I_{EB}}{I_{EB} + ITF} \right)^2 \right]}{q_b}$$

where  $ITF$  is the **Coefficient of TF**,  $ITF$  parameter value.

## Junction Charge Model

The PNP block lets you model junction charge. The collector-base charge  $Q_{cb}$  and the emitter-base charge  $Q_{eb}$  depend on an intermediate value,  $Q_{dep}$  as follows, after adjusting the applicable model parameters for temperature:

- For the internal base-emitter junctions:

$$Q_{eb} = TF_{mod} * I_{eb} + Q_{dep}$$

- For the internal base-collector junctions:

$$Q_{cb} = TR * I_{cb} + XCJC * Q_{dep}$$

- For the external base-collector junctions:

$$Q_{cb_{ext}} = (1 - XCJC) * Q_{dep}$$

$Q_{dep}$  depends on the junction voltage,  $V_{jct}$  ( $V_{EB}$  for the emitter-base junction and  $V_{CB}$  for the collector-base junction) as follows.

Applicable Range of $V_{jct}$ Values	Corresponding $Q_{dep}$ Equation
$V_{jct} < FC * VJ$	$Q_{dep} = C_{jct} * VJ * \frac{1 - (1 - V_{jct} / VJ)^{(1-MJ)}}{1 - MJ}$
$V_{jct} \geq FC * VJ$	$Q_{dep} = C_{jct} * \left[ F1 + \frac{F3 * (V_{jct} - FC * VJ) + \frac{MJ * [V_{jct}^2 - (FC * VJ)^2]}{2 * VJ}}{F2} \right]$

Where:

- $FC$  is the **Capacitance coefficient FC** parameter value.



- $VJ$  is:
  - The **B-E built-in potential**,  $VJE$  parameter value for the emitter-base junction.
  - The **B-C built-in potential**,  $VJC$  parameter value for the collector-base junction.
- $MJ$  is:
  - The **B-E exponential factor**,  $MJE$  parameter value for the emitter-base junction.
  - The **B-C exponential factor**,  $MJC$  parameter value for the collector-base junction.
- $C_{jct}$  is:
  - The **B-E depletion capacitance**,  $CJE$  parameter value for the emitter-base junction.
  - The **B-C depletion capacitance**,  $CJC$  parameter value for the collector-base junction.

$$F1 = VJ * \left( 1 - (1 - FC)^{(1 - MJ)} \right) / (1 - MJ)$$

$$F2 = (1 - FC)^{(1 + MJ)}$$

$$F3 = 1 - FC * (1 + MJ)$$

The collector-substrate charge  $Q_{sc}$  depends on the collector-substrate voltage  $V_{sc}$  as follows, after adjusting the applicable model parameters for temperature.

Applicable Range of $V_{sc}$ Values	Corresponding $Q_{sc}$ Equation
$V_{sc} < 0$	$Q_{sc} = CJS * VJS * \left( \frac{1 - (1 - V_{sc} / VJS)^{(1 - MJS)}}{1 - MJS} \right)$
$V_{sc} \geq 0$	$Q_{sc} = CJS * (1 + MJS * V_{sc} / (2 * VJS)) * V_{sc}$

where:

- $CJS$  is the **C-S junction capacitance**,  $CJS$  parameter value.

- $V_{JS}$  is the **Substrate built-in potential**, **VJS** parameter value.
- $M_{JS}$  is the **Substrate exponential factor**, **MJS** parameter value.

## Temperature Dependence

Several transistor parameters depend on temperature. There are two ways to specify the transistor temperature:

- When you select `Device` temperature for the **Model temperature dependence using** parameter, the transistor temperature is

$$T = T_C + T_O$$

where:

- $T_C$  is the **Circuit temperature** parameter value from the SPICE Environment Parameters block. If this block doesn't exist in the circuit,  $T_C$  is the default value of this parameter.
- $T_O$  is the **Offset local circuit temperature**, **TOFFSET** parameter value.
- When you select `Fixed` temperature for the **Model temperature dependence using** parameter, the transistor temperature is the **Fixed circuit temperature**, **TFIXED** parameter value.

The block provides the following relationship between the saturation current  $IS$  and the transistor temperature  $T$ :

$$IS(T) = IS * (T/T_{meas})^{XTI} * e^{\left(\frac{T}{T_{meas}} - 1\right) \frac{EG}{V_t}}$$

where:

- $IS$  is the **Transport saturation current**, **IS** parameter value.
- $T_{meas}$  is the **Parameter extraction temperature**, **TMEAS** parameter value.
- $XTI$  is the **Temperature exponent for IS**, **XTI** parameter value.
- $EG$  is the **Energy gap**, **EG** parameter value.
- $V_t = kT/q$ .

The block provides the following relationship between the base-emitter junction potential  $VJE$  and the transistor temperature  $T$ :

$$VJE(T) = VJE * \left( \frac{T}{T_{meas}} \right) - \frac{3 * k * T}{q} * \log \left( \frac{T}{T_{meas}} \right) - \left( \frac{T}{T_{meas}} \right) * EG_{T_{meas}} + EG_T$$

where:

- $VJE$  is the **B-E built-in potential**, **VJE** parameter value.

$$EG_{T_{meas}} = 1.16eV - (7.02e - 4 * T_{meas}^2) / (T_{meas} + 1108)$$

$$EG_T = 1.16eV - (7.02e - 4 * T^2) / (T + 1108)$$

The block uses the  $VJE(T)$  equation to calculate the base-collector junction potential by substituting  $VJC$  (the **B-C built-in potential**, **VJC** parameter value) for  $VJE$ .

The block provides the following relationship between the base-emitter junction capacitance  $CJE$  and the transistor temperature  $T$ :

$$CJE(T) = CJE * \left[ 1 + MJE * \left( 400e - 6 * (T - T_{meas}) - \frac{VJE(T) - VJE}{VJE} \right) \right]$$

where:

- $CJE$  is the **B-E depletion capacitance**, **CJE** parameter value.
- $MJE$  is the **B-E exponential factor**, **MJE** parameter value.

The block uses this equation to calculate the base-collector junction capacitance by substituting  $CJC$  (the **B-C depletion capacitance**, **CJC** parameter value) for  $CJE$  and  $MJC$  (the **B-C exponential factor**, **MJC** parameter value) for  $MJE$ .

The block provides the following relationship between the forward and reverse beta and the transistor temperature  $T$ :

$$\beta(T) = \beta * \left( \frac{T}{T_{meas}} \right)^{XTB}$$

where:

- $\beta$  is the **Forward beta, BF** or **Reverse beta, BR** parameter value.
- $XTB$  is the **Beta temperature exponent, XTB** parameter value.

The block provides the following relationship between the base-emitter leakage current  $ISE$  and the transistor temperature  $T$ :

$$ISE(T) = ISE * \left( \frac{T}{T_{meas}} \right)^{-XTB} * \left( \frac{IS(T)}{IS} \right)^{1/NE}$$

where:

- $ISE$  is the **B-E leakage current, ISE** parameter value.
- $NE$  is the **B-E emission coefficient, NE** parameter value.

The block uses this equation to calculate the base-collector leakage current by substituting  $ISC$  (the **B-C leakage current, ISC** parameter value) for  $ISE$  and  $NC$  (the **B-C emission coefficient, NC** parameter value) for  $NE$ .

## Basic Assumptions and Limitations

The model is based on the following assumptions:

- The PNP block does not support noise analysis.
- The PNP block applies initial conditions across junction capacitors and not across the block ports.

## Parameters

### Main Tab

#### Device area, AREA

The transistor area. This value multiplies the following parameter values:

- **Transport saturation current, IS**
- **Forward knee current, IKF**
- **B-E leakage current, ISE**
- **Reverse knee current, IKR**
- **B-C leakage current, ISC**
- **Half base resistance cur, IRB**
- **B-E depletion capacitance, CJE**
- **Coefficient of TF, ITF**
- **B-C depletion capacitance, CJC**
- **C-S junction capacitance, CJS**

It divides the following parameter values:

- **Zero-bias base resistance, RB**
- **Minimum base resistance, RBM**
- **Emitter resistance, RE**
- **Collector resistance, RC**

The default value is 1 m<sup>2</sup>. The value must be greater than 0.

### **Number of parallel devices, SCALE**

The number of parallel transistors the block represents. This value multiplies the output current and device charges. The default value is 1. The value must be greater than 0.

## **Forward Gain Tab**

### **Transport saturation current, IS**

The magnitude of the current at which the transistor saturates. The default value is 1e-16 A/m<sup>2</sup>. The value must be greater than or equal to 0.

### **Forward beta, BF**

The ideal maximum reverse beta. The default value is 100. The value must be greater than 0.

### **Forward emission coefficient, NF**

The reverse emission coefficient or ideality factor. The default value is 1. The value must be greater than 0.

**B-E leakage current, ISE**

The base-emitter leakage current. The default value is  $\theta$  A/m<sup>2</sup>. The value must be greater than or equal to  $\theta$ .

**B-E emission coefficient, NE**

The base-collector emission coefficient or ideality factor. The default value is 1.5. The value must be greater than  $\theta$ .

**Forward knee current, IKF**

The current value at which forward-beta high-current roll-off occurs. The default value is  $\theta$  A/m<sup>2</sup>. The value must be greater than or equal to  $\theta$ . For this parameter, the software interprets a value of  $\theta$  as infinity.

**Forward Early voltage, VAF**

The forward Early voltage. The default value is  $\theta$  V. The value must be greater than or equal to  $\theta$ . For this parameter, the software interprets a value of  $\theta$  as infinity.

**Reverse Gain Tab**

**Reverse beta, BR**

The ideal maximum reverse beta. The default value is 1. The value must be greater than  $\theta$ .

**Reverse emission coefficient, NR**

The reverse emission coefficient or ideality factor. The default value is 1. The value must be greater than  $\theta$ .

**B-C leakage current, ISC**

The base-collector leakage current. The default value is  $\theta$  A/m<sup>2</sup>. The value must be greater than or equal to  $\theta$ .

**B-C emission coefficient, NC**

The base-collector emission coefficient or ideality factor. The default value is 2. The value must be greater than  $\theta$ .

**Reverse knee current, IKR**

The current value at which reverse-beta high-current roll-off occurs. The default value is  $\theta$  A/m<sup>2</sup>. The value must be greater than or equal to  $\theta$ . For this parameter, the software interprets a value of  $\theta$  as infinity.

**Reverse Early voltage, VAR**

The reverse Early voltage. The default value is  $\theta$  V. The value must be greater than or equal to  $\theta$ . For this parameter, the software interprets a value of  $\theta$  as infinity.

**Resistors Tab****Emitter resistance, RE**

The resistance of the emitter. The default value is  $1e-4 \text{ m}^2 \cdot \Omega$ . The value must be greater than or equal to  $\theta$ .

**Collector resistance, RC**

The resistance of the collector. The default value is  $\theta.01 \text{ m}^2 \cdot \Omega$ . The value must be greater than or equal to  $\theta$ .

**Zero-bias base resistance, RB**

The resistance of the base. The default value is  $1 \text{ m}^2 \cdot \Omega$ . The value must be greater than or equal to  $\theta$ .

**Minimum base resistance, RBM**

The minimum resistance of the base. The default value is  $\theta \text{ m}^2 \cdot \Omega$ . The value must be less than or equal to the **Zero-bias base resistance, RB** parameter value.

**Half base resistance cur, IRB**

The base current at which the base resistance has dropped to half of its zero-bias value. The default value is  $\text{Inf A/m}^2$ . The value must be greater than or equal to  $\theta$ . Use the default value of  $\text{Inf}$  if you do not want to model the change in base resistance as a function of base current.

**Capacitance Tab****Model junction capacitance**

Select one of the following options for modeling the junction capacitance:

- **No** — Do not include junction capacitance in the model. This is the default option.
- **B-E Capacitance** — Model the junction capacitance across the base-emitter junction.
- **B-C Capacitance** — Model the junction capacitance across the base-collector junction.
- **C-S Capacitance** — Model the junction capacitance across the collector-substrate junction.

---

**Note** To include junction capacitance in the model:

- 1** Select **B-E Capacitance** and specify the base-emitter junction capacitance parameters.
- 2** Select **B-C Capacitance** and specify the base-collector junction capacitance parameters.
- 3** Select **C-S Capacitance** and specify the collector-substrate junction capacitance parameters.

You can specify or change any of the common parameters when you select any of the preceding options for the **Model junction capacitance** parameter.

---

### **B-E depletion capacitance, CJE**

The depletion capacitance across the base-emitter junction. This parameter is only visible when you select **B-E Capacitance** for the **Model junction capacitance** parameter. The default value is  $0 \text{ F/m}^2$ . The value must be greater than or equal to 0.

### **B-E built-in potential, VJE**

The base-emitter junction potential. This parameter is only visible when you select **B-E Capacitance** for the **Model junction capacitance** parameter. The default value is 0.75 V. The value must be greater than or equal to 0.01 V.

### **B-E exponential factor, MJE**

The grading coefficient for the base-emitter junction. This parameter is only visible when you select **B-E Capacitance** for the **Model junction capacitance** parameter. The default value is 0.33. The value must be greater than or equal to 0 and less than or equal to 0.9.

### **Forward transit time, TF**

The transit time of the minority carriers that cause diffusion capacitance when the base-emitter junction is forward-biased. This parameter is only visible when you select **B-E Capacitance** for the **Model junction capacitance** parameter. The default value is 0. The value must be greater than or equal to 0.

### **Coefficient of TF, XTF**

The coefficient for the base-emitter and base-collector bias dependence of the transit time, which produces a charge across the base-emitter junction. This parameter is only visible when you select **B-E Capacitance** for the **Model junction capacitance** parameter. The default value is 0. The value must be greater than or equal to 0. Use the default value of 0 if you do not want to model the effect of base-emitter bias on transit time.



**VBC dependence of TF, VTF**

The coefficient for the base-emitter bias dependence of the transit time. This parameter is only visible when you select B-E Capacitance for the **Model junction capacitance** parameter. The default value is 0 V. The value must be greater than or equal to 0. For this parameter, the software interprets a value of 0 as infinity.

**Coefficient of TF, ITF**

The coefficient for the dependence of the transit time on collector current. This parameter is only visible when you select B-E Capacitance for the **Model junction capacitance** parameter. The default value is 0 A/m<sup>2</sup>. The value must be greater than or equal to 0. Use the default value of 0 if you do not want to model the effect of collector current on transit time.

**B-C depletion capacitance, CJC**

The depletion capacitance across the base-collector junction. This parameter is only visible when you select B-C Capacitance for the **Model junction capacitance** parameter. The default value is 0 F/m<sup>2</sup>. The value must be greater than 0.

**B-C built-in potential, VJC**

The base-collector junction potential. This parameter is only visible when you select B-C Capacitance for the **Model junction capacitance** parameter. The default value is 0.75 V. The value must be greater than or equal to 0.01 V.

**B-C exponential factor, MJC**

The grading coefficient for the base-collector junction. This parameter is only visible when you select B-C Capacitance for the **Model junction capacitance** parameter. The default value is 0.33. The value must be greater than or equal to 0 and less than or equal to 0.9.

**B-C capacitance fraction, XCJC**

The fraction of the base-collector depletion capacitance that is connected between the internal base and the internal collector. The rest of the base-collector depletion capacitance is connected between the external base and the internal collector. This parameter is only visible when you select B-C Capacitance for the **Model junction capacitance** parameter. The default value is 0. The value must be greater than or equal to 0 and less than or equal to 1.

**Reverse transit time, TR**

The transit time of the minority carriers that cause diffusion capacitance when the base-collector junction is reverse-biased. This parameter is only visible when you select B-C Capacitance for the **Model junction capacitance** parameter. The default value is 0 s. The value must be greater than or equal to 0.

### Capacitance coefficient FC

The fitting coefficient that quantifies the decrease of the depletion capacitance with applied voltage. This parameter is only visible when you select B-E Capacitance or B-C Capacitance for the **Model junction capacitance** parameter. The default value is 0.5. The value must be greater than or equal to 0 and less than or equal to 0.95.

### Specify initial condition

Select one of the following options for specifying an initial condition:

- No — Do not specify an initial condition for the model. This is the default option.
- Yes — Specify the initial transistor conditions.

---

**Note** The PNP block applies the initial transistor voltages across the junction capacitors and not across the ports.

---

This parameter is only visible when you select B-E Capacitance or B-C Capacitance for the **Model junction capacitance** parameter.

### Initial condition voltage ICVBE

Base-emitter voltage at the start of the simulation. This parameter is only visible when you select B-E Capacitance or B-C Capacitance for the **Model junction capacitance** and Yes for the **Specify initial condition** parameter. The default value is 0 V.

### Initial condition voltage ICVCE

Base-collector voltage at the start of the simulation. This parameter is only visible when you select B-E Capacitance or B-C Capacitance for the **Model junction capacitance** and Yes for the **Specify initial condition** parameter. The default value is 0 V.

### C-S junction capacitance, CJS

The collector-substrate junction capacitance. This parameter is only visible when you select C-S Capacitance for the **Model junction capacitance** parameter. The default value is 0 F/m<sup>2</sup>. The value must be greater than or equal to 0.

### Substrate built-in potential, VJS

The potential of the substrate. This parameter is only visible when you select C-S Capacitance for the **Model junction capacitance** parameter. The default value is 0.75 V.

### Substrate exponential factor, MJS

The grading coefficient for the collector-substrate junction. This parameter is only visible when you select C-S Capacitance for the **Model junction capacitance** parameter. The default value is 0. The value must be greater than or equal to 0 and less than or equal to 0.9.

## Temperature Tab

### Model temperature dependence using

Select one of the following options for modeling the transistor temperature dependence:

- **Device temperature** — Use the device temperature, which is the **Circuit temperature** value plus the **Offset local circuit temperature, TOFFSET** value. The **Circuit temperature** value comes from the SPICE Environment Parameters block, if one exists in the circuit. Otherwise, it comes from the default value for this block.
- **Fixed temperature** — Use a temperature that is independent of the circuit temperature to model temperature dependence.

### Beta temperature exponent, XTB

The forward and reverse beta temperature exponent that models base current temperature dependence. This parameter is only visible when you select **Device temperature** for the **Model temperature dependence using** parameter. The default value is 0. The value must be greater than or equal to 0.

### Energy gap, EG

The energy gap that affects the increase in the saturation current as temperature increases. This parameter is only visible when you select **Device temperature** for the **Model temperature dependence using** parameter. The default value is 1.11 eV. The value must be greater than or equal to 0.1.

### Temperature exponent for IS, XTI

The order of the exponential increase in the saturation current as temperature increases. This parameter is only visible when you select **Device temperature** for the **Model temperature dependence using** parameter. The default value is 3. The value must be greater than or equal to 0.

### **Offset local circuit temperature, TOFFSET**

The amount by which the transistor temperature differs from the circuit temperature. This parameter is only visible when you select `Device temperature` for the **Model temperature dependence using** parameter. The default value is 0 K.

### **Parameter extraction temperature, TMEAS**

The temperature at which the transistor parameters were measured. The default value is 300.15 K. The value must be greater than 0.

### **Fixed circuit temperature, TFIXED**

The temperature at which to simulate the transistor. This parameter is only visible when you select `Fixed temperature` for the **Model temperature dependence using** parameter. The default value is 300.15 K. The value must be greater than 0.

## **Ports**

The block has the following ports:

**B**

Electrical conserving port associated with the transistor base terminal.

**C**

Electrical conserving port associated with the transistor collector terminal.

**E**

Electrical conserving port associated with the transistor emitter terminal.

**S**

Electrical conserving port associated with the transistor substrate terminal.

## **References**

- [1] G. Massobrio and P. Antognetti. *Semiconductor Device Modeling with SPICE*. 2nd Edition, McGraw-Hill, 1993. Chapter 2.

## **See Also**

Generic Linear Actuator

## SPICE Resistor

Model SPICE-compatible resistor



## Library

SPICE-Compatible Components/Passive Devices

## Description

The SPICE Resistor block represents a SPICE-compatible resistor. You can specify the resistance in one of the following ways:

- As a resistance value
- As process information that the block uses to calculate a resistance value

The block models temperature dependence. There are two ways to specify the resistor temperature:

- When you select `Device` temperature for the **Model temperature dependence using** parameter, the resistor temperature is

$$T = T_C + T_O$$

where:

- $T_C$  is the **Circuit temperature** parameter value from the SPICE Environment Parameters block. If this block doesn't exist in the circuit,  $T_C$  is the default value of this parameter.
- $T_O$  is the **Offset local circuit temperature, TOFFSET** parameter value.
- When you select `Fixed` temperature for the **Model temperature dependence using** parameter, the resistor temperature is the **Fixed circuit temperature, TFIXED** parameter value.

The block adjusts the specified or calculated resistance value for temperature using the following equation:

$$R = R_0(1 + TC1(T - T_{nom}) + TC2(T - T_{nom})^2)$$

Where

- $R_0$  is the specified or calculated resistance value.
- $TC1$  is the **First order temperature coefficient, TC1** parameter value.
- $TC2$  is the **Second order temperature coefficient, TC2** parameter value.
- $T_{nom}$  is the **Parameter extraction temperature, TMEAS** parameter value.

## Parameters

### Resistance Tab

#### Device scale factor, SCALE

The number of parallel resistors that the block represents. This value multiplies the output current. The default value is 1.

#### Resistor parameterization

Select one of the following options for specifying the resistor value:

- **Use specified resistance** — Provide the resistance value directly. This option is the default.
- **Calculate from process information** — Provide process parameters that the block uses to calculate the resistance value.

When you select this option, the block calculates the resistance using the following equation:

$$R = RSH * \frac{(LENGTH - NARROW)}{(WIDTH - NARROW)}$$

where:

- $RSH$  is the **Sheet resistance, RSH** parameter value.

- *LENGTH* is the **Resistor length, LENGTH** parameter value.
- *WIDTH* is the **Resistor width, WIDTH** parameter value.
- *NARROW* is the **Etch narrowing, NARROW** parameter value.

### **Resistance, R**

Resistance value. This parameter is only visible when you select `Use specified resistance` for the **Resistor parameterization** parameter. The default value is  $0 \Omega$ .

### **Sheet resistance, RSH**

Resistance per square of the resistor. This parameter is only visible when you select `Calculate from process information` for the **Resistor parameterization** parameter. The default value is  $0 \Omega$ .

### **Resistor length, LENGTH**

Length dimension of the resistor. This parameter is only visible when you select `Calculate from process information` for the **Resistor parameterization** parameter. The default value is  $1e-06$  m.

### **Resistor width, WIDTH**

Width dimension of the resistor. This parameter is only visible when you select `Calculate from process information` for the **Resistor parameterization** parameter. The default value is  $1e-06$  m.

### **Etch narrowing, NARROW**

Amount by which the resistor length and width are reduced due to side etching. This parameter is only visible when you select `Calculate from process information` for the **Resistor parameterization** parameter. The default value is  $0$  m.

## **Temperature Tab**

### **First order temperature coefficient, TC1**

Coefficient for the linear term in the equation that the block uses to adjust the specified or calculated resistance value for temperature. The default value is  $0$  1/K.

### **Second order temperature coefficient, TC2**

Coefficient for the quadratic term in the equation the block uses to adjust the specified or calculated resistance value for temperature. The default value is  $0$  1/K<sup>2</sup>.

### **Model temperature dependence using**

Select one of the following options for modeling the resistor temperature dependence:



- **Device temperature** — Use the device temperature, which is the **Circuit temperature** parameter value (from the SPICE Environment Parameters block, if one exists in the circuit, or the default value for this block otherwise) plus the **Offset local circuit temperature, TOFFSET** parameter value.
- **Fixed temperature** — Use a temperature that is independent of the circuit temperature to model temperature dependence.

### **Offset local circuit temperature, TOFFSET**

The amount by which the resistor temperature differs from the circuit temperature. This parameter is only visible when you select **Device temperature** for the **Model temperature dependence using** parameter. The default value is 0 K.

### **Parameter extraction temperature, TMEAS**

The temperature at which the resistor parameters were measured. The default value is 300.15 K. The value must be greater than 0.

### **Fixed circuit temperature, TFIXED**

The temperature at which to simulate the resistor. This parameter is only visible when you select **Fixed temperature** for the **Model temperature dependence using** parameter. The default value is 300.15 K. The value must be greater than 0.

## **Ports**

The block has the following ports:

+

Positive electrical voltage.

-

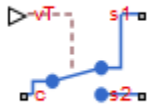
Negative electrical voltage.

## **See Also**

Diode

## SPDT Switch

Model single-pole double-throw switch



## Library

Passive Devices/Switches

## Description

The SPDT Switch block models a single-pole double-throw switch:

- When the switch is closed, port c is connected to port s2.
- When the switch is open, port c is connected to port s1.

Closed connections are modeled by a resistor with value equal to the **Closed resistance** parameter value. Open connections are modeled by a resistor with value equal to the reciprocal of the **Open conductance** parameter value.

If the **Threshold width** parameter is set to zero, the switch is closed if the voltage presented at the vT control port exceeds the value of the **Threshold** parameter.

If the **Threshold width** parameter is greater than zero, then switch conductance  $G$  varies smoothly between off-state and on-state values:

$$G = \frac{x}{R_{closed}} + (1 - x)G_{open}$$

$$\lambda = \frac{vT - \text{Threshold}}{\text{Threshold width}}$$

$$x = \begin{cases} 0 & \text{for } \lambda \leq 0 \\ 3\lambda^2 - 2\lambda^3 & \text{for } 0 < \lambda < 1 \\ 1 & \text{for } \lambda \geq 1 \end{cases}$$

The block uses the function  $3\lambda^2 - 2\lambda^3$  because its derivative is zero for  $\lambda = 0$  and  $\lambda = 1$ .

Defining a small positive **Threshold width** can help solver convergence in some models, particularly if the control port signal  $vT$  varies continuously as a function of other network variables. However, defining a nonzero threshold width precludes the solver making use of switched linear optimizations. Therefore, if the rest of your network is switched linear, MathWorks recommends that you set **Threshold width** to zero.

Optionally, you can add a delay between the point at which the voltage at  $vT$  passes the threshold and the switch opening or closing. To enable the delay, on the **Dynamics** tab, set the **Model dynamics** parameter to `Model turn-on and turn-off times`.

## Parameters

- “Main Tab” on page 1-1685
- “Dynamics Tab” on page 1-1686

### Main Tab

#### Closed resistance

Resistance between the c and s electrical ports when the switch is closed. The value must be greater than zero. The default value is  $0.01 \Omega$ .

#### Open conductance

Conductance between the c and s electrical ports when the switch is open. The value must be greater than zero. The default value is  $1e-6 \text{ S}$ .

#### Threshold

The threshold voltage for the control physical signal input  $vT$  above which the switch will turn on. The default value is  $0.5 \text{ V}$ .

#### Threshold width

The minimum increase in the control signal  $vT$  above the threshold value that will move the switch from fully open to fully closed. The default value is  $0 \text{ V}$ .

## Dynamics Tab

### Model dynamics

Select whether the block models a switching delay:

- **No dynamics** — Do not model the delay. This is the default option.
- **Model turn-on and turn-off times** — Use additional parameters to model a delay between the point at which the voltage at  $v_T$  passes the threshold and the switch opening or closing.

### Turn-on delay

Time between the input voltage exceeding the threshold voltage and the switch closing. This parameter is only visible when you select **Model turn-on and turn-off times** for the **Model dynamics** parameter. The value must be greater than zero. The default value is  $1e-3$  seconds.

### Turn-off delay

Time between the input voltage falling below the threshold voltage and the switch opening. This parameter is only visible when you select **Model turn-on and turn-off times** for the **Model dynamics** parameter. The value must be greater than zero. The default value is  $1e-3$  seconds.

### Initial input value, $v_T$

The value of the physical signal input  $v_T$  at time zero. This value is used to initialize the delayed control voltage parameter internally. This parameter is only visible when you select **Model turn-on and turn-off times** for the **Model dynamics** parameter. The default value is  $0$  V.

## Ports

This block has the following ports:

$v_T$

Physical signal that opens and closes the switch

$c, s1, s2$

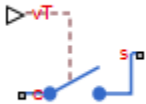
Electrical conserving ports

## **See Also**

[DPDT Switch](#) | [DPST Switch](#) | [SPST Switch](#) | [Switch](#)

## SPST Switch

Model single-pole single-throw switch



## Library

Passive Devices/Switches

## Description

The SPST Switch block models a single-pole single-throw switch:

- When the switch is closed, port **c** is connected to port **s** through a resistance with value equal to the **Closed resistance** parameter value.
- When the switch is open, port **c** is connected to port **s** through a resistance with value equal to the reciprocal of the **Open conductance** parameter value.

If the **Threshold width** parameter is set to zero, the switch is closed if the voltage presented at the **vT** control port exceeds the value of the **Threshold** parameter.

If the **Threshold width** parameter is greater than zero, then switch conductance  $G$  varies smoothly between off-state and on-state values:

$$G = \frac{x}{R_{closed}} + (1 - x)G_{open}$$

$$\lambda = \frac{vT - \text{Threshold}}{\text{Threshold width}}$$

$$x = \begin{cases} 0 & \text{for } \lambda \leq 0 \\ 3\lambda^2 - 2\lambda^3 & \text{for } 0 < \lambda < 1 \\ 1 & \text{for } \lambda \geq 1 \end{cases}$$

The block uses the function  $3\lambda^2 - 2\lambda^3$  because its derivative is zero for  $\lambda = 0$  and  $\lambda = 1$ .

Defining a small positive **Threshold width** can help solver convergence in some models, particularly if the control port signal  $vT$  varies continuously as a function of other network variables. However, defining a nonzero threshold width precludes the solver making use of switched linear optimizations. Therefore, if the rest of your network is switched linear, MathWorks recommends that you set **Threshold width** to zero.

Optionally, you can add a delay between the point at which the voltage at  $vT$  passes the threshold and the switch opening or closing. To enable the delay, on the **Dynamics** tab, set the **Model dynamics** parameter to `Model turn-on and turn-off times`.

## Parameters

- “Main Tab” on page 1-1689
- “Dynamics Tab” on page 1-1690

### Main Tab

#### Closed resistance

Resistance between the c and s electrical ports when the switch is closed. The value must be greater than zero. The default value is  $0.01 \Omega$ .

#### Open conductance

Conductance between the c and s electrical ports when the switch is open. The value must be greater than zero. The default value is  $1e-6 \text{ S}$ .

#### Threshold

The threshold voltage for the control physical signal input  $vT$  above which the switch will turn on. The default value is  $0.5 \text{ V}$ .

#### Threshold width

The minimum increase in the control signal  $vT$  above the threshold value that will move the switch from fully open to fully closed. The default value is  $0 \text{ V}$ .

## Dynamics Tab

### Model dynamics

Select whether the block models a switching delay:

- **No dynamics** — Do not model the delay. This is the default option.
- **Model turn-on and turn-off times** — Use additional parameters to model a delay between the point at which the voltage at  $v_T$  passes the threshold and the switch opening or closing.

### Turn-on delay

Time between the input voltage exceeding the threshold voltage and the switch closing. This parameter is only visible when you select **Model turn-on and turn-off times** for the **Model dynamics** parameter. The value must be greater than zero. The default value is  $1e-3$  seconds.

### Turn-off delay

Time between the input voltage falling below the threshold voltage and the switch opening. This parameter is only visible when you select **Model turn-on and turn-off times** for the **Model dynamics** parameter. The value must be greater than zero. The default value is  $1e-3$  seconds.

### Initial input value, $v_T$

The value of the physical signal input  $v_T$  at time zero. This value is used to initialize the delayed control voltage parameter internally. This parameter is only visible when you select **Model turn-on and turn-off times** for the **Model dynamics** parameter. The default value is  $0$  V.

## Ports

This block has the following ports:

$v_T$

Physical signal that opens and closes the switch

c, s

Electrical conserving ports



## **See Also**

[DPDT Switch](#) | [DPST Switch](#) | [SPDT Switch](#) | [Switch](#)

## SRM Commutation Logic

Commutation logic for switched reluctance machines

**Library:** Simscape / Electrical / Power Systems / Control / SRM Control



### Description

The commutation logic provides a logic signal for the Switched Reluctance Machine (SRM) block. The signal indicates when to switch the supply for each phase on and off.

The commutation signal for each phase is:

- 1 if  $\theta_{on} \leq \theta_{ph} \leq \theta_{off}$ .
- 0 if  $\theta_{ph} < \theta_{on}$ .
- 0 if  $\theta_{ph} > \theta_{off}$ .

where:

- $\theta_{ph}$  is the phase angle in the interval  $[0, \beta]$ .
- $\beta$  is the torque capability angle.
- $\theta_{on}$  is the switch-on angle.
- $\theta_{off}$  is the switch-off angle.

### Ports

#### Input

**theta** — Phase angle

scalar

Phase angle,  $\theta_{ph}$ , in the interval  $[0, \beta]$ .

Data Types: single | double

### **thetaON — Switch-on angle**

scalar

Lower threshold for turning on the switch.

Data Types: single | double

### **thetaOFF — Switch-off angle**

scalar

Upper threshold for turning off the switch.

Data Types: single | double

## **Output**

### **abcOnOff — Commutation signal**

vector

Commutation signal for each phase: 1 for the switch-on condition and 0 for switch-off condition

Data Types: single | double

## **Parameters**

### **Torque capability angle (rad) — Torque angle**

$\pi/2$  (default)

The torque production capability of one rotor pole, in radians.

### **Sample time (-1 for inherited) — Block sample time**

-1 (default) | positive scalar

Time, in s, between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

If this block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly using a positive scalar.

## **See Also**

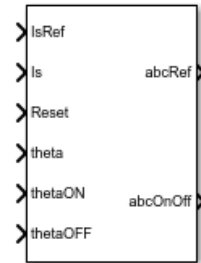
SRM Current Controller | SRM Current Controller with PWM Generation | SRM Hysteresis Current Controller | Switched Reluctance Machine

**Introduced in R2018a**

# SRM Current Controller

Current control for switched reluctance machines

**Library:** Simscape / Electrical / Power Systems / Control / SRM Control



## Description

The SRM Current Controller block performs discrete-time proportional-integral (PI) current control for the Switched Reluctance Machine (SRM) block.

## Equations

To determine the duty cycle, the block implements discrete-time proportional-integral (PI) current control in accordance with this equation.

$$D = \left( K_p + K_i \frac{T_s z}{z-1} \right) (I_{s\_ref} - I_s)$$

Where:

- $D$  is the duty cycle.
- $K_p$  is the proportional gain.
- $K_i$  is the integral gain.
- $T_s$  is the sample time.
- $I_{s\_ref}$  is the reference current.
- $I_s$  is the measured current.

To obtain control signals for the three-phases, the block then multiplies the duty cycle with the commutation signals. The resulting three control signals are normalized over the interval  $[0, 1]$ .

## Ports

### Input

#### **IsRef — Reference current**

scalar

Reference current for control.

Data Types: `single` | `double`

#### **Is — Measured current**

scalar

Measured current.

Data Types: `single` | `double`

#### **Reset — Integrator reset**

scalar

External reset signal (rising edge) for the integrator.

Data Types: `Boolean`

#### **theta — Theta-rotor angle**

scalar

Rotor angle in the interval  $[0, \beta]$ .

Data Types: `single` | `double`

#### **theta0N — Switch-on angle**

scalar

Angle for switching on the phase supply.

Data Types: `single` | `double`

**thetaOFF — Switch-off angle**

scalar

Angle for switching on the phase supply.

Data Types: `single` | `double`**Output****abcRef — Reference voltage**

vector

Control signal normalized in the interval [0, 1].

Data Types: `single` | `double`**abcOnOff — Switch signal**

vector

Switch signal for the *a*-, *b*-, and *c*-phases. 1 for the switch-on condition and 0 for switch-off conditionData Types: `single` | `double`**Parameters****Proportional gain — Controller proportional gain,  $K_p$** 

1 (default) | positive scalar

Proportional gain,  $K_p$ , of the controller.**Integral gain — Integral gain,  $K_i$** 

5 (default) | positive scalar

Integral gain,  $K_i$ , of the controller.**Anti-windup gain — Anti-windup gain,  $K_{aw}$** 

1 (default) | positive scalar

Anti-windup gain,  $K_{aw}$ , of the controller.

## **Sample time (-1 for inherited) — Block sample time**

-1 (default) | positive scalar

Time, in s, between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

If this block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly using a positive scalar.

## **References**

[1] Saha, N. and S. Panda. "Speed control with torque ripple reduction of switched reluctance motor by Hybrid Many Optimizing Liaison Gravitational Search technique." *Engineering Science and Technology*. Vol 20 (2017): 909-921.

## **See Also**

SRM Commutation Logic | SRM Current Controller with PWM Generation | SRM Hysteresis Current Controller | Switched Reluctance Machine

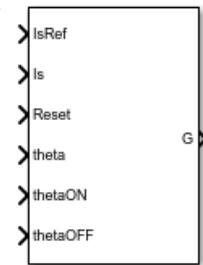
**Introduced in R2018a**



# SRM Current Controller with PWM Generation

Current controller with internal pulse width modulation for switched reluctance machines

**Library:** Simscape / Electrical / Power Systems / Control / SRM Control

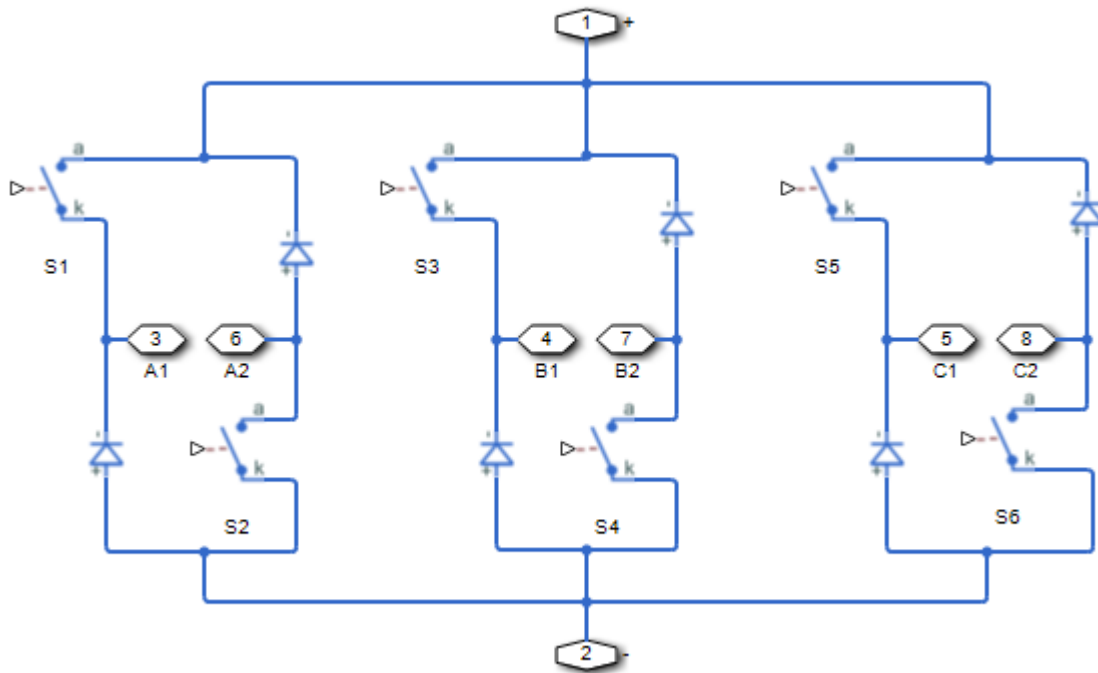


## Description

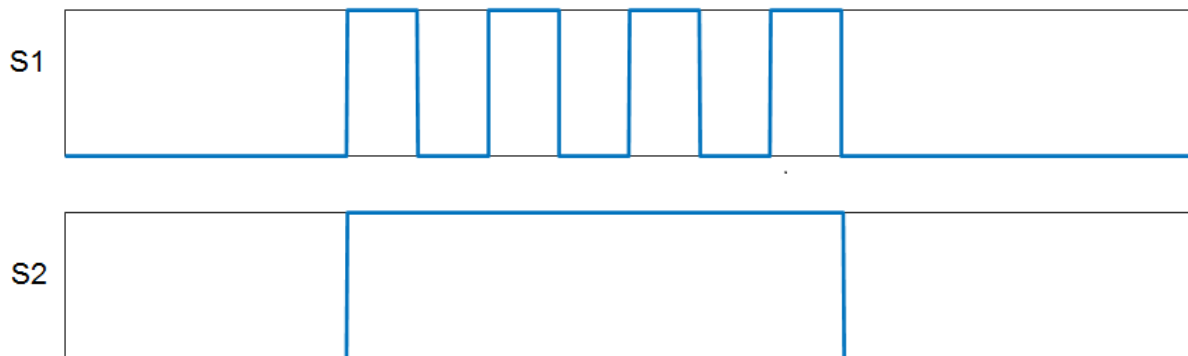
The SRM Current Controller block performs discrete-time proportional-integral (PI) current control for the Switched Reluctance Machine (SRM) block. The block includes pulse width modulation (PWM).

## PWM Generation Model

The figure shows the converter structure for an SRM.



As the figure shows, the PWM generation signal is for high side switching devices.



When the control signal is greater than the carrier counter value, the PWM generator outputs 1. Otherwise, it outputs 0.

## Equations

To determine the duty cycle, the block implements PI current control in the rotor reference frame in accordance with this equation.

$$D = \left( K_p + K_i \frac{T_s z}{z - 1} \right) (I_{s\_ref} - I_s)$$

Where:

- $D$  is the duty cycle.
- $K_p$  is the proportional gain.
- $K_i$  is the integral gain.
- $T_s$  is the sample time.
- $I_{s\_ref}$  is the reference current.
- $I_s$  is the measured current.

To obtain control signals for the three-phases, the block then multiplies the duty cycle with the commutation signals. The resulting three control signals are normalized over the interval  $[0, 1]$ .

## Ports

### Input

#### **IsRef — Reference current**

scalar

Reference current for control.

Data Types: `single` | `double`

#### **Is — Measured current**

scalar

Actual current.

Data Types: `single` | `double`

**Reset — Integrator reset**

scalar

External reset signal (rising edge) for the integrator.

Data Types: `Boolean`

**theta — Theta-rotor angle**

scalar

Rotor angle in the interval  $[0, \beta]$ .

Data Types: `single` | `double`

**thetaON — Switch-on angle**

scalar

Angle for switching on the phase supply.

Data Types: `single` | `double`

**thetaOFF — Switch-off angle**

scalar

Angle for switching on the phase supply.

Data Types: `single` | `double`

**Output**

**G — Gate control**

vector

Pulse waveforms that determine switching behavior.

Data Types: `single` | `double`

## Parameters

### Control Parameters

#### Proportional gain — Controller proportional gain, $K_p$

1 (default) | positive scalar

Proportional gain,  $K_p$ , of the controller.

#### Integral gain — Integral gain, $K_i$

5 (default) | positive scalar

Integral gain,  $K_i$ , of the controller.

#### Anti-windup gain — Anti-windup gain, $K_{aw}$

1 (default) | positive scalar

Anti-windup gain,  $K_{aw}$ , of the controller.

#### Sample time (-1 for inherited) — Block sample time

-1 (default) | positive scalar

Time, in s, between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

If this block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly using a positive scalar.

### PWM Generator

#### Carrier counter — Carrier counter model

Up (default) | Down | Up-Down

Use the carrier counter strategy to change the initial behavior of the PWM output:

- Up counter — PWM output begins at the start of the on state.
- Down counter — PWM output begins at the start of the off state.

- Up-down counter — PWM output begins in the middle of the on state.

### **Timer period (s) — PWM timer period**

0.001 (default) | positive scalar

Pulse width modulation timer period,  $T_{per}$ , in seconds.

### **Fundamental sample time (s) — Sample time for PWM generation**

0.0001 (default) | positive scalar

Time, in s, between consecutive PWM generator executions. During execution, the block produces PWM output and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

To ensure adequate resolution in the generated PWM signal, set the fundamental sample time so that  $0 < T_{s\_pwm} \leq 10T_{per}$ , where:

- $T_{s\_pwm}$  is the **Fundamental sample time (s)**.
- $T_{per}$  is the **Timer period (s)**.

## **References**

- [1] Saha, N. and S. Panda. "Speed control with torque ripple reduction of switched reluctance motor by Hybrid Many Optimizing Liaison Gravitational Search technique." *Engineering Science and Technology*. Vol 20 (2017): 909-921.

## **See Also**

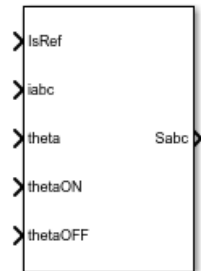
SRM Commutation Logic | SRM Current Controller | SRM Hysteresis Current Controller | Switched Reluctance Machine

**Introduced in R2018a**

# SRM Hysteresis Current Controller

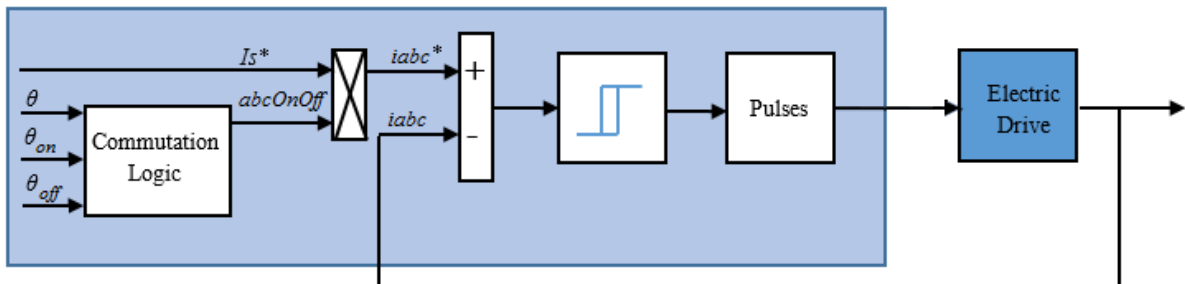
Hysteresis current control for switched reluctance machines

**Library:** Simscape / Electrical / Power Systems / Control / SRM Control



## Description

The SRM Hysteresis Current Controller block implements hysteresis current control for the Switched Reluctance Machine (SRM) block.



## Ports

### Input

**$I_s^{\text{Ref}}$  — Reference current**  
scalar

Reference current for control.

Data Types: single | double

**iabc — Measured current**

vector

Measured three-phase current.

Data Types: single | double

**Reset — Integrator reset**

scalar

External reset signal (rising edge) for the integrator.

Data Types: Boolean

**theta — Theta-rotor angle**

scalar

Rotor angle in the interval  $[0, \beta]$ .

Data Types: single | double

**theta0N — Switch-on angle**

scalar

Angle for switching on the phase supply.

Data Types: single | double

**theta0FF — Switch-off angle**

scalar

Angle for switching on the phase supply.

Data Types: single | double

**Output**

**Sabc — Voltage switch signal**

vector | 0 or 1

Signal for switching on and off the three-phase voltage.



Data Types: single | double

## Parameters

### **Torque capability angle (rad) – Torque capability angle**

$\pi/2$  (default) |  $[0, 2*\pi/\text{number of rotor poles}]$

The torque production capability of one rotor pole, in radians.

### **Hysteresis band – Hysteresis bandwidth**

5 (default)

Hysteresis band,  $h$ , for the current controller. The switch-on point is  $h/2$  and the switch-off point is  $-h/2$ .

### **Sample time (-1 for inherited) – Block sample time**

-1 (default) | positive scalar

Time, in s, between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

If this block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly using a positive scalar.

## See Also

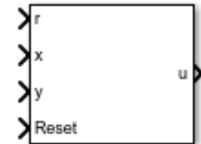
SRM Commutation Logic | SRM Current Controller | SRM Current Controller with PWM Generation | Switched Reluctance Machine

**Introduced in R2018a**

# State-Feedback Controller

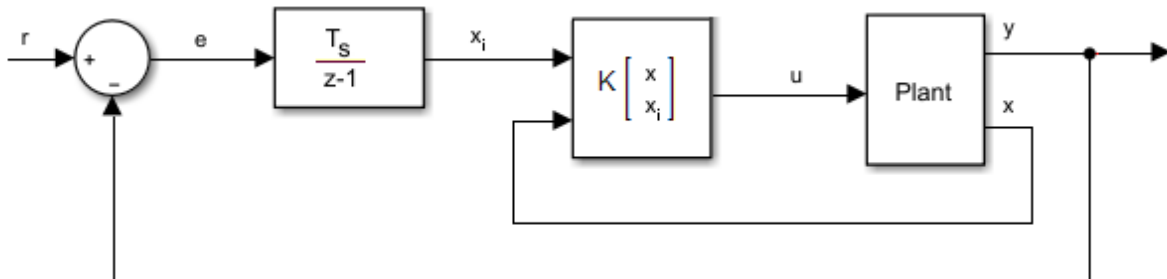
Discrete-time state-feedback controller with integral action

**Library:**           Simscape / Electrical / Power Systems / Control /  
                          General Control



## Description

The State-Feedback Controller block implements a discrete-time state-feedback controller with integral action. Use this block to control linear systems with single or multiple inputs and single or multiple outputs. The integral action serves to eliminate steady-state error in the controlled outputs. You can define the controller using a precomputed optimal gain or use the state-space model of your system to generate this gain using pole placement.



## Equations

The integral of the tracking error,  $x_i$ , is an additional state that ensures zero steady-state error for the closed-loop system. The extended state vector is

$$x_e = \begin{bmatrix} x \\ x_i \end{bmatrix},$$

Where:

- $x$  is the state vector.
- $x_i$  is the integral of the tracking error.
- $x_e$  is the extended state vector.

Therefore, the control action is

$$u = Kx_e,$$

Where:

- $K$  is the feedback matrix, that is, the pole placement.
- $u$  is the controller output.

## Assumptions

System state measurement and estimation occur outside the controller.

## Ports

### Input

#### **r — Plant reference**

scalar

Plant system reference signal.

Data Types: `single` | `double`

#### **x — State vector**

vector

Measured or estimated system state vector.

Data Types: `single` | `double`

**Reset — Integrator reset**

scalar

External reset signal (rising edge) for the integrator.

Data Types: `Boolean`

**y — Plant output**

scalar

Plant system output signal.

Data Types: `single` | `double`

**Output**

**u — Controller output**

scalar

Control system output signal.

Data Types: `single` | `double`

**Parameters**

**State-feedback design — Controller generation**

`State-feedback gain` (default) | `Desired eigenvalues`

Select the strategy for parameterizing controller gain:

- `State-feedback gain` — Specify the controller gain directly
- `Desired eigenvalues` — Specify the plant model and desired eigenvalues from which to generate the controller gain

**State-feedback parameterization — State-feedback parameterization**

`Discrete-time` (default) | `Continuous-time`

Select the strategy for parameterizing the state-space matrices and desired poles for the controller. The block implementation is discrete regardless of this parameterization.

**Dependencies**

To enable this parameter, set **State-feedback design** to Desired eigenvalues.

**Controller matrix — Controller matrix**

[1 1] (default) | matrix

Controller feedback matrix. To determine the controller matrix, if you have a license for Control System Toolbox, use the `lqr` or `lqi` function.

**Dependencies**

To enable this parameter, set **State-feedback design** to State-feedback gain.

**Discrete-time A matrix — A matrix in discrete time**

1 (default) | real scalar or matrix

State matrix of the discrete-time state-space model. The A matrix must be square, with the number of rows and columns equal to the order of the system.

**Dependencies**

To enable this parameter, set **State-feedback parameterization** to Discrete-time.

**Discrete-time B matrix — B matrix in discrete time**

1 (default) | real scalar or matrix

Input matrix of the discrete-time state-space model. The B matrix must have the number of rows equal to the order of the system, and the number of columns equal to the number of system inputs.

**Dependencies**

To enable this parameter, set **State-feedback parameterization** to Discrete-time.

**Discrete-time C matrix — C matrix in discrete time**

1 (default) | real scalar or matrix

Output matrix of the discrete-time state-space model. The C matrix must have the number of rows equal the number of outputs of the system, and the number of columns equal to the order of the system.

**Dependencies**

To enable this parameter, set **State-feedback parameterization** to Discrete-time.

## **Discrete-time D matrix — D matrix in discrete time**

1 (default) | real scalar or matrix

Feedthrough matrix of the discrete-time state-space model. The D matrix must have the number of rows equal to the number of system outputs, and the number of columns equal to the number of system inputs.

### **Dependencies**

To enable this parameter, set **State-feedback parameterization** to Discrete-time.

## **Continuous-time A matrix — A matrix in continuous time**

1 (default) | real scalar or matrix

State matrix of the continuous-time state-space model. The A matrix must be square, with the number of rows and columns equal to the order of the system.

### **Dependencies**

To enable this parameter, set **State-feedback parameterization** to Continuous-time.

## **Continuous-time B matrix — B matrix in continuous time**

1 (default) | real scalar or matrix

Input matrix of the continuous-time state-space model. The B matrix must have the number of rows equal to the order of the system, and the number of columns equal to the number of system inputs.

### **Dependencies**

To enable this parameter, set **State-feedback parameterization** to Continuous-time.

## **Continuous-time C matrix — C matrix in continuous time**

1 (default) | real scalar or matrix

Output matrix of the continuous-time state-space model. The C matrix must have the number of rows equal the number of outputs of the system, and the number of columns equal to the order of the system.

### **Dependencies**

To enable this parameter, set **State-feedback parameterization** to Continuous-time.

## **Continuous-time D matrix — D matrix in continuous time**

1 (default) | real scalar or matrix

Feedthrough matrix of the continuous-time state-space model. The D matrix must have the number of rows equal to the number of system outputs, and the number of columns equal to the number of system inputs.

#### Dependencies

To enable this parameter, set **State-feedback parameterization** to Continuous-time.

#### Discretization sample time — Discretization sample time

0.1 (default) | positive real number

Value used to discretize the state space matrices and also approximate the discrete-time eigenvalues.

#### Dependencies

To enable this parameter, set **State-feedback parameterization** to Continuous-time and **Sample time (-1 for inherited)** to -1.

#### Desired eigenvalues (discrete) — Observer eigenvalues

0 (default) | real vector

Specify the location of the eigenvalues to lie within the unit circle. The controller gain is then calculated based on these eigenvalues. The size of the vector must be equal to the system order plus the number of outputs.

#### Control action upper limit — $u_{max}$

5 (default) | scalar greater than the value of the **Control action lower limit** parameter

Upper limit for the control output signal.

#### Control action lower limit — $u_{min}$

0 (default) | scalar

Lower limit for the control output signal.

#### Sample time (-1 for inherited) — Sampling interval

-1 (default) | default value or a positive number

Time interval between samples. If the block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

## **See Also**

### **Blocks**

RST Controller | Smith Predictor Controller

**Introduced in R2017b**



# Stepper Motor

Model stepper motor

**Library:** Simscape / Electrical / Electronics and Mechatronics /  
Actuators & Drivers / Rotational Actuators  
Simscape / Electrical / Power Systems / Machines /  
Stepper Motor and Driver



## Description

The Stepper Motor block represents a stepper motor. It uses the input pulse trains, A and B, to control the mechanical output according to the following equations:

$$e_A = -K_m \omega \sin(N_r \theta)$$

$$e_B = K_m \omega \cos(N_r \theta)$$

$$\frac{di_A}{dt} = (v_A - Ri_A - e_A) / L$$

$$\frac{di_B}{dt} = (v_B - Ri_B - e_B) / L$$

$$J \frac{d\omega}{dt} + B\omega = T_e$$

see

$$T_e = -K_m \left( i_A - \frac{e_A}{R_m} \right) \sin(N_r \theta) + K_m \left( i_B - \frac{e_B}{R_m} \right) \cos(N_r \theta) - T_d \sin(4N_r \theta)$$

$$\frac{d\theta}{dt} = \omega$$

where:

- $e_A$  and  $e_B$  are the back electromotive forces (emfs) induced in the A and B phase windings, respectively.
- $i_A$  and  $i_B$  are the A and B phase winding currents.
- $v_A$  and  $v_B$  are the A and B phase winding voltages.
- $K_m$  is the motor torque constant.
- $N_r$  is the number of teeth on each of the two rotor poles. The **Full step size** parameter is  $(\pi/2)/N_r$ .
- $R$  is the winding resistance.
- $L$  is the winding inductance.
- $R_m$  is the magnetizing resistance.
- $B$  is the rotational damping.
- $J$  is the inertia.
- $\omega$  is the rotor speed.
- $\theta$  is the rotor angle.
- $T_d$  is the detent torque amplitude.

If the initial rotor is zero or some multiple of  $(\pi/2)/N_r$ , the rotor is aligned with the phase winding of pulse A. This happens when there is a positive current flowing from the **A+** to the **A-** ports and there is no current flowing from the **B+** to the **B-** ports.

Use the Stepper Motor Driver block to create the pulse trains for the Stepper Motor block.

The Stepper Motor block produces a positive torque acting from the mechanical **C** to **R** ports when the phase of pulse A leads the phase of pulse B.

## Averaged Mode

If you set the **Simulation mode** parameter to **Averaged**, both for a Stepper Motor block and for the Stepper Motor Driver block that controls it, then the individual steps are not simulated. This can be a good way to speed up simulation. In Averaged mode, under

nonslipping conditions, the motor and driver are represented by a second-order linear system that tracks the specified step rate. The demanded step rate is determined directly from voltage across **A+** and **A-**. So, for example, a voltage of +10 V across the **A+** and **A-** terminals is interpreted as a step rate demand of 10 steps per second. See the Stepper Motor Driver reference page for more information on how to connect the Stepper Motor Driver to your step angle controller.

Averaged mode includes a slip estimator to predict whether the stepper motor would have slipped if running in Stepping simulation mode. Slip is predicted if the motor torque exceeds the **Vector of maximum torque values** parameter value for longer than one step period, the step period being determined from the current step rate demand. Upon detecting slip, the simulation will proceed or stop with an error, according to the **Action on slipping** parameter value. If you choose the action that lets the simulation continue, note that simulation results may be incorrect. When slipping occurs, the torque generated by the motor is not generally the maximum available torque; the maximum torque is only achieved if the stepper controller detects slip and adjusts the step rate command accordingly.

The dynamics of the equivalent second-order system are determined from the values that you specify for the **Approximate total load inertia** and **Maximum step rate command** parameters. It is important that you set as accurate values as possible for these parameters, so that the step rate command is tracked, and the block does not generate false slipping warnings or errors.

If you run the motor in Averaged mode with the optional thermal ports exposed (see “Thermal Ports and Effects” on page 1-1717), then heat is added to the thermal ports, assuming that the windings are always powered even when the step rate command is zero. The block makes adjustments for half stepping and for reduced torque (and winding currents) at higher speeds. For these adjustments to be correct, the **Vector of maximum torque** parameter values must be correct. For half stepping, at zero speed the heat generated by the block is the average of that generated when stopped at a half step and at a full step.

To validate Averaged mode model configurations where you predict slip to occur, compare results with the same simulation performed in stepping mode.

## Thermal Ports and Effects

The block has three optional thermal ports, one for each of the two windings and one for the rotor. These ports are hidden by default. To expose the thermal ports, right-click the

block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal ports on the block icon, and adds the **Temperature Dependence** and **Thermal Port** tabs to the block dialog box. These tabs are described further on this reference page.

Use the thermal ports to simulate the effects of copper resistance and iron losses that convert electrical power to heat. If you expose these ports, winding resistance is assumed linearly dependent on temperature, and is given by:

$$R = R_0 (1 + \alpha (T - T_0))$$

where:

- $R$  is the resistance at temperature  $T$ .
- $R_0$  is the resistance at the measurement (or reference) temperature  $T_0$ . Specify the reference temperature using the **Measurement temperature** parameter.
- $\alpha$  is the resistance temperature coefficient, which you specify with the **Resistance temperature coefficients, [alpha\_A alpha\_B]** parameter. A typical value for copper is 0.00393/K.

The block calculates temperature of each of the windings and the rotor by

$$M \frac{dT}{dt} = Q$$

where

- $M$  is the thermal mass. Specify this value for the windings using the **Winding thermal masses, [M\_A M\_B]** parameter, and for the rotor using the **Rotor thermal mass** parameter.
- $T$  is the temperature. Specify the initial values for the windings using the **Winding initial temperatures, [T\_A T\_B]** parameter, and for the rotor using the **Rotor initial temperature** parameter.
- $Q$  is the heat flow, which is calculated from the iron losses of the windings:

$$Q_A = i_a^2 R_A (1 - \rho_m / 100)$$

$$Q_B = i_b^2 R_B (1 - \rho_m / 100)$$

$$Q_R = Q_A (\rho_m / 100) + Q_B (\rho_m / 100)$$

where  $\rho_m$  is the percentage of magnetizing resistance associated with the rotor. Specify this percentage using the **Percentage of magnetizing resistance associated with the rotor** parameter.

## Assumptions and Limitations

The model is based on the following assumptions:

- This model neglects magnetic saturation effects and any magnetic coupling between phases.
- When you select the **Start simulation from steady state** check box in the SimscapeSolver Configuration block, this block will not initialize an **Initial rotor angle** value between  $-\pi$  and  $\pi$ .
- To use Averaged mode, the Stepper Motor block must be directly connected to a Stepper Motor Driver block also running in Averaged mode.
- The Averaged mode is an approximation, and exact step tracking compared to the Stepping mode should not be expected.
- Slip detection in Averaged mode is approximate, and depends on a good estimate for load inertia and maximum step rate. Incorrect values may result in false slip detection.
- When simulating slip in Averaged mode, it is assumed that the stepper motor controller adjusts the step rate command so as to achieve maximum possible torque.

## Ports

### Conserving

#### **A+ — A-phase positive terminal**

electrical

Electrical conserving port associated with the A-phase positive terminal.

#### **A- — A-phase negative terminal**

electrical

Electrical conserving port associated with the A-phase negative terminal.

**B+ — B-phase positive terminal**

electrical

Electrical conserving port associated with the B-phase positive terminal.

**B- — B-phase negative terminal**

electrical

Electrical conserving port associated with the B-phase negative terminal.

**C — Machine case**

mechanical rotational

Mechanical rotational conserving port connected to the motor case.

**R — Machine rotor**

mechanical rotational

Mechanical rotational conserving port connected to the rotor.

**HA — Winding A thermal mass**

thermal

Thermal conserving port associated with the thermal mass of winding A. For more information, see “Thermal Ports and Effects” on page 1-1717.

**HB — Winding B thermal mass**

thermal

Thermal conserving port associated with the thermal mass of winding B. For more information, see “Thermal Ports and Effects” on page 1-1717.

**HR — Rotor thermal mass**

thermal

Thermal conserving port associated with the thermal mass of the rotor. For more information, see “Thermal Ports and Effects” on page 1-1717.

## Parameters

### Electrical Torque

#### Simulation mode — Simulation mode

Stepping (default) | Averaged

Use **Averaged** only if the block is connected directly to a Stepper Motor Driver block also running in Averaged mode.

#### Vector of rotational speeds — Rotor speeds

[0, 1000, 3000] rpm (default) | positive vector

Vector of rotational speeds at which to define maximum torque values, for slip prediction.

#### Dependencies

To enable this parameter, set **Simulation mode** to Averaged.

#### Vector of maximum torque values — Maximum torques

[2, 2, 1] N\*m (default) | positive vector

Vector of maximum torque values, to be used for slip prediction with the **Vector of rotational speeds** parameter. These values are often given on a datasheet, and correspond to the supply voltage and stepping type (half step or full step) specified in the driver.

#### Dependencies

To enable this parameter, set **Simulation mode** to Averaged.

#### Action on slipping — Slip response

none (default) | warn | error

Select the action for the block to perform during simulation upon detecting slip:

- **none** — Continue simulation, limiting the load torque according to the **Vector of maximum torque values**.
- **warn** — Continue simulation, limiting the load torque according to the **Vector of maximum torque values**, and generate a warning that the rotor is slipping.
- **error** — Stop the simulation and generate an error message that the rotor is slipping.

If you choose an action that lets the simulation continue, simulation results might be incorrect. When slipping occurs, the motor does not always generate maximum torque. The maximum torque is only achieved if the stepper controller detects slip and adjusts the step rate command accordingly.

### **Dependencies**

To enable this parameter, set **Simulation mode** to Averaged.

### **Approximate total load inertia — Total load inertia**

1e-4 kg\*m<sup>2</sup> (default) | positive number

Approximate total load inertia, including the rotor inertia. This value is used to help predict slip due to rapid acceleration demands.

### **Dependencies**

To enable this parameter, set **Simulation mode** to Averaged.

### **Maximum step rate command — Maximum command frequency**

10 Hz (default) | positive number

Maximum commanded step rate of the system. It is used to determine a suitable bandwidth for the second order system approximation to the stepper motor and driver.

### **Dependencies**

To enable this parameter, set **Simulation mode** to Averaged.

### **Phase winding resistance — Phase resistance**

0.55 Ohm (default) | positive number

Resistance of the A and B phase windings.

### **Dependencies**

To enable this parameter, set **Simulation mode** to Stepping.

### **Phase winding inductance — Phase inductance**

1.5e-3 Ohm (default) | positive number

Inductance of the A and B phase windings.



**Dependencies**

To enable this parameter, set **Simulation mode** to Stepping.

**Motor torque constant — Torque constant**

0.19 N\*m/A (default) | positive number

Motor torque constant,  $K_m$ .

**Dependencies**

To enable this parameter, set **Simulation mode** to Stepping.

**Detent torque — Torque variation amplitude**

0 N\*m (default) | positive number

Amplitude of the sinusoidal torque variation observed when rotating the shaft of the unpowered motor.

**Dependencies**

To enable this parameter, set **Simulation mode** to Stepping.

**Magnetizing resistance — Phase magnetizing resistance**

inf (default) | strictly positive number

Total magnetizing resistance seen from each of the phase windings. The value must be greater than zero. The default value is Inf, which implies that there are no iron losses.

**Dependencies**

To enable this parameter, set **Simulation mode** to Stepping.

**Full step size — Step size**

1.8 deg (default) | positive number

Step size when changing the polarity of either the A or B phase current.

**Mechanical****Rotor inertia — Rotational inertia**

4.5e-5 kg\*m<sup>2</sup> (default) | zero or positive number

Conservative force resisting rotor acceleration.

**Rotor damping — Rotational damping**

8.0e-4 N\*m/( rad/s) (default) | zero or positive number

Dissipative force resisting rotor speed.

**Initial rotor speed — Initial speed**

0 rpm (default) | real number

Speed of the rotor at the start of the simulation.

**Initial rotor angle — Initial angle**

0 rad (default) | real number

Angle of the rotor at the start of the simulation.

## Temperature Dependence

This set of parameters appears only for blocks with exposed thermal ports. For more information, see “Thermal Ports and Effects” on page 1-1717.

**Resistance temperature coefficients, [alpha\_A alpha\_B] — Temperature coefficients**

[.00393, .00393] 1/K (default) | positive vector

Two-element row vector defining the coefficient  $\alpha$  in the equation relating resistance to temperature, as described in “Thermal Ports and Effects” on page 1-1717. The first element corresponds to winding A, and the second to winding B. The default value is for copper.

**Measurement temperature — Reference temperature**

25 degC (default) | real number

Temperature for which motor parameters are defined.

## Thermal Port

This set of parameters appears only for blocks with exposed thermal ports. For more information, see “Thermal Ports and Effects” on page 1-1717.

**Winding thermal masses, [M\_A M\_B] — Winding thermal masses**

[100, 100] J/K (default) | positive vector

Two-element row vector defining the thermal mass for the A and B windings. The thermal mass is the energy required to raise the temperature by one degree.

**Winding initial temperatures, [T\_A T\_B] – Initial winding temperatures**

[25, 25] degC (default) | real number

Two-element row vector defining the temperature of the A and B thermal ports at the start of simulation.

**Rotor thermal mass – Rotor thermal mass**

50 J/K (default) | positive number

Thermal mass of the rotor, that is, the energy required to raise the temperature of the rotor by one degree.

**Rotor initial temperature – Initial rotor temperature**

25 degC (default) | real number

Temperature of the rotor at the start of simulation.

**Percentage of magnetizing resistance associated with the rotor – Magnetizing resistance rotor percentage**

90 (default) | positive number between 0 and 100

Percentage of the magnetizing resistance associated with the magnetic path through the rotor. It determines how much of the iron loss heating is attributed to the rotor thermal port, **HR**, and winding thermal ports, **HA** and **HB**.

## References

- [1] M. Bodson, J. N. Chiasson, R. T. Novotnak and R. B. Rekowski. "High-Performance Nonlinear Feedback Control of a Permanent Magnet Stepper Motor." IEEE Transactions on Control Systems Technology, Vol. 1, No. 1, March 1993.
- [2] P. P. Acarnley. *Stepping Motors: A Guide to Modern Theory and Practice*. New York: Peregrinus, 1982.
- [3] S.E. Lyshevski. *Electromechanical Systems, Electric Machines, and Applied Mechatronics*. CRC, 1999.

## **See Also**

Stepper Motor Driver | Unipolar Stepper Motor

**Introduced in R2008a**

# Stepper Motor Driver

Driver for stepper motor

**Library:** Simscape / Electrical / Electronics and Mechatronics /  
Actuators & Drivers / Rotational Actuators  
Simscape / Electrical / Power Systems / Machines /  
Stepper Motor and Driver



## Description

The Stepper Motor Driver block represents a driver for a stepper motor. It creates the pulse trains, A and B, required to control the motor. This block initiates a step each time the voltage at the ENA port rises above the **Enable threshold voltage** parameter value.

If the voltage at the REV port is less than or equal to the **Reverse threshold voltage** parameter value, pulse A leads pulse B by 90 degrees. If the voltage at the REV port is greater than the **Reverse threshold voltage** value, pulse B leads pulse A by 90 degrees and the motor direction is reversed.

At time zero, pulse A is positive and pulse B is negative.

If you set the **Stepping mode** parameter to **Half stepping**, the Stepper Motor Driver block can produce the output waveforms required for half stepping. In this mode, there is an intermediate state between the full steps, in which just one of the A or the B half-windings is powered. As a result, the step size is half of the stepper motor's full step size. At half steps, windings that are not powered are short-circuited. This approximates the effect of a freewheeling diode connected across the windings.

## Averaged Mode

If you set the **Simulation mode** parameter to **Averaged**, both for a Stepper Motor Driver block and for the Stepper Motor block connected to it, then the individual steps are not simulated. This can be a good way to speed up simulation. The Averaged mode assumes that the external controller provides a step rate demand. This step rate demand is determined from the voltage applied between the ENA and REF ports on the Stepper Motor Driver block, by multiplying this voltage by the value of the **Step rate sensitivity**

parameter. The rotation direction is set by the REF port in the same way as for the Stepping mode.

Averaged mode needs to communicate the step rate demand and also output voltage amplitude information to the Stepper Motor block. To do this, the step rate demand is applied as an equivalent voltage across the A+ and A- ports. Similarly the output voltage amplitude information is conveyed by applying a steady-state voltage across the B+ and B- ports with value equal to the **Output voltage amplitude** parameter.

## Assumptions and Limitations

The model is based on these assumptions:

- To use Averaged mode, the Stepper Motor Driver block must be directly connected to a Stepper Motor block also running in Averaged mode.
- When changing from Stepping to Averaged mode and back, you will need to modify your upstream blocks that provide the input voltages to the Stepper Motor Driver. One way to achieve this easily is to use Simulink variant subsystems.

## Ports

### Conserving

#### **A+ — A-phase positive terminal**

electrical

Electrical conserving port associated with the A-phase positive terminal.

#### **A- — A-phase negative terminal**

electrical

Electrical conserving port associated with the A-phase negative terminal.

#### **B+ — B-phase positive terminal**

electrical

Electrical conserving port associated with the B-phase positive terminal.

**B- — B-phase negative terminal**

electrical

Electrical conserving port associated with the B-phase negative terminal.

**ENA — Triggering input step voltage**

electrical

Electrical conserving port associated with the step trigger input.

**REF — Input floating reference voltage**

electrical

Electrical conserving port associated with the floating reference voltage.

**REV — Input voltage that controls motor direction**

electrical

Electrical conserving port associated with the motor direction input.

## Parameters

**Simulation mode — Simulation mode**

Stepping (default) | Averaged

Use **Averaged** only if the block is connected directly to a Stepper Motor block also running in Averaged mode.

**Step rate sensitivity — Step rate demand sensitivity**

10 Hz/V (default) | positive number

This parameter converts the voltage presented across the ENA and REF ports into a step rate demand.

**Dependencies**

To enable this parameter, set **Simulation mode** to Averaged.

**Enable threshold voltage — Step voltage threshold**

2.5 V (default) | positive number

When the voltage at the ENA port rises above this threshold, the Stepper Motor Driver block initiates a step.

### **Dependencies**

To enable this parameter, set **Simulation mode** to Stepping.

### **Reverse threshold voltage — Reversal voltage threshold**

2.5 V (default) | positive number

When the voltage at the REV port rises above this threshold, pulse B leads pulse A by 90 degrees, and the motor direction is reversed.

### **Output voltage amplitude — Pulse output amplitude**

10 V (default) | positive number

Amplitude of the output pulse trains.

### **Stepping mode — Step size**

Full stepping (default) | Half stepping

Select Full stepping or Half stepping.

## **See Also**

Controlled PWM Voltage | Stepper Motor

**Introduced in R2008a**



# Strain Gauge

Model deformation sensor



## Library

Sensors

## Description

The Strain Gauge block represents a sensor that generates a change in resistance as a function of strain using the following equation:

$$\frac{\Delta R}{R} = K\varepsilon$$

where:

- $\Delta R/R$  is the fractional change in resistance.
- $\varepsilon$  is the strain at port E.
- $K$  is the **Gauge factor** parameter value.

## Parameters

### Gauge resistance

The unstressed gauge resistance. The default value is 100  $\Omega$ .

### Gauge factor

The ratio  $K$  of the fractional change in resistance to the fractional change in length. The default value is 2.

## **Ports**

The block has the following ports:

E

Strain input

+

Positive electrical port

-

Negative electrical port

# Supercapacitor

Represent an electrochemical double-layer capacitor

**Library:** Simscape / Electrical / Power Systems / Sources



## Description

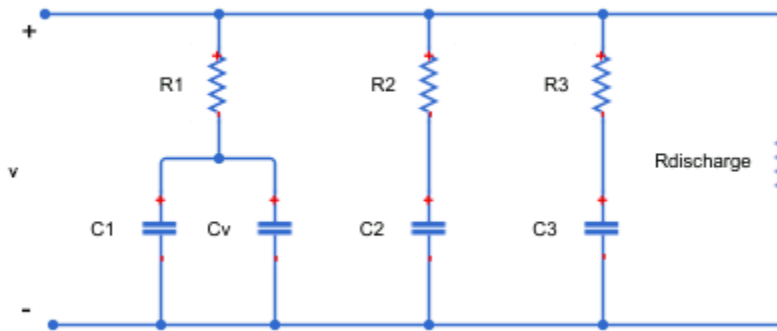
The Supercapacitor block represents an electrochemical double-layer capacitor (ELDC), which is commonly referred to as a supercapacitor or an ultracapacitor. The capacitance values for supercapacitors are orders of magnitude larger than the values for regular capacitors. Supercapacitors can provide bursts of energy because they can charge and discharge rapidly.

You can model any number of supercapacitor cells connected in series or in parallel using a single Supercapacitor block. To do so, set the relevant parameter, that is **Number of series cells** or **Number of parallel cells**, to a value larger than 1. Internally, the block simulates only the equations for a single supercapacitor cell, but it calculates:

- The output voltage according to the number of series-connected cells
- The current according to the number of parallel-connected cells

Calculating the output of a multiple-cell supercapacitor based on the output for a single cell is more efficient than simulating the equations for each cell individually.

The figure shows the equivalent circuit for a single cell in the Supercapacitor block. The circuit is a network of resistors and capacitors that is commonly used to model supercapacitor behavior.



Capacitors  $C_1$ ,  $C_2$ , and  $C_3$  have fixed capacitances. The capacitance of capacitor  $C_v$  depends on the voltage across it. Resistors  $R_1$ ,  $R_2$ , and  $R_3$  have fixed resistances. The voltage across each individual fixed capacitor in the Supercapacitor block is calculated as

$$V_{cn} = \frac{v}{N_{series}} - i_n R_n,$$

where:

- $v$  is the voltage across the block.
- $N_{series}$  is the number of cells in series.
- $n$  is the branch number.  $n = [1, 2, 3]$ .
- $i_n$  is the current through the  $n$ th branch.
- $R_n$  is the resistance in the  $n$ th branch.
- $V_{cn}$  is voltage across the capacitor in the  $n$ th branch.

The equation for the current through the first branch of the supercapacitor depends on the voltage across the capacitors in the branch. If the capacitors experience a positive voltage, that is

$$V_{c1} > 0,$$

then

$$i_1 = (C_1 + K_v V_{c1}) \frac{dV_{c1}}{dt},$$

else

$$i_1 = C_1 \frac{dV_{c1}}{dt},$$

where:

- $V_{c1}$  is voltage across the capacitors in the first branch.
- $C_1$  is the capacitance of the fixed capacitor in the first branch.
- $K_v$  is the voltage-dependent capacitance gain.
- $i_1$  is the current through the first branch.

For the remaining branches, the current is defined as

$$i_n = C_n \frac{dV_{cn}}{dt},$$

where:

- $n$  is the branch number.  $n = [2, 3]$ .
- $C_n$  is the capacitance of the  $n$ th branch.

The total current through the Supercapacitor block is

$$i = N_{parallel} \left( i_1 + i_2 + i_3 + \frac{v}{R_{discharge}} \right),$$

where:

- $N_{parallel}$  is the number of cells in parallel.
- $R_{discharge}$  is the self-discharge resistance of the supercapacitor.
- $i$  is the current through the supercapacitor.

## Ports

### Conserving

**+ — Positive electrical terminal**

electrical

Electrical conserving port associated with the positive terminal.

**- — Negative electrical terminal**

electrical

Electrical conserving port associated with the negative terminal.

## Parameters

### Cell Characteristics

**Fixed resistances, [R1 R2 R3] — Fixed resistance values for each branch**

[0.2, 90.0, 1000.0] Ohm (default)

Specify the resistances for the fixed resistors in the individual branches of the supercapacitor as an array.

**Fixed capacitances, [C1 C2 C3] — Fixed capacitance values for each branch**

[2.5, 1.5, 4.0] F (default)

Specify the individual capacitances for the fixed capacitors in the supercapacitor as an array.

**Voltage-dependent capacitor gain — Variable capacitance coefficient for the first branch**

0.95 F/V (default)

Specify the variable capacitance coefficient,  $K_v$ , for the voltage-dependent capacitor in the first branch of the supercapacitor. For information on determining the variable capacitance coefficient, see [1] on page 1-1737.

**Self-discharge resistance — Resistance to self-discharge**

inf (default)

Specify the self-discharge resistance of the supercapacitor that is connected between the two terminals.

### **Configuration**

#### **Number of series cells – Number of supercapacitor cells in series**

1 (default)

Specify the number of cells in the supercapacitor that are in a series configuration.

#### **Number of parallel cells – Number of supercapacitor cells in parallel**

1 (default)

Specify the number of cells in the supercapacitor that are in a parallel configuration.

### **Variables**

#### **Beginning Value – Initial target value**

0 (default)

Use the **Variables** tab to set the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

## **References**

- [1] Zubieta, L. and R. Bonert. “Characterization of Double-Layer Capacitors for Power Electronics Applications.” *IEEE Transactions on Industry Applications*, Vol. 36, No. 1, 2000, pp. 199-205.
- [2] Weddell, A. S., G. V. Merrett, T. J. Kazmierski, and B. M. Al-Hashimi. “Accurate Supercapacitor Modeling for Energy-Harvesting Wireless Sensor Nodes.” *IEEE Transactions on Circuits And Systems-II: Express Briefs*, Vol. 58, No. 12, 2011, pp. 911-915.

## **See Also**

### **Simscape Blocks**

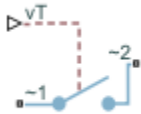
Battery | Controlled Current Source (Three-Phase) | Current Source (Three-Phase) | Voltage Source (Three-Phase)

**Introduced in R2016b**



# Switch

Three-phase single-throw switch



## Library

Simscape / Electrical / Power Systems / Switches & Breakers

## Description

The Switch block models a three-phase single-throw switch that uses an external signal to connect each phase of port ~1 with the corresponding phase of port ~2 via internal resistance.

The table shows how the external signal  $vT$  controls the block behavior.

Condition	Block Behavior	Resistance Parameter Used
$vT \leq \text{Threshold}$	The switch is open. Each phase in the composite three-phase port ~1 connects to the corresponding phase in the port ~2 via large internal resistance.	<b>Open conductance</b>
$vT > \text{Threshold}$	The switch is closed. Each phase in the composite three-phase port ~1 connects to the corresponding phase in the port ~2 via small internal resistance.	<b>Closed resistance</b>

## Parameters

### Closed resistance

Resistance between ports ~1 and ~2 when the switch is closed. The default value is 0.001 Ohm.

### Open conductance

Conductance between ports ~1 and ~2 when the switch is open. The default value is 1e-6 1/Ohm.

### Threshold

Threshold voltage for the control port vT. When the voltage is above the threshold, the switch is closed. The default value is 0.5 V.

## Ports

The block has the following ports:

~1

Expandable three-phase port

~2

Expandable three-phase port

vT

Scalar control port, which is either a physical signal or an electrical port.

## See Also

[Single-Phase Switch](#) | [Single-Phase Two-Way Switch](#) | [Two-Way Switch](#)

## Topics

[“Expand and Collapse Three-Phase Ports on a Block”](#)

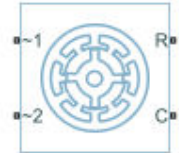
[“Switch Between Physical Signal and Electrical Ports”](#)

### Introduced in R2013b

# Switched Reluctance Machine

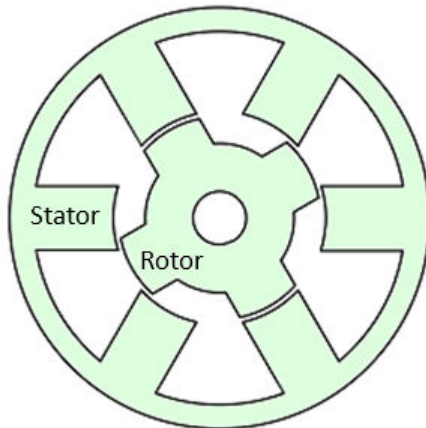
Switched reluctance machine (SRM)

**Library:** Simscape / Electrical / Power Systems / Machines / Reluctance



## Description

The Switched Reluctance Machine block represents a three-phase switched reluctance machine (SRM). The diagram shows the motor construction.



## Equations

The rotor stroke angle for a three-phase machine is

$$\theta_{st} = \frac{2\pi}{3N_r},$$

where:

- $\theta_{st}$  is the stoke angle.
- $N_r$  is the number of rotor poles.

The torque production capability,  $\beta$ , of one rotor pole is

$$\beta = \frac{2\pi}{N_r}.$$

The mathematical model for a switched reluctance machine (SRM) is highly nonlinear due to influence of the magnetic saturation on the flux linkage-to-angle,  $\lambda(\theta_{ph})$  curve. The phase voltage equation for an SRM is

$$v_{ph} = R_s i_{ph} + \frac{d\lambda_{ph}(i_{ph}, \theta_{ph})}{dt}$$

where:

- $v_{ph}$  is the voltage per phase.
- $R_s$  is the stator resistance per phase.
- $i_{ph}$  is the current per phase.
- $\lambda_{ph}$  is the flux linkage per phase.
- $\theta_{ph}$  is the angle per phase.

Rewriting the phase voltage equation in terms of partial derivatives yields this equation:

$$v_{ph} = R_s i_{ph} + \frac{\partial \lambda_{ph}}{\partial i_{ph}} \frac{di_{ph}}{dt} + \frac{\partial \lambda_{ph}}{\partial \theta_{ph}} \frac{d\theta_{ph}}{dt}.$$

Transient inductance is defined as

$$L_t(i_{ph}, \theta_{ph}) = \frac{\partial \lambda_{ph}(i_{ph}, \theta_{ph})}{\partial i_{ph}},$$

or more simply as

$$\frac{\partial \lambda_{ph}}{\partial i_{ph}}.$$

Back electromotive force is defined as

$$E_{ph} = \frac{\partial \lambda_{ph}}{\partial \theta_{ph}} \omega_r.$$

Substituting these terms into the rewritten voltage equation yields this voltage equation:

$$v_{ph} = R_s i_{ph} + L_t(i_{ph}, \theta_{ph}) \frac{di_{ph}}{dt} + E_{ph}.$$

Applying the co-energy formula to equations for torque,

$$T_{ph} = \frac{\partial W(\theta_{ph})}{\partial \theta_r},$$

and energy,

$$W(i_{ph}, \theta_{ph}) = \int_0^{i_{ph}} \lambda_{ph}(i_{ph}, \theta_{ph}) di_{ph}.$$

yields an integral equation that defines the instantaneous torque per phase, that is,

$$T_{ph}(i_{ph}, \theta_{ph}) = \int_0^{i_{ph}} \frac{\partial \lambda_{ph}(i_{ph}, \theta_{ph})}{\partial \theta_{ph}} di_{ph}.$$

Integrating over the phases give this equation, which defines the total instantaneous torque for a three-phase SRM:

$$T = \sum_{j=1}^3 T_{ph}(j).$$

The equation for motion is

$$J \frac{d\omega}{dt} = T - T_L - B_m \omega$$

where:

- $J$  is the rotor inertia.
- $\omega$  is the mechanical rotational speed.
- $T$  is the rotor torque. For the Switched Reluctance Machine block, torque flows from the machine case (block conserving port **C**) to the machine rotor (block conserving port **R**).
- $T_L$  is the load torque.
- $J$  is the rotor inertia.
- $B_m$  is the rotor damping.

For high-fidelity modeling and control development, use empirical data and finite element calculation to determine the flux linkage curve in terms of current and angle, that is,

$$\lambda_{ph}(i_{ph}, \theta_{ph}).$$

For low-fidelity modeling, you can also approximate the curve using analytical techniques. One such technique [2] uses this exponential function:

$$\lambda_{ph}(i_{ph}, \theta_{ph}) = \lambda_{sat} \left( 1 - e^{-i_{ph} f(\theta_{ph})} \right),$$

where:

- $\lambda_{sat}$  is the saturated flux linkage.
- $f(\theta_r)$  is obtained by Fourier expansion.

For the Fourier expansion, use the first two even terms of this equation:

$$f(\theta_{ph}) = a + b \cos(N_r \theta_{ph})$$

where  $a > b$ ,

$$a = \frac{L_{\min} + L_{\max}}{2\lambda_{sat}},$$

and

$$b = \frac{L_{\max} - L_{\min}}{2\lambda_{sat}}.$$

## Assumptions

The block assumes that a zero rotor angle corresponds to a rotor pole that is aligned perfectly with the  $a$ -phase.

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Ports

### Conserving

#### **R — Machine rotor**

mechanical rotational

Mechanical rotational conserving port associated with the machine rotor.

Data Types: double

#### **C — Machine case**

mechanical rotational

Mechanical rotational conserving port associated with the machine case.

Data Types: double

**~1 — Three-phase composite**

electrical

Expandable three-phase port.

Data Types: double

**~2 — Three-phase composite**

electrical

Expandable three-phase port.

Data Types: double

## Parameters

### Main

**Number of rotor poles — Rotor pole**

4 (default) | integer

Number of pole pairs on the rotor.

**Stator resistance per phase — Resistance**

3 Ohm (default)

Per-phase resistance of each of the stator windings.

**Stator parameterization — Parameterization method**

Specify saturated flux linkage (default) | Specify flux characteristic

Method for parameterizing the stator.

**Dependencies**

Selecting Specify saturated flux linkage enables these parameters:

- **Saturated flux linkage**



- **Aligned inductance**
- **Unaligned inductance**

Selecting Specify flux characteristic enables these parameters:

- **Current vector,  $i$**
- **Angle vector,  $\theta$**
- **Flux linkage matrix,  $\Phi(i,\theta)$**

#### **Saturated flux linkage – Flux linkage**

0.43 Wb (default)

Saturated flux linkage per phase.

#### **Dependencies**

To enable this parameter, set **Stator parameterization** to Specify saturated flux linkage.

#### **Aligned inductance – Inductance**

0.0046 H (default)

The value of this parameter must be greater than the value of the **Unaligned inductance** parameter.

#### **Dependencies**

To enable this parameter, set **Stator parameterization** to Specify saturated flux linkage.

#### **Unaligned inductance – Inductance**

6.7e-4 H (default)

The value of this parameter must be less than the value of the **Aligned inductance** parameter.

#### **Dependencies**

To enable this parameter, set **Stator parameterization** to Specify saturated flux linkage.

#### **Current vector, $i$ – Current**

[0, 50, 100] A (default)

Current vector used to identify the flux linkage curve family.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify flux characteristic.

**Angle vector, theta — Angle**

[0, 45, 90] deg (default)

Angle vector used to identify the flux linkage curve family.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify flux characteristic.

**Flux linkage matrix, Phi(i,theta) — Flux**

[0, 0, 0; .37, .06, .37; .43, .1, .43] Wb (default)

Flux linkage matrix that defines the flux linkage curve family.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify flux characteristic.

## Mechanical

**Rotor inertia — Inertia**

0.01 kg\*m<sup>2</sup> (default)

Inertia of the rotor attached to mechanical translational port **R**.

**Rotor Damping — Damping**

0 N\*m/(rad/s) (default)

Rotary damping.

## References

- [1] Boldea, I. and S. A. Nasar. *Electric Drives, Second Edition*. CRC Press, New York, 2005.

- [2] Ilic<sup>1</sup>-Spong, M., R. Marino, S. Peresada, and D. Taylor. "Feedback linearizing control of switched reluctance motors." *IEEE Transactions on Automatic Control*. Vol. 32, No. 5, 1987, pp. 371-379.

## **See Also**

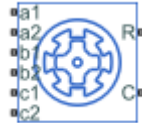
Brushless DC Motor | Permanent Magnet Synchronous Motor | Synchronous Machine Field Circuit (SI) | Synchronous Machine Field Circuit (pu) | Synchronous Machine Measurement | Synchronous Reluctance Machine

**Introduced in R2017b**

## Switched Reluctance Motor

Model a switched reluctance motor

**Library:** Simscape / Electrical / Electronics and Mechatronics / Actuators and Drivers / Rotational Actuators



### Description

The Switched Reluctance Motor implements a three-phase switched reluctance motor (SRM). The stator has three pole pairs, carrying the three motor windings, and the rotor has several nonmagnetic poles. The motor produces torque by energizing a stator pole pair, inducing a force on the closest rotor poles and pulling them toward alignment.

Choose this motor in your application to take advantage of these properties:

- Low cost
- Relatively safe failing currents
- Robustness to high temperature operation
- High torque-to-inertia ratio

Use this block to model an SRM using easily measurable or estimable parameters. To model an SRM using FEM data, see “Switched Reluctance Motor Parameterized with FEM Data”.

### Equations

The block calculates the voltage,  $v$ , across each of the windings by:

$$v = iR + L(\theta) \frac{di}{dt} + \frac{\partial \lambda}{\partial \theta} \omega_{mech},$$

where:

- $i$  is the winding current.
- $\lambda$  is the flux linkage.
- $\theta$  is the rotor angular position.
- $\omega_{mech}$  is the rotor speed.
- $R$  is the winding resistance. Set this value using the **Magnetizing resistance** parameter.
- $L$  is the inductance of the winding, given by the derivative of flux with respect to current:

$$L(\theta) = \frac{\partial \lambda(\theta)}{\partial i}.$$

The overall torque produced by the motor is:

$$\tau = \int_0^i \frac{\partial \lambda}{\partial \theta} di.$$

Because of the significant effect of saturation on flux linkage, the block approximates it for the above two equations using this relationship:

$$\lambda = \Phi_{sat} \left( 1 - e^{-L_0(\theta)i/\Phi_{sat}} \right).$$

Here:

- $\Phi_{sat}$  is the maximum saturated flux linkage.
- $L_0$  is the unsaturated inductance.

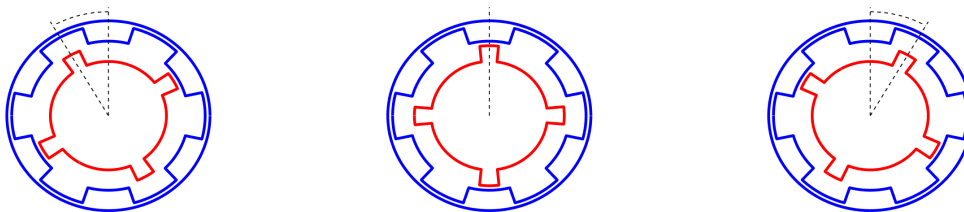
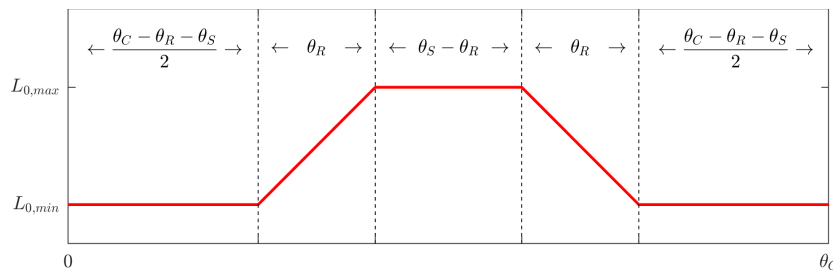
The effects of saturation are more prominent as the product of current and unsaturated inductance approach the saturated flux linkage value. Specify this value using the **Saturated flux linkage** parameter.

Differentiating the flux equation then gives the winding inductance:

$$L(\theta) = L_0(\theta) e^{-L_0(\theta)i/\Phi_{sat}}$$

The unsaturated inductance varies between a minimum and maximum value. The minimum value occurs when a rotor pole is directly between two stator poles. The

maximum occurs when the rotor pole is aligned with a stator pole. In between these two points, the block approximates the unsaturated inductance linearly as a function of rotor angle. This figure shows the unsaturated inductance as a rotor pole passes over a stator pole.



In the figure:

- $\theta_R$  corresponds to the angle subtended by the rotor pole. Set this using the **Angle subtended by each rotor pole** parameter.
- $\theta_S$  corresponds to the angle subtended by the stator pole. Set this using the **Angle subtended by each stator pole** parameter.
- $\theta_C$  corresponds to the angle subtended by this full cycle, determined by  $2\pi/2n$  where  $n$  is the number of stator pole pairs.

## Numerical Smoothing

In practice, magnetic edge effects prevent the inductance from taking a trapezoidal shape as a rotor pole passes over a stator pole. To model these effects, and to avoid gradient discontinuities that hinder solver convergence, the block smooths the gradient  $\partial L_0 / \partial \theta$  at inflection points. To change the angle over which this smoothing is applied, use the **Angle over which flux gradient changes are smoothed** parameter.

## Thermal Ports

The block has four optional thermal ports, one for each of the three windings and one for the rotor. These ports are hidden by default. To expose the ports, right-click the block and select **Simscape > Block choices > Show thermal port**. Thermal ports display on the block icon, and the **Thermal** tab appears in the block dialog box. These tabs are described further on this reference page.

Use the thermal ports to simulate the effects of copper resistance and iron losses that convert electrical power to heat. For more information on using thermal ports in actuator blocks, see “Simulating Thermal Effects in Rotational and Translational Actuators”.

## Ports

### Conserving

#### **a1 — Positive *a*-terminal**

electrical

Electrical conserving port associated with the positive terminal of stator winding *a*.

#### **a2 — Negative *a*-terminal**

electrical

Electrical conserving port associated with the negative terminal of stator winding *a*.

#### **b1 — Positive *b*-terminal**

electrical

Electrical conserving port associated with the positive terminal of stator winding *b*.

#### **b2 — Negative *b*-terminal**

electrical

Electrical conserving port associated with the negative terminal of stator winding *b*.

#### **c1 — Positive *c*-terminal**

electrical

Electrical conserving port associated with the positive terminal of stator winding *c*.

**c2 — Negative c-terminal**

electrical

Electrical conserving port associated with the negative terminal of stator winding *c*.

**R — Rotor**

mechanical

Mechanical rotational conserving port associated with the rotor.

**C — Casing**

mechanical

Mechanical rotational conserving port associated with the stator or casing.

**HA — a-terminal thermal port**

thermal

Thermal conserving port associated with stator winding *a*.

**Dependencies**

To enable this port, use the **Show thermal port** variant, as described in “Thermal Ports” on page 1-1753.

**HB — b-terminal thermal port**

thermal

Thermal conserving port associated with stator winding *b*.

**Dependencies**

To enable this port, use the **Show thermal port** variant, as described in “Thermal Ports” on page 1-1753.

**HC — c-terminal thermal port**

thermal

Thermal conserving port associated with stator winding *c*.

**Dependencies**

To enable this port, use the **Show thermal port** variant, as described in “Thermal Ports” on page 1-1753.



**HR — Rotor thermal port**

thermal

Thermal conserving port associated with the rotor.

**Dependencies**

To enable this port, use the **Show thermal port** variant, as described in “Thermal Ports” on page 1-1753.

## Parameters

### Main

**Number of rotor poles — Rotor pole**

4 (default) | positive integer

Number of poles of the rotor.

**Stator resistance per phase — Resistance**

3 0hm (default) | positive number

Per-phase resistance of each of the stator windings.

**Magnetizing resistance — Magnetic losses**

inf 0hm (default) | real number

The total magnetizing resistance for each of the phase windings. The default value of inf indicates that there are no iron losses.

**Aligned inductance — Inductance**

0.005 H (default) | positive number

Maximum inductance, achieved when a rotor and stator pole are aligned. The value of this parameter must be greater than the value of the **Unaligned inductance** parameter.

**Unaligned inductance — Inductance**

7e-4 H (default) | positive number

Maximum inductance, achieved when a rotor pole is midway between two stator poles. The value of this parameter must be less than the value of the **Aligned inductance** parameter.

### **Saturated flux linkage — Flux linkage**

inf Wb (default) | positive number

Saturated flux linkage per phase.

### **Angle subtended by each stator pole — Stator tooth angle**

45 deg (default) | positive number

Angle spanned by each stator tooth. This value must be greater than or equal to the value of **Angle subtended by each rotor pole**.

### **Angle subtended by each rotor pole — Rotor tooth angle**

42 deg (default) | positive number

Angle spanned by each rotor tooth.

### **Angle over which flux gradient changes are smoothed — Inflection point smoothing**

0.5 deg (default) | positive number

Angle over which sharp edges in trapezoidal inductance curve are smoothed. This value must be smaller than the value of **Angle subtended by each rotor pole**.

## **Mechanical**

### **Rotor inertia — Inertia**

0.1 kg\*m<sup>2</sup> (default) | zero or positive number

Moment of inertia of the rotor.

### **Rotor damping — Damping**

1e-3 N\*m/(rad/s) (default) | positive number

Damping of the rotor.

## Thermal

This tab appears only for blocks with exposed thermal ports. For more information, see “Thermal Ports” on page 1-1753.

### Resistance temperature coefficient — Temperature coefficient

3.93e-3 1/K (default) | positive number

Coefficient  $\alpha$  in the equation relating resistance to temperature for all three windings, as described in “Thermal Model for Actuator Blocks”. The default value, 3.93e-3 1/K, is for copper.

### Measurement temperature — Rated temperature

25 degC (default) | real number

The temperature for which motor parameters are quoted.

### Winding thermal mass — Winding thermal mass

100 J/K (default) | positive number

The thermal mass value for the *a*-, *b*-, and *c*-windings. The thermal mass is the energy required to raise the temperature by one degree.

### Rotor thermal mass — Rotor thermal mass

50 J/K (default) | positive number

The thermal mass of the rotor. The thermal mass is the energy required to raise the temperature of the rotor by one degree.

### Percentage of magnetizing resistance associated with the rotor — Iron loss heating distribution

50 (default) | positive number

The percentage of the magnetizing resistance associated with the magnetic path through the rotor. This parameter determines how much of the iron loss heating is attributed to:

- The rotor thermal port **HR**
- The three stator thermal ports **HA**, **HB**, and **HC**

## See Also

Switched Reluctance Machine

**Introduced in R2018a**

# Symmetrical-Components Transform

Implement *abc* to *+0* transform

**Library:** Simscape / Electrical / Power Systems / Control /  
Mathematical Transforms



## Description

The Symmetrical-Components Transform block implements a symmetrical transform of a set of phasors. The transform splits an unbalanced set of three phasors into three balanced sets of phasors.

In an unbalanced system with balanced impedances, use this block to decouple the system into three independent networks. In a balanced system, use this block to simplify the set of three-phasors to an equivalent one-line network. In this case, the positive set represents the one-line network.

Use the Power invariant property to choose between the Fortescue transform, and the alternative, power-invariant version.

## Equations

The symmetrical-components transform separates an unbalanced three-phase signal given in phasor quantities into three balanced sets of phasors:

$$\begin{bmatrix} v_a \\ v_b \\ v_c \end{bmatrix} = \begin{bmatrix} v_{a+} \\ v_{b+} \\ v_{c+} \end{bmatrix} + \begin{bmatrix} v_{a-} \\ v_{b-} \\ v_{c-} \end{bmatrix} + \begin{bmatrix} v_{a0} \\ v_{b0} \\ v_{c0} \end{bmatrix},$$

where:

- $v_a$ ,  $v_b$ , and  $v_c$  make up the original, unbalanced set of phasors.

- $v_{a+}$ ,  $v_{b+}$ , and  $v_{c+}$  make up the balanced, positive set of phasors.
- $v_{a-}$ ,  $v_{b-}$ , and  $v_{c-}$  make up the balanced, negative set of phasors.
- $v_{a0}$ ,  $v_{b0}$ , and  $v_{c0}$  make up the balanced, zero set of phasors.

The block calculates the symmetric  $a$ -phase using the transformation:

$$\begin{bmatrix} V_{a+} \\ V_{a-} \\ V_{a0} \end{bmatrix} = \frac{K}{3} \begin{bmatrix} 1 & a & a^2 \\ 1 & a^2 & a \\ 1 & 1 & 1 \end{bmatrix} \begin{bmatrix} V_a \\ V_b \\ V_c \end{bmatrix}.$$

where  $a$  is the complex rotation operator

$$a = e^{2\pi i/3},$$

and  $K$  is the constant that determines the type of transform:

$$\begin{cases} K = 1 & \text{Fortescue transform} \\ K = \sqrt{3} & \text{Power-invariant transform} \end{cases}$$

To select the power-invariant transform and simplify the power calculation in the  $+0$  domain, enable the `Power invariant` property.

Because the remaining two sets of symmetrical phasors are not often used in calculation, the block does not calculate them. However, they are given in terms of simple rotations of the first set:

$$\begin{bmatrix} V_{b+} \\ V_{b-} \\ V_{b0} \end{bmatrix} = \begin{bmatrix} a^2 & 0 & 0 \\ 0 & a & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} V_{a+} \\ V_{a-} \\ V_{a0} \end{bmatrix},$$

and

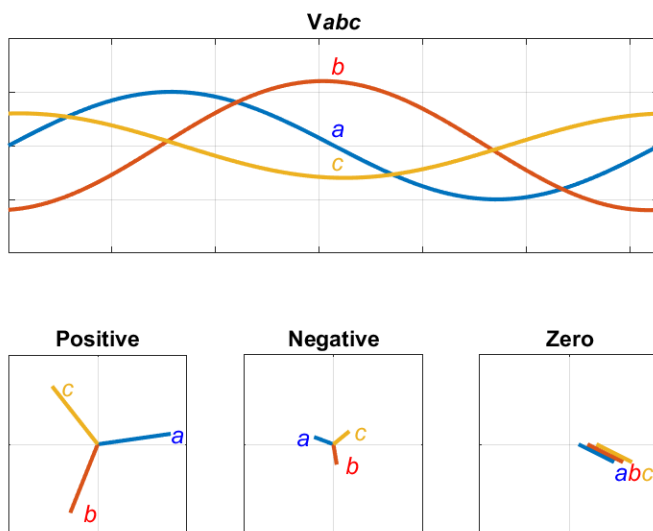
$$\begin{bmatrix} V_{c+} \\ V_{c-} \\ V_{c0} \end{bmatrix} = \begin{bmatrix} a & 0 & 0 \\ 0 & a^2 & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} V_{a+} \\ V_{a-} \\ V_{a0} \end{bmatrix}.$$

## Operating Principle

The three sets of balanced phasors generated by the transform have the following properties:

- The positive set has the same order as the unbalanced set of phasors  $a$ - $b$ - $c$ .
- The negative set has the opposite order as the unbalanced set of phasors  $a$ - $c$ - $b$ .
- The zero set has no order because all three phasor angles are equal.

This diagram visualizes the separation performed by the transform.



In the diagram, the top axis shows an unbalanced three-phase signal with components  $a$ ,  $b$ , and  $c$ . The bottom set of axes separates the three-phase signal into symmetrical positive, negative, and zero phasors.

Observe that in each case, the  $a$ ,  $b$ , and  $c$  components are symmetrical and are separated by:

- +120 degrees for the positive set.

- -120 degrees for the negative set.
- 0 degrees for the zero set.

## Inverse Transform

The symmetrical-components transform is unique and invertible:

$$\begin{bmatrix} V_a \\ V_b \\ V_c \end{bmatrix} = \frac{1}{K} \begin{bmatrix} 1 & 1 & 1 \\ a^2 & a & 1 \\ a & a^2 & 1 \end{bmatrix} \begin{bmatrix} V_{a+} \\ V_{a-} \\ V_{a0} \end{bmatrix}.$$

Use the Inverse Symmetrical-Components Transform block to perform this inverse transform.

## Ports

### Input

#### **abc — a, b, and c phasors**

vector

Three-phase set of unbalanced phasors to be separated, given as a complex signal.

Data Types: `single` | `double`

### Output

#### **+ - 0 — Balanced a phasor components**

vector

Positive, negative and zero *a* phasors, output as a complex signal. Use the rotations given in the equations section to compute the *b* and *c* phasor sets.

Data Types: `single` | `double`



## Parameters

### **Power invariant – Transform type**

off (default) | on

Power invariant toggle. Select this parameter to use the power-invariant alternative of the original Fortescue transform.

## References

[1] Anderson, P. M. *Analysis of Faulted Power Systems*. Hoboken, NJ: Wiley-IEEE Press, 1995.

## See Also

### **Blocks**

Clarke Transform | Clarke to Park Angle Transform | Inverse Symmetrical-Components Transform | Inverse Clarke Transform | Inverse Park Transform | Park to Clarke Angle Transform

**Introduced in R2017b**

## Synchronous Machine Field Circuit (pu)

Synchronous machine field circuit per-unit voltage supply and current measurement



### Library

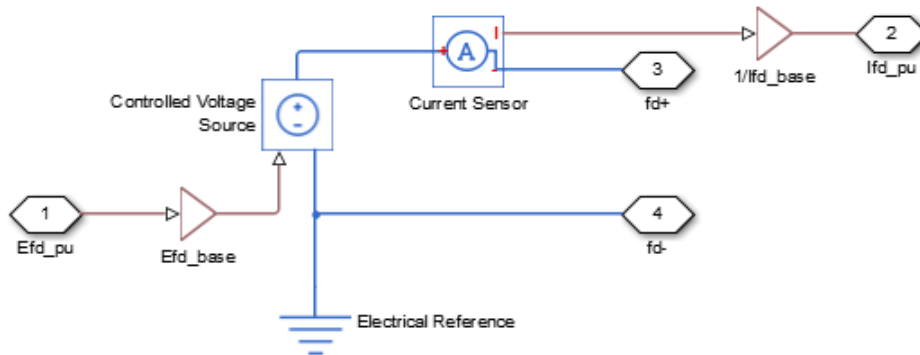
Simscape / Electrical / Power Systems / Machines

### Description

The Synchronous Machine Field Circuit (pu) block applies specified voltage to, and measures current through, the field circuit of the synchronous machine that it is connected to. It includes an electrical reference. The physical signal input `Efd_pu` defines the voltage and the physical signal output `Ifd_pu` provides the current, both in per-unit.

The per-unit bases are the nonreciprocal per-unit system, `Efd` and `Ifd`, rather than the reciprocal per-unit system, `efd` and `ifd`.

The figure shows the schematic for the Synchronous Machine Field Circuit (pu) block.



## Parameters

- “Main Tab” on page 1-1765
- “Machine Parameters Tab” on page 1-1766

## Main Tab

### Rated apparent power

Rated apparent power of the connected machine. The default value is 555e6 V\*A.

### Rated electrical frequency

Nominal electrical frequency at which rated apparent power of the connected machine is quoted. The default value is 60 Hz.

### Specify field circuit input required to produce rated terminal voltage at no load by

Choose between Field circuit voltage and Field circuit current. The default value is Field circuit current.

### Field circuit current

This value is used to calculate the per-unit bases for the field circuit (nonreciprocal per-unit system). This parameter is visible only when **Specify field circuit input required to produce rated terminal voltage at no load by** is set to Field circuit current. The default value is 1300 A.

### **Field circuit voltage**

This value is used to calculate the per-unit bases for the field circuit (nonreciprocal per-unit system). This parameter is visible only when **Specify field circuit input required to produce rated terminal voltage at no load by** is set to Field circuit voltage. The default value is 92.95 V.

## **Machine Parameters Tab**

### **Specify parameterization by**

Choose between Fundamental Parameters and Standard Parameters. The default value is Fundamental Parameters.

### **Stator d-axis mutual inductance (unsaturated), L<sub>adu</sub>**

Unsaturated stator d-axis mutual inductance. This parameter is visible only when **Specify parameterization by** is set to Fundamental Parameters. The default value is 1.66 pu.

### **Rotor field circuit resistance, R<sub>fd</sub>**

Rotor field-circuit resistance. This parameter is visible only when **Specify parameterization by** is set to Fundamental Parameters. The default value is 0.0006 pu.

### **Stator leakage reactance, X<sub>l</sub>**

Stator leakage reactance. This parameter is visible only when **Specify parameterization by** is set to Standard Parameters. The default value is 0.15 pu.

### **d-axis synchronous reactance, X<sub>d</sub>**

The d-axis synchronous reactance. This parameter is visible only when **Specify parameterization by** is set to Standard Parameters. The default value is 1.81 pu.

### **d-axis transient reactance, X<sub>d</sub>'**

The d-axis transient reactance. This parameter is visible only when **Specify parameterization by** is set to Standard Parameters. The default value is 0.3 pu.

### **Specify d-axis transient time constant by**

This parameter is visible only when **Specify parameterization by** is set to Standard Parameters. Choose between Open-circuit value and Short-circuit value. The default value is Open-circuit value.

**d-axis transient open-circuit, Td0'**

The d-axis transient open-circuit time constant. This parameter is visible only when **Specify d-axis transient time constant by** is set to `Open-circuit` value. The default value is 8 s.

**d-axis transient short-circuit, Td'**

The d-axis transient short-circuit time constant. This parameter is visible only when **Specify d-axis transient time constant by** is set to `Short-circuit` value. The default value is 1.326 s.

## Ports

The block has the following ports:

Efd\_pu

Field voltage input, per-unit

Ifd\_pu

Field current output, per-unit

fd+

Electrical conserving port corresponding to the field winding positive terminal

fd-

Electrical conserving port corresponding to the field winding negative terminal

## See Also

Synchronous Machine Salient Pole (standard) | Synchronous Machine Salient Pole (fundamental) | Synchronous Machine Field Circuit (SI) | Synchronous Machine Round Rotor (standard) | Synchronous Machine Round Rotor (fundamental)

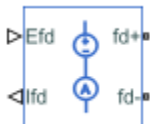
## Topics

Three-Phase Synchronous Machine Control

**Introduced in R2014b**

## Synchronous Machine Field Circuit (SI)

Synchronous machine field circuit voltage supply and current measurement in SI units



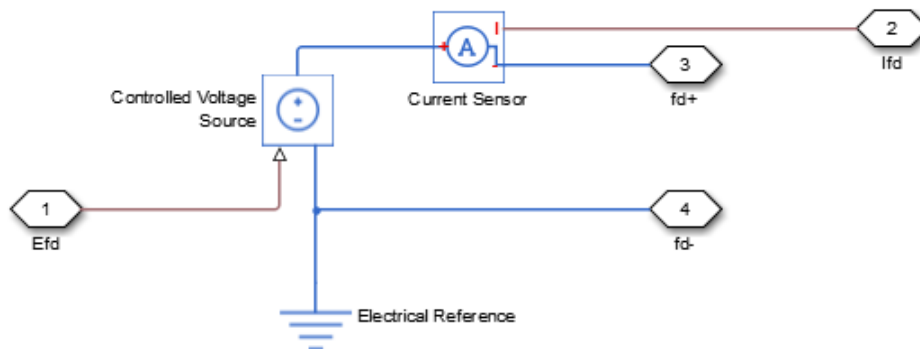
### Library

Simscape / Electrical / Power Systems / Machines

### Description

The Synchronous Machine Field Circuit (SI) block applies specified voltage to, and measures current through, the field circuit of the synchronous machine that it is connected to. It includes an electrical reference. The physical signal input  $E_{fd}$  defines the voltage, in Volts, and the physical signal output  $I_{fd}$  provides the current, in Amperes.

The figure shows the schematic for the Synchronous Machine Field Circuit (SI) block.



## Ports

The block has the following ports:

Efd

Field voltage input, V

Ifd

Field current output, A

fd+

Electrical conserving port corresponding to the field winding positive terminal

fd-

Electrical conserving port corresponding to the field winding negative terminal

## See Also

Synchronous Machine Salient Pole (standard) | Synchronous Machine Salient Pole (fundamental) | Synchronous Machine Field Circuit (pu) | Synchronous Machine Round Rotor (standard) | Synchronous Machine Round Rotor (fundamental)

**Introduced in R2014b**

# Synchronous Machine Measurement

Per-unit measurement from synchronous machine



## Library

Simscape / Electrical / Power Systems / Machines

## Description

The Synchronous Machine Measurement block outputs a per-unit measurement associated with a connected Synchronous Machine Round Rotor or Synchronous Machine Salient Pole block. The input of the Synchronous Machine Measurement block connects to the pu output port of the synchronous machine block.

You set the **Output** parameter to a per-unit measurement associated with the synchronous machine. Based on the value you select, the Synchronous Machine Measurement block:

- Directly outputs the value of an element in the input signal vector
- Calculates the per-unit measurement by using values of elements in the input signal vector in mathematical expressions

The Synchronous Machine Measurement block outputs a per-unit measurement from the synchronous machine according to the output value expressions in the table. For example, when you set **Output** to `Stator d-axis voltage`, the block directly outputs the value of the `pu_ed` element in the input signal vector. However, when you set **Output** to `Reactive power`, the block calculates the value from the `pu_ed`, `pu_eq`, `pu_id`, and `pu_iq` elements.



Output Parameter Setting	Output Value Expression
Field voltage (field circuit base, Efd)	pu_fd_Efd
Field current (field circuit base, Ifd)	pu_fd_Ifd
Electrical torque	pu_torque
Rotor velocity	pu_velocity
Stator d-axis voltage	pu_ed
Stator q-axis voltage	pu_eq
Stator zero-sequence voltage	pu_e0
Stator d-axis current	pu_id
Stator q-axis current	pu_iq
Stator zero-sequence current	pu_i0
Apparent power	$\sqrt{pu\_Pt^2 + pu\_Qt^2}$
Real power	$pu\_Pt = (pu\_ed*pu\_id) + (pu\_eq*pu\_iq) + 2(pu\_e0*pu\_i0)$
Reactive power	$pu\_Qt = (pu\_eq*pu\_id) - (pu\_ed*pu\_iq)$
Terminal voltage	$\sqrt{(pu\_ed^2 + pu\_eq^2)}$
Terminal current	$\sqrt{(pu\_id^2 + pu\_iq^2)}$
Power factor angle (rad)	$power\_factor\_angle = atan2(pu\_Qt, pu\_Pt)$
Power factor	$cos(power\_factor\_angle)$
Load angle (rad)	$load\_angle(rad) = atan2(pu\_ed, pu\_eq)$

## Parameters

### Output

Per-unit measurement from synchronous machine. The default value is Field voltage (field circuit base, Efd).

## Ports

The block has the following ports:

pu

Physical signal vector port associated with per-unit measurements from a connected synchronous machine. The vector elements are:

- pu\_fd\_Efd
- pu\_fd\_Ifd
- pu\_torque
- pu\_velocity
- pu\_ed
- pu\_eq
- pu\_e0
- pu\_id
- pu\_iq
- pu\_i0

o

Per-unit measurement output port.

## See Also

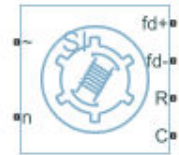
Synchronous Machine Round Rotor (fundamental) | Synchronous Machine Round Rotor (standard) | Synchronous Machine Salient Pole (fundamental) | Synchronous Machine Salient Pole (standard)

**Introduced in R2013b**

## Synchronous Machine Model 1.0 (SI parameters)

Synchronous machine with field circuit and no damper

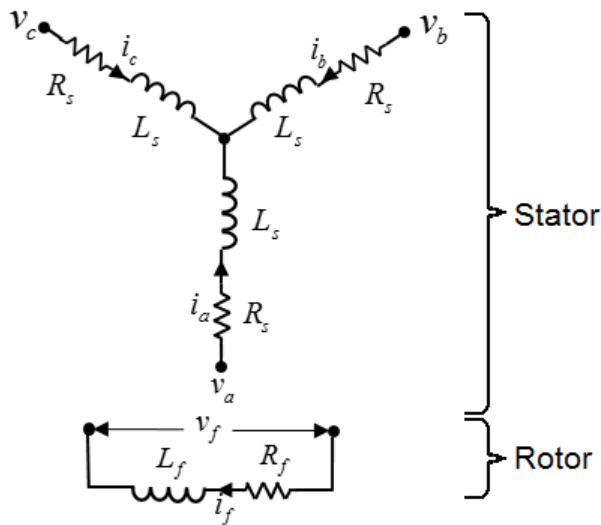
**Library:** Simscape / Electrical / Power Systems / Machines / Synchronous Machine (Simplified)



### Description

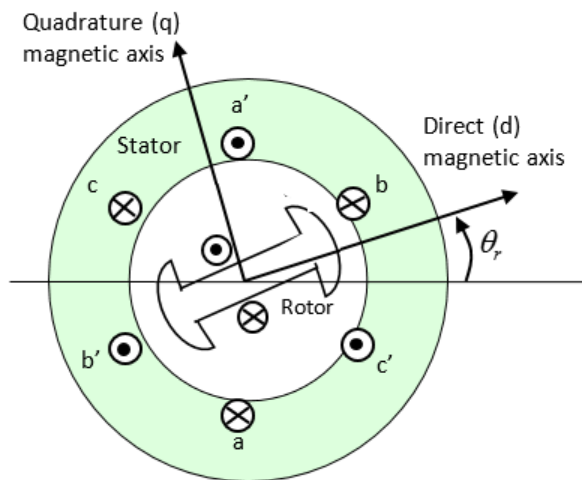
The Synchronous Machine Model 1.0 (SI parameters) block uses a simplified parameterization model for synchronous machines. Use the block to model synchronous machines with a field winding and no dampers.

The figure shows the equivalent electrical circuit for the stator and rotor windings.



## Motor Construction

The diagram shows the motor construction with a single pole pair on the rotor. For the axes convention, when rotor mechanical angle  $\theta_r$  is zero, the  $a$ -phase and permanent magnet fluxes are aligned. The block supports a second rotor axis definition for which rotor mechanical angle is defined as the angle between the  $a$ -phase magnetic axis and the rotor  $q$ -axis.



## Equations

Voltages across the stator windings are defined by

$$\begin{bmatrix} v_a \\ v_b \\ v_c \end{bmatrix} = \begin{bmatrix} R_s & 0 & 0 \\ 0 & R_s & 0 \\ 0 & 0 & R_s \end{bmatrix} \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix} + \begin{bmatrix} \frac{d\psi_a}{dt} \\ \frac{d\psi_b}{dt} \\ \frac{d\psi_c}{dt} \end{bmatrix},$$

where:

- $v_a$ ,  $v_b$ , and  $v_c$  are the individual phase voltages across the stator windings.
- $R_s$  is the equivalent resistance of each stator winding.
- $i_a$ ,  $i_b$ , and  $i_c$  are the currents flowing in the stator windings.
- 

$\frac{d\psi_a}{dt}$ ,  $\frac{d\psi_b}{dt}$ , and  $\frac{d\psi_c}{dt}$  are the rates of change of magnetic flux in each stator winding.

The voltage across the field winding is expressed as

$$v_f = R_f i_f + \frac{d\psi_f}{dt},$$

where:

- $v_f$  is the individual phase voltage across the field winding.
- $R_f$  is the equivalent resistance of the field winding.
- $i_f$  is the current flowing in the field winding.
- $\frac{d\psi_f}{dt}$  is the rate of change of magnetic flux in the field winding.

The permanent magnet, excitation winding, and the three star-wound stator windings contribute to the flux linking each winding. The total flux is defined by

$$\begin{bmatrix} \psi_a \\ \psi_b \\ \psi_c \end{bmatrix} = \begin{bmatrix} L_{aa} & L_{ab} & L_{ac} \\ L_{ba} & L_{bb} & L_{bc} \\ L_{ca} & L_{cb} & L_{cc} \end{bmatrix} \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix} + \begin{bmatrix} \psi_{am} \\ \psi_{bm} \\ \psi_{cm} \end{bmatrix} + \begin{bmatrix} L_{amf} \\ L_{bmf} \\ L_{cmf} \end{bmatrix} i_f,$$

where:

- $\psi_a$ ,  $\psi_b$ , and  $\psi_c$  are the total fluxes linking each stator winding.
- $L_{aa}$ ,  $L_{bb}$ , and  $L_{cc}$  are the self-inductances of the stator windings.
- $L_{ab}$ ,  $L_{ac}$ ,  $L_{ba}$ ,  $L_{bc}$ ,  $L_{ca}$ , and  $L_{cb}$  are the mutual inductances of the stator windings.
- $\psi_{am}$ ,  $\psi_{bm}$ , and  $\psi_{cm}$  are the magnetization fluxes linking the stator windings.
- $L_{amf}$ ,  $L_{bmf}$ , and  $L_{cmf}$  are the mutual inductances of the field winding.

The inductances in the stator windings are functions of rotor electrical angle and are defined by

$$\theta_e = N \theta_r,$$

$$L_{aa} = L_s + L_m \cos(2\theta_e),$$

$$L_{bb} = L_s + L_m \cos(2(\theta_e - 2\pi/3)),$$

$$L_{cc} = L_s + L_m \cos(2(\theta_e + 2\pi / 3)),$$

$$L_{ab} = L_{ba} = -M_s - L_m \cos(2(\theta_e + \pi / 6)),$$

$$L_{bc} = L_{cb} = -M_s - L_m \cos(2(\theta_e + \pi / 6 - 2\pi / 3)),$$

$$L_{ca} = L_{ac} = -M_s - L_m \cos(2(\theta_r + \pi / 6 + 2\pi / 3)),$$

where:

- $N$  is the number of rotor pole pairs.
- $\theta_r$  is the rotor mechanical angle.
- $\theta_e$  is the rotor electrical angle.
- $L_s$  is the stator self-inductance per phase. This value is the average self-inductance of each of the stator windings.
- $L_m$  is the stator inductance fluctuation. This value is the amplitude of the fluctuation in self-inductance and mutual inductance with changing rotor angle.
- $M_s$  is the stator mutual inductance. This value is the average mutual inductance between the stator windings.

The magnetization flux linking winding,  $a-a'$  is a maximum when  $\theta_r = 0^\circ$  and zero when  $\theta_r = 90^\circ$ . Therefore:

$$L_{mf} = \begin{bmatrix} L_{amf} \\ L_{bnf} \\ L_{cmf} \end{bmatrix} = \begin{bmatrix} L_{mf} \cos \theta_r \\ L_{mf} \cos(\theta_r - 2\pi / 3) \\ L_{mf} \cos(\theta_r + 2\pi / 3) \end{bmatrix}$$

and

$$\Psi_f = L_f i_f + L_{mf}^T \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix},$$

where:



- $L_{mf}$  is the mutual field armature inductance.
- $\psi_f$  is the flux linking the field winding.
- $L_f$  is the field winding inductance.
- $[L_{mf}]^T$  is the transform of the  $L_{mf}$  vector, that is,

$$[L_{mf}]^T = \begin{bmatrix} L_{amf} \\ L_{bmf} \\ L_{cmf} \end{bmatrix}^T = [L_{amf} \quad L_{bmf} \quad L_{cmf}].$$

## Simplified Equations

Applying the Park transformation to the block electrical defining equations produces an expression for torque that is independent of rotor angle.

The Park transformation is defined by

$$P = 2/3 \begin{bmatrix} \cos \theta_e & \cos(\theta_e - 2\pi/3) & \cos(\theta_e + 2\pi/3) \\ -\sin \theta_e & -\sin(\theta_e - 2\pi/3) & -\sin(\theta_e + 2\pi/3) \\ 0.5 & 0.5 & 0.5 \end{bmatrix}.$$

Applying the Park transformation to the first two electrical defining equations produces equations that define the block behavior:

$$v_d = R_s i_d + L_d \frac{di_d}{dt} + L_{mf} \frac{di_f}{dt} - N \omega i_q L_q,$$

$$v_q = R_s i_q + L_q \frac{di_q}{dt} + N \omega (i_d L_d + i_f L_{mf}),$$

$$v_0 = R_s i_0 + L_0 \frac{di_0}{dt},$$

$$v_f = R_f i_f + L_f \frac{di_f}{dt} + \frac{3}{2} L_{mf} \frac{di_d}{dt},$$

$$T = \frac{3}{2} N (i_q (i_d L_d + i_f L_{mf}) - i_d i_q L_q),$$

and

$$J \frac{d\omega}{dt} = T = T_L - B_m \omega.$$

where:

- $v_d$ ,  $v_q$ , and  $v_0$  are the  $d$ -axis,  $q$ -axis, and zero-sequence voltages. These voltages are defined by

$$\begin{bmatrix} v_d \\ v_q \\ v_0 \end{bmatrix} = P \begin{bmatrix} v_a \\ v_b \\ v_c \end{bmatrix}.$$

- $i_d$ ,  $i_q$ , and  $i_0$  are the  $d$ -axis,  $q$ -axis, and zero-sequence currents, defined by

$$\begin{bmatrix} i_d \\ i_q \\ i_0 \end{bmatrix} = P \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix}.$$

- $L_d$  is the stator  $d$ -axis inductance.  $L_d = L_s + M_s + 3/2 L_m$ .
- $\omega$  is the mechanical rotational speed.
- $L_q$  is the stator  $q$ -axis inductance.  $L_q = L_s + M_s - 3/2 L_m$ .
- $L_0$  is the stator zero-sequence inductance.  $L_0 = L_s - 2M_s$ .
- $T$  is the rotor torque. For the Synchronous Machine Model 1.0 (SI parameters) block, torque flows from the machine case (block conserving port **C**) to the machine rotor (block conserving port **R**).
- $J$  is the rotor inertia.
- $T_L$  is the load torque.

- $B_m$  is the rotor damping.

## Assumptions

The block assumes that the flux distribution is sinusoidal.

## Ports

### Conserving

#### **R — Machine rotor**

mechanical rotational

Mechanical rotational conserving port associated with the machine rotor.

#### **C — Machine case**

mechanical rotational

Mechanical rotational conserving port associated with the machine case.

#### **~ — Three-phase composite**

electrical

Expandable three-phase port associated with the stator windings.

#### **n — Neutral phase**

electrical

Electrical conserving port associated with the neutral phase.

#### **fd+ — Field winding positive terminal**

electrical

Electrical conserving port associated with the field winding positive terminal.

#### **fd- — Field winding negative terminal**

electrical

Electrical conserving port associated with the field winding negative terminal.

## Parameters

### Main

#### Number of pole pairs — Rotor pole pairs

4 (default) | integer

Number of permanent magnet pole pairs on the rotor.

#### Stator parameterization — Parameterization method

Specify Ld, Lq and L0 (default) | Specify Ls, Lm, and Ms

Method for parameterizing the stator.

#### Dependencies

Selecting Specify Ld, Lq and L0 enables these parameters:

- **Stator d-axis inductance, Ld**
- **Stator q-axis inductance, Lq**
- **Stator zero-sequence inductance, L0**

Selecting Specify Ls, Lm, and Ms enables these parameters:

- **Stator self-inductance per phase, Ls**
- **Stator inductance fluctuation, Lm**
- **Stator mutual inductance, Ms**

#### Stator d-axis inductance, Ld — Inductance

0.00015 H (default)

Direct-axis inductance of the machine stator.

#### Dependencies

To enable this parameter, set **Stator parameterization** to Specify Ld, Lq and L0.

#### Stator q-axis inductance, Lq — Inductance

0.00021 H (default)

Quadrature-axis inductance of the machine stator.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify Ld, Lq and L0.

**Stator zero-sequence inductance, L0 – Inductance**

0.000012 H (default)

Zero-axis inductance for the machine stator.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify Ld, Lq and L0.

**Stator self-inductance per phase, Ls – Inductance**

0.00016 H (default)

Average self-inductance of the three stator windings.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify Ls, Lm, and Ms.

**Stator inductance fluctuation, Lm – Inductance**

-0.00002 H (default)

Amplitude of the fluctuation in self-inductance and mutual inductance with the rotor angle.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify Ls, Lm, and Ms.

**Stator mutual inductance, Ms – Inductance**

0.00002 H (default)

Average mutual inductance between the stator windings.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify Ls, Lm, and Ms.

**Field winding inductance, Lf – Inductance**

0.05 H (default)

Inductance of the field winding.

**Mutual field armature inductance, Lmf — Inductance**

0.007 H (default)

Armature-field mutual inductance.

**Stator resistance per phase, Rs — Resistance**

0.08 Ohm (default)

Resistance of each of the stator windings.

**Field winding resistance, Rf — Resistance**

3 Ohm (default)

Resistance of the field winding.

**Zero sequence — Option to neglect zero-sequence terms**

Include (default) | Exclude

Option to neglect zero-sequence terms. Choices are:

- **Include** — Include zero-sequence terms. To prioritize model fidelity, use this default setting. Using this option results in an error for simulations that use the Partitioning solver. For more information, see “Increase Simulation Speed Using the Partitioning Solver” (Simscape).
- **Exclude** — Exclude zero-sequence terms. To prioritize simulation speed for desktop simulation or real-time deployment, select this option.

## **Mechanical**

**Rotor inertia — Inertia**

0.01 kg\*m<sup>2</sup> (default)

Inertia of the rotor.

**Rotor Damping — Damping**

0 N\*m/(rad/s) (default)

Damping of the rotor.

## Initial Conditions

### Initial currents, [*i<sub>d</sub>* *i<sub>q</sub>* *i<sub>0</sub>* *i<sub>f</sub>*] – Currents

1[0, 0, 0, 0] A (default) | vector

Initial *d*-, *q*-, 0-sequence and field winding currents.

### Rotor angle definition – Angle

Angle between the a-phase magnetic axis and the d-axis (default) | Angle between the a-phase magnetic axis and the q-axis

Reference point for the rotor angle measurement. If you select the default value, the rotor and *a*-phase fluxes are aligned for a zero-rotor angle. Otherwise, an *a*-phase current generates the maximum torque value for a zero-rotor angle.

### Initial rotor angle – Angle

0 deg (default) | 0-360 deg

Rotor angle at simulation start time.

### Initial rotor speed – Angular speed

0 rpm (default)

Rotor speed at simulation start time. If the rotor inertia, *J*, is zero, the initial speed of the rotor is zero rpm and the initial rotor speed is ignored.

## References

- [1] Kundur, P. *Power System Stability and Control*. New York, NY: McGraw Hill, 1993.
- [2] Anderson, P. M. *Analysis of Faulted Power Systems*. IEEE Press, Power Systems Engineering, 1995.
- [3] Retif, J. M., X. Lin-Shi, A. M. Llor, and F. Morand “New hybrid direct-torque control for a winding rotor synchronous machine.” *2004 IEEE 35th Annual Power Electronics Specialists Conference*. Vol. 2 (2004): 1438-1442.
- [4] IEEE Power Engineering Society. IEEE Std 1110-2002. *IEEE Guide for Synchronous Generator Modeling Practices and Applications in Power System Stability Analyses*. Piscataway, NJ: IEEE, 2002.

## **See Also**

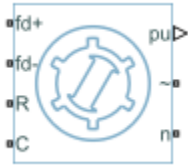
Brushless DC Motor | Permanent Magnet Synchronous Motor | Switched Reluctance Machine | Synchronous Machine Field Circuit (SI) | Synchronous Machine Field Circuit (pu) | Synchronous Machine Measurement | Synchronous Reluctance Machine

**Introduced in R2018a**



# Synchronous Machine Model 2.1 (fundamental)

Synchronous machine with simplified transformation, simplified representation, and fundamental parameterization



## Library

Simscape / Electrical / Power Systems / Machines / Synchronous Machine (Simplified)

## Description

The Synchronous Machine Model 2.1 (fundamental) block models a synchronous machine with one field winding and one damper on the  $d$ -axis and one damper on the  $q$ -axis. You use fundamental parameters to define the characteristics of the machine. This block contains a  $dq$  Park transformation, so use it only for balanced operation.

## Equations

The synchronous machine equations are expressed with respect to a rotating reference frame, defined by

$$\theta_e(t) = N\theta_r(t),$$

where:

- $\theta_e$  is the electrical angle.

- $N$  is the number of pole pairs.
- $\theta_r$  is the rotor angle.

The Park transformation maps the synchronous machine equations to the rotating reference frame with respect to the electrical angle. The Park transformation is defined by

$$P_s = \frac{2}{3} \begin{bmatrix} \cos \theta_e & \cos(\theta_e - \frac{2\pi}{3}) & \cos(\theta_e + \frac{2\pi}{3}) \\ -\sin \theta_e & -\sin(\theta_e - \frac{2\pi}{3}) & -\sin(\theta_e + \frac{2\pi}{3}) \end{bmatrix}.$$

The Park transformation is used to define the per-unit synchronous machine equations. The stator voltage equations are defined by

$$e_d = e_d'' - R_a i_d + x_q'' \dot{i}_q$$

and

$$e_q = e_q'' - x_d'' \dot{i}_d - R_a i_q,$$

where:

- $e_d''$  and  $e_q''$  are the  $d$ -axis and  $q$ -axis voltages behind subtransient reactances.
- $R_a$  is the stator resistance.
- $i_d$  and  $i_q$  are the  $d$ -axis and  $q$ -axis stator currents, defined by

$$\begin{bmatrix} i_d \\ i_q \end{bmatrix} = P_s \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix}.$$

$i_a$ ,  $i_b$ , and  $i_c$  are the stator currents flowing from port  $\sim$  to port  $n$ .

- $x_d''$  and  $x_q''$  are the  $d$ -axis and  $q$ -axis subtransient reactances.
- $e_d$  and  $e_q$  are the  $d$ -axis and  $q$ -axis stator voltages, defined by

$$\begin{bmatrix} e_d \\ e_q \end{bmatrix} = P_s \begin{bmatrix} v_a \\ v_b \\ v_c \end{bmatrix}.$$

$v_a$ ,  $v_b$ , and  $v_c$  are the stator voltages measured from port  $\sim$  to neutral port n.

The rotor voltage equation is defined by

$$e_{fd} = R_{fd} \cdot i_{fd},$$

where:

- $R_{fd}$  is the resistance of rotor field circuit.
- $i_{fd}$  is the per-unit field current using the synchronous machine model reciprocal per-unit system.
- $e_{fd}$  is the per-unit field voltage using the synchronous machine model reciprocal per-unit system.

The voltage-behind-transient-reactance equations are defined by

$$\frac{de_d''}{dt} = \frac{(x_q - x_q'')i_q - e_d''}{T_{q0}''},$$

$$\frac{de_q'}{dt} = \frac{E_{fd} - (x_d - x_d')i_d - e_q'}{T_{d0}'},$$

and

$$\frac{de_q''}{dt} = \frac{e_q' - (x_d' - x_d'')i_d - e_q''}{T_{d0}''},$$

where:

- $x_d$  and  $x_q$  are the  $d$ -axis and  $q$ -axis synchronous reactances.

- $T''_{d0}$  and  $T''_{q0}$  are the  $d$ -axis and  $q$ -axis subtransient open-circuit time constants.
- $E_{fd}$  is the per-unit field voltage using the exciter model nonreciprocal per-unit system.
- $x'_d$  is the  $d$ -axis transient reactance.
- $e'_q$  is the  $q$ -axis voltage behind transient reactance.
- $T'_{d0}$  is the  $d$ -axis transient open-circuit time constant.

The rotor torque is defined by

$$T_e = e''_d i_d + e''_q i_q - (x''_d - x''_q) i_d i_q.$$

These defining equations do not describe the parameters you can set in the dialog box. To see their relationship with the equation coefficients, see [1].

## Display Options

You can perform display actions using the **Electrical** menu on the block context menu.

Right-click the block and, from the **Electrical** menu, select an option:

- **Display Base Values** displays the machine per-unit base values in the MATLAB Command Window.
- **Display Associated Base Values** displays associated per-unit base values in the MATLAB Command Window.
- **Display Associated Initial Conditions** displays associated initial conditions in the MATLAB Command Window.

## Ports

### fd+

Electrical conserving port corresponding to the field winding positive terminal.

### fd-

Electrical conserving port corresponding to the field winding negative terminal.

### R

Mechanical rotational conserving port associated with the machine rotor.

**C**

Mechanical rotational conserving port associated with the machine case.

**pu**

Physical signal vector port associated with the machine per-unit measurements. The vector elements are:

- pu\_fd\_Efd
- pu\_fd\_Ifd
- pu\_torque
- pu\_velocity
- pu\_ed
- pu\_eq
- pu\_e0 — This port is provided to maintain a compatible interface for existing machine models. Its value is always zero.
- pu\_id
- pu\_iq
- pu\_i0 — This port is provided to maintain a compatible interface for existing machine models. Its value is always zero.

~

Expandable three-phase port associated with the stator windings.

**n**

Electrical conserving port associated with the neutral point of the wye winding configuration. This port is provided to maintain a compatible interface for existing machine models. The voltage and current on this port are ignored.

## Parameters

- “Main” on page 1-1792
- “Impedances” on page 1-1793
- “Initial Conditions” on page 1-1793

## **Main**

### **Rated apparent power**

Rated apparent power. The default value is 555e6 V\*A.

### **Rated voltage**

RMS rated line-line voltage. The default value is 24e3 V.

### **Rated electrical frequency**

Nominal electrical frequency at which rated apparent power is quoted. The default value is 60 Hz.

### **Number of pole pairs**

Number of machine pole pairs. The default value is 1.

### **Specify field circuit input required to produce rated terminal voltage at no load by**

Select between Field circuit voltage and Field circuit current. The default value is Field circuit current.

### **Field circuit current**

This parameter is visible only when **Specify field circuit input required to produce rated terminal voltage at no load by** is set to Field circuit current. The default value is 1300 A.

### **Field circuit voltage**

This parameter is visible only when **Specify field circuit input required to produce rated terminal voltage at no load by** is set to Field circuit voltage. The default value is 92.95 V.

### **Rotor angle definition**

Reference point for the rotor angle measurement.

The default option is Angle between the a-phase magnetic axis and the d-axis. When you select this value, the rotor *d*-axis and stator *a*-phase magnetic axis are aligned when the rotor angle is zero.

The other value you can choose for this parameter is Angle between the a-phase magnetic axis and the q-axis. When you select this value, the rotor *q*-axis and stator *a*-phase magnetic axis are aligned when the rotor angle is zero.

## Impedances

### **Stator d-axis mutual inductance (unsaturated), $L_{adu}$**

Unsaturated stator  $d$ -axis mutual inductance. The default value is 1.66 pu.

### **Stator q-axis mutual inductance (unsaturated), $L_{aqu}$**

Unsaturated stator  $q$ -axis mutual inductance. The default value is 1.61 pu.

### **Stator leakage inductance, $L_l$**

Stator leakage inductance. The default value is 0.15 pu.

### **Stator resistance, $R_a$**

Stator resistance. The default value is 0.003 pu.

### **Rotor field circuit inductance, $L_{fd}$**

Rotor field circuit inductance. The default value is 0.165 pu.

### **Rotor field circuit resistance, $R_{fd}$**

Rotor field circuit resistance. The default value is 0.0006 pu.

### **Rotor d-axis damper winding 1 inductance, $L_{1d}$**

Rotor  $d$ -axis damper winding 1 inductance. The default value is 0.1713 pu.

### **Rotor d-axis damper winding 1 resistance, $R_{1d}$**

Rotor  $d$ -axis damper winding 1 resistance. The default value is 0.0284 pu.

### **Rotor q-axis damper winding 1 inductance, $L_{1q}$**

Rotor  $q$ -axis damper winding 1 inductance. The default value is 0.1066 pu.

### **Rotor q-axis damper winding 1 resistance, $R_{1q}$**

Rotor  $q$ -axis damper winding 1 resistance. The default value is 0.0650 pu.

## Initial Conditions

### **Specify initialization by**

Select between Electrical power and voltage output and Mechanical and voltage states. The default value is Electrical power and voltage output.

### **Terminal voltage magnitude**

Initial RMS line-line voltage. This parameter is visible only when you set **Specify initialization by** to Electrical power and voltage output. The default value is 24e3 V.

## Terminal voltage angle

Initial voltage angle. This parameter is visible only when you set **Specify initialization by** to Electrical power and voltage output. The default value is 0 deg.

## Terminal active power

Initial active power. This parameter is visible only when you set **Specify initialization by** to Electrical power and voltage output. The default value is 500e6 V\*A.

## Terminal reactive power

Initial reactive power. This parameter is visible only when you set **Specify initialization by** to Electrical power and voltage output. The default value is 0 V\*A.

## Initial rotor angle

Initial rotor angle. During steady-state operation, set this parameter to the sum of the load angle and required terminal voltage offset. This parameter is visible only when you set **Specify initialization by** to Mechanical and voltage states. The default value is 0 deg.

## Initial voltage behind d-axis subtransient reactance

Initial voltage behind  $d$ -axis subtransient reactance. This parameter is visible only when you set **Specify initialization by** to Mechanical and voltage states. The default value is 0 pu.

## Initial voltage behind q-axis transient reactance

Initial voltage behind  $q$ -axis transient reactance. This parameter is visible only when you set **Specify initialization by** to Mechanical and voltage states. The default value is 0 pu.

## Initial voltage behind q-axis subtransient reactance

Initial voltage behind  $q$ -axis subtransient reactance. This parameter is visible only when you set **Specify initialization by** to Mechanical and voltage states. The default value is 0 pu.

## References

- [1] Kundur, P. *Power System Stability and Control*. New York, NY: McGraw Hill, 1993.
- [2] Lyshevski, S. E. *Electromechanical Systems, Electric Machines and Applied Mechatronics*. Boca Raton, FL: CRC Press, 1999.



[3] Pal, M. K. *Lecture Notes on Power System Stability*. <http://www.mkpalconsulting.com/files/stabilitybook.pdf>. June, 2007.

## See Also

Synchronous Machine Measurement | Synchronous Machine Model 2.1 (standard) | Synchronous Machine Round Rotor (fundamental) | Synchronous Machine Round Rotor (standard) | Synchronous Machine Salient Pole (fundamental) | Synchronous Machine Salient Pole (standard)

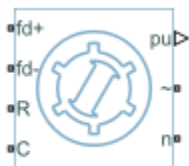
## Topics

“Expand and Collapse Three-Phase Ports on a Block”  
Three-Phase Synchronous Machine Control

**Introduced in R2015a**

## Synchronous Machine Model 2.1 (standard)

Synchronous machine with simplified transformation, simplified representation, and standard parameterization



### Library

Simscape / Electrical / Power Systems / Machines / Synchronous Machine (Simplified)

### Description

The Synchronous Machine Model 2.1 (standard) block models a synchronous machine with one field winding and one damper on the  $d$ -axis and one damper on the  $q$ -axis. You use standard parameters to define the characteristics of the machine. The block converts the standard parameters to fundamental parameters. This block contains a  $dq$  Park transformation, so use it only for balanced operation.

### Equations

The synchronous machine equations are expressed with respect to a rotating reference frame, defined by

$$\theta_e(t) = N\theta_r(t),$$

where:

- $\theta_e$  is the electrical angle.

- $N$  is the number of pole pairs.
- $\theta_r$  is the rotor angle.

The Park transformation maps the synchronous machine equations to the rotating reference frame with respect to the electrical angle. The Park transformation is defined by

$$P_s = \frac{2}{3} \begin{bmatrix} \cos \theta_e & \cos(\theta_e - \frac{2\pi}{3}) & \cos(\theta_e + \frac{2\pi}{3}) \\ -\sin \theta_e & -\sin(\theta_e - \frac{2\pi}{3}) & -\sin(\theta_e + \frac{2\pi}{3}) \end{bmatrix}.$$

The Park transformation is used to define the per-unit synchronous machine equations. The stator voltage equations are defined by

$$e_d = e_d'' - R_a i_d + x_q'' \dot{i}_q$$

and

$$e_q = e_q'' - x_d'' \dot{i}_d - R_a \dot{i}_q,$$

where:

- $e_d''$  and  $e_q''$  are the  $d$ -axis and  $q$ -axis voltages behind subtransient reactances.
- $R_a$  is the stator resistance.
- $i_d$  and  $i_q$  are the  $d$ -axis and  $q$ -axis stator currents, defined by

$$\begin{bmatrix} i_d \\ i_q \end{bmatrix} = P_s \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix}.$$

$i_a$ ,  $i_b$ , and  $i_c$  are the stator currents flowing from port  $\sim$  to port  $n$ .

- $x_d''$  and  $x_q''$  are the  $d$ -axis and  $q$ -axis subtransient reactances.
- $e_d$  and  $e_q$  are the  $d$ -axis and  $q$ -axis stator voltages, defined by

$$\begin{bmatrix} e_d \\ e_q \end{bmatrix} = P_s \begin{bmatrix} v_a \\ v_b \\ v_c \end{bmatrix}.$$

$v_a$ ,  $v_b$ , and  $v_c$  are the stator voltages measured from port  $\sim$  to neutral port n.

The rotor voltage equation is defined by

$$e_{fd} = R_{fd} \cdot i_{fd},$$

where:

- $R_{fd}$  is the resistance of rotor field circuit.
- $i_{fd}$  is the per-unit field current using the synchronous machine model reciprocal per-unit system.
- $e_{fd}$  is the per-unit field voltage using the synchronous machine model reciprocal per-unit system.

The voltage-behind-transient-reactance equations are defined by

$$\frac{de_d''}{dt} = \frac{(x_q - x_q'')i_q' - e_d''}{T_{q0}''},$$

$$\frac{de_q'}{dt} = \frac{E_{fd} - (x_d - x_d')i_d' - e_q'}{T_{d0}'},$$

and

$$\frac{de_q''}{dt} = \frac{e_q' - (x_d' - x_d'')i_d' - e_q''}{T_{d0}''},$$

where:

- $x_d$  and  $x_q$  are the  $d$ -axis and  $q$ -axis synchronous reactances.

- $T''_{d0}$  and  $T''_{q0}$  are the  $d$ -axis and  $q$ -axis subtransient open-circuit time constants.
- $E_{fd}$  is the per-unit field voltage using the exciter model nonreciprocal per-unit system.
- $x'_d$  is the  $d$ -axis transient reactance.
- $e'_q$  is the  $q$ -axis voltage behind transient reactance.
- $T'_{d0}$  is the  $d$ -axis transient open-circuit time constant.

The rotor torque is defined by

$$T_e = e''_d i_d + e''_q i_q - (x''_d - x''_q) i_d i_q.$$

These defining equations do not describe the short-circuit time constants you can set in the dialog box. To see their relationship with the equation coefficients, see [1], [4].

## Display Options

You can perform display actions using the **Electrical** menu on the block context menu.

Right-click the block and, from the **Electrical** menu, select an option:

- **Display Base Values** displays the machine per-unit base values in the MATLAB Command Window.
- **Display Associated Base Values** displays associated per-unit base values in the MATLAB Command Window.
- **Display Associated Initial Conditions** displays associated initial conditions in the MATLAB Command Window.

## Ports

### fd+

Electrical conserving port corresponding to the field winding positive terminal.

### fd-

Electrical conserving port corresponding to the field winding negative terminal.

### R

Mechanical rotational conserving port associated with the machine rotor.

**C**

Mechanical rotational conserving port associated with the machine case.

**pu**

Physical signal vector port associated with the machine per-unit measurements. The vector elements are:

- `pu_fd_Efd`
- `pu_fd_Ifd`
- `pu_torque`
- `pu_velocity`
- `pu_ed`
- `pu_eq`
- `pu_e0` — This port is provided to maintain a compatible interface for existing machine models. Its value is always zero.
- `pu_id`
- `pu_iq`
- `pu_i0` — This port is provided to maintain a compatible interface for existing machine models. Its value is always zero.

~

Expandable three-phase port associated with the stator windings.

**n**

Electrical conserving port associated with the neutral point of the wye winding configuration. This port is provided to maintain a compatible interface for existing machine models. The voltage and current on this port are ignored.

## Parameters

- “Main” on page 1-1801
- “Impedances” on page 1-1802
- “Time Constants” on page 1-1802
- “Initial Conditions” on page 1-1803

## Main

### Rated apparent power

Rated apparent power. The default value is 555e6 VA.

### Rated voltage

RMS rated line-line voltage. The default value is 24e3 V.

### Rated electrical frequency

Nominal electrical frequency at which rated apparent power is quoted. The default value is 60 Hz.

### Number of pole pairs

Number of machine pole pairs. The default value is 1.

### Specify field circuit input required to produce rated terminal voltage at no load by

Select between Field circuit voltage and Field circuit current. The default value is Field circuit current.

### Field circuit current

This parameter is visible only when **Specify field circuit input required to produce rated terminal voltage at no load by** is set to Field circuit current. The default value is 1300 A.

### Field circuit voltage

This parameter is visible only when **Specify field circuit input required to produce rated terminal voltage at no load by** is set to Field circuit voltage. The default value is 92.95 V.

### Rotor angle definition

Reference point for the rotor angle measurement.

The default option is Angle between the a-phase magnetic axis and the d-axis. When you select this value, the rotor *d*-axis and stator *a*-phase magnetic axis are aligned when the rotor angle is zero.

The other value you can choose for this parameter is Angle between the a-phase magnetic axis and the q-axis. When you select this value, the rotor *q*-axis and stator *a*-phase magnetic axis are aligned when the rotor angle is zero.

## Impedances

### Stator resistance, $R_a$

Stator resistance. The default value is 0.003 pu.

### Stator leakage reactance, $X_l$

Stator leakage reactance. The default value is 0.15 pu.

### d-axis synchronous reactance, $X_d$

*d*-axis synchronous reactance. The default value is 1.81 pu.

### q-axis synchronous reactance, $X_q$

*q*-axis synchronous reactance. The default value is 1.76 pu.

### d-axis transient reactance, $X_d'$

*d*-axis transient reactance. The default value is 0.3 pu.

### d-axis subtransient reactance, $X_d''$

*d*-axis subtransient reactance. The default value is 0.23 pu.

### q-axis subtransient reactance, $X_q''$

*q*-axis subtransient reactance. The default value is 0.25 pu.

## Time Constants

### Specify d-axis transient time constant

Select between Open-circuit value and Short-circuit value. The default value is Open-circuit value.

### d-axis transient open-circuit, $T_{d0}'$

*d*-axis transient open-circuit time constant. This parameter is visible only when **Specify d-axis transient time constant** is set to Open-circuit value. The default value is 8 s.

### d-axis transient short-circuit, $T_d'$

*d*-axis transient short-circuit time constant. This parameter is visible only when **Specify d-axis transient time constant** is set to Short-circuit value. The default value is 1.326 s.

### Specify d-axis subtransient time constant

Select between Open-circuit value and Short-circuit value. The default value is Open-circuit value.



**d-axis subtransient open-circuit, Td0''**

*d*-axis subtransient open-circuit time constant. This parameter is visible only when **Specify d-axis subtransient time constant** is set to Open-circuit value. The default value is 0.03 s.

**d-axis subtransient short-circuit, Td''**

*d*-axis subtransient short-circuit time constant. This parameter is visible only when **Specify d-axis subtransient time constant** is set to Short-circuit value. The default value is 0.023 s.

**Specify q-axis subtransient time constant**

Select between Open-circuit value and Short-circuit value. The default value is Open-circuit value.

**q-axis subtransient open-circuit, Tq0''**

*q*-axis subtransient open-circuit time constant. This parameter is visible only when **Specify q-axis subtransient time constant** is set to Open-circuit value. The default value is 0.07 s.

**q-axis subtransient short-circuit, Tq''**

*q*-axis subtransient short-circuit time constant. This parameter is visible only when **Specify q-axis subtransient time constant** is set to Short-circuit value. The default value is 0.0269 s.

**Initial Conditions****Specify initialization by**

Select between Electrical power and voltage output and Mechanical and voltage states. The default value is Electrical power and voltage output.

**Terminal voltage magnitude**

Initial RMS line-line voltage. This parameter is visible only when you set **Specify initialization by** to Electrical power and voltage output. The default value is 24e3 V.

**Terminal voltage angle**

Initial voltage angle. This parameter is visible only when you set **Specify initialization by** to Electrical power and voltage output. The default value is 0 deg.

### **Terminal active power**

Initial active power. This parameter is visible only when you set **Specify initialization by** to Electrical power and voltage output. The default value is 500e6 V\*A.

### **Terminal reactive power**

Initial reactive power. This parameter is visible only when you set **Specify initialization by** to Electrical power and voltage output. The default value is 0 V\*A.

### **Initial rotor angle**

Initial rotor angle. During steady-state operation, set this parameter to the sum of the load angle and required terminal voltage offset. This parameter is visible only when you set **Specify initialization by** to Mechanical and voltage states. The default value is 0 deg.

### **Initial voltage behind d-axis subtransient reactance**

Initial voltage behind *d*-axis subtransient reactance. This parameter is visible only when you set **Specify initialization by** to Mechanical and voltage states. The default value is 0 pu.

### **Initial voltage behind q-axis transient reactance**

Initial voltage behind *q*-axis transient reactance. This parameter is visible only when you set **Specify initialization by** to Mechanical and voltage states. The default value is 0 pu.

### **Initial voltage behind q-axis subtransient reactance**

Initial voltage behind *q*-axis subtransient reactance. This parameter is visible only when you set **Specify initialization by** to Mechanical and voltage states. The default value is 0 pu.

## **References**

- [1] Kundur, P. *Power System Stability and Control*. New York, NY: McGraw Hill, 1993.
- [2] Lyshevski, S. E. *Electromechanical Systems, Electric Machines and Applied Mechatronics*. Boca Raton, FL: CRC Press, 1999.
- [3] Pal, M. K. *Lecture Notes on Power System Stability*. <http://www.mkpalconsulting.com/files/stabilitybook.pdf>. June, 2007.

[4] Canay, I. M. "*Modelling of alternating-current machines having multiple rotor circuits.*" *IEEE Transactions on Energy Conversion* , vol. 8, no. 2, 280-296, 1993.

## See Also

Synchronous Machine Measurement | Synchronous Machine Model 2.1 (fundamental) | Synchronous Machine Round Rotor (fundamental) | Synchronous Machine Round Rotor (standard) | Synchronous Machine Salient Pole (fundamental) | Synchronous Machine Salient Pole (standard)

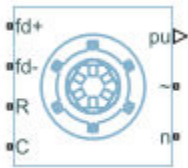
## Topics

"Expand and Collapse Three-Phase Ports on a Block"  
Three-Phase Synchronous Machine Control

**Introduced in R2015a**

## Synchronous Machine Round Rotor (fundamental)

Round-rotor synchronous machine with fundamental parameterization



### Library

Simscape / Electrical / Power Systems / Machines / Synchronous Machine (Round Rotor)

### Description

The Synchronous Machine Round Rotor (fundamental) block models a round-rotor synchronous machine using fundamental parameters.

### Equations

The synchronous machine equations are expressed with respect to a rotating reference frame, defined by

$$\theta_e(t) = N\theta_r(t),$$

where:

- $\theta_e$  is the electrical angle.
- $N$  is the number of pole pairs.
- $\theta_r$  is the rotor angle.

The Park transformation maps the synchronous machine equations to the rotating reference frame with respect to the electrical angle. The Park transformation is defined by

$$P_s = \frac{2}{3} \begin{bmatrix} \cos \theta_e & \cos(\theta_e - \frac{2\pi}{3}) & \cos(\theta_e + \frac{2\pi}{3}) \\ -\sin \theta_e & -\sin(\theta_e - \frac{2\pi}{3}) & -\sin(\theta_e + \frac{2\pi}{3}) \\ \frac{1}{2} & \frac{1}{2} & \frac{1}{2} \end{bmatrix}.$$

The Park transformation is used to define the per-unit synchronous machine equations. The stator voltage equations are defined by

$$e_d = \frac{1}{\omega_{base}} \frac{d\Psi_d}{dt} - \Psi_q \omega_r - R_a i_d,$$

$$e_q = \frac{1}{\omega_{base}} \frac{d\Psi_q}{dt} + \Psi_d \omega_r - R_a i_q,$$

and

$$e_0 = \frac{1}{\omega_{base}} \frac{d\Psi_0}{dt} - R_a i_0,$$

where:

- $e_d$ ,  $e_q$ , and  $e_0$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator voltages, defined by

$$\begin{bmatrix} e_d \\ e_q \\ e_0 \end{bmatrix} = P_s \begin{bmatrix} v_a \\ v_b \\ v_c \end{bmatrix}.$$

$v_a$ ,  $v_b$ , and  $v_c$  are the stator voltages measured from port  $\sim$  to neutral port  $\mathbf{n}$ .

- $\omega_{base}$  is the per-unit base electrical speed.

- $\psi_d$ ,  $\psi_q$ , and  $\psi_0$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator flux linkages.
- $\omega_r$  is the per-unit rotor rotational speed.
- $R_a$  is the stator resistance.
- $i_d$ ,  $i_q$ , and  $i_0$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator currents, defined by

$$\begin{bmatrix} i_d \\ i_q \\ i_0 \end{bmatrix} = P_s \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix}.$$

$i_a$ ,  $i_b$ , and  $i_c$  are the stator currents flowing from port  $\sim$  to port **n**.

The rotor voltage equations are defined by

$$e_{fd} = \frac{1}{\omega_{base}} \frac{d\Psi_{fd}}{dt} + R_{fd}i_{fd},$$

$$e_{1d} = \frac{1}{\omega_{base}} \frac{d\Psi_{1d}}{dt} + R_{1d}i_{1d} = 0,$$

$$e_{1q} = \frac{1}{\omega_{base}} \frac{d\Psi_{1q}}{dt} + R_{1q}i_{1q} = 0,$$

and

$$e_{2q} = \frac{1}{\omega_{base}} \frac{d\Psi_{2q}}{dt} + R_{2q}i_{2q} = 0,$$

where:

- $e_{fd}$  is the field voltage.
- $e_{1d}$ ,  $e_{1q}$ , and  $e_{2q}$  are the voltages across the  $d$ -axis damper winding 1,  $q$ -axis damper winding 1, and  $q$ -axis damper winding 2. They are all equal to 0.
- $\Psi_{fd}$ ,  $\Psi_{1d}$ ,  $\Psi_{1q}$ , and  $\Psi_{2q}$  are the magnetic fluxes linking the field circuit,  $d$ -axis damper winding 1,  $q$ -axis damper winding 1, and  $q$ -axis damper winding 2.

- $R_{fd}$ ,  $R_{1d}$ ,  $R_{1q}$ , and  $R_{2q}$  are the resistances of rotor field circuit,  $d$ -axis damper winding 1,  $q$ -axis damper winding 1, and  $q$ -axis damper winding 2.
- $i_{fd}$ ,  $i_{1d}$ ,  $i_{1q}$ , and  $i_{2q}$  are the currents flowing in the field circuit,  $d$ -axis damper winding 1,  $q$ -axis damper winding 1, and  $q$ -axis damper winding 2.

The saturation equations are defined by

$$\psi_{ad} = \psi_d + L_l i_d,$$

$$\psi_{aq} = \psi_q + L_l i_q,$$

$$\psi_{at} = \sqrt{\psi_{ad}^2 + \psi_{aq}^2},$$

$K_s = 1$  (If saturation is disabled),

$K_s = f(\psi_{at})$  (If saturation is enabled),

$$L_{ad} = K_s * L_{adu},$$

and

$$L_{aq} = K_s * L_{aqu},$$

where:

- $\psi_{ad}$  is the  $d$ -axis air-gap or mutual flux linkage.
- $\psi_{aq}$  is the  $q$ -axis air-gap or mutual flux linkage.
- $\psi_{at}$  is the air-gap flux linkage.
- $K_s$  is the saturation factor.
- $L_{adu}$  is the unsaturated mutual inductance of the stator  $d$ -axis.
- $L_{ad}$  is the mutual inductance of the stator  $d$ -axis.
- $L_{aqu}$  is the unsaturated mutual inductance of the stator  $q$ -axis.
- $L_{aq}$  is the mutual inductance of the stator  $q$ -axis.

The saturation factor function,  $f$ , is calculated from the per-unit open-circuit lookup table as:

$$L_{ad} = \frac{d\psi_{at}}{di_{fd}},$$

$$V_{ag} = g(i_{fd}),$$

and

$$L_{ad} = \frac{dg(i_{fd})}{di_{fd}} = \frac{dV_{ag}}{di_{fd}},$$

where  $V_{ag}$  is the per-unit air-gap voltage.

In per-unit,

$$K_s = \frac{L_{ad}}{L_{adu}},$$

and

$$\psi_{at} = V_{ag}$$

can be rearranged to

$$K_s = f(\psi_{at}).$$

The stator flux linkage equations are defined by

$$\Psi_d = -(L_{ad} + L_l)i_d + L_{ad}i_{fd} + L_{ad}\dot{i}_d,$$

$$\Psi_q = -(L_{aq} + L_l)i_q + L_{aq}\dot{i}_q + L_{aq}i_{2q},$$

and



$$\Psi_0 = -L_0 i_0,$$

where:

- $L_l$  is the stator leakage inductance.
- $L_{ad}$  and  $L_{aq}$  are the mutual inductances of the stator  $d$ -axis and  $q$ -axis.

The rotor flux linkage equations are defined by

$$\Psi_{fd} = L_{ffd} i_{fd} + L_{f1d} i_{1d} - L_{ad} i_d,$$

$$\Psi_{1d} = L_{f1d} i_{fd} + L_{11d} i_{1d} - L_{ad} i_d,$$

$$\Psi_{1q} = L_{11q} i_{1q} + L_{aq} i_{2q} - L_{aq} i_q,$$

and

$$\Psi_{2q} = L_{aq} i_{1q} + L_{22q} i_{2q} - L_{aq} i_q,$$

where:

- $L_{ffd}$  is the self-inductances of the rotor field circuit.
- $L_{ffq}$  is the self-inductance of the rotor field circuit.
- $L_{11d}$  is the self-inductance of the  $d$ -axis damper winding 1.
- $L_{11q}$  is the self-inductance of the  $q$ -axis damper winding 1.
- $L_{22q}$  is the self-inductance of the  $q$ -axis damper winding 2.
- $L_{f1d}$  is the rotor field circuit and  $d$ -axis damper winding 1 mutual inductance.

The inductances are defined by these equations:

$$L_{ffd} = L_{ad} + L_{fd}$$

$$L_{f1d} = L_{ffd} - L_{fd}$$

$$L_{11d} = L_{f1d} + L_{1d}$$

$$L_{11q} = L_{aq} + L_{1q}$$

$$L_{22q} = L_{aq} + L_{2q}$$

The inductance equations assume that per-unit mutual inductance  $L_{12q} = L_{aq}$ , that is, the stator and rotor currents in the  $q$ -axis all link a single mutual flux represented by  $L_{aq}$ .

The rotor torque is defined by

$$T_e = \Psi_d i_q - \Psi_q i_d.$$

## Plotting and Display Options

You can perform plotting and display actions using the **Electrical** menu on the block context menu.

Right-click the block and, from the **Electrical** menu, select an option:

- **Display Base Values** — Displays the machine per-unit base values in the MATLAB Command Window.
- **Display Associated Base Values** — Displays associated per-unit base values in the MATLAB Command Window.
- **Display Associated Initial Conditions** — Displays associated initial conditions in the MATLAB Command Window.
- **Plot Open-Circuit Saturation (pu)** — Plots air-gap voltage,  $V_{ag}$ , versus field current,  $i_{fd}$ , both measured in per-unit, in a MATLAB figure window. The plot contains three traces:
  - Unsaturated — **Stator d-axis mutual inductance (unsaturated)**, **Ladu** you specify
  - Saturated — **Per-unit open-circuit lookup table (Vag versus ifd)** you specify
  - Derived — Open-circuit lookup table (per-unit) derived from the **Per-unit open-circuit lookup table (Vag versus ifd)** you specify. This data is used to calculate the saturation factor,  $K_s$ , versus magnetic flux linkage,  $\psi_{at}$ , characteristic.
- **Plot Saturation Factor (pu)** — Plots saturation factor,  $K_s$ , versus magnetic flux linkage,  $\psi_{at}$ , both measured in per-unit, in a MATLAB figure window using the machine parameters. This parameter is derived from other parameters that you specify:

- **Stator d-axis mutual inductance (unsaturated),  $L_{adu}$**
- **Per-unit field current saturation data,  $ifd$**
- **Per-unit air-gap voltage saturation data,  $V_{ag}$**

## Ports

### **fd+**

Electrical conserving port corresponding to the field winding positive terminal

### **fd-**

Electrical conserving port corresponding to the field winding negative terminal

### **R**

Mechanical rotational conserving port associated with the machine rotor

### **C**

Mechanical rotational conserving port associated with the machine case

### **pu**

Physical signal vector port associated with the machine per-unit measurements. The vector elements are:

- `pu_fd_Efd`
- `pu_fd_Ifd`
- `pu_torque`
- `pu_velocity`
- `pu_ed`
- `pu_eq`
- `pu_e0`
- `pu_id`
- `pu_iq`
- `pu_i0`

### **~**

Expandable three-phase port associated with the stator windings

**n**

Electrical conserving port associated with the neutral point of the wye winding configuration

## Parameters

- “Main” on page 1-1814
- “Impedances” on page 1-1815
- “Saturation” on page 1-1816
- “Initial Conditions” on page 1-1817

### Main

#### Rated apparent power

Rated apparent power. The default value is 555e6 V\*A.

#### Rated voltage

RMS rated line-line voltage. The default value is 24e3 V.

#### Rated electrical frequency

Nominal electrical frequency at which rated apparent power is quoted. The default value is 60 Hz.

#### Number of pole pairs

Number of machine pole pairs. The default value is 1.

#### Specify field circuit input required to produce rated terminal voltage at no load by

Select one of these options:

- Field circuit voltage
- Field circuit current

The default value is Field circuit current.

#### Field circuit current

This parameter is visible only when **Specify field circuit input required to produce rated terminal voltage at no load by** is set to Field circuit current. The default value is 1300 A.

**Field circuit voltage**

This parameter is visible only when **Specify field circuit input required to produce rated terminal voltage at no load by** is set to `Field circuit voltage`. The default value is 92.95 V.

**Zero sequence**

Option to include or exclude zero-sequence terms.

- **Include** — Include zero-sequence terms. To prioritize model fidelity, use this default setting. Using this option results in an error for simulations that use the Partitioning solver. For more information, see “Increase Simulation Speed Using the Partitioning Solver” (Simscape).
- **Exclude** — Exclude zero-sequence terms. To prioritize simulation speed for desktop simulation or real-time deployment, select this option.

**Rotor angle definition**

Reference point for the rotor angle measurement.

The default option is `Angle between the a-phase magnetic axis and the d-axis`. When you select this value, the rotor *d*-axis and stator *a*-phase magnetic axis are aligned when the rotor angle is zero.

The other value you can choose for this parameter is `Angle between the a-phase magnetic axis and the q-axis`. When you select this value, the rotor *q*-axis and stator *a*-phase magnetic axis are aligned when the rotor angle is zero.

**Impedances****Stator d-axis mutual inductance (unsaturated), Ladu**

Unsaturated stator *d*-axis mutual inductance. If **Magnetic saturation representation** is set to `None`, this is equivalent to the stator *d*-axis mutual inductance. The default value is 1.66 pu.

**Stator q-axis mutual inductance (unsaturated), Laqu**

Unsaturated stator *q*-axis mutual inductance. If **Magnetic saturation representation** is set to `None`, this is equivalent to the stator *q*-axis mutual inductance. The default value is 1.61 pu.

**Stator zero-sequence inductance, L0**

Stator zero-sequence inductance. The default value is 0.15 pu.

**Stator leakage inductance, L<sub>l</sub>**

Stator leakage inductance. The default value is 0.15 pu.

**Stator resistance, R<sub>a</sub>**

Stator resistance. The default value is 0.003 pu.

**Rotor field circuit inductance, L<sub>fd</sub>**

Rotor field circuit inductance. The default value is 0.165 pu.

**Rotor field circuit resistance, R<sub>fd</sub>**

Rotor field circuit resistance. The default value is 0.0006 pu.

**Rotor d-axis damper winding 1 inductance, L<sub>1d</sub>**

Rotor *d*-axis damper winding 1 inductance. The default value is 0.1713 pu.

**Rotor d-axis damper winding 1 resistance, R<sub>1d</sub>**

Rotor *d*-axis damper winding 1 resistance. The default value is 0.0284 pu.

**Rotor q-axis damper winding 1 inductance, L<sub>1q</sub>**

Rotor *q*-axis damper winding 1 inductance. The default value is 0.7252 pu.

**Rotor q-axis damper winding 1 resistance, R<sub>1q</sub>**

Rotor *q*-axis damper winding 1 resistance. The default value is 0.00619 pu.

**Rotor q-axis damper winding 2 inductance, L<sub>2q</sub>**

Rotor *q*-axis damper winding 2 inductance. The default value is 0.125 pu.

**Rotor q-axis damper winding 2 resistance, R<sub>2q</sub>**

Rotor *q*-axis damper winding 2 resistance. The default value is 0.02368 pu.

## Saturation

**Magnetic saturation representation**

Block magnetic saturation representation. Options are:

- None
- Per-unit open-circuit lookup table ( $V_{ag}$  versus  $i_{fd}$ )

The default value is None.

**Per-unit field current saturation data, ifd**

Field current,  $i_{fd}$ , data that populates the air-gap voltage,  $V_{ag}$ , versus field current,  $i_{fd}$ , lookup table. This parameter is visible only when you set **Magnetic saturation**

**representation** to Per-unit open-circuit lookup table ( $V_{ag}$  versus  $i_{fd}$ ). This parameter must contain a vector with at least five elements. The default value is [0.00, 0.48, 0.76, 1.38, 1.79] pu.

#### **Per-unit air-gap voltage saturation data, $V_{ag}$**

Air-gap voltage,  $V_{ag}$ , data that populates the air-gap voltage,  $V_{ag}$ , versus field current,  $i_{fd}$ , lookup table. This parameter is visible only when you set **Magnetic saturation representation** to Per-unit open-circuit lookup table ( $V_{ag}$  versus  $i_{fd}$ ). This parameter must contain a vector with at least five elements. The default value is [0.00, 0.80, 1.08, 1.31, 1.40] pu.

## **Initial Conditions**

### **Specify initialization by**

Initialization method. Select one of these options:

- Electrical power and voltage output, the default value
- Mechanical and magnetic states

### **Terminal voltage magnitude**

Initial RMS line-line voltage. This parameter is visible only when you set **Specify initialization by** to Electrical power and voltage output. The default value is 24e3 V.

### **Terminal voltage angle**

Initial voltage angle. This parameter is visible only when you set **Specify initialization by** to Electrical power and voltage output. The default value is 0 deg.

### **Terminal active power**

Initial active power. This parameter is visible only when you set **Specify initialization by** to Electrical power and voltage output. The default value is 500e6 V\*A.

### **Terminal reactive power**

Initial reactive power. This parameter is visible only when you set **Specify initialization by** to Electrical power and voltage output. The default value is 0 V\*A.

### **Initial rotor angle**

Initial rotor angle. During steady-state operation, set this parameter to the sum of the load angle and required terminal voltage offset. This parameter is visible only when

you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 deg.

### **Initial stator d-axis magnetic flux linkage**

Stator  $d$ -axis initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

### **Initial stator q-axis magnetic flux linkage**

Stator  $q$ -axis initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

### **Initial stator zero-sequence magnetic flux linkage**

Zero-sequence initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

### **Initial field circuit magnetic flux linkage**

Field circuit initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

### **Initial d-axis damper winding 1 magnetic flux linkage**

$d$ -axis damper winding 1 initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

### **Initial q-axis damper winding 1 magnetic flux linkage**

$q$ -axis damper winding 1 initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

### **Initial q-axis damper winding 2 magnetic flux linkage**

$q$ -axis damper winding 2 initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

## **References**

[1] Kundur, P. *Power System Stability and Control*. New York, NY: McGraw Hill, 1993.



[2] Lyshevski, S. E. *Electromechanical Systems, Electric Machines and Applied Mechatronics*. Boca Raton, FL: CRC Press, 1999.

## See Also

Synchronous Machine Measurement | Synchronous Machine Model 2.1 (fundamental) | Synchronous Machine Model 2.1 (standard) | Synchronous Machine Round Rotor (standard) | Synchronous Machine Salient Pole (fundamental) | Synchronous Machine Salient Pole (standard)

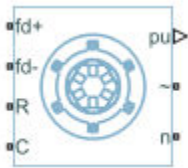
## Topics

“Expand and Collapse Three-Phase Ports on a Block”  
Three-Phase Synchronous Machine Control

**Introduced in R2013b**

## Synchronous Machine Round Rotor (standard)

Round-rotor synchronous machine with standard parameterization



### Library

Simscape / Electrical / Power Systems / Machines / Synchronous Machine (Round Rotor)

### Description

The Synchronous Machine Round Rotor (standard) block models a round-rotor synchronous machine using standard parameters.

### Equations

The synchronous machine equations are expressed with respect to a rotating reference frame, defined by

$$\theta_e(t) = N\theta_r(t),$$

where:

- $\theta_e$  is the electrical angle.
- $N$  is the number of pole pairs.
- $\theta_r$  is the rotor angle.

The Park transformation maps the synchronous machine equations to the rotating reference frame with respect to the electrical angle. The Park transformation is defined by

$$P_s = \frac{2}{3} \begin{bmatrix} \cos \theta_e & \cos(\theta_e - \frac{2\pi}{3}) & \cos(\theta_e + \frac{2\pi}{3}) \\ -\sin \theta_e & -\sin(\theta_e - \frac{2\pi}{3}) & -\sin(\theta_e + \frac{2\pi}{3}) \\ \frac{1}{2} & \frac{1}{2} & \frac{1}{2} \end{bmatrix}.$$

The Park transformation is used to define the per-unit synchronous machine equations. The stator voltage equations are defined by

$$e_d = \frac{1}{\omega_{base}} \frac{d\Psi_d}{dt} - \Psi_q \omega_r - R_a i_d,$$

$$e_q = \frac{1}{\omega_{base}} \frac{d\Psi_q}{dt} + \Psi_d \omega_r - R_a i_q,$$

and

$$e_0 = \frac{1}{\omega_{base}} \frac{d\Psi_0}{dt} - R_a i_0,$$

where:

- $e_d$ ,  $e_q$ , and  $e_0$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator voltages, defined by

$$\begin{bmatrix} e_d \\ e_q \\ e_0 \end{bmatrix} = P_s \begin{bmatrix} v_a \\ v_b \\ v_c \end{bmatrix}.$$

where  $v_a$ ,  $v_b$ , and  $v_c$  are the stator voltages measured from port  $\sim$  to neutral port  $\mathbf{n}$ .

- $\omega_{base}$  is the per-unit base electrical speed.

- $\psi_d$ ,  $\psi_q$ , and  $\psi_0$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator flux linkages.
- $\omega_r$  is the per-unit rotor rotational speed.
- $R_a$  is the stator resistance.
- $i_d$ ,  $i_q$ , and  $i_0$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator currents, defined by

$$\begin{bmatrix} i_d \\ i_q \\ i_0 \end{bmatrix} = P_s \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix}.$$

where  $i_a$ ,  $i_b$ , and  $i_c$  are the stator currents flowing from port  $\sim$  to port  $\mathbf{n}$ .

The rotor voltage equations are defined by

$$e_{fd} = \frac{1}{\omega_{base}} \frac{d\Psi_{fd}}{dt} + R_{fd}i_{fd},$$

$$e_{1d} = \frac{1}{\omega_{base}} \frac{d\Psi_{1d}}{dt} + R_{1d}i_{1d} = 0,$$

$$e_{1q} = \frac{1}{\omega_{base}} \frac{d\Psi_{1q}}{dt} + R_{1q}i_{1q} = 0,$$

and

$$e_{2q} = \frac{1}{\omega_{base}} \frac{d\Psi_{2q}}{dt} + R_{2q}i_{2q} = 0,$$

where:

- $e_{fd}$  is the field voltage.
- $e_{1d}$ ,  $e_{1q}$ , and  $e_{2q}$  are the voltages across the  $d$ -axis damper winding 1,  $q$ -axis damper winding 1, and  $q$ -axis damper winding 2. They are all equal to 0.
- $\Psi_{fd}$ ,  $\Psi_{1d}$ ,  $\Psi_{1q}$ , and  $\Psi_{2q}$  are the magnetic fluxes linking the field circuit,  $d$ -axis damper winding 1,  $q$ -axis damper winding 1, and  $q$ -axis damper winding 2.

- $R_{fd}$ ,  $R_{1d}$ ,  $R_{1q}$ , and  $R_{2q}$  are the resistances of rotor field circuit,  $d$ -axis damper winding 1,  $q$ -axis damper winding 1, and  $q$ -axis damper winding 2.
- $i_{fd}$ ,  $i_{1d}$ ,  $i_{1q}$ , and  $i_{2q}$  are the currents flowing in the field circuit,  $d$ -axis damper winding 1,  $q$ -axis damper winding 1, and  $q$ -axis damper winding 2.

The saturation equations are defined by

$$\psi_{ad} = \psi_d + L_l i_d,$$

$$\psi_{aq} = \psi_q + L_l i_q,$$

$$\psi_{at} = \sqrt{\psi_{ad}^2 + \psi_{aq}^2},$$

$K_s = 1$  (If saturation is disabled),

$K_s = f(\psi_{at})$  (If saturation is enabled),

$$L_{ad} = K_s * L_{adu},$$

and

$$L_{aq} = K_s * L_{aqu},$$

where:

- $\psi_{ad}$  is the  $d$ -axis air-gap or mutual flux linkage.
- $\psi_{aq}$  is the  $q$ -axis air-gap or mutual flux linkage.
- $\psi_{at}$  is the air-gap flux linkage.
- $K_s$  is the saturation factor.
- $L_{adu}$  is the unsaturated mutual inductance of the stator  $d$ -axis.
- $L_{ad}$  is the mutual inductance of the stator  $d$ -axis.
- $L_{aqu}$  is the unsaturated mutual inductance of the stator  $q$ -axis.
- $L_{aq}$  is the mutual inductance of the stator  $q$ -axis.

The saturation factor function,  $f$ , is calculated from the per-unit open-circuit lookup table as:

$$L_{ad} = \frac{d\psi_{at}}{di_{fd}},$$

$$V_{ag} = g(i_{fd}),$$

and

$$L_{ad} = \frac{dg(i_{fd})}{di_{fd}} = \frac{dV_{ag}}{di_{fd}},$$

where  $V_{ag}$  is the per-unit air-gap voltage.

In per-unit,

$$K_s = \frac{L_{ad}}{L_{adu}},$$

and

$$\psi_{at} = V_{ag}$$

can be rearranged to

$$K_s = f(\psi_{at}).$$

The stator flux linkage equations are defined by

$$\Psi_d = -(L_{ad} + L_l)i_d + L_{ad}i_{fd} + L_{ad}i_{1d},$$

$$\Psi_q = -(L_{aq} + L_l)i_q + L_{aq}i_{1q} + L_{aq}i_{2q},$$

and

$$\Psi_0 = -L_0 i_0,$$

where:

- $L_l$  is the stator leakage inductance.
- $L_{ad}$  and  $L_{aq}$  are the mutual inductances of the stator  $d$ -axis and  $q$ -axis.

The rotor flux linkage equations are defined by

$$\Psi_{fd} = L_{ffd} i_{fd} + L_{f1d} i_{1d} - L_{ad} i_d,$$

$$\Psi_{1d} = L_{f1d} i_{fd} + L_{11d} i_{1d} - L_{ad} i_d,$$

$$\Psi_{1q} = L_{11q} i_{1q} + L_{aq} i_{2q} - L_{aq} i_q,$$

and

$$\Psi_{2q} = L_{aq} i_{1q} + L_{22q} i_{2q} - L_{aq} i_q,$$

where:

- $L_{ffd}$  is the self-inductances of the rotor field circuit.
- $L_{ffq}$  is the self-inductance of the rotor field circuit.
- $L_{11d}$  is the self-inductance of the  $d$ -axis damper winding 1.
- $L_{11q}$  is the self-inductance of the  $q$ -axis damper winding 1.
- $L_{22q}$  is the self-inductance of the  $q$ -axis damper winding 2.
- $L_{f1d}$  is the rotor field circuit and  $d$ -axis damper winding 1 mutual inductance.

The inductances are defined by the these equations:

$$L_{ffd} = L_{ad} + L_{fd}$$

$$L_{f1d} = L_{ffd} - L_{fd}$$

$$L_{11d} = L_{f1d} + L_{1d}$$

$$L_{11q} = L_{aq} + L_{1q}$$

$$L_{22q} = L_{aq} + L_{2q}$$

The inductance equations assume that per-unit mutual inductance  $L_{12q} = L_{aq}$ , that is, the stator and rotor currents in the  $q$ -axis all link a single mutual flux represented by  $L_{aq}$ .

The rotor torque is defined by

$$T_e = \Psi_d i_q - \Psi_q i_d.$$

These equations do not describe the time constants you can set in the dialog box. To see their relationship with the equation coefficients, see [1], [3].

## Plotting and Display Options

You can perform plotting and display actions using the **Electrical** menu on the block context menu.

Right-click the block and, from the **Electrical** menu, select an option:

- **Display Base Values** — Displays the machine per-unit base values in the MATLAB Command Window.
- **Display Associated Base Values** — Displays associated per-unit base values in the MATLAB Command Window.
- **Display Associated Initial Conditions** — Displays associated initial conditions in the MATLAB Command Window.
- **Plot Open-Circuit Saturation (pu)** — Plots air-gap voltage,  $V_{ag}$ , versus field current,  $i_{fd}$ , both measured in per-unit, in a MATLAB figure window. The plot contains three traces:
  - Unsaturated — **Stator d-axis mutual inductance (unsaturated)**, **Ladu** you specify
  - Saturated — **Per-unit open-circuit lookup table (Vag versus ifd)** you specify
  - Derived — Open-circuit lookup table (per-unit) derived from the **Per-unit open-circuit lookup table (Vag versus ifd)** you specify. This data is used to calculate the saturation factor,  $K_s$ , versus magnetic flux linkage,  $\psi_{at}$ , characteristic.



- **Plot Saturation Factor (pu)** — Plots saturation factor,  $K_s$ , versus magnetic flux linkage,  $\psi_{at}$ , both measured in per-unit, in a MATLAB figure window using the present machine parameters. This parameter is derived from other parameters that you specify:
  - **Stator d-axis mutual inductance (unsaturated),  $L_{adu}$**
  - **Per-unit field current saturation data,  $ifd$**
  - **Per-unit air-gap voltage saturation data,  $V_{ag}$**

## Ports

### fd+

Electrical conserving port corresponding to the field winding positive terminal

### fd-

Electrical conserving port corresponding to the field winding negative terminal

### R

Mechanical rotational conserving port associated with the machine rotor

### C

Mechanical rotational conserving port associated with the machine case

### pu

Physical signal vector port associated with the machine per-unit measurements. The vector elements are:

- pu\_fd\_Efd
- pu\_fd\_Ifd
- pu\_torque
- pu\_velocity
- pu\_ed
- pu\_eq
- pu\_e0
- pu\_id
- pu\_iq

- pu\_i0

~

Expandable three-phase port associated with the stator windings

**n**

Electrical conserving port associated with the neutral point of the wye winding configuration

## Parameters

- “Main” on page 1-1828
- “Impedances” on page 1-1829
- “Time Constants” on page 1-1830
- “Saturation” on page 1-1831
- “Initial Conditions” on page 1-1832

### Main

#### Rated apparent power

Rated apparent power. The default value is 555e6 V\*A.

#### Rated voltage

RMS rated line-line voltage. The default value is 24e3 V.

#### Rated electrical frequency

Nominal electrical frequency at which rated apparent power is quoted. The default value is 60 Hz.

#### Number of pole pairs

Number of machine pole pairs. The default value is 1.

#### Specify field circuit input required to produce rated terminal voltage at no load by

Choose between Field circuit voltage and Field circuit current. The default value is Field circuit current.

**Field circuit current**

Field circuit current. This parameter is visible only when **Specify field circuit input required to produce rated terminal voltage at no load by** is set to Field circuit current. The default value is 1300 A.

**Field circuit voltage**

Field circuit voltage. This parameter is visible only when **Specify field circuit input required to produce rated terminal voltage at no load by** is set to Field circuit voltage. The default value is 92.95 V.

**Zero sequence**

Option to include or exclude zero-sequence terms.

- **Include** — Include zero-sequence terms. To prioritize model fidelity, use this default setting. Using this option results in an error for simulations that use the Partitioning solver. For more information, see “Increase Simulation Speed Using the Partitioning Solver” (Simscape).
- **Exclude** — Exclude zero-sequence terms. To prioritize simulation speed for desktop simulation or real-time deployment, select this option.

**Rotor angle definition**

Reference point for the rotor angle measurement.

The default option is **Angle between the a-phase magnetic axis and the d-axis**. When you select this value, the rotor *d*-axis and stator *a*-phase magnetic axis are aligned when the rotor angle is zero.

The other value you can choose for this parameter is **Angle between the a-phase magnetic axis and the q-axis**. When you select this value, the rotor *q*-axis and stator *a*-phase magnetic axis are aligned when the rotor angle is zero.

**Impedances****Stator resistance,  $R_a$** 

Stator resistance. The default value is 0.003 pu.

**Stator leakage reactance,  $X_l$** 

Stator leakage reactance. The default value is 0.15 pu.

***d*-axis synchronous reactance,  $X_d$** 

*d*-axis synchronous reactance. The default value is 1.81 pu.

**q-axis synchronous reactance,  $X_q$**

*q*-axis synchronous reactance. The default value is 1.76 pu.

**zero-sequence reactance,  $X_0$**

Zero-sequence reactance. The default value is 0.15 pu.

**d-axis transient reactance,  $X_d'$**

*d*-axis transient reactance. The default value is 0.3 pu.

**q-axis transient reactance,  $X_q'$**

*q*-axis transient reactance. The default value is 0 pu.

**d-axis subtransient reactance,  $X_d''$**

*d*-axis subtransient reactance. The default value is 0.23 pu.

**q-axis subtransient reactance,  $X_q''$**

*q*-axis subtransient reactance. The default value is 0.25 pu.

## Time Constants

**Specify d-axis transient time constant**

Select between `Open-circuit value` and `Short-circuit value`. The default value is `Open-circuit value`.

**d-axis transient open-circuit,  $T_{d0}'$**

*d*-axis transient open-circuit time constant. This parameter is visible only when **Specify d-axis transient time constant** is set to `Open-circuit value`. The default value is 8 s.

**d-axis transient short-circuit,  $T_d'$**

*d*-axis transient short-circuit time constant. This parameter is visible only when **Specify d-axis transient time constant** is set to `Short-circuit value`. The default value is 1.326 s.

**Specify d-axis subtransient time constant**

Select between `Open-circuit value` and `Short-circuit value`. The default value is `Open-circuit value`.

**d-axis subtransient open-circuit,  $T_{d0}''$**

*d*-axis subtransient open-circuit time constant. This parameter is visible only when **Specify d-axis subtransient time constant** is set to `Open-circuit value`. The default value is 0.03 s.

**d-axis subtransient short-circuit,  $T_d''$** 

$d$ -axis subtransient short-circuit time constant. This parameter is visible only when **Specify d-axis subtransient time constant** is set to Short-circuit value. The default value is 0.023 s.

**Specify q-axis transient time constant**

Select between Open-circuit value and Short-circuit value. The default value is Open-circuit value.

**q-axis transient open-circuit,  $T_{q0}'$** 

$q$ -axis transient open-circuit time constant. This parameter is visible only when **Specify q-axis transient time constant** is set to Open-circuit value. The default value is 1 s.

**q-axis transient short-circuit,  $T_q'$** 

$q$ -axis transient short-circuit time constant. This parameter is visible only when **Specify q-axis transient time constant** is set to Short-circuit value. The default value is 0.3693 s.

**Specify q-axis subtransient time constant**

Select between Open-circuit value and Short-circuit value. The default value is Open-circuit value.

**q-axis subtransient open-circuit,  $T_{q0}''$** 

$q$ -axis subtransient open-circuit time constant. This parameter is visible only when **Specify q-axis subtransient time constant** is set to Open-circuit value. The default value is 0.07 s.

**q-axis subtransient short-circuit,  $T_q''$** 

$q$ -axis subtransient short-circuit time constant. This parameter is visible only when **Specify q-axis subtransient time constant** is set to Short-circuit value. The default value is 0.0269 s.

**Saturation****Magnetic saturation representation**

Block magnetic saturation representation. Options are:

- None
- Per-unit open-circuit lookup table (Vag versus ifd)

The default value is None.

### **Per-unit field current saturation data, ifd**

Field current,  $i_{fd}$ , data that populates the air-gap voltage,  $V_{ag}$ , versus field current,  $i_{fd}$ , lookup table. This parameter is visible only when you set **Magnetic saturation representation** to Per-unit open-circuit lookup table (Vag versus ifd). This parameter must contain a vector with at least five elements. The default value is [0.00, 0.48, 0.76, 1.38, 1.79] pu.

### **Per-unit air-gap voltage saturation data, Vag**

Air-gap voltage,  $V_{ag}$ , data that populates the air-gap voltage,  $V_{ag}$ , versus field current,  $i_{fd}$ , lookup table. This parameter is visible only when you set **Magnetic saturation representation** to Per-unit open-circuit lookup table (Vag versus ifd). This parameter must contain a vector with at least five elements. The default value is [0.00, 0.80, 1.08, 1.31, 1.40] pu.

## **Initial Conditions**

### **Specify initialization by**

Initialization method. Select one of these options:

- Electrical power and voltage output, the default value
- Mechanical and magnetic states

### **Terminal voltage magnitude**

Initial RMS line-line voltage. This parameter is visible only when you set **Specify initialization by** to Electrical power and voltage output. The default value is 24e3 V.

### **Terminal voltage angle**

Initial voltage angle. This parameter is visible only when you set **Specify initialization by** to Electrical power and voltage output. The default value is 0 deg.

### **Terminal active power**

Initial active power. This parameter is visible only when you set **Specify initialization by** to Electrical power and voltage output. The default value is 500e6 V\*A.

### **Terminal reactive power**

Initial reactive power. This parameter is visible only when you set **Specify initialization by** to Electrical power and voltage output. The default value is 0 V\*A.

**Initial rotor angle**

Initial rotor angle. During steady-state operation, set this parameter to the sum of the load angle and required terminal voltage offset. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 deg.

**Initial stator d-axis magnetic flux linkage**

Stator *d*-axis initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

**Initial stator q-axis magnetic flux linkage**

Stator *q*-axis initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

**Initial stator zero-sequence magnetic flux linkage**

Zero-sequence initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

**Initial field circuit magnetic flux linkage**

Field circuit initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

**Initial d-axis damper winding 1 magnetic flux linkage**

*d*-axis damper winding 1 initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

**Initial q-axis damper winding 1 magnetic flux linkage**

*q*-axis damper winding 1 initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

**Initial q-axis damper winding 2 magnetic flux linkage**

*q*-axis damper winding 2 initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

## References

- [1] Kundur, P. *Power System Stability and Control*. New York, NY: McGraw Hill, 1993.
- [2] Lyshevski, S. E. *Electromechanical Systems, Electric Machines and Applied Mechatronics*. Boca Raton, FL: CRC Press, 1999.
- [3] Canay, I. M. "Modelling of alternating-current machines having multiple rotor circuits." *IEEE Transactions on Energy Conversion* , vol. 8, no. 2, 280-296, 1993.

## See Also

Synchronous Machine Measurement | Synchronous Machine Model 2.1 (fundamental) | Synchronous Machine Model 2.1 (standard) | Synchronous Machine Round Rotor (fundamental) | Synchronous Machine Salient Pole (fundamental) | Synchronous Machine Salient Pole (standard)

## Topics

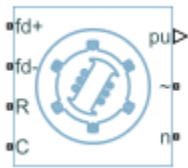
"Expand and Collapse Three-Phase Ports on a Block"  
"Marine Full Electric Propulsion Power System"  
"Three-Phase Synchronous Machine Control"  
"Three-Phase Synchronous Machine Governor Control Design"

## Introduced in R2013b



# Synchronous Machine Salient Pole (fundamental)

Salient-pole synchronous machine with fundamental parameterization



## Library

Simscape / Electrical / Power Systems / Machines / Synchronous Machine (Salient Pole)

## Description

The Synchronous Machine Salient Pole (fundamental) block models a salient-pole synchronous machine using fundamental parameters.

## Equations

The synchronous machine equations are expressed with respect to a rotating reference frame, defined by

$$\theta_e(t) = N\theta_r(t),$$

where:

- $\theta_e$  is the electrical angle.
- $N$  is the number of pole pairs.
- $\theta_r$  is the rotor angle.

The Park transformation maps the synchronous machine equations to the rotating reference frame with respect to the electrical angle. The Park transformation is defined by

$$P_s = \frac{2}{3} \begin{bmatrix} \cos \theta_e & \cos(\theta_e - \frac{2\pi}{3}) & \cos(\theta_e + \frac{2\pi}{3}) \\ -\sin \theta_e & -\sin(\theta_e - \frac{2\pi}{3}) & -\sin(\theta_e + \frac{2\pi}{3}) \\ \frac{1}{2} & \frac{1}{2} & \frac{1}{2} \end{bmatrix}.$$

The Park transformation is used to define the per-unit synchronous machine equations. The stator voltage equations are defined by

$$e_d = \frac{1}{\omega_{base}} \frac{d\Psi_d}{dt} - \Psi_q \omega_r - R_a i_d,$$

$$e_q = \frac{1}{\omega_{base}} \frac{d\Psi_q}{dt} + \Psi_d \omega_r - R_a i_q,$$

and

$$e_0 = \frac{1}{\omega_{base}} \frac{d\Psi_0}{dt} - R_a i_0,$$

where:

- $e_d$ ,  $e_q$ , and  $e_0$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator voltages, defined by

$$\begin{bmatrix} e_d \\ e_q \\ e_0 \end{bmatrix} = P_s \begin{bmatrix} v_a \\ v_b \\ v_c \end{bmatrix}.$$

$v_a$ ,  $v_b$ , and  $v_c$  are the stator voltages measured from port  $\sim$  to neutral port  $\mathbf{n}$ .

- $\omega_{base}$  is the per-unit base electrical speed.

- $\psi_d$ ,  $\psi_q$ , and  $\psi_0$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator flux linkages.
- $\omega_r$  is the per-unit rotor rotational speed.
- $R_a$  is the stator resistance.
- $i_d$ ,  $i_q$ , and  $i_0$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator currents, defined by

$$\begin{bmatrix} i_d \\ i_q \\ i_0 \end{bmatrix} = P_s \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix}.$$

$i_a$ ,  $i_b$ , and  $i_c$  are the stator currents flowing from port  $\sim$  to port  $n$ .

The rotor voltage equations are defined by

$$e_{fd} = \frac{1}{\omega_{base}} \frac{d\Psi_{fd}}{dt} + R_{fd}i_{fd},$$

$$e_{1d} = \frac{1}{\omega_{base}} \frac{d\Psi_{1d}}{dt} + R_{1d}i_{1d} = 0,$$

and

$$e_{1q} = \frac{1}{\omega_{base}} \frac{d\Psi_{1q}}{dt} + R_{1q}i_{1q} = 0,$$

where:

- $e_{fd}$  is the field voltage.
- $e_{1d}$ , and  $e_{1q}$  are the voltages across the  $d$ -axis damper winding 1 and  $q$ -axis damper winding 1. They are equal to 0.
- $\Psi_{fd}$ ,  $\Psi_{1d}$ , and  $\Psi_{1q}$  are the magnetic fluxes linking the field circuit,  $d$ -axis damper winding 1, and  $q$ -axis damper winding 1.
- $R_{fd}$ ,  $R_{1d}$ , and  $R_{1q}$  are the resistances of rotor field circuit,  $d$ -axis damper winding 1, and  $q$ -axis damper winding 1.
- $i_{fd}$ ,  $i_{1d}$ , and  $i_{1q}$  are the currents flowing in the field circuit,  $d$ -axis damper winding 1, and  $q$ -axis damper winding 1.

The saturation equations are defined by

$$\psi_{ad} = \psi_d + L_l i_d,$$

$$\psi_{aq} = \psi_q + L_l i_q,$$

$$\psi_{at} = \sqrt{\psi_{ad}^2 + \psi_{aq}^2},$$

$K_s = 1$  (If saturation is disabled),

$K_s = f(\psi_{at})$  (If saturation is enabled),

and

$$L_{ad} = K_s * L_{adu},$$

where:

- $\psi_{ad}$  is the  $d$ -axis air-gap or mutual flux linkage.
- $\psi_{aq}$  is the  $q$ -axis air-gap or mutual flux linkage.
- $\psi_{at}$  is the air-gap flux linkage.
- $K_s$  is the saturation factor.
- $L_{adu}$  is the unsaturated mutual inductance of the stator  $d$ -axis.
- $L_{ad}$  is the mutual inductance of the stator  $d$ -axis.

The saturation factor function,  $f$ , is calculated from the per-unit open-circuit lookup table as:

$$L_{ad} = \frac{d\psi_{at}}{di_{fd}},$$

$$V_{ag} = g(i_{fd}),$$

and

$$L_{ad} = \frac{dg(i_{fd})}{di_{fd}} = \frac{dV_{ag}}{di_{fd}},$$

where:

- $V_{ag}$  is the per-unit air-gap voltage.

In per-unit,

$$K_s = \frac{L_{ad}}{L_{adu}},$$

and

$$\psi_{at} = V_{ag}$$

can be rearranged to

$$K_s = f(\psi_{at}).$$

The stator flux linkage equations are defined by

$$\Psi_d = -(L_{ad} + L_l)i_d + L_{ad}i_{fd} + L_{ad}\dot{i}_d,$$

$$\Psi_q = -(L_{aq} + L_l)i_q + L_{aq}\dot{i}_q,$$

and

$$\Psi_0 = -L_0\dot{i}_0,$$

where:

- $L_l$  is the stator leakage inductance.
- $L_{ad}$  and  $L_{aq}$  are the mutual inductances of the stator  $d$ -axis and  $q$ -axis.

The rotor flux linkage equations are defined by

$$\psi_{fd} = L_{ffd}i_{fd} + L_{f1d}i_{1d} - L_{ad}i_d,$$

$$\psi_{1d} = L_{f1d}i_{fd} + L_{11d}i_{1d} - L_{ad}i_d,$$

and

$$\psi_{1q} = L_{11q}i_{1q} - L_{aq}i_q,$$

where:

- $L_{ffd}$  is the self-inductance of the rotor field circuit.
- $L_{11d}$  is the self-inductance of the  $d$ -axis damper winding 1.
- $L_{11q}$  is the self-inductance of the  $q$ -axis damper winding 1.
- $L_{f1d}$  is the rotor field circuit and  $d$ -axis damper winding 1 mutual inductance.

The inductances are defined by these equations:

$$L_{ffd} = L_{ad} + L_{fd}$$

$$L_{f1d} = L_{ffd} - L_{fd}$$

$$L_{11d} = L_{f1d} + L_{1d}$$

$$L_{11q} = L_{aq} + L_{1q}$$

The inductance equations assume that per-unit mutual inductance  $L_{12q} = L_{aq}$ , that is, the stator and rotor currents in the  $q$ -axis all link a single mutual flux represented by  $L_{aq}$ .

The rotor torque is defined by

$$T_e = \Psi_d i_q - \Psi_q i_d.$$

## Plotting and Display Options

You can perform plotting and display actions using the **Electrical** menu on the block context menu.

Right-click the block and, from the **Electrical** menu, select one of these options:

- **Display Base Values** — Displays the machine per-unit base values in the MATLAB Command Window.
- **Display Associated Base Values** — Displays associated per-unit base values in the MATLAB Command Window.
- **Display Associated Initial Conditions** — Displays associated initial conditions in the MATLAB Command Window.
- **Plot Open-Circuit Saturation (pu)** — Plots air-gap voltage,  $V_{ag}$ , versus field current,  $i_{fd}$ , both measured in per-unit, in a MATLAB figure window. The plot contains three traces:
  - Unsaturated — **Stator d-axis mutual inductance (unsaturated),  $L_{ad}$**  you specify
  - Saturated — **Per-unit open-circuit lookup table (Vag versus ifd)** you specify
  - Derived — Open-circuit lookup table (per-unit) derived from the **Per-unit open-circuit lookup table (Vag versus ifd)** you specify. This data is used to calculate the saturation factor,  $K_s$ , versus magnetic flux linkage,  $\psi_{at}$ , characteristic.
- **Plot Saturation Factor (pu)** — Plots saturation factor,  $K_s$ , versus magnetic flux linkage,  $\psi_{at}$ , both measured in per-unit, in a MATLAB figure window using the present machine parameters. This parameter is derived from other parameters that you specify:
  - **Stator d-axis mutual inductance (unsaturated),  $L_{ad}$**
  - **Per-unit field current saturation data, ifd**
  - **Per-unit air-gap voltage saturation data, Vag**

## Ports

### fd+

Electrical conserving port corresponding to the field winding positive terminal

### fd-

Electrical conserving port corresponding to the field winding negative terminal

### R

Mechanical rotational conserving port associated with the machine rotor

## C

Mechanical rotational conserving port associated with the machine case

## pu

Physical signal vector port associated with the machine per-unit measurements. The vector elements are:

- pu\_fd\_Efd
- pu\_fd\_Ifd
- pu\_torque
- pu\_velocity
- pu\_ed
- pu\_eq
- pu\_e0
- pu\_id
- pu\_iq
- pu\_i0

~

Expandable three-phase port associated with the stator windings

## n

Electrical conserving port associated with the neutral point of the wye winding configuration

## Parameters

- “Main” on page 1-1843
- “Impedances” on page 1-1844
- “Saturation” on page 1-1845
- “Initial Conditions” on page 1-1845



## Main

### Rated apparent power

Rated apparent power. The default value is  $300e6 \text{ V}\cdot\text{A}$ .

### Rated voltage

RMS rated line-line voltage. The default value is  $24e3 \text{ V}$ .

### Rated electrical frequency

Nominal electrical frequency at which rated apparent power is quoted. The default value is  $60 \text{ Hz}$ .

### Number of pole pairs

Number of machine pole pairs. The default value is  $10$ .

### Specify field circuit input required to produce rated terminal voltage at no load by

Choose between `Field circuit voltage` and `Field circuit current`. The default value is `Field circuit current`.

### Field circuit current

This parameter is visible only when **Specify field circuit input required to produce rated terminal voltage at no load by** is set to `Field circuit current`. The default value is  $1000 \text{ A}$ .

### Field circuit voltage

This parameter is visible only when **Specify field circuit input required to produce rated terminal voltage at no load by** is set to `Field circuit voltage`. The default value is  $216.54 \text{ V}$ .

### Zero sequence

Option to include or exclude zero-sequence terms.

- **Include** — Include zero-sequence terms. To prioritize model fidelity, use this default setting. Using this option results in an error for simulations that use the Partitioning solver. For more information, see “Increase Simulation Speed Using the Partitioning Solver” (Simscape).
- **Exclude** — Exclude zero-sequence terms. To prioritize simulation speed for desktop simulation or real-time deployment, select this option.

### Rotor angle definition

Reference point for the rotor angle measurement.

The default option is **Angle between the a-phase magnetic axis and the d-axis**. When you select this value, the rotor  $d$ -axis and stator  $a$ -phase magnetic axis are aligned when the rotor angle is zero.

The other value you can choose for this parameter is **Angle between the a-phase magnetic axis and the q-axis**. When you select this value, the rotor  $q$ -axis and stator  $a$ -phase magnetic axis are aligned when the rotor angle is zero.

## Impedances

### **Stator d-axis mutual inductance (unsaturated), Ladu**

Unsaturated stator  $d$ -axis mutual inductance,  $L_{adu}$ . If **Magnetic saturation representation** is set to **None**, this is equivalent to the stator  $d$ -axis mutual inductance,  $L_{ad}$ . The default value is 0.9 pu.

### **Stator q-axis mutual inductance, Laq**

Stator  $q$ -axis mutual inductance,  $L_{aq}$ . The default value is 0.55 pu.

### **Stator zero-sequence inductance, L0**

Stator zero-sequence inductance,  $L_0$ . The default value is 0.15 pu.

### **Stator leakage inductance, Ll**

Stator leakage inductance. The default value is 0.15 pu.

### **Stator resistance, Ra**

Stator resistance. The default value is 0.011 pu.

### **Rotor field circuit inductance, Lfd**

Rotor field circuit inductance. The default value is 0.2571 pu.

### **Rotor field circuit resistance, Rfd**

Rotor field circuit resistance. The default value is 0.0006 pu.

### **Rotor d-axis damper winding 1 inductance, L1d**

Rotor  $d$ -axis damper winding 1 inductance. The default value is 0.2 pu.

### **Rotor d-axis damper winding 1 resistance, R1d**

Rotor  $d$ -axis damper winding 1 resistance. The default value is 0.0354 pu.

### **Rotor q-axis damper winding 1 inductance, L1q**

Rotor  $q$ -axis damper winding 1 inductance. The default value is 0.2567 pu.

### **Rotor q-axis damper winding 1 resistance, R1q**

Rotor  $q$ -axis damper winding 1 resistance. The default value is 0.0428 pu.

## Saturation

### Magnetic saturation representation

Block magnetic saturation representation. Options are:

- None
- Per-unit open-circuit lookup table ( $V_{ag}$  versus  $i_{fd}$ )

The default value is None.

### Per-unit field current saturation data, $i_{fd}$

Field current,  $i_{fd}$ , data that populates the air-gap voltage,  $V_{ag}$ , versus field current,  $i_{fd}$ , lookup table. This parameter is visible only when you set **Magnetic saturation representation** to Per-unit open-circuit lookup table ( $V_{ag}$  versus  $i_{fd}$ ). This parameter must contain a vector with at least five elements. The default value is [0.00, 0.48, 0.76, 1.38, 1.79] pu.

### Per-unit air-gap voltage saturation data, $V_{ag}$

Air-gap voltage,  $V_{ag}$ , data that populates the air-gap voltage,  $V_{ag}$ , versus field current,  $i_{fd}$ , lookup table. This parameter is visible only when you set **Magnetic saturation representation** to Per-unit open-circuit lookup table ( $V_{ag}$  versus  $i_{fd}$ ). This parameter must contain a vector with at least five elements. The default value is [0.00 0.43 0.59 0.71 0.76] pu.

## Initial Conditions

### Specify initialization by

Initialization method. Select one of these options:

- Electrical power and voltage output, the default value
- Mechanical and magnetic states

### Terminal voltage magnitude

Initial RMS line-line voltage. This parameter is visible only when you set **Specify initialization by** to Electrical power and voltage output. The default value is 24e3 V.

### Terminal voltage angle

Initial voltage angle. This parameter is visible only when you set **Specify initialization by** to Electrical power and voltage output. The default value is 0 deg.

### **Terminal active power**

Initial active power. This parameter is visible only when **Specify initialization by** is set to Electrical power and voltage output. The default value is 270e6 V\*A.

### **Terminal reactive power**

Initial reactive power. This parameter is visible only when you set **Specify initialization by** to Electrical power and voltage output. The default value is 0 V\*A.

### **Initial rotor angle**

Initial rotor angle. During steady-state operation, set this parameter to the sum of the load angle and required terminal voltage offset. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 deg.

### **Initial stator d-axis magnetic flux linkage**

Stator *d*-axis initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

### **Initial stator q-axis magnetic flux linkage**

Stator *q*-axis initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

### **Initial stator zero-sequence magnetic flux linkage**

Zero-sequence initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

### **Initial field circuit magnetic flux linkage**

Field circuit initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

### **Initial d-axis damper winding 1 magnetic flux linkage**

*d*-axis damper winding 1 initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

**Initial q-axis damper winding 1 magnetic flux linkage**

*q*-axis damper winding 1 initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

**Initial q-axis damper winding 2 magnetic flux linkage**

*q*-axis damper winding 2 initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

**References**

- [1] Kundur, P. *Power System Stability and Control*. New York, NY: McGraw Hill, 1993.
- [2] Lyshevski, S. E. *Electromechanical Systems, Electric Machines and Applied Mechatronics*. Boca Raton, FL: CRC Press, 1999.

**See Also**

Synchronous Machine Measurement | Synchronous Machine Model 2.1 (fundamental) | Synchronous Machine Model 2.1 (standard) | Synchronous Machine Round Rotor (fundamental) | Synchronous Machine Round Rotor (standard) | Synchronous Machine Salient Pole (standard)

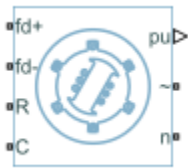
**Topics**

“Expand and Collapse Three-Phase Ports on a Block”  
“SM Torque Control”  
“SM Velocity Control”

**Introduced in R2013b**

## Synchronous Machine Salient Pole (standard)

Salient-pole synchronous machine with standard parameterization



### Library

Simscape / Electrical / Power Systems / Machines / Synchronous Machine (Salient Pole)

### Description

The Synchronous Machine Salient Pole (standard) block models a salient-pole synchronous machine using standard parameters.

### Equations

The synchronous machine equations are expressed with respect to a rotating reference frame, defined by

$$\theta_e(t) = N\theta_r(t),$$

where:

- $\theta_e$  is the electrical angle.
- $N$  is the number of pole pairs.
- $\theta_r$  is the rotor angle.

The Park transformation maps the synchronous machine equations to the rotating reference frame with respect to the electrical angle. The Park transformation is defined by

$$P_s = \frac{2}{3} \begin{bmatrix} \cos \theta_e & \cos(\theta_e - \frac{2\pi}{3}) & \cos(\theta_e + \frac{2\pi}{3}) \\ -\sin \theta_e & -\sin(\theta_e - \frac{2\pi}{3}) & -\sin(\theta_e + \frac{2\pi}{3}) \\ \frac{1}{2} & \frac{1}{2} & \frac{1}{2} \end{bmatrix}.$$

The Park transformation is used to define the per-unit synchronous machine equations. The stator voltage equations are defined by

$$e_d = \frac{1}{\omega_{base}} \frac{d\Psi_d}{dt} - \Psi_q \omega_r - R_a i_d,$$

$$e_q = \frac{1}{\omega_{base}} \frac{d\Psi_q}{dt} + \Psi_d \omega_r - R_a i_q,$$

and

$$e_0 = \frac{1}{\omega_{base}} \frac{d\Psi_0}{dt} - R_a i_0,$$

where:

- $e_d$ ,  $e_q$ , and  $e_0$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator voltages, defined by

$$\begin{bmatrix} e_d \\ e_q \\ e_0 \end{bmatrix} = P_s \begin{bmatrix} v_a \\ v_b \\ v_c \end{bmatrix}.$$

$v_a$ ,  $v_b$ , and  $v_c$  are the stator voltages measured from port  $\sim$  to neutral port  $\mathbf{n}$ .

- $\omega_{base}$  is the per-unit base electrical speed.

- $\psi_d$ ,  $\psi_q$ , and  $\psi_0$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator flux linkages.
- $\omega_r$  is the per-unit rotor rotational speed.
- $R_a$  is the stator resistance.
- $i_d$ ,  $i_q$ , and  $i_0$  are the  $d$ -axis,  $q$ -axis, and zero-sequence stator currents, defined by

$$\begin{bmatrix} i_d \\ i_q \\ i_0 \end{bmatrix} = P_s \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix}.$$

$i_a$ ,  $i_b$ , and  $i_c$  are the stator currents flowing from port  $\sim$  to port  $\mathbf{n}$ .

The rotor voltage equations are defined by

$$e_{fd} = \frac{1}{\omega_{base}} \frac{d\Psi_{fd}}{dt} + R_{fd}i_{fd},$$

$$e_{1d} = \frac{1}{\omega_{base}} \frac{d\Psi_{1d}}{dt} + R_{1d}i_{1d} = 0,$$

and

$$e_{1q} = \frac{1}{\omega_{base}} \frac{d\Psi_{1q}}{dt} + R_{1q}i_{1q} = 0,$$

where:

- $e_{fd}$  is the field voltage.
- $e_{1d}$ , and  $e_{1q}$  are the voltages across the  $d$ -axis damper winding 1 and  $q$ -axis damper winding 1. They are equal to 0.
- $\Psi_{fd}$ ,  $\Psi_{1d}$ , and  $\Psi_{1q}$  are the magnetic fluxes linking the field circuit,  $d$ -axis damper winding 1, and  $q$ -axis damper winding 1.
- $R_{fd}$ ,  $R_{1d}$ , and  $R_{1q}$  are the resistances of rotor field circuit,  $d$ -axis damper winding 1, and  $q$ -axis damper winding 1.
- $i_{fd}$ ,  $i_{1d}$ , and  $i_{1q}$  are the currents flowing in the field circuit,  $d$ -axis damper winding 1, and  $q$ -axis damper winding 1.



The saturation equations are defined by

$$\psi_{ad} = \psi_d + L_l i_d,$$

$$\psi_{aq} = \psi_q + L_l i_q,$$

$$\psi_{at} = \sqrt{\psi_{ad}^2 + \psi_{aq}^2},$$

$K_s = 1$  (If saturation is disabled),

$K_s = f(\psi_{at})$  (If saturation is enabled),

and

$$L_{ad} = K_s * L_{adu},$$

where:

- $\psi_{ad}$  is the  $d$ -axis air-gap or mutual flux linkage.
- $\psi_{aq}$  is the  $q$ -axis air-gap or mutual flux linkage.
- $\psi_{at}$  is the air-gap flux linkage.
- $K_s$  is the saturation factor.
- $L_{adu}$  is the unsaturated mutual inductance of the stator  $d$ -axis.
- $L_{ad}$  is the mutual inductance of the stator  $d$ -axis.

The saturation factor function,  $f$ , is calculated from the per-unit open-circuit lookup table as:

$$L_{ad} = \frac{d\psi_{at}}{di_{fd}},$$

$$V_{ag} = g(i_{fd}),$$

and

$$L_{ad} = \frac{dg(i_{fd})}{di_{fd}} = \frac{dV_{ag}}{di_{fd}},$$

where:

- $V_{ag}$  is the per-unit air-gap voltage.

In per-unit,

$$K_s = \frac{L_{ad}}{L_{adu}},$$

and

$$\psi_{at} = V_{ag}$$

can be rearranged to

$$K_s = f(\psi_{at}).$$

The stator flux linkage equations are defined by

$$\Psi_d = -(L_{ad} + L_l)i_d + L_{ad}i_{fd} + L_{ad}\dot{i}_d,$$

$$\Psi_q = -(L_{aq} + L_l)i_q + L_{aq}\dot{i}_q,$$

and

$$\Psi_0 = -L_0\dot{i}_0,$$

where:

- $L_l$  is the stator leakage inductance.
- $L_{ad}$  and  $L_{aq}$  are the mutual inductances of the stator  $d$ -axis and  $q$ -axis.

The rotor flux linkage equations are defined by

$$\psi_{fd} = L_{ffd}i_{fd} + L_{f1d}i_{1d} - L_{ad}i_d,$$

$$\psi_{1d} = L_{f1d}i_{fd} + L_{11d}i_{1d} - L_{ad}i_d,$$

and

$$\psi_{1q} = L_{11q}i_{1q} - L_{aq}i_q,$$

where:

- $L_{ffd}$  is the self-inductance of the rotor field circuit.
- $L_{11d}$  is the self-inductance of the  $d$ -axis damper winding 1.
- $L_{11q}$  is the self-inductance of the  $q$ -axis damper winding 1.
- $L_{f1d}$  is the rotor field circuit and  $d$ -axis damper winding 1 mutual inductance.

The inductance equations are:

$$L_{ffd} = L_{ad} + L_{fd}$$

$$L_{f1d} = L_{ffd} - L_{fd}$$

$$L_{11d} = L_{f1d} + L_{1d}$$

$$L_{11q} = L_{aq} + L_{1q}$$

The inductance equations assume that per-unit mutual inductance  $L_{12q} = L_{aq}$ , that is, the stator and rotor currents in the  $q$ -axis all link a single mutual flux represented by  $L_{aq}$ .

The rotor torque is defined by

$$T_e = \Psi_d i_q - \Psi_q i_d.$$

These defining equations do not describe the time constants you can set in the dialog box. To see their relationship with the equation coefficients, see [1], [3].

## Plotting and Display Options

You can perform plotting and display actions using the **Electrical** menu on the block context menu.

Right-click the block and, from the **Electrical** menu, select an option:

- **Display Base Values** — Displays the machine per-unit base values in the MATLAB Command Window.
- **Display Associated Base Values** — Displays associated per-unit base values in the MATLAB Command Window.
- **Display Associated Initial Conditions** — Displays associated initial conditions in the MATLAB Command Window.
- **Plot Open-Circuit Saturation (pu)** — Plots air-gap voltage,  $V_{ag}$ , versus field current,  $i_{fd}$ , both measured in per-unit, in a MATLAB figure window. The plot contains three traces:
  - Unsaturated — **Stator d-axis mutual inductance (unsaturated),  $L_{ad}$**  you specify
  - Saturated — **Per-unit open-circuit lookup table (Vag versus ifd)** you specify
  - Derived — Open-circuit lookup table (per-unit) derived from the **Per-unit open-circuit lookup table (Vag versus ifd)** you specify. This data is used to calculate the saturation factor,  $K_s$ , versus magnetic flux linkage,  $\psi_{at}$ , characteristic.
- **Plot Saturation Factor (pu)** — Plots saturation factor,  $K_s$ , versus magnetic flux linkage,  $\psi_{at}$ , both measured in per-unit, in a MATLAB figure window using the present machine parameters. This parameter is derived from other parameters that you specify:
  - **Stator d-axis mutual inductance (unsaturated),  $L_{ad}$**
  - **Per-unit field current saturation data, ifd**
  - **Per-unit air-gap voltage saturation data, Vag**

## Ports

**fd+**

Electrical conserving port corresponding to the field winding positive terminal

**fd-**

Electrical conserving port corresponding to the field winding negative terminal

**R**

Mechanical rotational conserving port associated with the machine rotor

**C**

Mechanical rotational conserving port associated with the machine case

**pu**

Physical signal vector port associated with the machine per-unit measurements. The vector elements are:

- pu\_fd\_Efd
- pu\_fd\_Ifd
- pu\_torque
- pu\_velocity
- pu\_ed
- pu\_eq
- pu\_e0
- pu\_id
- pu\_iq
- pu\_i0

**~**

Expandable three-phase port associated with the stator windings

**n**

Electrical conserving port associated with the neutral point of the wye winding configuration

## Parameters

- “Main” on page 1-1856
- “Impedances” on page 1-1857
- “Time Constants” on page 1-1857

- “Saturation” on page 1-1858
- “Initial Conditions” on page 1-1859

## **Main**

### **Rated apparent power**

Rated apparent power. The default value is  $300e6 \text{ V}\cdot\text{A}$ .

### **Rated voltage**

RMS rated line-line voltage. The default value is  $24e3 \text{ V}$ .

### **Rated electrical frequency**

Nominal electrical frequency at which rated apparent power is quoted. The default value is  $60 \text{ Hz}$ .

### **Number of pole pairs**

Number of machine pole pairs. The default value is  $10$ .

### **Specify field circuit input required to produce rated terminal voltage at no load by**

Choose between `Field circuit voltage` and `Field circuit current`. The default value is `Field circuit current`.

### **Field circuit current**

This parameter is visible only when **Specify field circuit input required to produce rated terminal voltage at no load by** is set to `Field circuit current`. The default value is  $1000 \text{ A}$ .

### **Field circuit voltage**

This parameter is visible only when **Specify field circuit input required to produce rated terminal voltage at no load by** is set to `Field circuit voltage`. The default value is  $216.54 \text{ V}$ .

### **Zero sequence**

Option to include or exclude zero-sequence terms.

- **Include** — Include zero-sequence terms. To prioritize model fidelity, use this default setting. Using this option results in an error for simulations that use the Partitioning solver. For more information, see “Increase Simulation Speed Using the Partitioning Solver” (Simscape).
- **Exclude** — Exclude zero-sequence terms. To prioritize simulation speed for desktop simulation or real-time deployment, select this option.

**Rotor angle definition**

Reference point for the rotor angle measurement.

The default option is **Angle between the a-phase magnetic axis and the d-axis**. When you select this value, the rotor *d*-axis and stator *a*-phase magnetic axis are aligned when the rotor angle is zero.

The other value you can choose for this parameter is **Angle between the a-phase magnetic axis and the q-axis**. When you select this value, the rotor *q*-axis and stator *a*-phase magnetic axis are aligned when the rotor angle is zero.

**Impedances****Stator resistance, Ra**

Stator resistance. The default value is 0.011 pu.

**Stator leakage reactance, Xl**

Stator leakage reactance. The default value is 0.15 pu.

**d-axis synchronous reactance, Xd**

*d*-axis synchronous reactance. The default value is 1.05 pu.

**q-axis synchronous reactance, Xq**

*q*-axis synchronous reactance. The default value is 0.7 pu.

**zero-sequence reactance, X0**

Zero-sequence reactance. The default value is 0.15 pu.

**d-axis transient reactance, Xd'**

*d*-axis transient reactance. The default value is 0.35 pu.

**d-axis subtransient reactance, Xd''**

*d*-axis subtransient reactance. The default value is 0.25 pu.

**q-axis subtransient reactance, Xq''**

*q*-axis subtransient reactance. The default value is 0.325 pu.

**Time Constants****Specify d-axis transient time constant**

Select between **Open-circuit value** and **Short-circuit value**. The default value is **Open-circuit value**.

**d-axis transient open-circuit, Td0'**

*d*-axis transient open-circuit time constant. This parameter is visible only when **Specify d-axis transient time constant** is set to `Open-circuit` value. The default value is 5.25 s.

**d-axis transient short-circuit, Td'**

*d*-axis transient short-circuit time constant. This parameter is visible only when **Specify d-axis transient time constant** is set to `Short-circuit` value. The default value is 1.75 s.

**Specify d-axis subtransient time constant**

Select between `Open-circuit` value and `Short-circuit` value. The default value is `Open-circuit` value.

**d-axis subtransient open-circuit, Td0''**

*d*-axis subtransient open-circuit time constant. This parameter is visible only when **Specify d-axis subtransient time constant** is set to `Open-circuit` value. The default value is 0.03 s.

**d-axis subtransient short-circuit, Td''**

*d*-axis subtransient short-circuit time constant. This parameter is visible only when **Specify d-axis subtransient time constant** is set to `Short-circuit` value. The default value is 0.0214 s.

**Specify q-axis subtransient time constant**

Select between `Open-circuit` value and `Short-circuit` value. The default value is `Open-circuit` value.

**q-axis subtransient open-circuit, Tq0''**

*q*-axis subtransient open-circuit time constant. This parameter is visible only when **Specify q-axis subtransient time constant** is set to `Open-circuit` value. The default value is 0.05 s.

**q-axis subtransient short-circuit, Tq''**

*q*-axis subtransient short-circuit time constant. This parameter is visible only when **Specify q-axis subtransient time constant** is set to `Short-circuit` value. The default value is 0.0232 s.

## Saturation

### Magnetic saturation representation

Block magnetic saturation representation. Options are:



- None
- Per-unit open-circuit lookup table ( $V_{ag}$  versus  $i_{fd}$ )

The default value is None.

#### **Per-unit field current saturation data, $i_{fd}$**

Field current,  $i_{fd}$ , data that populates the air-gap voltage,  $V_{ag}$ , versus field current,  $i_{fd}$ , lookup table. This parameter is visible only when you set **Magnetic saturation representation** to Per-unit open-circuit lookup table ( $V_{ag}$  versus  $i_{fd}$ ). This parameter must contain a vector with at least five elements. The default value is [0.00, 0.48, 0.76, 1.38, 1.79] pu.

#### **Per-unit air-gap voltage saturation data, $V_{ag}$**

Air-gap voltage,  $V_{ag}$ , data that populates the air-gap voltage,  $V_{ag}$ , versus field current,  $i_{fd}$ , lookup table. This parameter is visible only when you set **Magnetic saturation representation** to Per-unit open-circuit lookup table ( $V_{ag}$  versus  $i_{fd}$ ). This parameter must contain a vector with at least five elements. The default value is [0.00 0.43 0.59 0.71 0.76] pu.

## **Initial Conditions**

### **Specify initialization by**

Initialization method. Select one of these options:

- Electrical power and voltage output, the default value
- Mechanical and magnetic states

### **Terminal voltage magnitude**

Initial RMS line-line voltage. This parameter is visible only when you set **Specify initialization by** to Electrical power and voltage output. The default value is 24e3 V.

### **Terminal voltage angle**

Initial voltage angle. This parameter is visible only when you set **Specify initialization by** to Electrical power and voltage output. The default value is 0 deg.

### **Terminal active power**

Initial active power. This parameter is visible only when **Specify initialization by** is set to Electrical power and voltage output. The default value is 270e6 V\*A.

### **Terminal reactive power**

Initial reactive power. This parameter is visible only when you set **Specify initialization by** to Electrical power and voltage output. The default value is 0 V\*A.

### **Initial rotor angle**

Initial rotor angle. During steady-state operation, set this parameter to the sum of the load angle and required terminal voltage offset. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 deg.

### **Initial stator d-axis magnetic flux linkage**

Stator *d*-axis initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

### **Initial stator q-axis magnetic flux linkage**

Stator *q*-axis initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

### **Initial stator zero-sequence magnetic flux linkage**

Zero-sequence initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

### **Initial field circuit magnetic flux linkage**

Field circuit initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

### **Initial d-axis damper winding 1 magnetic flux linkage**

*d*-axis damper winding 1 initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

### **Initial q-axis damper winding 1 magnetic flux linkage**

*q*-axis damper winding 1 initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

### **Initial q-axis damper winding 2 magnetic flux linkage**

*q*-axis damper winding 2 initial flux linkage. This parameter is visible only when you set **Specify initialization by** to Mechanical and magnetic states. The default value is 0 pu.

## **References**

- [1] Kundur, P. *Power System Stability and Control*. New York, NY: McGraw Hill, 1993.
- [2] Lyshevski, S. E. *Electromechanical Systems, Electric Machines and Applied Mechatronics*. Boca Raton, FL: CRC Press, 1999.
- [3] Canay, I. M. "Modelling of alternating-current machines having multiple rotor circuits." *IEEE Transactions on Energy Conversion* , vol. 8, no. 2, 280-296, 1993.

## **See Also**

Synchronous Machine Measurement | Synchronous Machine Model 2.1 (fundamental) | Synchronous Machine Model 2.1 (standard) | Synchronous Machine Round Rotor (fundamental) | Synchronous Machine Salient Pole (fundamental) | Synchronous Machine Salient Pole (fundamental)

## **Topics**

"Expand and Collapse Three-Phase Ports on a Block"  
"Marine Full Electric Propulsion Power System"

**Introduced in R2013b**

# Synchronous Reluctance Machine

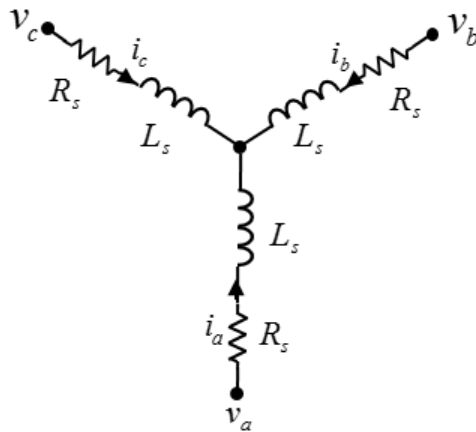
Synchronous reluctance machine with sinusoidal flux distribution

**Library:** Simscape / Electrical / Power Systems / Machines / Reluctance



## Description

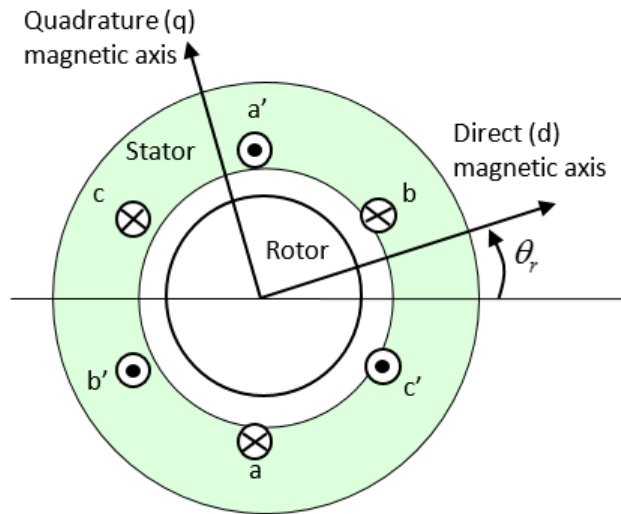
The Synchronous Reluctance Machine block represents a synchronous reluctance machine (SynRM) with sinusoidal flux distribution. The figure shows the equivalent electrical circuit for the stator windings.



## Motor Construction

The diagram shows the motor construction with a single pole-pair on the rotor. For the axes convention shown, when rotor mechanical angle  $\theta_r$  is zero, the  $a$ -phase and

permanent magnet fluxes are aligned. The block supports a second rotor axis definition for which rotor mechanical angle is defined as the angle between the  $a$ -phase magnetic axis and the rotor  $q$ -axis.



## Equations

The combined voltage across the stator windings is

$$\begin{bmatrix} v_a \\ v_b \\ v_c \end{bmatrix} = \begin{bmatrix} R_s & 0 & 0 \\ 0 & R_s & 0 \\ 0 & 0 & R_s \end{bmatrix} \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix} + \begin{bmatrix} \frac{d\psi_a}{dt} \\ \frac{d\psi_b}{dt} \\ \frac{d\psi_c}{dt} \end{bmatrix},$$

where:

- $v_a$ ,  $v_b$ , and  $v_c$  are the individual phase voltages across the stator windings.
- $R_s$  is the equivalent resistance of each stator winding.

- $i_a$ ,  $i_b$ , and  $i_c$  are the currents flowing in the stator windings.
- $\psi_a$ ,  $\psi_b$ , and  $\psi_c$  are the magnetic fluxes that link each stator winding.

The permanent magnet, excitation winding, and the three stator windings contribute to the flux that links each winding. The total flux is defined as

$$\begin{bmatrix} \psi_a \\ \psi_b \\ \psi_c \end{bmatrix} = \begin{bmatrix} L_{aa} & L_{ab} & L_{ac} \\ L_{ba} & L_{bb} & L_{bc} \\ L_{ca} & L_{cb} & L_{cc} \end{bmatrix} \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix}$$

where:

- $L_{aa}$ ,  $L_{bb}$ , and  $L_{cc}$  are the self-inductances of the stator windings.
- $L_{ab}$ ,  $L_{ac}$ ,  $L_{ba}$ ,  $L_{bc}$ ,  $L_{ca}$ , and  $L_{cb}$  are the mutual inductances of the stator windings.

The inductances in the stator windings are functions of rotor electrical angle and are defined as

$$L_{aa} = L_s + L_m \cos(2\theta_r),$$

$$L_{bb} = L_s + L_m \cos\left(2\left(\theta_r - \frac{2\pi}{3}\right)\right),$$

$$L_{cc} = L_s + L_m \cos\left(2\left(\theta_r + \frac{2\pi}{3}\right)\right),$$

$$L_{ab} = L_{ba} = -M_s - L_m \cos\left(\theta_r + \frac{\pi}{6}\right),$$

$$L_{bc} = L_{cb} = -M_s - L_m \cos\left(\theta_r + \frac{\pi}{6} - \frac{2\pi}{3}\right),$$

$$L_{ca} = L_{ac} = -M_s - L_m \cos\left(\theta_r + \frac{\pi}{6} + \frac{2\pi}{3}\right),$$

where:

- $L_s$  is the stator self-inductance per phase. This value is the average self-inductance of each of the stator windings.
- $L_m$  is the stator inductance fluctuation. This value is the amplitude of the fluctuation in self-inductance and mutual inductance with changing rotor angle.
- $\theta_r$  is the rotor mechanical angle.
- $M_s$  is the stator mutual inductance. This value is the average mutual inductance between the stator windings.

## Simplified Equations

Applying the Park transformation to the block electrical defining equations produces an expression for torque that is independent of rotor angle.

The Park transformation,  $P$ , is defined as

$$P = \frac{2}{3} \begin{bmatrix} \cos \theta_e & \cos \left( \theta_e - \frac{2\pi}{3} \right) & \cos \left( \theta_e + \frac{2\pi}{3} \right) \\ -\sin \theta_e & -\sin \left( \theta_e - \frac{2\pi}{3} \right) & -\sin \left( \theta_e + \frac{2\pi}{3} \right) \\ \frac{1}{2} & \frac{1}{2} & \frac{1}{2} \end{bmatrix},$$

where  $\theta_e$  is the electrical angle. The electrical angle depends on the rotor mechanical angle and the number of pole pairs such that

$$\theta_e = N \theta_r,$$

where:

- $N$  is the number of pole pairs.
- $\theta_r$  is the rotor mechanical angle.

Applying the Park transformation to the first two electrical defining equations produces equations that define the behavior of the block:

$$v_d = R_s i_d + L_d \frac{di_d}{dt} - N \omega_i L_q,$$

$$v_q = R_s i_q + L_q \frac{di_q}{dt} + N \omega i_d L_d,$$

$$v_0 = R_s i_0 + L_0 \frac{di_0}{dt},$$

$$T = \frac{3}{2} N (i_q i_d L_d - i_d i_q L_q)$$

$$J \frac{d\omega}{dt} = T - T_L - B_m \omega,$$

where:

- $i_d$ ,  $i_q$ , and  $i_0$  are the  $d$ -axis,  $q$ -axis, and zero-sequence currents, defined by

$$\begin{bmatrix} i_d \\ i_q \\ i_0 \end{bmatrix} = P \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix},$$

where  $i_a$ ,  $i_b$ , and  $i_c$  are the stator currents.

- $v_d$ ,  $v_q$ , and  $v_0$  are the  $d$ -axis,  $q$ -axis, and zero-sequence currents, defined by

$$\begin{bmatrix} v_d \\ v_q \\ v_0 \end{bmatrix} = P \begin{bmatrix} v_a \\ v_b \\ v_c \end{bmatrix},$$

where  $v_a$ ,  $v_b$ , and  $v_c$  are the stator currents.

- The  $dq0$  inductances are defined, respectively as

- $$L_d = L_s + M_s + \frac{3}{2} L_m$$

- $$L_q = L_s + M_s - \frac{3}{2} L_m$$



- $L_0 = L_s - 2M_s$ .
- $R_s$  is the stator resistance per phase.
- $N$  is the number of rotor pole pairs.
- $T$  is the rotor torque. For the Synchronous Reluctance Machine block, torque flows from the machine case (block conserving port **C**) to the machine rotor (block conserving port **R**).
- $T_L$  is the load torque.
- $B_m$  is the rotor damping.
- $\omega$  is the rotor mechanical rotational speed.
- $J$  is the rotor inertia.

## Assumptions

The block assumes that the flux distribution is sinusoidal.

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Ports

### Conserving

#### **R — Machine rotor**

mechanical rotational

Mechanical rotational conserving port associated with the machine rotor.

### **C — Machine case**

mechanical rotational

Mechanical rotational conserving port associated with the machine case.

### **~ — Three-phase composite**

electrical

Expandable three-phase port associated with the stator windings.

### **n — Neutral phase**

electrical

Electrical conserving port associated with the neutral phase.

## **Parameters**

### **Main**

#### **Number of pole pairs — Rotor pole pairs**

6 (default) | integer

Number of permanent magnet pole pairs on the rotor.

#### **Stator parameterization — Parameterization method**

Specify Ld, Lq and L0 (default) | Specify Ls, Lm, and Ms

Method for parameterizing the stator.

#### **Dependencies**

Selecting Specify Ld, Lq and L0 enables these parameters:

- **Stator d-axis inductance, Ld**
- **Stator q-axis inductance, Lq**
- **Stator zero-sequence inductance, L0**

Selecting Specify Ls, Lm, and Ms enables these parameters:

- **Stator self-inductance per phase, Ls**

- **Stator inductance fluctuation,  $L_m$**
- **Stator mutual inductance,  $M_s$**

**Stator d-axis inductance,  $L_d$  – Inductance**

0.0031 H (default)

Direct-axis inductance of the machine stator.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify  $L_d$ ,  $L_q$  and  $L_0$ .

**Stator q-axis inductance,  $L_q$  – Inductance**

0.004 H (default)

Quadrature-axis inductance of the machine stator.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify  $L_d$ ,  $L_q$  and  $L_0$ .

**Stator zero-sequence inductance,  $L_0$  – Inductance**

0.0005 H (default)

Zero-axis inductance for the machine stator.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify  $L_d$ ,  $L_q$  and  $L_0$ .

**Stator self-inductance per phase,  $L_s$  – Inductance**

0.0025 H (default)

Average self-inductance of the three stator windings.

**Dependencies**

To enable this parameter, set **Stator parameterization** to Specify  $L_s$ ,  $L_m$ , and  $M_s$ .

**Stator inductance fluctuation,  $L_m$  – Inductance**

-0.0003 H (default)

Amplitude of the fluctuation in self-inductance and mutual inductance with the rotor angle.

## Dependencies

To enable this parameter, set **Stator parameterization** to Specify Ls, Lm, and Ms.

## Stator mutual inductance, Ms — Inductance

0.0010 H (default)

Average mutual inductance between the stator windings.

## Dependencies

To enable this parameter, set **Stator parameterization** to Specify Ls, Lm, and Ms.

## Stator resistance per phase, Rs — Resistance

0.7 Ohm (default)

Resistance of each of the stator windings.

## Zero sequence — Option to neglect zero-sequence terms

Include (default) | Exclude

Option to neglect zero-sequence terms. Choices are:

- **Include** — Include zero-sequence terms. To prioritize model fidelity, use this default setting. Using this option results in an error for simulations that use the Partitioning solver. For more information, see “Increase Simulation Speed Using the Partitioning Solver” (Simscape).
- **Exclude** — Exclude zero-sequence terms. To prioritize simulation speed for desktop simulation or real-time deployment, select this option.

## Mechanical

### Rotor inertia — Inertia

0.01 kg\*m<sup>2</sup> (default)

Inertia of the rotor attached to mechanical translational port **R**.

### Rotor Damping — Damping

0 N\*m/(rad/s) (default)

Rotary damping.

**Rotor angle definition – Angle**

Angle between the a-phase magnetic axis and the d-axis (default) | Angle between the a-phase magnetic axis and the q-axis

Reference point for the rotor angle measurement. If you select the default value, the rotor and *a*-phase fluxes are aligned for a zero-rotor angle. Otherwise, an *a*-phase current generates the maximum torque value for a zero-rotor angle.

**References**

- [1] Kundur, P. *Power System Stability and Control*. New York, NY: McGraw Hill, 1993.
- [2] Anderson, P. M. *Analysis of Faulted Power Systems*. Hoboken, NJ: Wiley-IEEE Press, 1995.
- [3] Moghaddam, R. *Synchronous Reluctance Machine (SynRM) in Variable Speed Drives (VSD) Applications - Theoretical and Experimental Reevaluation*. KTH School of Electrical Engineering, Stockholm, Sweden, 2011.

**See Also****Simscape Blocks**

Brushless DC Motor | Hybrid Excitation Synchronous Machine | Permanent Magnet Synchronous Motor | Switched Reluctance Machine | Synchronous Machine Field Circuit (pu) | Synchronous Machine Measurement

**Introduced in R2017b**

# Tap-Changing Transformer

Single-phase tap-changing transformer

**Library:** Simscape / Electrical / Power Systems / Passive Devices / Fundamental Components



## Description

The Tap-Changing Transformer block represents a single-phase tap-changing transformer. You can vary the turns-ratio of the transformer during the simulation using the control input.

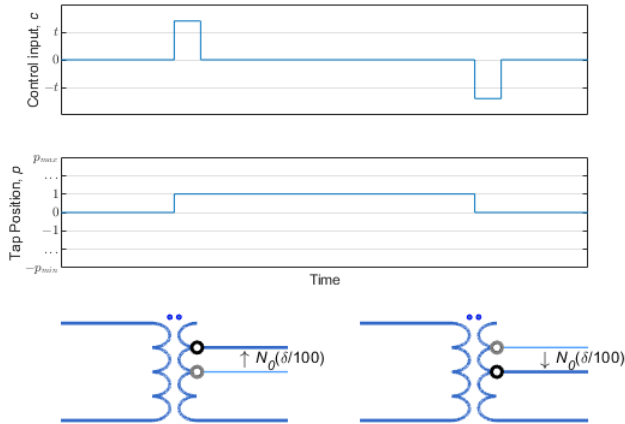
Use this component to regulate or change the output voltage of a linear transformer during a simulation. To model the effects of saturation, consider using the Nonlinear Transformer block.

## Operating Principle

Use the control input **c** to change the tap position of the transformer.

- Increase **c** above the **Control threshold**,  $t$ , to increase the tap position  $p$  by one.
- Decrease **c** below the negative of the **Control threshold** to decrease the tap position  $p$  by one.

This figure shows the tap-response to a control input.



In the diagram:

- $N_0$  is the nominal number of turns for the tap-changer winding.
- $\delta$  is the percent change in turns per tap position step. Specify this value using the **Change per tap (%)** parameter.
- $p_{min}$  and  $p_{max}$  are the minimum and maximum allowable tap indices. Specify these values with the **Minimum tap index (nominal=0)** and **Maximum tap index (nominal=0)** parameters.

To select the tap-changer winding, use the **Tap-changer location** parameter. The overall turns ratio,  $n$ , depends on this location:

- Primary

$$n = \frac{N_p(N_0, \delta, p)}{N_s} = n_0 \left( 1 + \frac{\delta p}{100} \right)$$

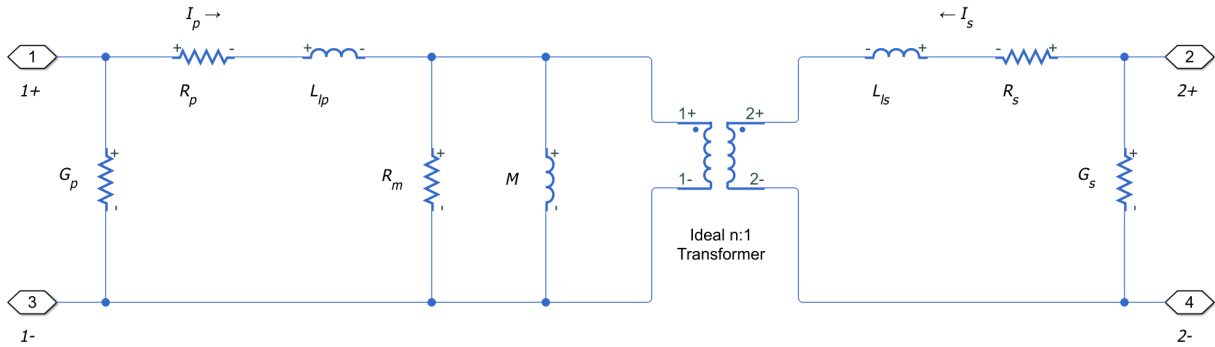
- Secondary

$$n = \frac{N_p}{N_s(N_0, \delta, p)} = n_0 \left( 1 + \frac{\delta p}{100} \right)^{-1}$$

where  $n_0$  is the nominal turns ratio for the transformer.

## Equivalent Circuit

The equivalent circuit is shown in the diagram.



Here:

- $R_p$  and  $R_s$  are the primary and secondary series resistances, respectively. Changes in the tap position affect these values.
- $L_{lp}$  and  $L_{ls}$  are the primary and secondary leakage inductances, respectively. Changes in the tap position affect these values.
- $R_m$  and  $M$  are the magnetization resistance and inductance, respectively. Changes in the tap position affect these values.
- $n$  is the turns ratio of the transformer.
- $G_p$  and  $G_s$  are the primary and secondary leakage conductances, respectively. Changes in the tap position do not affect these values.

## Ports

### Conserving

**1+ — Line 1 positive terminal**  
electrical



Electrical conserving port associated with the positive terminal of line 1.

**1- — Line 1 negative terminal**

electrical

Electrical conserving port associated with the negative terminal of line 1.

**2+ — Line 2 positive terminal**

electrical

Electrical conserving port associated with the positive terminal of line 2.

**2- — Line 2 negative terminal**

electrical

Electrical conserving port associated with the negative terminal of line 2.

**c — Control input**

physical

Input physical signal that specifies the input control value.

## Parameters

### Nominal

**Turns ratio (primary/secondary) — Turns ratio**

1 (default) | positive number

Turns ratio of the transformer in the nominal tap position. The turns ratio is defined as the primary number of turns divided by the secondary number of turns.

**Primary leakage inductance — Line 1 leakage inductance**

1e-6 H (default) | positive number

Leakage inductance of the primary winding.

**Secondary leakage inductance — Line 2 leakage inductance**

1e-6 H (default) | positive number

Leakage inductance of the secondary winding.

**Core-loss resistance — Mutual resistance**

1e6 Ohm (default) | zero or positive number

Mutual resistance of the transformer.

**Magnetization inductance — Magnetization inductance**

100e-6 H (default) | positive number

Magnetization inductance of the transformer.

**Primary series resistance — Line 1 series resistance**

0 Ohm (default) | zero or positive number

Series resistance of the primary winding.

**Secondary series resistance — Line 2 series resistance**

0 Ohm (default) | zero or positive number

Series resistance of the secondary winding.

**Primary leakage conductance — Line 1 leakage conductance**

0 1/Ohm (default) | zero or positive number

Leakage conductance of the primary winding. Set this value to be nonzero for achieving numerical convergence in some circuit topologies.

**Secondary leakage conductance — Line 2 leakage conductance**

0 1/Ohm (default) | zero or positive number

Leakage conductance of the secondary winding. Set this value to be nonzero for achieving numerical convergence in some circuit topologies.

**Tap**

**Tap-changer location — Tap-changer winding**

Primary (default) | Secondary

Specify whether the tap-changer is on the primary or secondary winding.

**Minimum tap index (nominal=0) — Minimum tap position**

-5 (default) | zero or negative integer

Minimum allowable position of the tap-changer.

**Maximum tap index (nominal=0) – Maximum tap position**

5 (default) | zero or positive integer

Maximum allowable position of the tap-changer.

**Change per tap (%) – Tap change step**

1 % (default) | positive number

Percent change in the number of turns per tap position step of the tap-changer winding. Set this value such that the absolute percent change at the minimum and maximum tap indices is less than 100 %.

**Control threshold – Shift threshold**

0.5 (default)

Control value at which the tap changes position. To lower the tap position, apply a control signal **c** smaller than the negative of this value. To increase the tap position, apply a control signal **c** greater than this value.

**See Also**

Nonlinear Transformer

**Introduced in R2018a**

# Thermal Resistor

Heat transfer by conduction through a layer of material



## Library

Simscape / Electrical / Power Systems / Semiconductors / Fundamental Components / Thermal

## Description

The Thermal Resistor block represents heat transfer by conduction through a layer of material. The heat transfer is:

- Governed by Fourier's law
- Proportional to the temperature difference across the layer of material
- Inversely proportional to the thermal resistance of the material

The equation for conductive heat transfer is:

$$Q_{AB} = \frac{T_{AB}}{R_{thermal}},$$

where:

- $Q_{AB}$  is the heat flow through the material.
- $T_{AB}$  is the temperature difference across the layer of material.
- $R_{thermal}$  is the thermal resistance of the material.

Thermal resistance can be calculated as:

$$R_{thermal} = \frac{D}{kA},$$

where:

- $D$  is the thickness of the layer of material.
- $k$  is the thermal conductivity of the material.
- $A$  is the area normal to the heat flow direction.

Use the Thermal Resistor block to parameterize an equivalent component in terms of thermal resistance of the material layer. To parameterize an equivalent component in terms of the thickness, thermal conductivity, and area of the material layer, use the Conductive Heat Transfer block from the Simscape Foundation library.

## Parameters

- “Parameters Tab” on page 1-1879
- “Variables Tab” on page 1-1879

## Parameters Tab

### Thermal resistance

The default value for the thermal resistance,  $R_{thermal}$ , is  $1e-3$  K/W.

## Variables Tab

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Ports

The block has the following ports:

A

Thermal conserving port associated with surface A of the material that the heat flows through.

B

Thermal conserving port associated with surface B of the material that the heat flows through.

## See Also

Cauer Thermal Model Element | Foster Thermal Model

**Introduced in R2016a**

# Thermistor

Model NTC thermistor using B-parameter equation



## Library

Sensors

## Description

The Thermistor block represents an NTC thermistor using the B-parameter equation. The resistance at temperature  $T$  is

$$R = R_0 e^{B(1/T - 1/T_0)}$$

where:

- $R_0$  is the nominal resistance at the reference temperature  $T_0$ .
- $B$  is the characteristic temperature constant.

The following equation describes the thermal behavior of the block:

$$Q = K_d t_c \frac{dT}{dt}$$

where:

- $Q$  is the net heat flow into port A.
- $K_d$  is the **Dissipation factor** parameter value.

- $t_c$  is the **Thermal time constant** parameter value.
- $dT/dt$  is the rate of change of the temperature.

To model the thermistor in free space:

- 1 Connect the thermistor to the B port of a Simscape Convective Heat Transfer block.
- 2 Connect the A port of the Convective Heat Transfer block to a Simscape Ideal Temperature Source block whose temperature is set to the ambient temperature.
- 3 Set the **Area** parameter of the Convective Heat Transfer block to an approximate area  $A_{\text{nom}}$ .
- 4 Set the **Heat transfer coefficient** parameter of the Convective Heat Transfer block to  $K_d/A_{\text{nom}}$ .

## Variables

Use the **Variables** section of the block interface to set the priority and initial target values for the block variables prior to simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

## Parameters

- “Electrical Tab” on page 1-1882
- “Thermal Tab” on page 1-1883

## Electrical Tab

### Nominal resistance $R_0$ at reference temperature $T_0$

The nominal resistance of the thermistor at the reference temperature. Many datasheets quote the nominal resistance at 25°C and list it as R25. The default value is 1000  $\Omega$ .

### Characteristic temperature constant $B$

The coefficient  $B$  in the equation that describes resistance as a function of temperature. The default value is 3500 K.

### Reference temperature $T_0$

The temperature at which the nominal resistance was measured. The default value is 298.15 K.



## Thermal Tab

### Thermal time constant

The time it takes the sensor temperature to reach 63% of the final temperature change when a step change in ambient temperature occurs. The default value is 5 s.

### Dissipation factor

The thermal power required to raise the thermistor temperature by one K. The default value is  $0.75 \times 10^{-4}$  W/K.

## Ports

The block has the following ports:

A

Thermal port

+

Positive electrical port

-

Negative electrical port

## See Also

PTC Thermistor | Resistor | Thermal Resistor

# Thermocouple

Model sensor that converts thermal potential difference into electrical potential difference



## Library

Sensors

## Description

The Thermocouple block represents a thermocouple using the standard polynomial parameterization defined in the NIST ITS-90 Thermocouple Database [1].

For thermocouples Type B, E, J, K ( $t \leq 0$  degC), N, R, S or T, the voltage  $E$  across the device in mV is

$$E(mV) = c_0 + c_1 t + \dots + c_n t^n$$

where:

- $c_i$  is the  $i^{\text{th}}$  element of the **Coefficients [c0 c1 ... cn]** parameter value.
- $t$  is the temperature difference in degrees Celsius between the temperature at the thermal port A and the **Reference temperature** parameter value.

---

**Note** The equation for voltage across the device as a function of temperature difference is defined in mV. The units of the voltage across the actual device is V.

---

For thermocouples Type K ( $t \geq 0$  degC), the equation contains an additional exponential term:

$$E(mV) = c_0 + c_1t + \dots + c_nt^n + a_0e^{a_1(t-a_2)^2}$$

where  $a_0$ ,  $a_1$ , and  $a_2$  are additional coefficients, required only by the Type K thermocouple, defined by the **Coefficients [a0 a1 a2]** parameter value.

The following equation describes the thermal behavior of the block:

$$Q = K_d t_c \frac{dT}{dt}$$

where:

- $T$  is the temperature at port A.
- $Q$  is the net heat flow into port A.
- $K_d$  is the **Dissipation factor** parameter value.
- $t_c$  is the **Thermal time constant** parameter value.
- $dT/dt$  is the rate of change of the temperature.

To model the thermocouple in free space:

- 1 Connect the thermocouple to the B port of a Simscape Convective Heat Transfer block.
- 2 Connect the A port of the Convective Heat Transfer block to a Simscape Ideal Temperature Source block whose temperature is set to the ambient temperature.
- 3 Set the **Area** parameter of the Convective Heat Transfer block to an approximate area  $A_{nom}$ .
- 4 Set the **Heat transfer coefficient** parameter of the Convective Heat Transfer block to  $K_d/A_{nom}$ .

## Basic Assumptions and Limitations

The model is based on the following assumptions:

- The high-order polynomials this block uses are very sensitive to the number of significant figures used for computation. Use all available significant figures when specifying the **Coefficients [c0 c1 ... cn]** parameter.

- **Coefficients [c0 c1 ... cn]** are defined for use over a specified temperature range.
- The maximum supported value for  $n$  in the **Coefficients [c0 c1 ... cn]** parameter is 14, that is, the vector cannot have more than 15 elements.

## Parameters

- “Electrical Tab” on page 1-1886
- “Thermal Tab” on page 1-1887

## Electrical Tab

### Thermocouple model

Select one of the modeling options:

- Type B, E, J, K ( $t \leq 0$  degC), N, R, S or T — This option is equivalent to the block functionality in previous releases.
- Type K ( $t \geq 0$  degC) — This option adds an exponential term to the block equations when the temperature difference is greater than 0 degrees Celsius.

### Coefficients [c0 c1 ... cn]

The vector of coefficients  $c$  in the equation that describes voltage as a function of temperature. The maximum length of the vector is 15 elements. The default value is [ 0 0 0 0 0 0 0 0 0 ].

---

**Note** You can download parameters for standard thermocouple types from the NIST database [1]. For information on how to do this, see the Simulink Approximating Nonlinear Relationships: Type S Thermocouple example.

---

### Coefficients [a0 a1 a2]

The vector of additional coefficients  $a_0$ ,  $a_1$ , and  $a_2$ , required only by the Type K thermocouple. This parameter is only visible when you select Type K ( $t \geq 0$  degC) for the **Thermocouple model** parameter. The default value is [ 0 0 0 ].

## Thermal Tab

### Reference temperature

The temperature the block subtracts from the temperature at the thermal port in calculating the voltage across the device. The default value is 0 °C.

### Thermal time constant

The time it takes the thermocouple temperature to reach 63% of the final temperature change when a step change in ambient temperature occurs. The default value is 1 s.

### Dissipation factor

The thermal power required to raise the thermocouple temperature by one K. The default value is 0.001 W/K.

### Initial temperature

The temperature of the thermocouple at the start of the simulation. The default value is 25 °C.

## Ports

The block has the following ports:

A

Thermocouple thermal port

+

Positive electrical port

-

Negative electrical port

## References

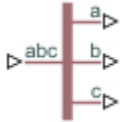
[1] NIST ITS-90 Thermocouple Database <https://srdata.nist.gov/its90/main>

## **See Also**

Thermal Resistor

# Three Element Demux

Convert three-element physical signal vector into scalar physical signals



## Library

Simscape / Electrical / Power Systems / Sensors

## Description

The Three Element Demux block splits a three-element physical signal vector into three scalar physical signals.

## Ports

The block has the following ports:

abc

Three-element physical signal input port.

a

Scalar physical signal output port.

b

Scalar physical signal output port.

c

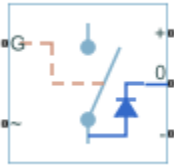
Scalar physical signal output port.

**Introduced in R2013b**



# Three-Level Converter

Twelve-pulse three-phase three-level neutral-point clamped controlled converter.



## Library

Simscape / Electrical / Power Systems / Semiconductors

## Description

The Three-Level Converter block models a twelve-pulse three-phase three-level neutral-point clamped controlled converter. You can use this block to connect a three-phase AC network to a three-level DC network.

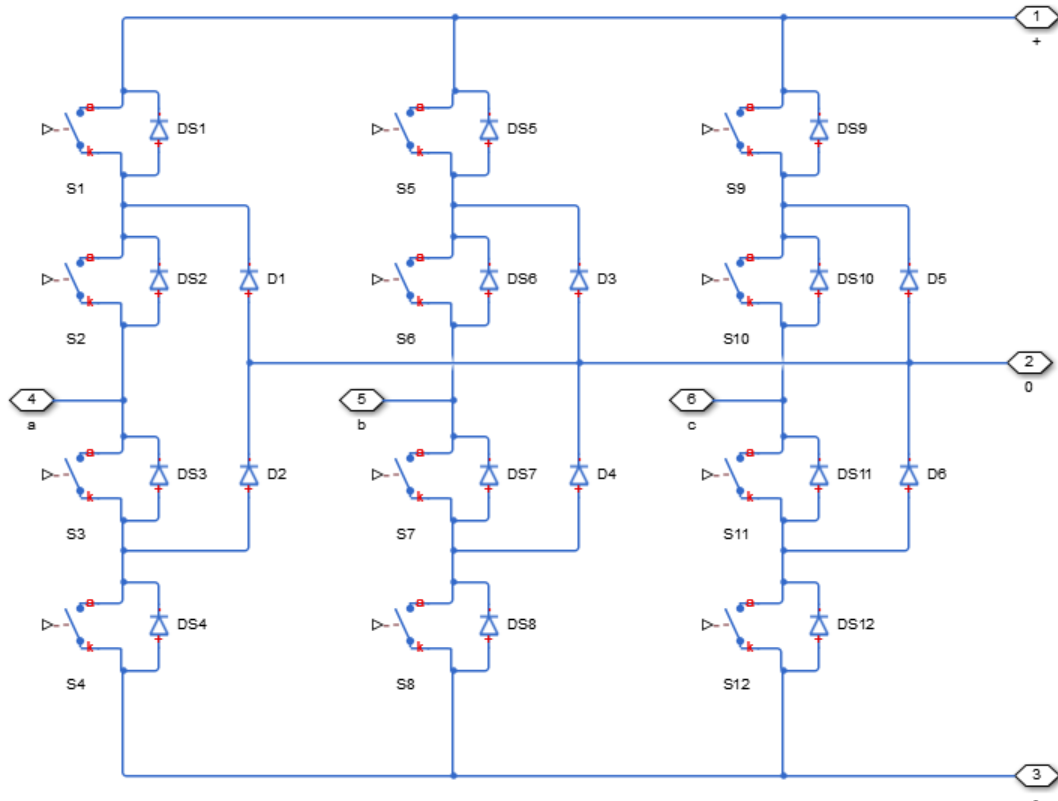
## Model

The block contains three bridge arms, each of which has four switching devices and the associated anti-parallel diodes. Options for the type of switching devices are:

- GTO
- Ideal Semiconductor Switch
- IGBT
- MOSFET
- Thyristor

Each component in the three-arm circuit is the same switching device, which you specify using an option in the Converter block dialog box. The switching devices are in the **Semiconductors > Fundamental Components** sublibrary.

The figure shows the equivalent circuit for the block using an Ideal Semiconductor block as the switching device.



You control the gate ports of the 12 switching devices via an input to the Three-Level Converter block **G** port.

- 1 Use a Twelve-Pulse Gate Multiplexer block to multiplex all 12 gate signals into a single vector.
- 2 Connect the output of the Twelve-Pulse Gate Multiplexer block to the Three-Level Converter block **G** port.

You use the **Diodes** tab of the block dialog box to include an integral protection diode for each switching device. An integral diode protects the semiconductor device by providing

a conduction path for reverse current. An inductive load can produce a high reverse-voltage spike when the semiconductor device suddenly switches off the voltage supply to the load.

The table shows how to set the **Integral protection diode** parameter based on your goals.

Goal	Value to Select	Block Behavior
Prioritize simulation speed.	Protection diode with no dynamics	The block includes an integral copy of the Diode block. To parameterise the internal Diode block, use the <b>Protection</b> parameters.
Precisely specify reverse-mode charge dynamics.	Protection diode with charge dynamics	The block includes an integral copy of the Commutation Diode block. To parameterise the internal Commutation Diode block, use the <b>Protection</b> parameters.

You use the **Snubbers** tab of the block dialog box to include a snubber circuit for each switching device. Each snubber consists of a resistor and capacitor connected in series. Typically, a snubber circuit protects a switching device against very high voltages produced by an inductive load when the device turns off the voltage supply to the load. Snubber circuits also prevent excessive rates of change of current when a switching device turns on.

## Parameters

- “Switching Devices Tab” on page 1-1893
- “Diodes Tab” on page 1-1896
- “Snubbers Tab” on page 1-1899

## Switching Devices Tab

### Switching device

Converter switching device. The default value is Ideal Semiconductor Switch.

The switching devices you can select are:

- GTO
- Ideal Semiconductor Switch
- IGBT
- MOSFET

When you select GTO, parameters for the GTO block appear.

### **Additional GTO Parameters**

#### **Forward voltage, Vf**

Minimum voltage required across the anode and cathode block ports for the gradient of the device i-v characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**. The default value is 0.8 V.

#### **On-state resistance**

Rate of change of voltage versus current above the forward voltage. The default value is 0.001 Ohm.

#### **Off-state conductance**

Anode-cathode conductance when the device is off. The value must be less than  $1/R$ , where  $R$  is the value of **On-state resistance**. The default value is  $1e-6$  1/Ohm.

#### **Gate trigger voltage, Vgt**

Gate-cathode voltage threshold. The device turns on when the gate-cathode voltage is above this value. The default value is 1 V.

#### **Gate turn-off voltage, Vgt\_off**

Gate-cathode voltage threshold. The device turns off when the gate-cathode voltage is below this value. The default value is -1 V.

#### **Holding current**

Current threshold. The device stays on when the current is above this value, even when the gate-cathode voltage falls below the gate trigger voltage. The default value is 1 A.

For more information, see GTO.

When you select Ideal Semiconductor Switch, parameters for the Ideal Semiconductor Switch block appear.

## Additional Ideal Semiconductor Switch Parameters

### On-state resistance

Anode-cathode resistance when the device is on. The default value is 0.001 Ohm.

### Off-state conductance

Anode-cathode conductance when the device is off. The value must be less than  $1/R$ , where  $R$  is the value of **On-state resistance**. The default value is  $1e-6$  1/Ohm.

### Threshold voltage, $V_{th}$

Gate-cathode voltage threshold. The device turns on when the gate-cathode voltage is above this value. The default value is 6 V.

For more information, see Ideal Semiconductor Switch.

When you select IGBT, parameters for the IGBT block appear.

## Additional IGBT Parameters

### Forward voltage, $V_f$

Minimum voltage required across the collector and emitter block ports for the gradient of the diode i-v characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**. The default value is 0.8 V.

### On-state resistance

Collector-emitter resistance when the device is on. The default value is 0.001 Ohm.

### Off-state conductance

Collector-emitter conductance when the device is off. The value must be less than  $1/R$ , where  $R$  is the value of **On-state resistance**. The default value is  $1e-6$  1/Ohm.

### Threshold voltage, $V_{th}$

Collector-emitter voltage at which the device turns on. The default value is 6 V.

For more information, see IGBT.

When you select MOSFET, parameters for the MOSFET block appear.

## **Additional MOSFET Parameters**

### **On-state resistance, $R_{DS(on)}$**

Drain-source resistance when the device is on. The default value is 0.001 Ohm.

### **Off-state conductance**

Drain-source conductance when the device is off. The value must be less than  $1/R$ , where  $R$  is the value of **On-state resistance**. The default value is  $1e-6$  1/Ohm.

### **Threshold voltage, $V_{th}$**

Gate-source voltage threshold. The device turns on when the gate-source voltage is above this value. The default value is 6 V.

For more information, see MOSFET.

## **Diodes Tab**

### **Integral protection diode**

Integral protection diode for each switching device. Choose between `Diode with no dynamics` and `Diode with charge dynamics`. The default value is `Diode with no dynamics`.

When you select `Diode with no dynamics`, additional parameters appear.

### **Additional Parameters for Diode with no dynamics**

#### **Forward voltage**

Minimum voltage required across the + and - block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**. The default value is 0.8 V.

#### **On resistance**

Rate of change of voltage versus current above the **Forward voltage**. The default value is 0.001 Ohm.

#### **Off conductance**

Conductance of the reverse-biased diode. The default value is  $1e-5$  1/Ohm.

For more information on these parameters, see Diode.

When you select Protection diode with charge dynamics, additional parameters appear.

### **Additional Parameters for Diode with charge dynamics**

#### **Forward voltage**

Minimum voltage required across the + and - block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**. The default value is 0.8 V.

#### **On resistance**

Rate of change of voltage versus current above the **Forward voltage**. The default value is 0.001 Ohm.

#### **Off conductance**

Conductance of the reverse-biased diode. The default value is  $1e-5$  1/Ohm.

#### **Junction capacitance**

Diode junction capacitance. The default value is 50 nF.

#### **Peak reverse current, iRM**

Peak reverse current measured by an external test circuit. This value must be less than zero. The default value is -235 A.

#### **Initial forward current when measuring iRM**

Initial forward current when measuring peak reverse current. This value must be greater than zero. The default value is 300 A.

#### **Rate of change of current when measuring iRM**

Rate of change of current when measuring peak reverse current. This value must be less than zero. The default value is -50 A/ $\mu$ s.

#### **Reverse recovery time parameterization**

Determines how you specify reverse recovery time in the block. The default value is Specify reverse recovery time directly.

If you select Specify stretch factor or Specify reverse recovery charge, you specify a value that the block uses to derive the reverse recovery time. For more information on these options, see "Alternatives to Specifying trr Directly" on page 1-339.

### **Reverse recovery time, trr**

Interval between the time when the current initially goes to zero (when the diode turns off) and the time when the current falls to less than 10% of the peak reverse current. The default value is 15  $\mu$ s.

This parameter is visible only if you set **Reverse recovery time parameterization** to **Specify reverse recovery time directly**.

The value of the **Reverse recovery time, trr** parameter must be greater than the value of the **Peak reverse current, iRM** parameter divided by the value of the **Rate of change of current when measuring iRM** parameter.

### **Reverse recovery time stretch factor**

Value that the block uses to calculate **Reverse recovery time, trr**. This value must be greater than 1. The default value is 3.

This parameter is visible only if you set **Reverse recovery time parameterization** to **Specify stretch factor**.

Specifying the stretch factor is an easier way to parameterize the reverse recovery time than specifying the reverse recovery charge. The larger the value of the stretch factor, the longer it takes for the reverse recovery current to dissipate.

### **Reverse recovery charge, Qrr**

Value that the block uses to calculate **Reverse recovery time, trr**. Use this parameter if the data sheet for your diode device specifies a value for the reverse recovery charge instead of a value for the reverse recovery time.

The reverse recovery charge is the total charge that continues to dissipate when the

diode turns off. The value must be less than  $-\frac{i_{RM}^2}{2a}$ ,

where:

- $i_{RM}$  is the value specified for **Peak reverse current, iRM**.
- $a$  is the value specified for **Rate of change of current when measuring iRM**.

The default value is 1500  $\mu$ As.

The parameter is visible only if you set **Reverse recovery time parameterization** to **Specify reverse recovery charge**.



For more information on these parameters, see Commutation Diode.

## Snubbers Tab

### Snubber

Snubber for each switching device. The default value is None.

### Snubber resistance

This parameter is visible only if you set **Snubber** to RC snubber. The default value is 0.1 Ohm.

### Snubber capacitance

This parameter is visible only if you set **Snubber** to RC snubber. The default value is 1e-7 F.

## Ports

The block has the following ports:

G

Vector input port associated with the gate terminals of the switching devices. Connect this port to a Twelve-Pulse Gate Multiplexer block.

~

Expandable three-phase port.

+

Electrical conserving port associated with the DC positive terminal.

0

Electrical conserving port associated with the DC neutral terminal.

-

Electrical conserving port associated with the DC negative terminal.

## See Also

Average-Value Inverter | Average-Value Rectifier | Converter | Rectifier | Twelve-Pulse Gate Multiplexer

## **Topics**

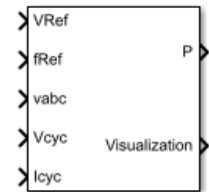
“Expand and Collapse Three-Phase Ports on a Block”

**Introduced in R2014b**

# Three-Phase Bridge Cycloconverter Voltage Controller

PI-based RMS voltage control for three-phase bridge cycloconverters

**Library:** Simscape / Electrical / Power Systems / Control / Converter Control



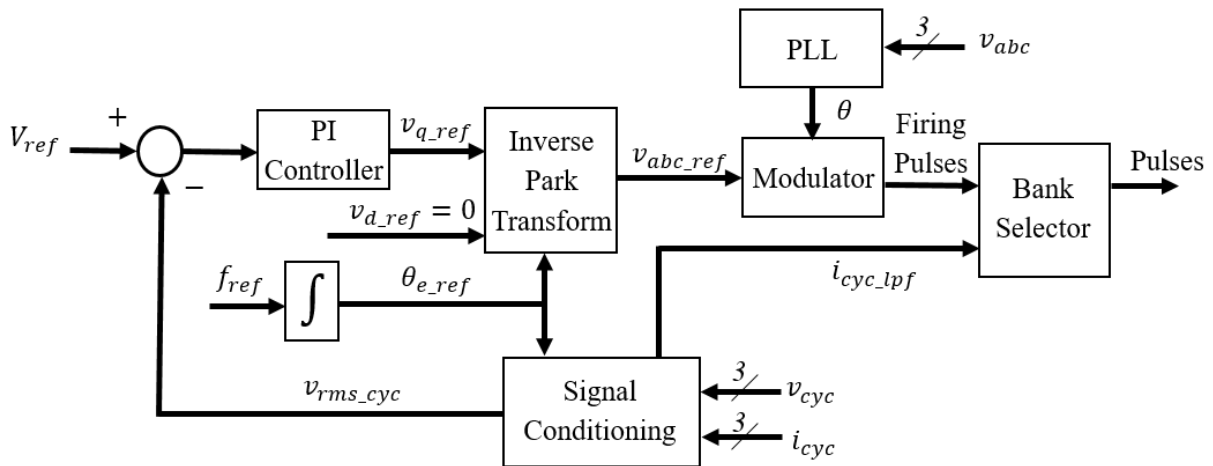
## Description

The Three-Phase Bridge Cycloconverter Voltage Controller block implements a PI-based root-mean-square (RMS) voltage controller for three-phase bridge cycloconverters.

To convert a three-phase signal directly from a higher frequency to a lower frequency, use this block with a three-phase bridge cycloconverter. Refer to “Three-Phase Bridge Cycloconverter” for an example of such a conversion.

## Operating Principle

The controller regulates the cycloconverter line-to-neutral RMS voltage to a given value and a given electrical frequency. The structure of the cycloconverter controller is illustrated in this diagram.

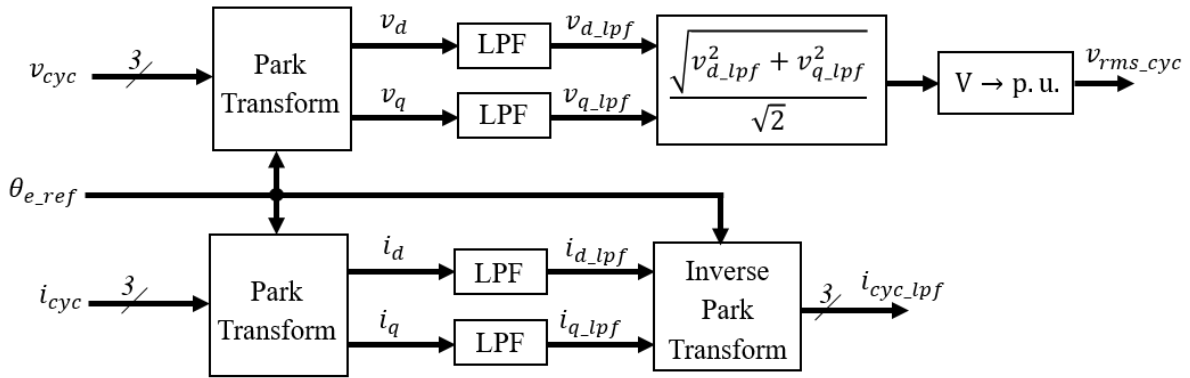


In the diagram:

- The controller integrates the desired output frequency  $f_{ref}$  to produce the reference electrical angle  $\theta_{e\_ref}$ .
- The Signal Conditioning block filters the cycloconverter line-to-neutral voltage  $v_{cyc}$  and current  $i_{cyc}$  to produce the per-unit RMS voltage  $v_{rms\_cyc}$  and smoothed current signal  $i_{cyc\_lpf}$ .
- The PI Controller generates a reference phase voltage in the  $q$ -axis from the error between the desired output RMS voltage  $V_{ref}$  and  $v_{rms\_cyc}$ .
- The Inverse Park Transform block converts the reference phase voltage in  $dq0$ -coordinates to a phase voltage  $v_{abc\_ref}$  in  $abc$ -coordinates.
- The Three-Phase Sinusoidal Measurement (PLL) block estimates the phase angle  $\theta$  of the input voltage signal  $v_{abc}$ .

The Modulator and Bank Selector blocks create the 36 pulses to drive the cycloconverter using the reference phase voltage  $v_{abc\_ref}$ , estimated phase angle  $\theta$ , and filtered cycloconverter current  $i_{cyc\_lpf}$ . To generate the firing angles, the controller uses the cosine wave crossing method.

This diagram shows the signal conditioning logic.



In the diagram:

- The Park Transform blocks convert the measured cycloconverter voltage  $v_{cyc}$  and current  $i_{cyc}$  into  $d$ - and  $q$ -axis components ( $v_d, v_q, i_d, i_q$ ) using the reference electrical angle  $\theta_{e\_ref}$ .
- The Low-Pass Filter (LPF) blocks remove the high-frequency noise from each of the  $d$ - and  $q$ -axis voltage and currents to produce the filtered components ( $v_{d\_lpf}, v_{q\_lpf}, i_{d\_lpf}, i_{q\_lpf}$ ).
- The block calculates the cycloconverter per-unit RMS voltage  $v_{rms\_cyc}$  by taking the squared sum of the  $dq$  components, dividing by  $\sqrt{2}$ , and finally converting from SI to per-unit representation.
- The Inverse Park Transform converts the  $dq$  filtered current back to the  $abc$ -axis and outputs it as  $i_{cyc\_lpf}$ .

The cycloconverter reference line-to-neutral rms voltage output is given in per-unit representation.

## Visualization

The block outputs a bus containing six signals for visualization:

- The estimated phase angle  $\theta$  of the input voltage signal  $v_{abc}$
- The desired RMS voltage  $V_{ref}$  of the output signal

- The reference phase voltages  $v_{abc\_ref}$  of the desired output signal
- The filtered line-to-neutral cycloconverter RMS voltage  $v_{rms\_cyc}$
- The filtered cycloconverter phase currents  $i_{cyc\_lpf}$
- The filtered cycloconverter phase voltages  $v_{cyc\_lpf}$

## Ports

### Input

#### **VRef — Reference voltage**

scalar

Reference line-to-neutral RMS voltage, expressed in per-unit representation.

Data Types: `single` | `double`

#### **fRef — Reference frequency**

scalar

Reference electrical frequency, in Hz.

Data Types: `single` | `double`

#### **vabc — Phase voltages**

vector

Measured phase voltages of the source, in V.

Data Types: `single` | `double`

#### **Vcyc — Cycloconverter voltages**

vector

Measured cycloconverter phase voltages, in V.

Data Types: `single` | `double`

#### **Icyc — Cycloconverter currents**

vector

Measured cycloconverter phase currents, in A.

Data Types: single | double

## Output

### **P – Pulses**

vector

Thyristor pulse vector to control a three-phase bridge cycloconverter.

Data Types: single | double

### **Visualization – Visualization bus**

bus

Bus containing internal signals for visualization. For a full list of signals, refer to the “Visualization” on page 1-1903 section.

Data Types: single | double

## Parameters

### **Rated voltage (phase-to-phase RMS) – Rated RMS voltage**

6000 (default) | positive number

Rated RMS voltage for per-unit conversion calculations, in V.

### **Loop filter proportional gain – LF proportional gain**

2 (default) | positive number

Loop filter proportional gain for the phase-locked loop (PLL) estimating the phase of the input signal. This value determines the aggressiveness of the PLL in tracking and locking to the phase angle. Increase this value to improve reaction time of the tracking to step changes in the phase angle.

### **Loop filter integral gain – LF integral gain**

20 (default) | positive number

Loop filter integral gain for the phase-locked loop (PLL) estimating the phase of the input signal. Increase this value to increase the rate at which steady-state error is eliminated in the phase angle. This value also determines the aggressiveness of the PLL in tracking and locking to the phase.

## **Filters time constant (s) — Time constant**

1e-2 (default) | positive number

Time constant of the low-pass filters in the Signal Conditioning block of the controller. These filters reduce undesired high-frequency noise in the cycloconverter phase voltage and current measurements.

## **Controller proportional gain — Proportional gain**

1 (default) | positive number

Proportional gain for the PI-controller that generates the reference phase voltage for the cycloconverter. Increase this value to increase the aggressiveness of the controller.

## **Controller integral gain — Integral gain**

12 (default) | positive number

Integral gain of the PI-controller that generates the reference phase voltage for the cycloconverter. Increase this value to increase the rate at which steady-state error is eliminated in the phase voltage signal.

## **Controller anti-windup gain — Anti-windup gain**

10 (default) | positive number

Anti-windup gain of the PI-controller that generates the reference phase voltage for the cycloconverter.

## **Thyristor pulse width (rad) — Pulse width**

5\*pi/6 (default) | positive number

Angular width of pulses sent to the cycloconverter.

## **Bank selector current threshold (A) — Current threshold**

5 (default) | positive number

Current threshold for switching between positive and negative converters.

## **Pulse ordering — Pulse ordering rule**

Sequential device order (default) | Natural order of commutation

Strategy used for the ordering of generated pulses.

## **Sample time (-1 for inherited) — Sample time**

-1 (default) | positive number



Sample time for the block (-1 for inherited). If you use this block inside a triggered subsystem, set the sample time to -1. If you use this block in a continuous variable-step model, set the sample time explicitly.

## References

- [1] Chen, H., M. H. Johnson, and D. C. Aliprantis. "Low-frequency AC transmission for offshore wind power." *IEEE Transactions on Power Delivery*. Vol. 28, Number 4, 2013, pp. 2236-2244.

## See Also

### Blocks

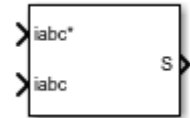
Integrator with Wrapped State (Discrete or Continuous) | Low-Pass Filter (Discrete or Continuous) | Park Transform | Three-Phase Sinusoidal Measurement (PLL)

**Introduced in R2017b**

# Three-Phase Hysteresis Current Controller

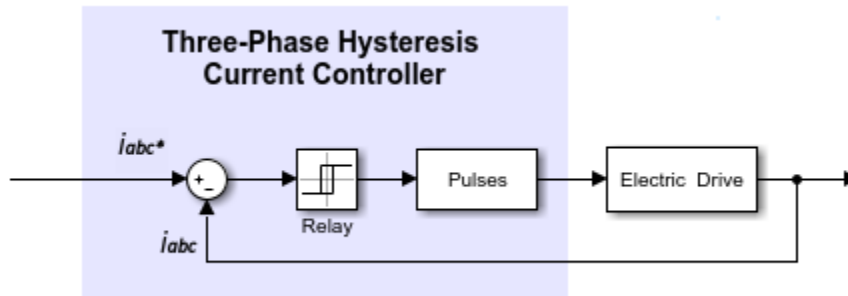
Three-phase hysteresis current control

**Library:** Simscape / Electrical / Power Systems / Control / General Machine Control



## Description

The Three-Phase Hysteresis Current Controller block implements three-phase hysteresis current control for power converters.



## Ports

### Input

**$i_{abc}^*$  — Current**  
vector

Three-phase reference currents.

Data Types: single | double

**iabc — Current**

vector

Measured three-phase currents.

Data Types: `single` | `double`**Output****S — Controller output**

vector

Six-pulse vector for power converter control.

Data Types: `single` | `double`**Parameters****Current hysteresis band (A) — Hysteresis band**

1 (default) | positive number

Hysteresis band,  $h$ , for the current controller. The switch-on point is  $h/2$  and the switch-off point is  $-h/2$ .

**Sample time (-1 for inherited) — Block sample time**

-1 (default) | positive scalar

Time, in s, between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

If this block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly using a positive scalar.

**See Also**

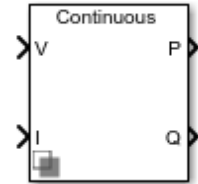
Brushless DC Motor | Velocity Controller

**Introduced in R2018a**

# Three-Phase Power Measurement

Measure three-phase real and reactive power

**Library:** Simscape / Electrical / Power Systems / Control / Measurements



## Description

The Three-Phase Power Measurement block measures the real and reactive power of an element in a three-phase network. The block outputs the power quantities for each frequency component you specify in the selected symmetrical sequence.

Use this block to measure power for both sinusoidal and nonsinusoidal periodic signals. For single-phase power measurement, consider using the Power Measurement block.

Set the **Sample time** parameter to 0 for continuous-time operation, or explicitly for discrete-time operation.

Specify a vector of all frequency components to include in the power output using the **Harmonic numbers** parameter:

- To output the DC component, specify 0.
- To output the component corresponding to the fundamental frequency, specify 1.
- To output components corresponding to higher-order harmonics, specify  $n > 1$ .

## Equations

For each specified harmonic  $k$ , the block calculates the real power  $P_k$  and reactive power  $Q_k$  for the specified sequence from the phasor equation:

$$P_k + jQ_k = \frac{3}{2} (V_k e^{j\theta_k}) (\overline{I_k e^{j\theta_k}}),$$

where:

- $V_k e^{j\theta_{V_k}}$  is the phasor representing the  $k$ -component voltage of the selected sequence.
- $\overline{I_k e^{j\theta_{I_k}}}$  is the complex conjugate of  $I_k e^{j\theta_{I_k}}$ , the phasor representing the  $k$ -component current of the selected sequence.

Select the symmetrical sequence used in the power calculation using the **Sequence** parameter:

- **Positive:**

$$V_k e^{j\theta_{V_k}} = V_{k+} e^{j\theta_{V_{k+}}}, I_k e^{j\theta_{I_k}} = I_{k+} e^{j\theta_{I_{k+}}}$$

- **Negative:**

$$V_k e^{j\theta_{V_k}} = V_{k-} e^{j\theta_{V_{k-}}}, I_k e^{j\theta_{I_k}} = I_{k-} e^{j\theta_{I_{k-}}}$$

- **Zero:**

$$V_k e^{j\theta_{V_k}} = V_{k0} e^{j\theta_{V_{k0}}}, I_k e^{j\theta_{I_k}} = I_{k0} e^{j\theta_{I_{k0}}}$$

The block calculates the symmetrical set of  $+0$  voltage phasors from the set of  $abc$  voltage phasors using the symmetrical components transform  $S$ :

$$\begin{bmatrix} V_{k+} e^{j\theta_{V_{k+}}} \\ V_{k-} e^{j\theta_{V_{k-}}} \\ V_{k0} e^{j\theta_{V_{k0}}} \end{bmatrix} = S \begin{bmatrix} V_{ka} e^{j\theta_{V_{ka}}} \\ V_{kb} e^{j\theta_{V_{kb}}} \\ V_{kc} e^{j\theta_{V_{kc}}} \end{bmatrix}.$$

For more information about this transform, see Symmetrical Components Transform.

The block obtains this set of  $abc$  voltage phasors from the three-phase input voltage  $V(t)$  as:

$$\begin{bmatrix} V_{ka} e^{j\theta_{V_{ka}}} \\ V_{kb} e^{j\theta_{V_{kb}}} \\ V_{kc} e^{j\theta_{V_{kc}}} \end{bmatrix} = \frac{2}{T} \int_{t-T}^t V(t) \sin(2\pi k F t) dt + j \frac{2}{T} \int_{t-T}^t V(t) \cos(2\pi k F t) dt,$$

where  $T$  is the period of the input signal, or equivalently the inverse of its base frequency  $F$ .

The block calculates the symmetrical set of current phasors in the same way as it does the voltage.

If the input signals have a finite number of harmonics  $n$ , the total real power  $P$  and total reactive power  $Q$  for the specified sequence can be calculated from their components:

$$P = \sum_{k=0}^n P_k$$

$$Q = \sum_{k=1}^n Q_k.$$

The summation for  $Q$  does not include the DC component ( $k = 0$ ), because this component only contributes to real power.

## Ports

### Input

#### **V** — Input voltage

vector

Three-phase voltage across element from which to measure power, in V.

Data Types: single | double

#### **I** — Input current

vector

Three-phase current through element from which to measure power, in A.

Data Types: `single` | `double`

## Output

### **P — Real power**

vector

Real power for selected frequency components, in W.

Data Types: `single` | `double`

### **P — Reactive power**

vector

Reactive power for selected frequency components, in var.

Data Types: `single` | `double`

## Parameters

### **Base frequency (Hz) — Fundamental frequency**

60 (default) | positive number

Fundamental frequency corresponding to component  $k=1$ .

### **Harmonic numbers — Frequency components**

[0 1 2 3] (default) | vector

Frequency components to include in the output. Specify either a scalar value corresponding to the desired component or a vector of all desired components.

- The value  $k = 0$  corresponds to the DC component.
- The value  $k = 1$  corresponds to the fundamental frequency.
- Values  $k > 1$  correspond to higher-level harmonics.

If you specify a vector, the order of the power outputs correspond to the order of this vector.



**Sequence — Symmetrical sequence**

Positive (default) | Negative | Zero

Symmetrical sequence of the power output.

**Sample time — Block sample time** $\emptyset$  (default) | positive number

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For continuous operation, set this property to  $\emptyset$ . For discrete operation, specify the sample time explicitly as a positive number. This block does not support inherited sample time.

If this block is in a masked subsystem, or other variant subsystem that allows either continuous and discrete operation, promote the sample time parameter. Promoting the sample time parameter ensures correct switching between the continuous and discrete implementations of the block. For more information, see “Promote Parameter to Mask” (Simulink).

## See Also

**Blocks**

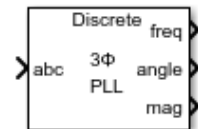
Power Measurement | RMS Measurement | Sinusoidal Measurement (PLL) | Three-Phase Sinusoidal Measurement (PLL)

**Introduced in R2017b**

## Three-Phase Sinusoidal Measurement (PLL)

Estimate three-phase sinusoidal characteristics using a phase-locked loop

**Library:** Simscape / Electrical / Power Systems / Control / Measurements



### Description

The Three-Phase Sinusoidal Measurement (PLL) block estimates the frequency characteristics of a balanced three-phase sinusoidal signal. The block uses a standard phase-locked loop (PLL) strategy to estimate the frequency and phase angle of the input signal. It also outputs the magnitude of the input signal.

Use this block in control applications when the frequency, phase angle, or magnitude are required and cannot be measured directly. To estimate the frequency characteristics of a non-three-phase or unbalanced sinusoidal signal, use the Sinusoidal Measurement (PLL) block instead.

### Equations

The phase-locked loop generates a sinusoid that approximates the input signal  $u(t)$  with the form:

$$y(t) = A(t) \sin\left(\phi_0 + \int 2\pi f(t) dt\right),$$

where:

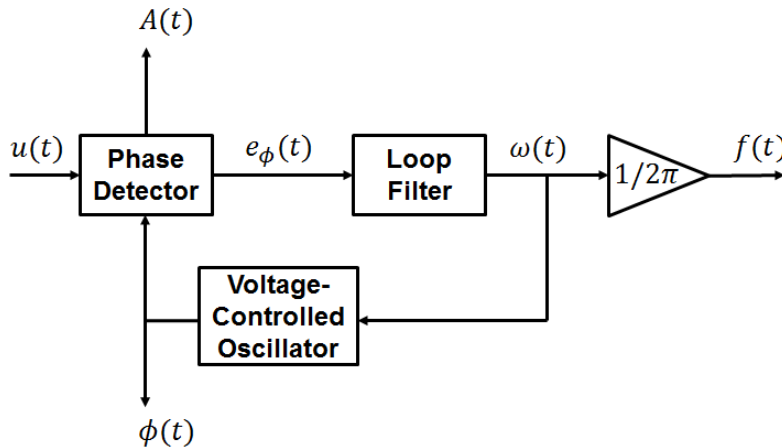
- $y$  is the estimate of the input signal.
- $A$  is the amplitude of the input signal.
- $\phi_0$  is the initial phase angle of the input signal.

Because the input signal is assumed to be balanced, the block calculates the amplitude directly from the instantaneous amplitude of the three phases. The estimated phase angle  $\phi$  is the angle of this generated sinusoid:

$$\phi(t) = \phi_0 + \int 2\pi f(t) dt,$$

where  $f$  is the frequency of the sinusoid, and  $\phi_0$  is the initial phase angle.

This diagram shows the overall structure of the phase-locked loop.



In the diagram:

- The phase detector produces an error signal relative to the phase difference  $e_\phi$  between the input sinusoid  $u$  and the synthesized sinusoid  $y$ . It also outputs the amplitude  $A$ .
- The loop filter provides an estimate of the input angular frequency  $\omega$  by filtering out the high-frequency components of the phase difference. The block also outputs the converted frequency  $f$  in Hz.
- The voltage-controlled oscillator integrates the angular speed to produce the phase estimate  $\phi$  which it sends to the Phase Detector for comparison.

## Ports

### Input

#### **abc — Input signal**

vector

Three-phase input signal.

Data Types: `single` | `double`

### Output

#### **freq — Frequency**

scalar

Estimated frequency of the input signal, in Hz.

Data Types: `single` | `double`

#### **angle — Phase angle**

scalar

Estimated phase angle of the first phase of the input signal, in rad.

Data Types: `single` | `double`

#### **mag — Magnitude**

scalar

Magnitude of the input signal.

Data Types: `single` | `double`

## Parameters

#### **Loop filter proportional gain — LF proportional gain**

200 (default) | positive scalar

Proportional gain for the loop filter. Increase this value to increase the rate at which steady-state error is eliminated in the phase angle. This value also determines the aggressiveness of the PLL in tracking and locking to the phase angle.

**Loop filter integral gain – LF integral gain**

2000 (default) | positive scalar

Integral gain for the loop filter. This determines the aggressiveness of the PLL in tracking and locking to the phase. Increase this value to reduce and eliminate steady-state error in the phase angle.

**Initial frequency (Hz) – Initial frequency**

60 Hz (default) | scalar

Initial estimate of the input frequency.

**Initial phase angle (rad) – Initial phase**

0 rad (default) | scalar or vector

Initial estimate of the phase angle.

**Sample time (-1 for inherited) – Block sample time**

-1 (default) | 0 | positive scalar

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For inherited discrete-time operation, specify -1. For discrete-time operation, specify a positive integer. For continuous-time operation, specify 0.

If this block is in a masked subsystem, or other variant subsystem that allows you to switch between continuous operation and discrete operation, promote the sample time parameter. Promoting the sample time parameter ensures correct switching between the continuous and discrete implementations of the block. For more information, see “Promote Parameter to Mask” (Simulink).

## See Also

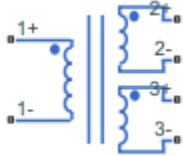
**Blocks**

RMS Measurement | Sinusoidal Measurement (PLL)

**Introduced in R2017b**

# Three-Winding Mutual Inductor

Model three coupled inductors



## Library

Passive Devices

## Description

The Three-Winding Mutual Inductor block represents a set of three coupled inductors or windings. The voltage across the three windings is

$$V_1 = L_1 \frac{dI_1}{dt} + M_{12} \frac{dI_2}{dt} + M_{13} \frac{dI_3}{dt}$$

$$V_2 = M_{12} \frac{dI_1}{dt} + L_2 \frac{dI_2}{dt} + M_{23} \frac{dI_3}{dt}$$

$$V_3 = M_{13} \frac{dI_1}{dt} + M_{23} \frac{dI_2}{dt} + L_3 \frac{dI_3}{dt}$$

where:

- $V_i$  is voltage across the  $i$ th winding.
- $I_i$  is current through the  $i$ th winding.
- $L_i$  is self inductance of the  $i$ th winding.
- 

$M_{ij}$  is mutual inductance of the  $i$ th and  $j$ th windings,  $M_{ij} = K_{ij} \sqrt{L_i L_j}$ .

In the preceding equations, currents are positive when flowing into the positive node of their respective inductor terminals.

When you run a simulation that includes this block, the software checks the specified parameter values to ensure that the resulting device is passive. If it is not, the software issues an error.

## Parameters

### Inductance L1

The self inductance of the first winding. The default value is 0.001 H.

### Inductance L2

The self inductance of the second winding. The default value is 0.001 H.

### Inductance L3

The self inductance of the third winding. The default value is 0.001 H.

### Coefficient of coupling, K12

The coefficient that defines the mutual inductance between the first and second windings. The default value is 0.9. The absolute value must be between 0 and 1, exclusive.

### Coefficient of coupling, K13

The coefficient that defines the mutual inductance between the first and third windings. The default value is 0.9. The absolute value must be between 0 and 1, exclusive.

### Coefficient of coupling, K23

The coefficient that defines the mutual inductance between the second and third windings. The default value is 0.9. The absolute value must be between 0 and 1, exclusive.

### Specify initial condition

Select one of the following options for specifying an initial condition:

- No — Do not specify an initial condition for the model. This is the default option.
- Yes — Specify the initial inductor currents.



**Initial current port 1, IC1**

The current flowing through the first winding at the start of the simulation. This parameter is only visible when you select **Yes** for the **Specify initial condition** parameter. The default value is 0 A.

**Initial current port 2, IC2**

The current flowing through the second winding at the start of the simulation. This parameter is only visible when you select **Yes** for the **Specify initial condition** parameter. The default value is 0 A.

**Initial current port 3, IC3**

The current flowing through the third winding at the start of the simulation. This parameter is only visible when you select **Yes** for the **Specify initial condition** parameter. The default value is 0 A.

## Ports

The block has the following ports:

1+

Positive electrical voltage of the first mutual inductor

1-

Negative electrical voltage of the first mutual inductor

2+

Positive electrical voltage of the second mutual inductor

2-

Negative electrical voltage of the second mutual inductor

3+

Positive electrical voltage of the third mutual inductor

3-

Negative electrical voltage of the third mutual inductor

# Thyristor

Model thyristor using NPN and PNP transistors



## Library

Semiconductor Devices

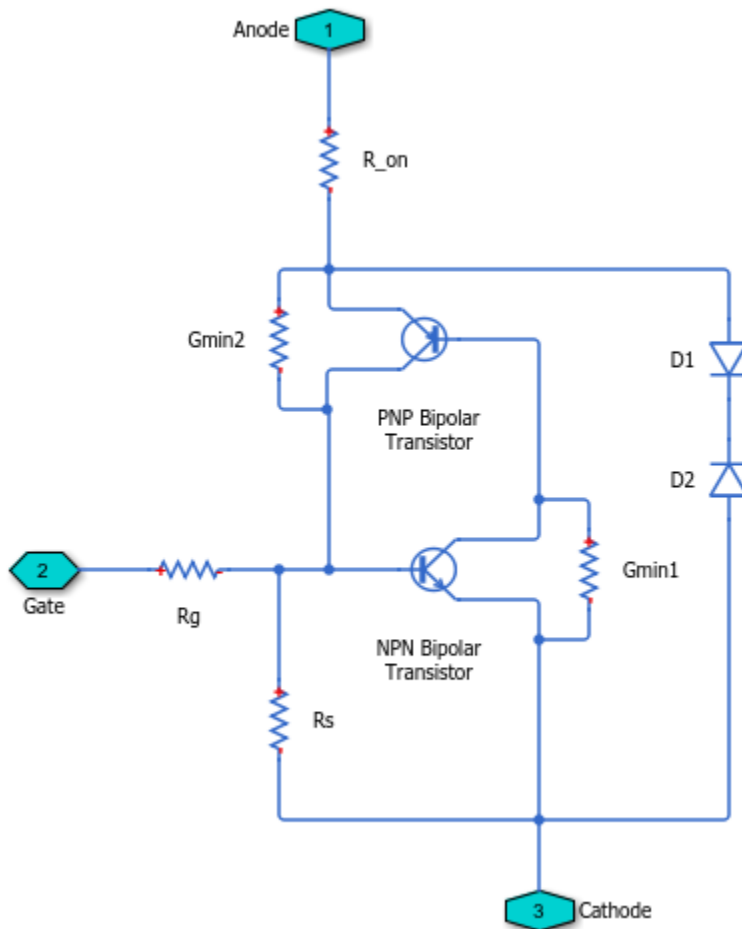
## Description

The Thyristor block provides two ways of modeling a thyristor:

- As an equivalent circuit based on NPN and PNP bipolar transistors
- By a lookup table approximation to the on-state I-V (current-voltage) curve

## Representation by Equivalent Circuit

The equivalent circuit contains a pair of NPN and PNP bipolar transistors, as shown in the following illustration.



The P-N-P-N structure of a thyristor is matched by the P-N-P and N-P-N structures of the bipolar transistors, the base of each device being connected to the collector of the other device. Ensuring that this circuit behaves like a thyristor is primarily picking suitable parameter values of the NPN and PNP devices, plus external resistors. For example, for the circuit to latch into the on-state, once triggered by a suitable gate current, the total gain of the two transistors must be greater than one. This model structure replicates the behavior of a thyristor in typical application circuits, while at the same time presenting a minimum number of equations to the solver, to improve simulation speed.

---

**Note** It is extremely important that you parameterize the thyristor component correctly before using it in your model. To help you do this, there are two test harnesses in the Simscape Electrical Electronics and Mechatronics examples, Thyristor Static Behavior Validation and Thyristor Dynamic Behavior Validation. Follow the help text for these two examples, plus a datasheet for your device, to re-parameterize the thyristor component so that it replicates the required behavior. You can then copy the parameterized component into your model. Take care to model the gate drive circuitry correctly, including circuit series resistance. Connecting a controlled voltage source directly to the thyristor gate gives nonphysical results because it clamps the gate to the cathode voltage when the gate demand is zero.

---

The model captures the following thyristor behaviors:

- Off-state currents,  $I_{DRM}$  and  $I_{RRM}$ . These are typically quoted for the maximum off-state voltages  $V_{DRM}$  and  $V_{RRM}$ . It is assumed, as is the case for most thyristors, that  $I_{DRM} = I_{RRM}$  and  $V_{DRM} = V_{RRM}$ .
- The gate trigger voltage is equal to the **Gate trigger voltage, v\_GT** parameter value when the gate current is equal to the **Gate trigger current, i\_GT** parameter value.
- The thyristor latches on when the gate current is equal to the **Gate trigger current, i\_GT**. The thyristor does not latch on until the gate current reaches this value. To ensure this is the case, you must set the **Internal shunt resistor, Rs** parameter correctly. If the resistance is too high, then the gate triggers before the gate current reaches  $i_{GT}$ . If the resistance is too small, then the gate does not trigger.

You can determine the value of the internal shunt resistor  $R_s$  by running the simulation. To see how this can be done, refer to the Thyristor Static Behavior Validation example. Alternatively, if you are using the thyristor in a circuit where there is an external resistor  $R_{GK}$  connected from gate to cathode, then the effect of  $R_s$  is usually very small, and it can be set to `inf`.

- With the thyristor in the on-state, if the gate current is removed, the thyristor stays in the on-state, provided that the load current is higher than the latching current. You do not specify the latching current directly because its value is primarily determined by other block parameters.

However, the latching current can be influenced by the **Product of NPN and PNP forward current gains** parameter on the **Advanced** tab. Reducing the gain increases the latching current.

- The on-state voltage is equal to the **On-state voltage, V\_T** parameter value when the load current is equal to the **On-state current, I\_T** parameter value. This is ensured

by the  $R_{on}$  resistance value, which takes into account the voltage drop seen across the PNP and NPN devices.

- Triggering by rate of rise of off-state voltage. A rapid change in anode-cathode voltage induces a current in the base-collector capacitance terms. If this current is large enough, it triggers the thyristor into the on-state. The thyristor initialization routine calculates a suitable value for the base-collector capacitance, so that when the rate of change of voltage is equal to the **Critical rate of rise of off-state voltage,  $dV/dt$**  parameter value, the thyristor triggers on. This calculation is based on the approximation that the required current is  $v_{GT} / R_{GK}$  where  $R_{GK}$  is the gate-cathode resistance value used when quoting the critical  $dV/dt$  value.
- A nonzero gate-controlled turn-on time. This is primarily influenced by the **NPN transistor forward transit time, TF**. You either specify this parameter directly, or calculate an approximate value for TF from the turn-on time.
- A nonzero commuted turn-off time. This is primarily influenced by the **PNP transistor forward transit time, TF**. You can either specify this parameter directly, or set it to be equal to the forward transit time for the NPN transistor.

Resistors  $G_{min1}$  and  $G_{min2}$  improve numerical robustness at large forward and reverse voltages. Their values influence the off-state currents by no more than 1% at the maximum off-state forward and reverse voltages.

---

**Note** Because this block implementation includes a charge model, you must model the impedance of the circuit driving the gate to obtain representative turn-on and turn-off dynamics. Therefore, if you are simplifying the gate drive circuit by representing it as a controlled voltage source, you must include a suitable series resistor between the voltage source and the gate.

---

## Representation by Lookup Table

If using the lookup table representation, you provide tabulated values for anode-cathode current as a function of anode-cathode voltage when in the on state. The main advantages of using this option are simulation speed and ease of parameterization. To further simplify the underlying model, this representation does not model:

- Device triggering due to rate of rise of off-state voltage
- Commuted turn-off time

The turn-on delay is represented by an input gate-cathode capacitor, the value of which is calculated so that the delay between gate voltage rising and the device starting to turn on is equal to the value specified by the **Turn-on delay time** parameter. The turn-on rise time for the load current is implemented by ramping nonlinearly between zero and the current determined by on-state current-voltage profile over a time period specified by the value of the **Turn-on rise time** parameter. Note that the resulting turn-on current profile is an approximation to an actual device.

## Thermal Port

The block has an optional thermal port, hidden by default. To expose the thermal port, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal port H on the block icon, and adds the **Thermal Port** tab to the block dialog box.

Use the thermal port to simulate the effects of generated heat and device temperature. For more information on using thermal ports and on the **Thermal Port** tab parameters, see “Simulating Thermal Effects in Semiconductors”.

## Basic Assumptions and Limitations

The Thyristor block has the following limitations:

- This block does not model temperature-dependent effects. Simscape Electrical Electronics and Mechatronics simulates the block at the temperature at which the component behavior was measured, as specified by the **Measurement temperature** parameter value. All parameters must be quoted for this temperature.
- If you use the equivalent circuit representation:
  - In sensitive gate circuits (that is, where there is no external gate-cathode resistor  $R_{GK}$ ), you must set the value of the **Internal shunt resistor,  $R_s$**  parameter to ensure correct triggering. If the internal shunt resistance is too high, then the thyristor triggers for currents less than  $i_{GT}$ . If the internal shunt resistance is too low, the thyristor does not trigger for an input current of  $i_{GT}$ . For details on using simulation to determine the acceptable internal shunt resistance value, see the Thyristor Static Behavior Validation example.
  - Triggering by exceeding the break-over voltage is not modeled.
  - Numerically the thyristor can be demanding to simulate, given the very small gate currents in comparison to the load current, and also the very steep current

gradients during switching. However, for most typical thyristor-based circuits, you can use the default simulation parameters. In some cases you may need to tighten the **Absolute Tolerance** and **Relative Tolerance** parameters on the **Solver** tab of the Configuration Parameters dialog box, to ensure convergence. In such cases, changing the default value of **Absolute Tolerance** from `auto` to `1e-4` or `1e-5` is usually sufficient, because it prevents adaptive changing of this parameter during simulation.

- The leakage currents are approximated by the diodes D1 and D2, as shown in the equivalent circuit. This approach assumes that the leakage via the two transistors is small in comparison. This assumption is not valid for values of  $v_{GT}$  that are significantly smaller than the typical forward voltage drop of 0.6 V.
- If you use the lookup table representation:
  - Triggering by exceeding the break-over voltage or by rate of change of off-state voltage is not modeled.
  - Commutated turn-off time is not modeled. Check that your circuit does not violate the stated commutated turn-off time for the thyristor.
  - When you specify a turn-on rise time, the resulting current-time profile is an approximation.

## Parameters

- “Main Tab” on page 1-1929
- “Gate Triggering Tab” on page 1-1931
- “dV/dt Triggering Tab” on page 1-1931
- “Time Constants Tab” on page 1-1932
- “Advanced Tab” on page 1-1933

## Main Tab

### I-V characteristics defined by

Select the thyristor representation:

- `Fundamental nonlinear equations` — Use an equivalent circuit based on NPN and PNP bipolar transistors. This is the default.

- `Lookup table` — Use a lookup table approximation to the on-state I-V curve.

**On-state voltage,  $V_T$** 

The anode-cathode static voltage drop when in the on-state, and the current flowing is equal to the on-state current  $I_T$ . The default value is 1.2 V. This parameter is visible only when you select `Fundamental nonlinear equations` for the **I-V characteristics defined by** parameter.

**On-state current,  $I_T$** 

Static load (or equivalently anode) current that flows when the anode-cathode voltage is equal to the on-state voltage  $V_T$ . The default value is 1 A. This parameter is visible only when you select `Fundamental nonlinear equations` for the **I-V characteristics defined by** parameter.

**Vector of on-state voltages,  $V_T$** 

The vector of on-state voltages, to be used for table lookup. The vector values must be strictly increasing, and the first value must be greater than zero. The values can be nonuniformly spaced. The default values, in V, are [0.75 1 1.25 1.5 1.75 2 2.25]. This parameter is visible only when you select `Lookup table` for the **I-V characteristics defined by** parameter.

**Vector of corresponding currents,  $I_T$** 

The vector of currents corresponding to the on-state voltages vector values, to be used for 1D table lookup. The two vectors must be of the same size. The default values, in A, are [0.015 0.22 0.75 1.4 2 2.75 3.45]. This parameter is visible only when you select `Lookup table` for the **I-V characteristics defined by** parameter.

**Off-state current,  $I_{DRM}$** 

The off-state anode current  $I_{DRM}$  that flows when the anode-cathode voltage is equal to the off-state voltage  $V_{DRM}$ . The default value is 0.01 mA.

**Corresponding off-state voltage,  $V_{DRM}$** 

Corresponding off-state voltage,  $V_{DRM}$ . The anode-cathode voltage  $V_{DRM}$  applied with the thyristor in the off-state when quoting the off-state current  $I_{DRM}$ . The default value is 400 V.

**Holding current**

Minimum current at which the thyristor stays in the off state when gate voltage is not commanding the on state. The default value is 1 mA. This parameter is visible only when you select `Lookup table` for the **I-V characteristics defined by** parameter.



### Measurement temperature

The device simulation temperature. You must specify all block parameter values for this temperature. The default value is 25 °C. This parameter is visible only when you select Fundamental nonlinear equations for the **I-V characteristics defined by** parameter.

## Gate Triggering Tab

### Gate trigger current, I\_GT

Critical gate current  $i_{GT}$  required to turn the transistor on, resulting in a gate voltage equal to the gate trigger voltage  $v_{GT}$ . You must set the value of the **Internal shunt resistor, Rs** parameter on the **Advanced** tab to ensure that the gate triggers at  $i_{GT}$ , and not for currents less than  $i_{GT}$ . The default value is 3  $\mu$ A.

### Corresponding gate voltage, V\_GT

Gate-cathode voltage  $v_{GT}$  when the gate current is equal to the gate trigger current  $i_{GT}$ . The default value is 0.6 V.

### Test voltage, V\_D

Supply voltage used when quoting values for  $v_{GT}$  and  $i_{GT}$ . The default value is 12 V. This parameter is visible only when you select Fundamental nonlinear equations for the **I-V characteristics defined by** parameter.

### Test load resistor

Load resistor used when quoting values for  $v_{GT}$  and  $i_{GT}$ . The default value is 120  $\Omega$ . This parameter is visible only when you select Fundamental nonlinear equations for the **I-V characteristics defined by** parameter.

## dV/dt Triggering Tab

### Critical rate of rise of off-state voltage, dV/dt

Rate at which the off-state anode-cathode voltage must be increased for the thyristor to turn on. The default value is 150 V/ $\mu$ s. This parameter is visible only when you select Fundamental nonlinear equations for the **I-V characteristics defined by** parameter.

### Test gate-cathode resistor, R\_GK

Gate-cathode resistor used when quoting the critical rate of rise off off-state voltage. The default value is 1 K $\Omega$ . This parameter is visible only when you select

Fundamental nonlinear equations for the **I-V characteristics defined by** parameter.

## Time Constants Tab

### NPN device forward transit time parameterization

Select one of the following options:

- Derive approximate value from gate-controlled turn-on time — The block calculates the NPN device forward transit time based on the values for the gate-controlled turn-on time and corresponding gate current that you specify.
- Specify directly — Provide the value directly by using the **NPN device forward transit time** parameter.

This parameter is visible only when you select Fundamental nonlinear equations for the **I-V characteristics defined by** parameter.

### Gate-controlled turn-on time

Time for the gate to turn from the off to the on state when a gate current is applied. The default value is 2  $\mu\text{s}$ . This parameter is visible only when you select Fundamental nonlinear equations for the **I-V characteristics defined by** parameter and Derive approximate value from gate-controlled turn-on time for **NPN device forward transit time parameterization**.

### Corresponding gate current

The gate current used when quoting the gate-controlled turn-on time. The gate current and turn-on time are used to calculate an approximate value for the NPN transistor forward transit time on the assumption that all of the input charge is used to raise the gate voltage to the gate trigger voltage  $v_{GT}$ . The default value is 10 mA. This parameter is visible only when you select Fundamental nonlinear equations for the **I-V characteristics defined by** parameter and Derive approximate value from gate-controlled turn-on time for **NPN device forward transit time parameterization**.

### NPN device forward transit time

Represents the mean time for the minority carriers to cross the base region from the emitter to the collector of the NPN device [1]. The default value is 0.3  $\mu\text{s}$ . This parameter is visible only when you select Fundamental nonlinear equations for the **I-V characteristics defined by** parameter and Specify directly for **NPN device forward transit time parameterization**.

### PNP device forward transit time parameterization

Select one of the following options:

- `Set equal to NPN device forward transit time` — The block uses the NPN device forward transit time value.
- `Specify directly` — Provide the value directly by using the **PNP device forward transit time** parameter.

This parameter is visible only when you select `Fundamental nonlinear equations` for the **I-V characteristics defined by** parameter.

### PNP device forward transit time

Represents the mean time for the minority carriers to cross the base region from the emitter to the collector of the PNP device [1]. The default value is  $0.3 \mu\text{s}$ . This parameter is visible only when you select `Fundamental nonlinear equations` for the **I-V characteristics defined by** parameter and `Specify directly` for **PNP device forward transit time parameterization**.

### Turn-on delay time

Time delay before the device starts to turn on following a step in current on the gate from zero to the value specified by the **Gate current for turn-on delay time** parameter. The default value is  $0 \text{ s}$ . This parameter is visible only when you select `Lookup table` for the **I-V characteristics defined by** parameter.

### Gate current for turn-on delay time

The gate current used when measuring the turn-on delay time. The default value is  $1 \text{ mA}$ . This parameter is visible only when you select `Lookup table` for the **I-V characteristics defined by** parameter.

### Turn-on rise time

Time it takes for the thyristor to turn on fully following the turn-on delay event. The default value is  $0 \text{ s}$ . This parameter is visible only when you select `Lookup table` for the **I-V characteristics defined by** parameter.

## Advanced Tab

### Internal shunt resistor, $R_s$

Represents the gate-cathode shunt resistance. It is important to set this parameter value to ensure that the gate triggers at  $i_{GT}$ , and not for currents less than  $i_{GT}$ . For details, see the Thyristor Static Behavior Validation example. If you are using the thyristor in a circuit where there is an external gate-cathode resistor  $R_{GK}$ , then usually

the effect of  $R_s$  is small, and it can be set to `inf`. The default value is 87 k $\Omega$ . This parameter is visible only when you select `Fundamental nonlinear equations` for the **I-V characteristics defined by** parameter.

### **Internal series gate resistor, $R_g$**

Represents the resistance associated with the gate connection. A typical value is of the order of a few ohms, and its impact on static and dynamic characteristics is small. Therefore, its precise value is not important, but its presence helps avoid numerical simulation issues if the gate is driven directly by a voltage source. You can specify any positive value. The default value is 10  $\Omega$ .

### **Product of NPN and PNP forward current gains**

This is the product of the NPN forward gain  $BF_{NPN}$  and the PNP forward gain  $BF_{PNP}$ . The value must be greater than one for latching to occur. The smaller the value, the larger the latching current becomes. However, latching current is primarily set by other block parameters, and the total gain has only a small effect. The default value is 10. This parameter is visible only when you select `Fundamental nonlinear equations` for the **I-V characteristics defined by** parameter.

## **Ports**

The block has the following ports:

G

Electrical conserving port associated with the gate

A

Electrical conserving port associated with the anode

K

Electrical conserving port associated with the cathode

## **Examples**

See the Thyristor Static Behavior Validation and Thyristor Dynamic Behavior Validation examples.

## References

- [1] G. Massobrio and P. Antognetti. *Semiconductor Device Modeling with SPICE*. 2nd Edition, McGraw-Hill, 1993.

## See Also

NPN Bipolar Transistor | PNP Bipolar Transistor

# Thyristor (Piecewise Linear)

Thyristor

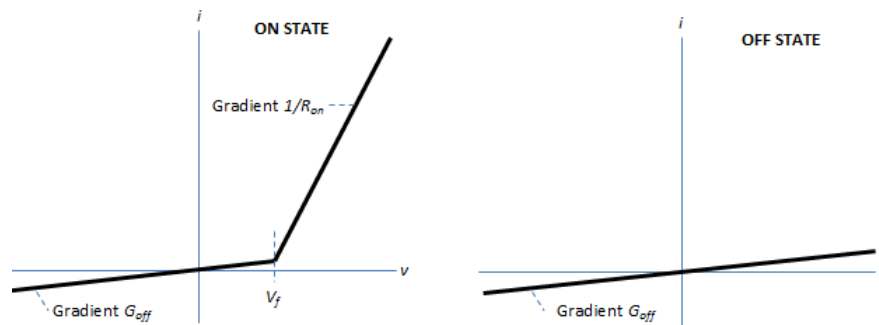


## Library

Simscape / Electrical / Power Systems / Semiconductors / Fundamental Components

## Description

The Thyristor (Piecewise Linear) block models a thyristor. The I-V characteristic for a thyristor is such that the thyristor turns on if the gate-cathode voltage exceeds the specified gate trigger voltage. The device turns off if the load current falls below the specified holding-current value.



In the on state, the anode-cathode path behaves like a linear diode with forward-voltage drop,  $V_f$ , and on-resistance,  $R_{on}$ .

In the off state, the anode-cathode path behaves like a linear resistor with a low off-state conductance,  $G_{off}$ .

The defining Simscape equations for the block are:

```

if (v > Vf)&&((G>Vgt)|| (i>Ih))
    i == (v - Vf*(1-Ron*Goff))/Ron;
else
    i == v*Goff;
end

```

where:

- $v$  is the anode-cathode voltage.
- $V_f$  is the forward voltage.
- $G$  is the gate voltage.
- $V_{gt}$  is the gate trigger voltage.
- $i$  is the anode-cathode current.
- $I_h$  is the holding current.
- $R_{on}$  is the on-state resistance.
- $G_{off}$  is the off-state conductance.

Using the Integral Diode tab of the block dialog box, you can include an integral cathode-anode diode. An integral diode protects the semiconductor device by providing a conduction path for reverse current. An inductive load can produce a high reverse-voltage spike when the semiconductor device suddenly switches off the voltage supply to the load.

The table shows you how to set the **Integral protection diode** parameter based on your goals.

Goal	Value to Select	Block Behavior
Prioritize simulation speed.	Protection diode with no dynamics	The block includes an integral copy of the Diode block. To parameterise the internal Diode block, use the <b>Protection</b> parameters.

Goal	Value to Select	Block Behavior
Precisely specify reverse-mode charge dynamics.	Protection diode with charge dynamics	The block includes an integral copy of the Commutation Diode block. To parameterise the internal Commutation Diode block, use the <b>Protection</b> parameters.

## Modeling Variants

The block provides four modeling variants. To select the desired variant, right-click the block in your model. From the context menu, select **Simscape > Block choices**, and then one of these variants:

- **PS Control Port** — Contains a physical signal port that is associated with the gate terminal. This variant is the default.
- **Electrical Control Port** — Contains an electrical conserving port that is associated with the gate terminal.
- **PS Control Port | Thermal Port** — Contains a thermal port and a physical signal port that is associated with the gate terminal.
- **Electrical Control Port | Thermal Port** — Contains a thermal port and an electrical conserving port that is associated with the gate terminal.

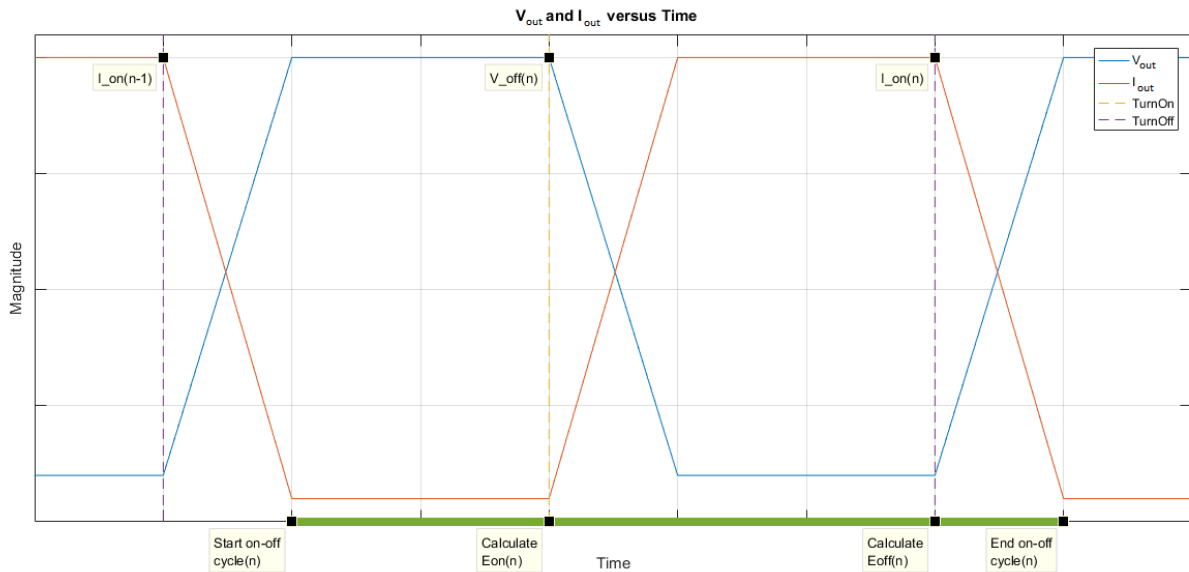
The variants of this block without the thermal port do not simulate heat generation in the device.

The variants with the thermal port allow you to model the heat that switching events and conduction losses generate. For numerical efficiency, the thermal state does not affect the electrical behavior of the block. The thermal port is hidden by default. To enable the thermal port, select a thermal block variant.

## Thermal Loss Equations

The figure shows an idealized representation of the output voltage,  $V_{out}$ , and the output current,  $I_{out}$ , of the semiconductor device. The interval shown includes the entire  $n$ th switching cycle, during which the block turns off and then on.





When the semiconductor turns on during the  $n$ th switching cycle, the amount of thermal energy that the device dissipates increments by a discrete amount. If you select **Voltage, current, and temperature** for the **Thermal loss dependent on** parameter, the equation for the incremental change is

$$E_{on(n)} = \frac{V_{off(n)}}{V_{off\_data}} fcn(T, I_{on(n-1)}),$$

where:

- $E_{on(n)}$  is the switch-on loss at the  $n$ th switch-on event.
- $V_{off(n)}$  is the off-state output voltage,  $V_{out}$ , just before the device switches on during the  $n$ th switching cycle.
- $V_{off\_data}$  is the **Off-state voltage for losses data** parameter value.
- $T$  is the device temperature.
- $I_{on(n-1)}$  is the on-state output current,  $I_{out}$ , just before the device switches off during the cycle that precedes the  $n$ th switching cycle.

The function  $fcn$  is a 2-D lookup table with linear interpolation and linear extrapolation:

$$E = \text{tablelookup}(T_{j\_data}, I_{out\_data}, E_{on\_data}, T, I_{on(n-1)}),$$

where:

- $T_{j\_data}$  is the **Temperature vector, Tj** parameter value.
- $I_{out\_data}$  is the **Output current vector, Iout** parameter value.
- $E_{on\_data}$  is the **Switch-on loss, Eon=fcn(Tj,Iout)** parameter value.

If you select **Voltage** and **current** for the **Thermal loss dependent on** parameter, when the semiconductor turns on during the  $n$ th switching cycle, the equation that the block uses to calculate the incremental change in the discrete amount of thermal energy that the device dissipates is

$$E_{on(n)} = \left( \frac{V_{off(n)}}{V_{off\_data}} \right) \left( \frac{I_{on(n-1)}}{I_{out\_scalar}} \right) (E_{on\_scalar})$$

where:

- $I_{out\_scalar}$  is the **Output current, Iout** parameter value.
- $E_{on\_scalar}$  is the **Switch-on loss** parameter value.

When the semiconductor turns off during the  $n$ th switching cycle, the amount of thermal energy that the device dissipates increments by a discrete amount. If you select **Voltage**, **current**, and **temperature** for the **Thermal loss dependent on** parameter, the equation for the incremental change is

$$E_{off(n)} = \frac{V_{off(n)}}{V_{off\_data}} \text{fcn}(T, I_{on(n)}),$$

where:

- $E_{off(n)}$  is the switch-off loss at the  $n$ th switch-off event.
- $V_{off(n)}$  is the off-state output voltage,  $V_{out}$ , just before the device switches on during the  $n$ th switching cycle.
- $V_{off\_data}$  is the **Off-state voltage for losses data** parameter value.
- $T$  is the device temperature.

- $I_{on(n)}$  is the on-state output current,  $I_{out}$ , just before the device switches off during the  $n$ th switching cycle.

The function  $fcn$  is a 2-D lookup table with linear interpolation and linear extrapolation:

$$E = \text{tablelookup}(T_{j\_data}, I_{out\_data}, E_{off\_data}, T, I_{on(n)}),$$

where:

- $T_{j\_data}$  is the **Temperature vector**, **Tj** parameter value.
- $I_{out\_data}$  is the **Output current vector**, **Iout** parameter value.
- $E_{off\_data}$  is the **Switch-off loss**, **Eoff=fcn(Tj,Iout)** parameter value.

If you select **Voltage** and **current** for the **Thermal loss dependent on** parameter, when the semiconductor turns off during the  $n$ th switching cycle, the equation that the block uses to calculate the incremental change in the discrete amount of thermal energy that the device dissipates is

$$E_{off(n)} = \left( \frac{V_{off(n)}}{V_{off\_data}} \right) \left( \frac{I_{on(n-1)}}{I_{out\_scalar}} \right) (E_{off\_scalar})$$

where:

- $I_{out\_scalar}$  is the **Output current**, **Iout** parameter value.
- $E_{off\_scalar}$  is the **Switch-off loss** parameter value.

If you select **Voltage**, **current**, and **temperature** for the **Thermal loss dependent on** parameter, then, for both the on state and the off state, the heat loss due to electrical conduction is

$$E_{conduction} = \int fcn(T, I_{out}) dt,$$

where:

- $E_{conduction}$  is the heat loss due to electrical conduction.
- $T$  is the device temperature.
- $I_{out}$  is the device output current.

The function  $fcn$  is a 2-D lookup table:

$$Q_{conduction} = \text{tablelookup}(T_{j\_data}, I_{out\_data}, I_{out\_data\_repmat} \cdot *V_{on\_data}, T, I_{out}),$$

where:

- $T_{j\_data}$  is the **Temperature vector, Tj** parameter value.
- $I_{out\_data}$  is the **Output current vector, Iout** parameter value.
- $I_{out\_data\_repmat}$  is a matrix that contains length,  $T_{j\_data}$ , copies of  $I_{out\_data}$ .
- $V_{on\_data}$  is the **On-state voltage, Von=fcn(Tj,Iout)** parameter value.

If you select **Voltage** and **current** for the **Thermal loss dependent on** parameter, then, for both the on state and the off state, the heat loss due to electrical conduction is

$$E_{conduction} = \int (I_{out} * V_{on\_scalar}) dt,$$

where  $V_{on\_scalar}$  is the **On-state voltage** parameter value.

The block uses the **Energy dissipation time constant** parameter to filter the amount of heat flow that the block outputs. The filtering allows the block to:

- Avoid discrete increments for the heat flow output
- Handle a variable switching frequency

The filtered heat flow is

$$Q = \frac{1}{\tau} \left( \sum_{i=1}^n E_{on(i)} + \sum_{i=1}^n E_{off(i)} + E_{conduction} - \int Q dt \right),$$

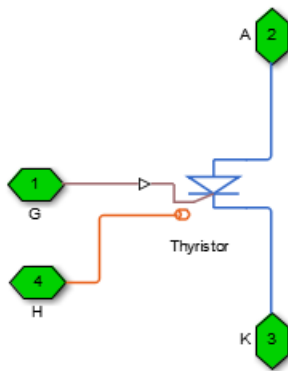
where:

- $Q$  is the heat flow from the component.
- $\tau$  is the **Energy dissipation time constant** parameter value.
- $n$  is the number of switching cycles.
- $E_{on(i)}$  is the switch-on loss at the  $i$ th switch-on event.

- $E_{off(i)}$  is the switch-off loss at the  $i$ th switch-off event.
- $E_{conduction}$  is the heat loss due to electrical conduction.
- $\int Qdt$  is the total heat previously dissipated from the component.

## Ports

The figure shows the block port names.



G

Port associated with the gate terminal. You can set the port to either a physical signal or electrical port.

A

Electrical conserving port associated with the anode terminal.

K

Electrical conserving port associated with the cathode terminal.

H

Thermal conserving port. The thermal port is optional and is hidden by default. To enable this port, select a variant that includes a thermal port.

## Parameters

- “Main Tab” on page 1-1944
- “Integral Diode Tab” on page 1-1944
- “Thermal Model Tab” on page 1-1947

### Main Tab

#### Forward voltage, Vf

Forward voltage at which the device turns on. The default value is 0.8 V.

#### On-state resistance

Anode-cathode resistance when the device is on. The default value is 0.001 Ohm.

#### Off-state conductance

Anode-cathode conductance when the device is off. The value must be less than  $1/R$ , where R is the value of **On-state resistance**. The default value is  $1e-5$  1/Ohm.

#### Gate trigger voltage, Vgt

Gate-cathode voltage threshold. The device turns on when the gate-cathode voltage is above this value. The default value is 6 V.

#### Holding current

Current threshold. The device stays on when the current is above this value, even when the gate-cathode voltage falls below the gate trigger voltage. The default value is 1 A.

### Integral Diode Tab

#### Integral protection diode

Block integral protection diode. The default value is None.

The diodes you can select are:

- Protection diode with no dynamics
- Protection diode with charge dynamics

When you select Protection diode with no dynamics, additional parameters appear.

**Additional Parameters for Protection diode with no dynamics****Forward voltage**

Minimum voltage required across the + and - block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**. The default value is 0.8 V.

**On resistance**

Rate of change of voltage versus current above the **Forward voltage**. The default value is 0.001 Ohm.

**Off conductance**

Conductance of the reverse-biased diode. The default value is  $1e-5$  1/Ohm.

For more information on these parameters, see Diode.

When you select Protection diode with charge dynamics, additional parameters appear.

**Additional Parameters for Protection diode with charge dynamics****Forward voltage**

Minimum voltage required across the + and - block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**. The default value is 0.8 V.

**On resistance**

Rate of change of voltage versus current above the **Forward voltage**. The default value is 0.001 Ohm.

**Off conductance**

Conductance of the reverse-biased diode. The default value is  $1e-5$  1/Ohm.

**Junction capacitance**

Diode junction capacitance. The default value is 50 nF.

**Peak reverse current, iRM**

Peak reverse current measured by an external test circuit. This value must be less than zero. The default value is -235 A.

**Initial forward current when measuring iRM**

Initial forward current when measuring peak reverse current. This value must be greater than zero. The default value is 300 A.

### **Rate of change of current when measuring $i_{RM}$**

Rate of change of current when measuring peak reverse current. This value must be less than zero. The default value is  $-50 \text{ A}/\mu\text{s}$ .

### **Reverse recovery time parameterization**

Determines how you specify reverse recovery time in the block. The default value is **Specify reverse recovery time directly**.

If you select **Specify stretch factor** or **Specify reverse recovery charge**, you specify a value that the block uses to derive the reverse recovery time. For more information on these options, see “Alternatives to Specifying  $t_{rr}$  Directly” on page 1-339.

### **Reverse recovery time, $t_{rr}$**

Interval between the time when the current initially goes to zero (when the diode turns off) and the time when the current falls to less than 10% of the peak reverse current. The default value is  $15 \mu\text{s}$ .

This parameter is visible only if you set **Reverse recovery time parameterization** to **Specify reverse recovery time directly**.

The value of the **Reverse recovery time,  $t_{rr}$**  parameter must be greater than the value of the **Peak reverse current,  $i_{RM}$**  parameter divided by the value of the **Rate of change of current when measuring  $i_{RM}$**  parameter.

### **Reverse recovery time stretch factor**

Value that the block uses to calculate **Reverse recovery time,  $t_{rr}$** . This value must be greater than 1. The default value is 3.

This parameter is visible only if you set **Reverse recovery time parameterization** to **Specify stretch factor**.

Specifying the stretch factor is an easier way to parameterize the reverse recovery time than specifying the reverse recovery charge. The larger the value of the stretch factor, the longer it takes for the reverse recovery current to dissipate.

### **Reverse recovery charge, $Q_{rr}$**

Value that the block uses to calculate **Reverse recovery time,  $t_{rr}$** . Use this parameter if the data sheet for your diode device specifies a value for the reverse recovery charge instead of a value for the reverse recovery time.



The reverse recovery charge is the total charge that continues to dissipate when the

diode turns off. The value must be less than  $-\frac{i_{RM}^2}{2a}$ ,

where:

- $i_{RM}$  is the value specified for **Peak reverse current, iRM**.
- $a$  is the value specified for **Rate of change of current when measuring iRM**.

The default value is 1500  $\mu$ As.

The parameter is visible only if you set **Reverse recovery time parameterization** to **Specify reverse recovery charge**.

For more information on these parameters, see Commutation Diode.

## Thermal Model Tab

The **Thermal Model** tab is enabled only when you select a block variant that includes a thermal port.

### Thermal loss dependent on

Select a parameterization method. The option that you select determines which other parameters are enabled. Options are:

- **Voltage and current** — Use scalar values to specify the output current, switch-on loss, switch-off loss, and on-state voltage data.
- **Voltage, current, and temperature** — Use vectors to specify the output current, switch-on loss, switch-off loss, on-state voltage, and temperature data. This is the default parameterization method.

### Off-state voltage for losses data

The output voltage of the device during the off state. This is the blocking voltage at which the switch-on loss and switch-off loss data are defined. The default value is 300 V.

### Energy dissipation time constant

Time constant used to average the switch-on losses, switch-off losses, and conduction losses. This value is equal to the period of the minimum switching frequency. The default value is  $1e-4$  s.

**Additional Parameters for Parameterizing by Voltage, Current, and Temperature****Temperature vector, Tj**

Temperature values at which the switch-on loss, switch-off loss, and on-state voltage are specified. Specify this parameter using a vector quantity. The default value is [ 298.15 398.15 ] K.

**Output current vector, Iout**

Output currents for which the switch-on loss, switch-off loss and on-state voltage are defined. The first element must be zero. Specify this parameter using a vector quantity. The default value is [ 0 10 50 100 200 400 600 ] A.

**Switch-on loss, Eon=fcn(Tj,Iout)**

Energy dissipated during a single switch on event. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a vector quantity. The default value is [ 0 2.9e-4 0.00143 0.00286 0.00571 0.01314 0.02286; 0 5.7e-4 0.00263 0.00514 0.01029 0.02057 0.03029 ] J.

**Switch-off loss, Eoff=fcn(Tj,Iout)**

Energy dissipated during a single switch-off event. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a vector quantity. The default value is [ 0 2.1e-4 0.00107 0.00214 0.00429 0.009859999999999999 0.01714; 0 4.3e-4 0.00197 0.00386 0.00771 0.01543 0.02271 ] J.

**On-state voltage, Von=fcn(Tj,Iout)**

Voltage drop across the device while it is in a triggered conductive state.. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a vector quantity. The default value is [ 0 1.1 1.3 1.45 1.75 2.25 2.7; 0 1 1.15 1.35 1.7 2.35 3 ] V.

**Additional Parameters for Parameterizing by Voltage and Current****Output current, Iout**

Output currents for which the switch-on loss, switch-off loss, and on-state voltage are defined. The first element must be zero. Specify this parameter using a scalar quantity. The default value is 600 A.

**Switch-on loss**

Energy dissipated during a single switch-on event. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a scalar quantity. The default value is  $0.02286$  J.

**Switch-off loss**

Energy dissipated during a single switch-off event. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a scalar quantity. The default value is  $0.01714$  J.

**On-state voltage**

Voltage drop across the block while it is in a triggered conductive state. This parameter is defined as a function of temperature and final on-state output current. Specify this parameter using a scalar quantity. The default value is  $2.7$  V.

**See Also**

Commutation Diode | Diode (Piecewise Linear) | GTO | IGBT | Ideal Semiconductor Switch | MOSFET

**Topics**

“Quantifying IGBT Thermal Losses”

“Simulate Thermal Losses in Semiconductors”

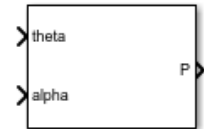
“Switch Between Physical Signal and Electrical Ports”

**Introduced in R2013b**

## Thyristor 6-Pulse Generator

Generate thyristor 6-pulse waveform in single-pulsing mode

**Library:** Simscape / Electrical / Power Systems / Control /  
Pulse Width Modulation



### Description

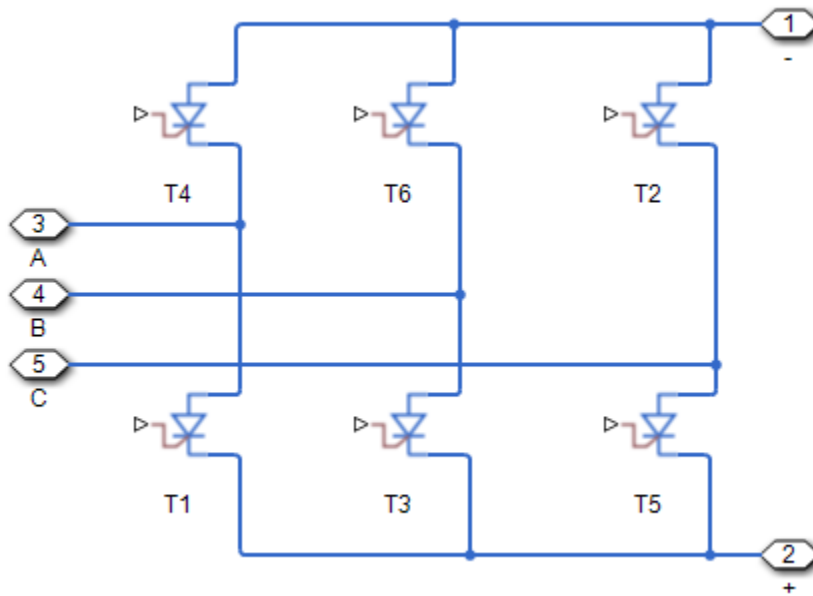
The Thyristor 6-Pulse Generator block implements a thyristor 6-pulse waveform generator in single-pulsing mode.

You can use this block to perform phase-controlled AC-to-DC conversion by:

- Measuring the synchronization angle of the AC signal with a phase-locked loop
- Controlling a thyristor converter network with the pulses generated by this block

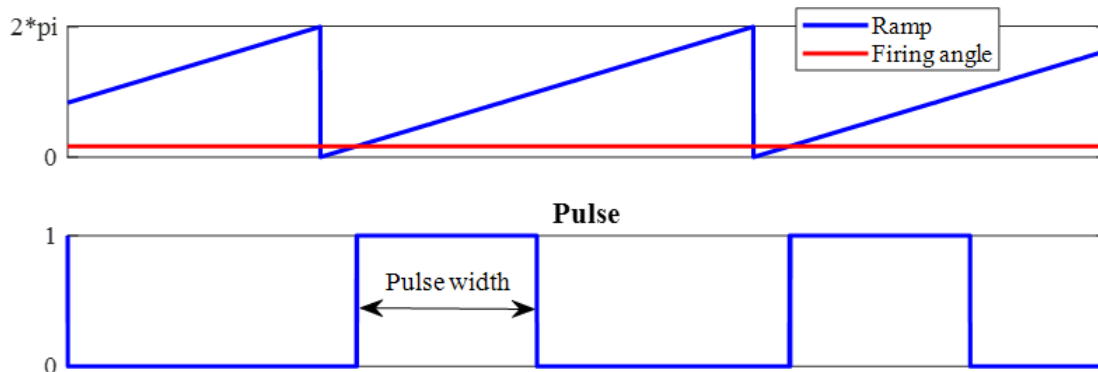
### Model

The figure shows the equivalent circuit for the Thyristor 6-Pulse Generator.



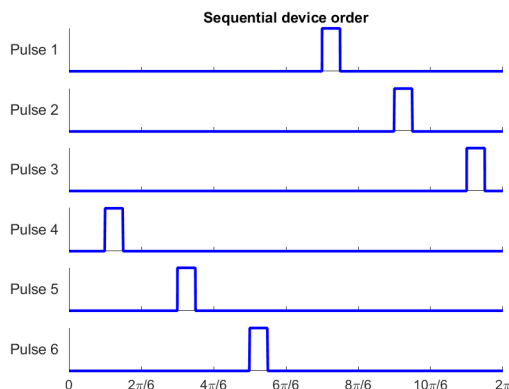
Based on the synchronization angle,  $\theta$ , and the firing angle,  $\alpha$ , the block internally generates six ramps, one for each of the pulse elements in its output vector.

The block generates a pulse at one of the outputs when the associated ramp meets or crosses the specified firing angle in the upward direction. This figure shows such a pulse generation mechanism for one of the outputs.

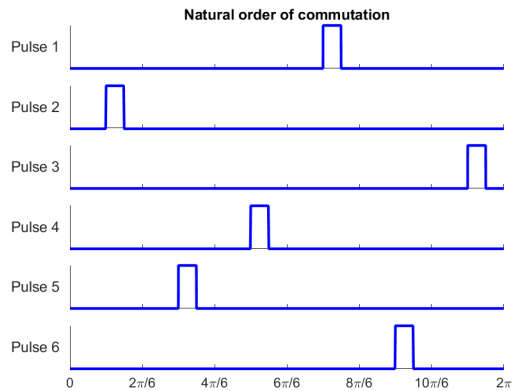


Set the pulse ordering strategy to modify the distinct phase-shift of each ramp, and as a result, the order of generated pulses:

- Set the Pulse ordering property to Sequential device order to generate pulses in sequential order. Use this strategy to generate pulses for the Converter block or other thyristor networks that use sequential ordering.



- Set the Pulse ordering property to Natural order of commutation to generate pulses in the natural order. Use this strategy to generate pulses for thyristor networks that use natural ordering.



## Ports

### Input

**theta — Synchronization angle, radians**

scalar

Synchronization angle in the range  $[0, 2\pi]$ , in radians.

Data Types: `single` | `double`

**alpha — Firing angle, radians**

scalar

Thyristor firing angle in radians.

Data Types: `single` | `double`

### Output

**P — Pulse vector**

vector

Thyristor pulse vector.

Data Types: `single` | `double`

## Parameters

### **Pulse ordering — Pulse ordering strategy**

Sequential device order (default) | Natural order of commutation

Specify the rule for pulse ordering based on the configuration of the thyristor network you are controlling. Use the `Sequential device order` strategy to generate pulses for the Converter block.

### **Pulse width (rad) — Pulse width**

$5\pi/6$  rad (default) | positive scalar in range  $[0, \pi]$

Specify the width of each pulse in the range  $[0, \pi]$ .

### **Sample time (-1 for inherited) — Block sample time**

$1e-5$  (default) | positive scalar

Time, in s, between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

If this block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly using a positive scalar.

## References

- [1] Pelly, B. R. *Thyristor Phase-Controlled Converters and Cycloconverters: Operation, Control, and Performance*. New York, NY: John Wiley & Sons, Inc., 1971.

## See Also

### **Blocks**

Three-Phase Sinusoidal Measurement (PLL) | Thyristor 12-Pulse Generator | Thyristor Rectifier Voltage Controller

### **Simscape Blocks**

Converter | Six-Pulse Gate Multiplexer | Thyristor

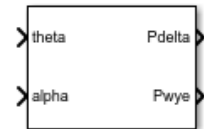


**Introduced in R2017b**

# Thyristor 12-Pulse Generator

Generate thyristor 12-pulse waveform in single-pulsing mode

**Library:** Simscape / Electrical / Power Systems / Control / Pulse Width Modulation



## Description

The Thyristor 12-Pulse Generator block implements a thyristor 12-pulse waveform generator in single-pulsing mode.

You can use this block to perform phase-controlled AC to DC conversion by:

- Measuring the synchronization angle of the AC signal with a phase-locked loop
- Controlling a thyristor converter network with the pulses generated by this block

## Model

The Thyristor 12-Pulse Generator outputs six pulses for a delta gate driver and six pulses for a wye gate driver. The delta connection can lead (Delta11) or lag (Delta1). For information on the control model, see the Thyristor 6-Pulse Generator block.

## Ports

### Input

**theta** — Synchronization angle, rad

scalar

Synchronization angle in the range  $[0, 2\pi]$ , in rad.

Data Types: single | double

**alpha — Firing angle, rad**

scalar

Thyristor firing angle in rad.

Data Types: `single` | `double`**Output****Pdelta — Delta pulse vector**

vector

Thyristor pulse vector for the delta connection.

Data Types: `single` | `double`**Pwye — Wye pulse vector**

vector

Thyristor pulse vector for the wye connection.

Data Types: `single` | `double`**Parameters****Pulse ordering — Pulse ordering strategy**`Sequential device order` (default) | `Natural order of commutation`

Specify the rule for pulse ordering based on the configuration of the thyristor network you are controlling. Use the `Sequential device order` strategy to generate pulses for the Converter block.

**Delta connection — Delta sinusoidal phase variance model**`Leading (Delta11)` (default) | `Lagging (Delta1)`

Sinusoidal phase variance model for the delta connection.

**Pulse width (rad) — Pulse width**`5*pi/6 rad` (default) | positive scalar in range `[0, pi]`Specify the width of each pulse in the range `[0, pi]`.

## **Sample time (-1 for inherited) — Block sample time**

1e-5 (default) | positive scalar

Time, in s, between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

If this block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly using a positive scalar.

## **References**

[1] Pelly, B. R. *Thyristor Phase-Controlled Converters and Cycloconverters: Operation, Control, and Performance*. New York, NY: John Wiley & Sons, Inc., 1971.

## **See Also**

### **Blocks**

Three-Phase Sinusoidal Measurement (PLL) | Thyristor 6-Pulse Generator | Thyristor Rectifier Voltage Controller

### **Simscape Blocks**

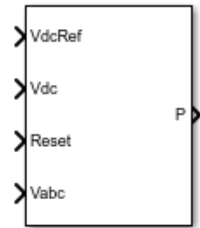
Converter | Thyristor | Twelve-Pulse Gate Multiplexer | Wye-Delta1-Wye Transformer | Wye-Delta11-Wye Transformer

### **Introduced in R2018a**

# Thyristor Rectifier Voltage Controller

Discrete-time PI-based DC-link voltage control for thyristor rectifiers

**Library:** Simscape / Electrical / Power Systems / Control / Converter Control

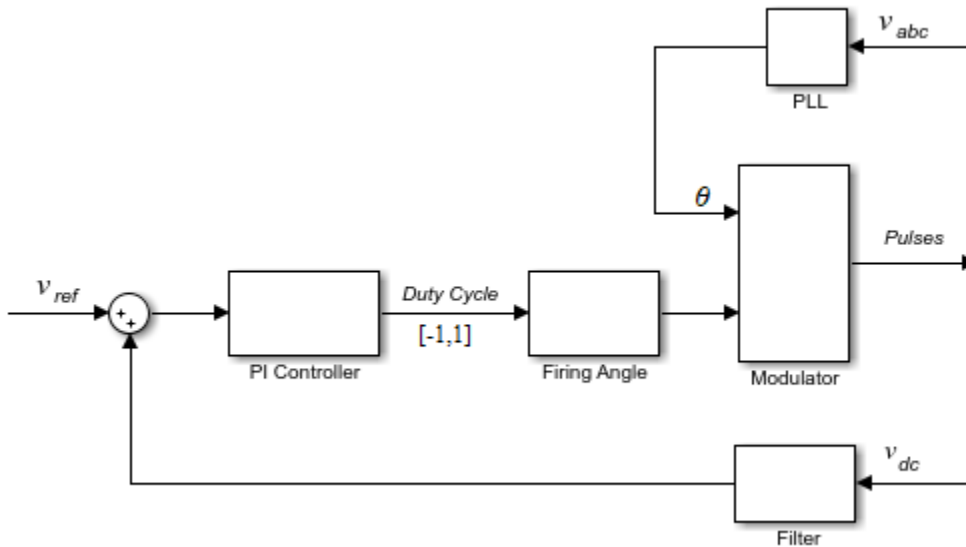


## Description

The Thyristor Rectifier Voltage Controller block implements a discrete-time proportional-integral (PI) based DC-link voltage controller for thyristor rectifiers.

## Model

To regulate the output DC-link voltage of a thyristor rectifier, the Thyristor Rectifier Voltage Controller block determines firing angles using the cosine wave-crossing method. The figure shows the control structure, which includes a three-phase phase-locked loop (PLL).



## Ports

### Input

#### VdcRef — Reference DC voltage, V

scalar

Desired DC-link voltage.

Data Types: single | double

#### Vdc — DC-link voltage, V

scalar

Measured DC-link voltage.

Data Types: single | double

#### Reset — External reset

scalar

External reset signal (rising edge) for integrators.

Data Types: `single` | `double`

### **Vabc — Three-phase voltage, V**

vector

Three-phase voltage specified as a vector of that contains the *a*-, *b*-, and *c*-phase voltages.

Data Types: `single` | `double`

## **Output**

### **P — Pulses**

vector

Pulse control signal.

Data Types: `single` | `double`

## **Parameters**

### **Loop filter proportional gain — Loop filter $K_p$**

2 (default) | positive scalar

Loop filter proportional gain for the phase-locked loop (PLL).

### **Loop filter integral gain — Loop filter $K_i$**

20 (default) | positive scalar

Loop filter integral gain for the phase-locked loop (PLL).

### **Proportional gain — Controller $K_p$**

1 (default) | positive scalar

Proportional gain for the PI-controller that generates the reference phase voltage.

### **Integral gain — Controller $K_i$**

1 (default) | positive scalar

Integral gain for the PI-controller that generates the reference phase voltage.

**Thyristor pulse width (rad) — Pulse width**

5\*pi/6 (default) | positive scalar

Angular width of pulses.

**Pulse ordering — Pulse ordering rule**

Sequential device order (default) | Natural order of commutation

Model for the ordering the generated pulses.

**Sample time (-1 for inherited) — Block sample time**

-1 (default) | positive scalar

Time, in s, between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

If this block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly using a positive scalar.

**Time constant voltage filter — DC voltage filter  $\tau$**

0.001 (default) | positive scalar

Time constant for the DC voltage filter.

**Dependencies**

The **Time constant voltage filter** parameter is only visible when the **Filter DC voltage** check box is selected.

**Filter DC voltage — DC voltage filter option**

on (default) | off

When the check box is:

- Selected — The block filters the DC voltage signal.
- Cleared — The block does not filter the DC voltage signal.

**Dependencies**

The **Time constant voltage filter** parameter is only visible when the **Filter DC voltage** check box is selected.



## References

- [1] Pelly, B. R. *Thyristor Phase-Controlled Converters and Cycloconverters: Operation, Control, and Performance*. New York, NY: John Wiley & Sons, Inc., 1971.

## See Also

Converter | Six-Pulse Gate Multiplexer | Three-Phase Sinusoidal Measurement (PLL) | Thyristor 12-Pulse Generator | Thyristor 6-Pulse Generator | Twelve-Pulse Gate Multiplexer

**Introduced in R2018a**

## Time-Based Fault

Time-based single-phase, two-phase, or three-phase grounded or ungrounded fault



## Library

Simscape / Electrical / Power Systems / Passive Devices / Faults

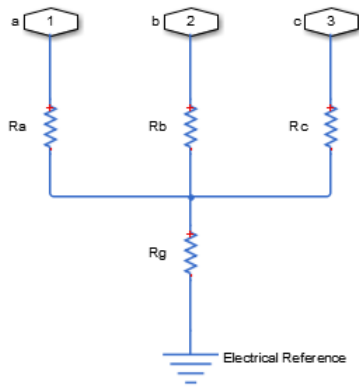
## Description

The Time-Based Fault block models any permutation of a single-phase, two-phase, or three-phase grounded or ungrounded fault. You specify the fault activation time using the block **Fault start time** parameter. The fault becomes inactive when the fault duration that you specify elapses.

You can set the Time-Based Fault block to represent any of these permutations:

- Single-phase-to-ground fault (a-g, b-g, or c-g)
- Two-phase fault (a-b, b-c, or c-a)
- Two-phase-to-ground fault (a-b-g, b-c-g, or c-a-g)
- Three-phase fault (a-b-c)
- Three-phase-to-ground fault (a-b-c-g)

The figure shows the equivalent circuit diagram for the Time-Based Fault block.



You can determine the resistance in the equivalent circuit using the equations in the table.

Fault type	Value of $R_a$	Value of $R_b$	Value of $R_c$	Value of $R_g$
None / inactive	$\frac{1}{G_{pn}}$	$\frac{1}{G_{pn}}$	$\frac{1}{G_{pn}}$	Infinity / open circuit
a-g	$R_{pn}$	$\frac{1}{G_{pn}}$	$\frac{1}{G_{pn}}$	$R_{ng}$
b-g	$\frac{1}{G_{pn}}$	$R_{pn}$	$\frac{1}{G_{pn}}$	$R_{ng}$
c-g	$\frac{1}{G_{pn}}$	$\frac{1}{G_{pn}}$	$R_{pn}$	$R_{ng}$
a-b	$R_{pn}$	$R_{pn}$	$\frac{1}{G_{pn}}$	Infinity / open circuit
b-c	$\frac{1}{G_{pn}}$	$R_{pn}$	$R_{pn}$	Infinity / open circuit

<b>Fault type</b>	<b>Value of <math>R_a</math></b>	<b>Value of <math>R_b</math></b>	<b>Value of <math>R_c</math></b>	<b>Value of <math>R_g</math></b>
c-a	$R_{pn}$	$\frac{1}{G_{pn}}$	$R_{pn}$	Infinity / open circuit
a-b-g	$R_{pn}$	$R_{pn}$	$\frac{1}{G_{pn}}$	$R_{ng}$
b-c-g	$\frac{1}{G_{pn}}$	$R_{pn}$	$R_{pn}$	$R_{ng}$
c-a-g	$R_{pn}$	$\frac{1}{G_{pn}}$	$R_{pn}$	$R_{ng}$
a-b-c	$R_{pn}$	$R_{pn}$	$R_{pn}$	Infinity / open circuit
a-b-c-g	$R_{pn}$	$R_{pn}$	$R_{pn}$	$R_{ng}$

where:

- $R_a$  is the resistance between the a-phase and the neutral point of a wye connection.
- $R_b$  is the resistance between the b-phase and the neutral point of a wye connection.
- $R_c$  is the resistance between the c-phase and the neutral point of a wye connection.
- $R_g$  is the resistance between the neutral point of a wye connection and electrical reference.
- $R_{pn}$  is the value of the **Faulted phase-neutral resistance** parameter.
- $R_{ng}$  is the value of the **Faulted neutral-ground resistance** parameter.
- $G_{pn}$  is the value of the **Unfaulted phase-neutral conductance** parameter.

## Parameters

- “Main Tab” on page 1-1967
- “Parasitics Tab” on page 1-1968

## Main Tab

### Fault type

Select one of the following:

- None — Specifies that the fault is not active. This is the default value.
- Single-phase to ground (a-g)
- Single-phase to ground (b-g)
- Single-phase to ground (c-g)
- Two-phase (a-b)
- Two-phase (b-c)
- Two-phase (c-a)
- Two-phase to ground (a-b-g)
- Two-phase to ground (b-c-g)
- Two-phase to ground (c-a-g)
- Three-phase (a-b-c)
- Three-phase to ground (a-b-c-g)

### Faulted phase-neutral resistance

Resistance between the phase connection and the neutral point when the fault is active. This parameter is visible if the **Fault type** parameter is set to anything other than None. The default value is  $1e-3$  Ohm.

### Faulted neutral-ground resistance

Resistance between the neutral point and the electrical reference when fault is active. This parameter is visible if the **Fault type** parameter is set to any fault which includes a ground connection. The default value is  $1e-3$  Ohm.

### Fault start time

Simulation time when the fault becomes active. This parameter is visible if the **Fault type** parameter is set to anything other than None. The default value is 1 s.

### Fault duration

Period of time that the fault is active. This parameter is visible if the **Fault type** parameter is set to anything other than None. The default value is 0.1 s.

## **Parasitics Tab**

### **Unfaulted phase-neutral conductance**

Conductance between the phase connections and the neutral point when a phase is not involved in the fault. The default value is  $1e-6$  1/Ohm.

## **Ports**

The block has one expandable three-phase port for connecting the fault to the system.

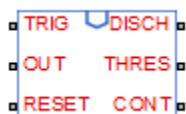
## **See Also**

Enabled Fault

**Introduced in R2014a**

# Timer

Model timer integrated circuit behaviorally



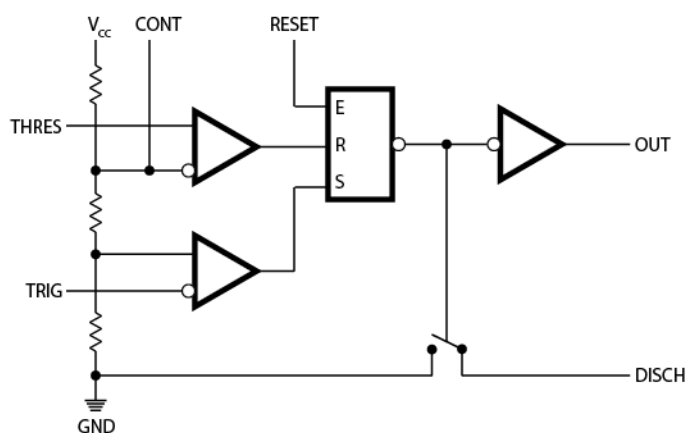
## Library

Integrated Circuits

## Description

The Timer block is a behavioral model of a timer integrated circuit such as the NE555.

The following figure shows the implementation structure.



The **Potential divider component resistance** parameter sets the values of the three resistors creating the potential divider. The two comparator inputs have infinite input resistance and zero input capacitance. The S-R Latch block provides the functionality of

the set-reset latch. It includes an output capacitor and a resistor with values set to match the **Propagation delay** parameter value. The block models the output stage inverter using a CMOS NOT block. You define the output resistance, low-level output voltage, and high-level output voltage for the CMOS gate in the Timer block dialog box. The discharge switch approximates the NPN bipolar transistor on a real timer as a switch with defined switch on-resistance and off-resistance values.

## Basic Assumptions and Limitations

This block has the following limitations:

- The behavior is abstracted. Results are not as accurate as a transistor-level model.
- Delay in response to changing inputs depends solely upon the RC time constant of the resistor-capacitor network at the output of the latch. In practice, the delay has a more complex dependency on the device structure. Set this value based on the output-pulse rise and fall times.
- The drop in output voltage is a linear function of output current. In practice, the relationship is that of a bipolar transistor push-pull pair.
- The controlled switch arrangement used by the block is an approximation of an open-collector arrangement.
- The power supply connects internally within the component, and the block assumes that the GND pin is grounded.

## Parameters

- “Supply Tab” on page 1-1970
- “Outputs Tab” on page 1-1971
- “Discharge Tab” on page 1-1971
- “Potential Divider Tab” on page 1-1971

## Supply Tab

### Power supply voltage

The voltage value  $V_{cc}$  that the block applies internally to the timer component. The default value is 15 V.



## Outputs Tab

### Low level output voltage

The output voltage when the timer output is low and no output current is drawn. The default value is 0 V.

### High level output voltage

The output voltage  $V_{OH}$  when the timer output is high and no current is drawn. The default value is 14.1 V.

### Output resistance

The ratio of output voltage drop to output current. Set this parameter to

$(V_{OH} - V_{OH1}) / I_{OH1}$ , where  $V_{OH1}$  is the reduced output high voltage when the output current is  $I_{OH1}$ . The default value is 8  $\Omega$ .

### Propagation delay

Set this value to the input-pulse or output-pulse rise time. The default value is 1e-07 s.

## Discharge Tab

### Discharge switch on-resistance

A representative value is the discharge pin saturation voltage divided by the corresponding current. The default value is 12  $\Omega$ .

### Discharge switch off-resistance

A representative value is the discharge pin leakage current divided by the corresponding pin voltage. The default value is 5e+08  $\Omega$ .

## Potential Divider Tab

### Potential divider component resistance

A typical value for a 555-type timer is 5 k $\Omega$ . You can measure it directly across the positive supply and control pins when the chip does not connect to a circuit. The default value is 5 k $\Omega$ .

## Ports

This block has the following ports:

THRES

Electrical port corresponding to the threshold pin

TRIG

Electrical port corresponding to the trigger pin

CONT

Electrical port corresponding to the control pin

RESET

Electrical port corresponding to the reset pin

OUT

Electrical port corresponding to the output pin

DISCH

Electrical port corresponding to the discharge pin

## See Also

Comparator | S-R Latch

# Transmission Line

Model delay-based or lumped parameter transmission line



## Library

Passive Devices

## Description

The Transmission Line block lets you choose between the following models of a transmission line:

- 1 Delay-based and lossless
- 2 Delay-based and lossy
- 3 Lumped parameter L-section
- 4 Lumped parameter pi-section

The first option provides the best simulation performance, with options 2, 3 and 4 requiring progressively more computing power.

### Delay-Based and Lossless

This first option, **Delay-based and lossless**, models the transmission line as a fixed impedance, irrespective of frequency, plus a delay term. The defining equations are:

$$v_1(t) - i_1(t) Z_0 = v_2(t - \tau) + i_2(t - \tau) Z_0$$

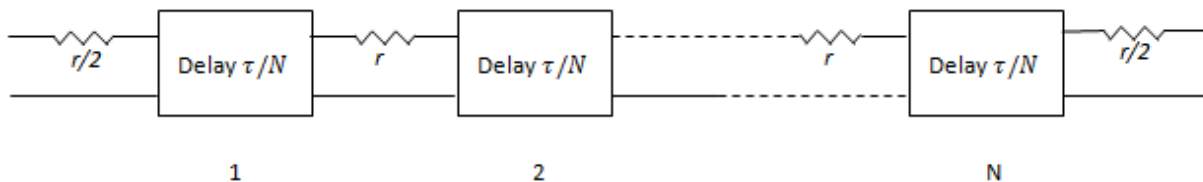
$$v_2(t) - i_2(t) Z_0 = v_1(t - \tau) + i_1(t - \tau) Z_0$$

where:

- $v_1$  is the voltage across the left-hand end of the transmission line.
- $i_1$  is the current into the left-hand end of the transmission line.
- $v_2$  is the voltage across the right-hand end of the transmission line.
- $i_2$  is the current into the right-hand end of the transmission line.
- $\tau$  is the transmission line delay.
- $Z_0$  is the line characteristic impedance.

## Delay-Based and Lossy

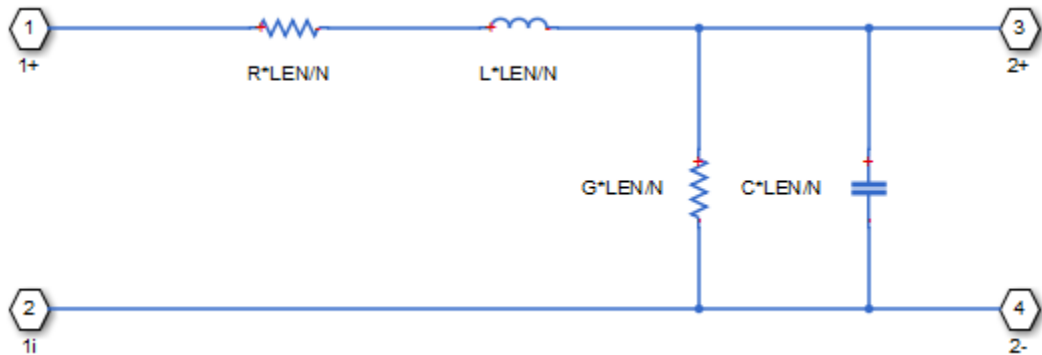
To introduce losses, the second option, `Delay-based and lossy`, connects  $N$  delay-based components, each defined by the above equations, in series via a set of resistors, as shown in the following illustration.



$N$  is an integer greater than or equal to 1.  $r = R \cdot LEN / N$ , where  $R$  is the line resistance per unit length and  $LEN$  is the line length.

## Lumped Parameter L-Section

The following block diagram shows the model of one L-line segment.



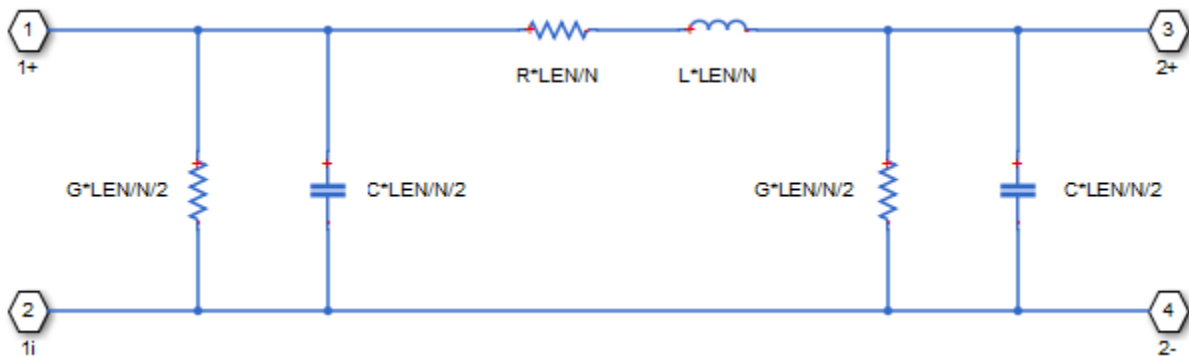
The lumped parameter parameterization uses  $N$  copies of the above segment model connected in series.

Parameters are as follows:

- $R$  is line resistance per unit length.
- $L$  is the line inductance per unit length.
- $C$  is the line capacitance per unit length.
- $G$  is the line conductance per unit length.
- $LEN$  is the length of the line.
- $N$  is the number of series segments.

### Lumped Parameter Pi-Section

The following block diagram shows the model of one pi-line segment.



The lumped parameter parameterization uses  $N$  copies of the above segment model connected in series. The parameters are as defined for the L-section transmission line model. Unlike the L-section model, the pi-section model is symmetric.

## Lumped Parameter Line Model Parameterization

The lumped-parameter models (L-section or pi-section) are the most challenging to simulate, typically needing many more segments (greater  $N$ ) than for the delay-based and lossy model [1 on page 1-1979].

Cable manufacturers do not typically quote an inductance value per unit length, but instead give the characteristic impedance. The inductance, capacitance, and characteristic impedance are related by:

$$L = \frac{C \cdot Z_0^2}{\dots}$$

The block lets you specify either  $L$  or  $Z_0$  when using the lumped parameter model.

## Basic Assumptions and Limitations

The Transmission Line model has the following limitations:

- For the lumped parameter options, MathWorks recommends that you use a trapezoidal solver such as `ode23t`. This is because lumped parameter transmission models have very lightly damped internal dynamics, which are best suited to trapezoidal solvers for numerical accuracy.

- The lumped parameter pi-section model has a parallel capacitor at both ends. This means that you should not connect it directly to an ideal voltage source, that is, a source with no internal resistance. The lumped parameter L-section model, however, has a series input resistor, and therefore you can connect it directly to an ideal voltage source.

## Parameters

### Model type

Select one of the following transmission line models:

- **Delay-based and lossless** — Model the transmission line as a fixed impedance, irrespective of frequency, plus a delay term, as described in “Delay-Based and Lossless” on page 1-1973. This is the default method. It provides the best simulation performance.
- **Delay-based and lossy** — Model the transmission line as a number of delay-based components, connected in series via a set of resistors, as described in “Delay-Based and Lossy” on page 1-1974.
- **Lumped parameter L-section** — Model the transmission line as a number of L-line segments, connected in series, as described in “Lumped Parameter L-Section” on page 1-1974.
- **Lumped parameter pi-section** — Model the transmission line as a number of pi-line segments, connected in series, as described in “Lumped Parameter Pi-Section” on page 1-1975.

### Transmission delay

The total transmission line delay. This parameter appears for delay-based models only. The parameter value must be greater than zero. The default value is 5 ns, which is a typical value for a one-meter coaxial cable.

### Characteristic impedance

The characteristic impedance of the transmission line. This parameter appears for delay-based models, and for lumped parameter models where **Parameterization** is By characteristic impedance and capacitance. The parameter value must be greater than zero. The default value is 50  $\Omega$ .

### **Parameterization**

This parameter appears for lumped parameter models only. Select the model parameterization method, as described in “Lumped Parameter Line Model Parameterization” on page 1-1976:

- By characteristic impedance and capacitance — Specify values for the **Characteristic impedance** and **Capacitance per unit length** parameters. This is the default method.
- By inductance and capacitance — Specify values for the **Inductance per unit length** and **Capacitance per unit length** parameters.

### **Inductance per unit length**

The effective inductance of the transmission line per unit length. For lumped parameter models where **Parameterization** is By inductance and capacitance, this parameter appears instead of the **Characteristic impedance** parameter. The parameter value must be greater than zero. The default value is 220  $\mu\text{H}/\text{m}$ .

### **Capacitance per unit length**

The transmission line capacitance per unit length. This parameter appears for lumped parameter models only. The parameter value must be greater than zero. The default value is 90  $\text{pF}/\text{m}$ .

### **Resistance per unit length**

The total transmission line resistance (that is, the sum of the resistance for the two conducting paths) per unit length. This parameter appears for Delay-based and lossy and for lumped parameter models. The parameter value must be greater than zero. The default value is 0.3  $\Omega/\text{m}$ .

### **Insulation conductance per unit length**

The conductance between the two transmission line conductors per unit length. This parameter appears for lumped parameter models only. The parameter value must be greater than, or equal to, zero. The default value is  $5\text{e}-6$   $\text{S}/\text{m}$ .

### **Line length**

The total transmission line length. This parameter appears for Delay-based and lossy and for lumped parameter models. The parameter value must be greater than zero. The default value is 1 m.

### **Number of segments**

The number of model segments used to represent the transmission line. This parameter appears for Delay-based and lossy and for lumped parameter models.



The parameter value must be an integer greater than, or equal to, 1. The default value is 1.

## Ports

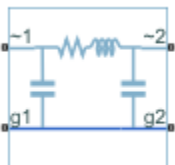
The block has four conserving electrical ports. For a coaxial cable, the two top ports correspond to the inner conductor, and the two lower ports to the external shielding conductor.

## References

- [1] Sussman-Fort, S.E. and J.C. Hantgan. "SPICE Implementation of Lossy Transmission Line and Schottky Diode Models." *IEEE Transactions on Microwave Theory and Techniques*. Vol. 36, No. 1, January, 1988.

## Transmission Line (Three-Phase)

Three-phase transmission line using lumped-parameter pi-line model



### Library

Simscape / Electrical / Power Systems / Passive Devices

### Description

The Transmission Line (Three-Phase) block models a three-phase transmission line using the lumped-parameter pi-line model. This model takes into account phase resistance, phase self-inductance, line-line mutual inductance and resistance, line-line capacitance, and line-ground capacitance.

To simplify the block-defining equations, Clarke's transformation is used. The resulting equations are:

$$V'_1 - V'_2 = \begin{bmatrix} R + 2R_m & & \\ & R - R_m & \\ & & R - R_m \end{bmatrix} I'_1 + \begin{bmatrix} L + 2M & & \\ & L - M & \\ & & L - M \end{bmatrix} \frac{dI'_1}{dt}$$

$$I'_1 + I'_2 = \begin{bmatrix} C_g & & \\ & C_g + 3C_l & \\ & & C_g + 3C_l \end{bmatrix} \frac{dV'_2}{dt}$$

$$I'_1 = T'I_1$$

$$I'_2 = T I_2$$

$$V'_1 = T V_1$$

$$V'_2 = T V_2$$

$$T = \frac{1}{\sqrt{3}} \begin{bmatrix} 1 & \sqrt{2} & 0 \\ 1 & -1/\sqrt{2} & \sqrt{3/2} \\ 1 & -1/\sqrt{2} & -\sqrt{3/2} \end{bmatrix}$$

where:

- $R$  is the line resistance for the segment.
- $R_m$  is the mutual resistance for the segment.
- $L$  is the line inductance for the segment.
- $C_g$  is the line-ground capacitance for the segment.
- $C_l$  is the line-line capacitance for the segment.
- $T$  is the Clarke's transformation matrix.
- $I_1$  is the three-phase current flowing into the ~1 port.
- $I_2$  is the three-phase current flowing into the ~2 port.
- $V_1$  is the three-phase voltage at the ~1 port.
- $V_2$  is the three-phase voltage at the ~2 port.

The positive and zero-sequence parameters are defined by the diagonal terms in the transformed equations:

$$R_0 = R + 2R_m$$

$$R_1 = R - R_m$$

$$L_0 = L + 2M$$

$$L_1 = L - M$$

$$C_0 = C_g$$

$$C_1 = C_g + 3C_l$$

Rearranging these equations gives the physical line quantities in terms of positive and zero-sequence parameters:

$$R = \frac{2R_1 + R_0}{3}$$

$$R_m = \frac{R_0 - R_1}{3}$$

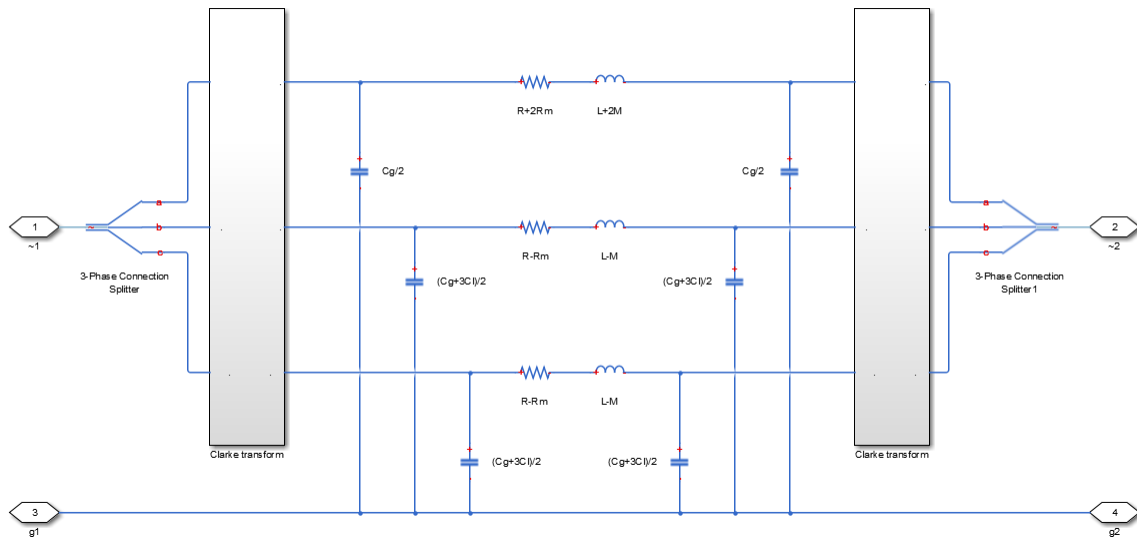
$$L = \frac{2L_1 + L_0}{3}$$

$$M = \frac{L_0 - L_1}{3}$$

$$C_l = \frac{C_1 - C_0}{3}$$

$$C_g = C_0$$

The figure shows the equivalent electrical circuit for a single-segment pi-line model using Clarke's transformation.



To increase fidelity, you can use the **Number of segments** parameter to repeat the pi-section N times, resulting in an N-segment transmission line model. More segments significantly slows down your simulation.

To improve numerical performance, you can add parasitic resistance and conductance components. Choosing large values for these components improves simulation speed but decreases simulation accuracy.

## Parameters

- “Main tab” on page 1-1983
- “Parasitics tab” on page 1-1984

### Main tab

#### Line length

Length of the transmission line. The default value is 1 km.

### **Resistance**

Resistance of the transmission line per phase per-unit length. The default value is 0.02 Ohm/km.

### **Inductance**

Self-inductance of the transmission line per phase per-unit length. The default value is 0.5 mH/km.

### **Mutual inductance**

Line-line mutual inductance per-unit length. Set this to 0 to remove mutual inductance. The default value is 0.1 mH/km.

### **Line-line capacitance**

Line-line capacitance per-unit length. The default value is 0.3  $\mu$ F/km.

### **Line-ground capacitance**

Line-ground capacitance per-unit length. The default value is 0  $\mu$ F/km (no line-ground capacitance).

### **Mutual Resistance**

Line-line mutual resistance per unit length. The default value is 0 Ohm/km (no line-line mutual resistance).

### **Number of segments**

Number of segments in the pi-line model. The default value is 1.

## **Parasitics tab**

### **Parasitic series resistance**

Resistance value, divided by the number of segments, that is added in series with every capacitor in the model. The default value is 1e-6 Ohm.

### **Parasitic parallel conductance**

Conductance value, divided by the number of segments, that is added in parallel with every series resistor and inductor in the model. The default value is 1e-6 1/Ohm.

## **Ports**

The block has the following ports:

~1

Expandable three-phase port

~2

Expandable three-phase port

g1

Electrical conserving port corresponding to ground connection at ~1 end of the transmission line

g2

Electrical conserving port corresponding to ground connection at ~2 end of the transmission line

## See Also

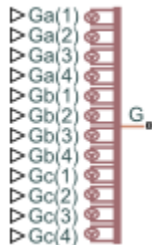
### Topics

“Expand and Collapse Three-Phase Ports on a Block”

**Introduced in R2013b**

## Twelve-Pulse Gate Multiplexer

Multiplex gate input signals to Three-Level Converter block



### Library

Simscape / Electrical / Power Systems / Semiconductors

### Description

The Twelve-Pulse Gate Multiplexer block routes gate voltage signals to the 12 switching devices in a Three-Level Converter block. The block multiplexes the 12 gate signals into a single vector. Gate signals are ordered as *a*-phase, *b*-phase, and then *c*-phase, with four gate signals per phase.

If you want to use Simscape Electrical Electronics and Mechatronics to model the electronics that drive the Three-Level Converter block, you can switch the input ports of the Twelve-Pulse Gate Multiplexer block from physical signal ports to electrical ports.

When you switch the block inputs to electrical ports, the block shows 12 pairs of electrical connections, each pair corresponding to the gate and cathode of a switching device.

### Ports

The block has the following ports:



**Ga(1), Ga(2), Ga(3), Ga(4)**

Ports associated with the gate terminals of the Three-Level Converter *a*-phase switching devices. You can set the ports to either physical signal or electrical ports.

**Gb(1), Gb(2), Gb(3), Gb(4)**

Ports associated with the gate terminals of the Three-Level Converter *b*-phase switching devices. You can set the ports to either physical signal or electrical ports.

**Gc(1), Gc(2), Gc(3), Gc(4)**

Ports associated with the gate terminals of the Three-Level Converter *c*-phase switching devices. You can set the ports to either physical signal or electrical ports.

**G**

Vector output port associated with the multiplexed gate signals. Connect this port to the G port of the Three-Level Converter block.

**Ka(1), Ka(2), Ka(3), Ka(4)**

Electrical conserving ports associated with the individual cathode terminals corresponding to the Three-Level Converter block *a*-phase switching devices. These ports are visible only if you set the input ports of the Twelve-Pulse Gate Multiplexer block to electrical ports.

**Kb(1), Kb(2), Kb(3), Kb(4)**

Electrical conserving ports associated with the individual cathode terminals corresponding to the Three-Level Converter block *b*-phase switching devices. These ports are visible only if you set the input ports of the Twelve-Pulse Gate Multiplexer block to electrical ports.

**Kc(1), Kc(2), Kc(3), Kc(4)**

Electrical conserving ports associated with the individual cathode terminals corresponding to the Three-Level Converter block *c*-phase switching devices. These ports are visible only if you set the input ports of the Twelve-Pulse Gate Multiplexer block to electrical ports.

## See Also

Six-Pulse Gate Multiplexer | Three-Level Converter

## Topics

“Switch Between Physical Signal and Electrical Ports”

**Introduced in R2014b**

# Two-Pulse Gate Multiplexer

Combine two gate-pulse voltage signals into a single vector

**Library:** Simscape / Electrical / Power Systems / Semiconductors



## Description

The Two-Pulse Gate Multiplexer block multiplexes two separate voltage signals into a single vector. The vectorized signal can control the gates of two switching devices in a converter, such as a Two-Quadrant Chopper block.

## Model

Each of two model variants for the Two-Pulse Gate Multiplexer block corresponds to a **Block choice** option. To access the block choices, in the model window, right-click the block, and then use either of these methods:

- From the context menu, select **Simscape > Block choices**.
- On the Simulink Editor menu bar, select **View > Property Inspector**. In the **Property Inspector** window, click the value of the **Block choice**.

The model variants are:

- PS ports — Two-pulse gate multiplexer with physical signal ports. Select this default option to control switching device gates in a converter block using Simulink gate-control voltage signals. To multiplex and connect Simulink signals to the gate-control inport of a converter block:
  - 1 Convert each voltage signal using a Simulink-PS Converter block.
  - 2 Multiplex the converted gate signals into a single vector using the multiplexor block.
  - 3 Connect the vector signal to the **G** port of the converter.
- Electrical ports — Two-pulse gate multiplexer with electrical conserving ports. To control switching device gates in a converter block using Simscape Electrical

Electronics and Mechatronics blocks, select this option. The electrical ports include pairs of electrical connections. Each pair corresponds to the gate and cathode of a switching device in the connected converter block.

## Ports

### Input

#### **G1 — Gate-control voltage signal 1**

physical signal

Physical signal port associated with the gate terminal of the first switching device in a connected converter block.

#### **Dependencies**

This port only appears for the PS ports block choice.

Data Types: double

#### **G2 — Gate-control voltage signal 2**

physical signal

Physical signal port associated with the gate terminal of the second switching device in a connected converter block.

#### **Dependencies**

This port only appears for the PS ports block choice.

Data Types: double

### Conserving

#### **G1 — Gate-control voltage signal 1**

electrical

Electrical conserving port associated with the gate terminal of the first switching device in a connected converter block.

**Dependencies**

This port only appears for the `Electrical ports` block choice.

**a — A-phase AC reference point**

electrical

Electrical conserving port associated with the A-phase for the high-side switching device.

**Dependencies**

This port only appears for the `Electrical ports` block choice.

**G2 — Gate-control voltage signal 2**

electrical

Electrical conserving port associated with the gate terminal of the second switching device in a connected converter block.

**Dependencies**

This port only appears for the `Electrical ports` block choice.

**L — DC reference point**

electrical

Electrical conserving port associated with the DC negative connection for the low-side switching device.

**Dependencies**

This port only appears for the `Electrical ports` block choice.

**See Also**

[Four-Pulse Gate Multiplexer](#) | [Six-Pulse Gate Multiplexer](#) | [Twelve-Pulse Gate Multiplexer](#) | [Two-Quadrant Chopper](#)

**Topics**

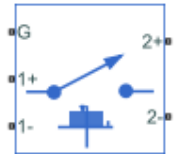
[Switch Between Physical Signal and Electrical Ports](#)

**Introduced in R2018a**

## Two-Quadrant Chopper

Two-quadrant controlled DC-DC chopper

**Library:** Simscape / Electrical / Power Systems / Semiconductors



### Description

The Two-Quadrant Chopper block represents a two-quadrant controlled chopper for converting a fixed DC input to a variable DC output. The block contains two switching devices. Options for the type of switching devices are:

- GTO — Gate turn-off thyristor. For information on the I-V characteristic of the device, see GTO.
- Ideal semiconductor switch — For information on the I-V characteristic of the device, see Ideal Semiconductor Switch.
- IGBT — Insulated-gate bipolar transistor. For information on the I-V characteristic of the device, see IGBT.
- MOSFET — N-channel metal-oxide-semiconductor field-effect transistor. For information on the I-V characteristic of the device, see MOSFET.
- Thyristor — For information on the I-V characteristic of the device, see Thyristor.

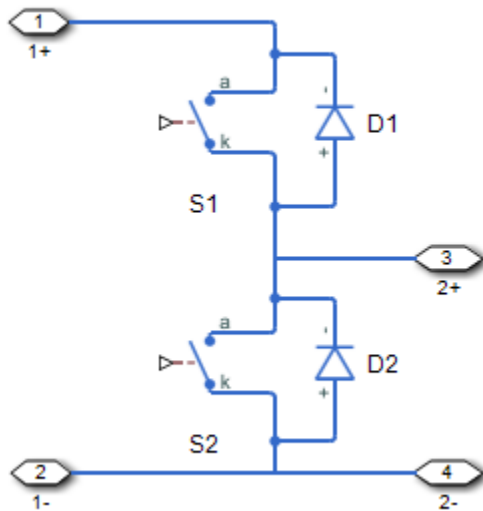
### Model

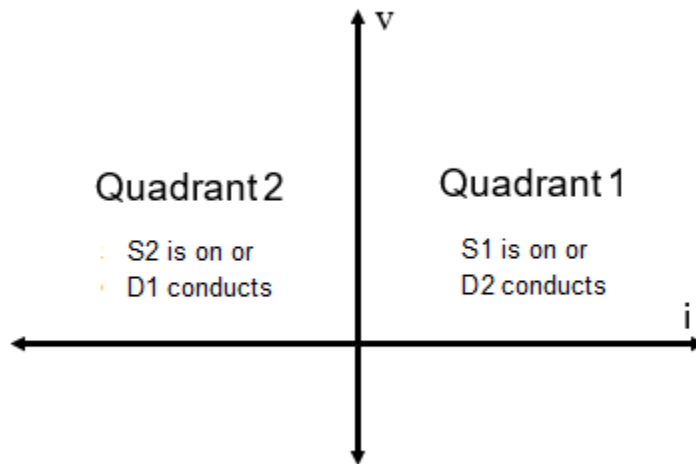
Each of two model variants for the Two-Quadrant Chopper block corresponds to a **Block choice** option. To access the block choices, in the model window, right-click the block, and then use either of these methods:

- From the context menu, select **Simscape > Block choices**.
- On the Simulink Editor menu bar, select **View > Property Inspector**. In the **Property Inspector** window, click the value of the **Block choice**.

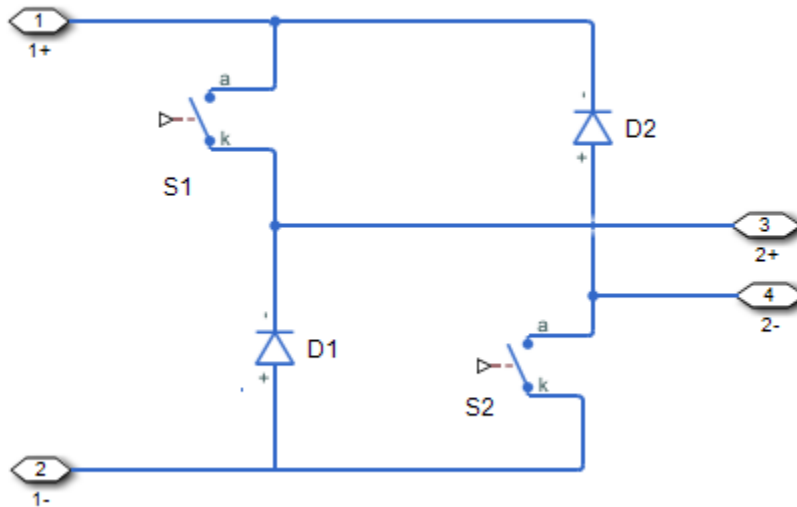
The model variants are:

- First- and second- quadrant chopper. This block choice is the default. The figures show the equivalent circuit and the operation for the first- and second- quadrant model.

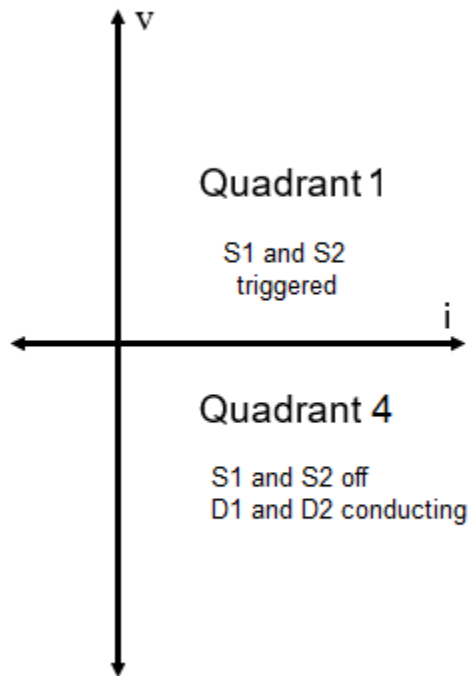




- First- and fourth- quadrant chopper. The figures show the equivalent circuit and the operation for the first- and fourth- quadrant model.







## Protection

The block contains an integral protection diode for each switching device. The integral diode protects the semiconductor device by providing a conduction path for reverse current. An inductive load can produce a high reverse-voltage spike when the semiconductor device suddenly switches off the voltage supply to the load.

To configure the internal protection diode block, use the **Protection Diode** parameters. This table shows how to set the **Model dynamics** parameter based on your goals.

Goals	Value to Select	Integral Protection Diode
Prioritize simulation speed.	Protection diode with no dynamics	The Diode block

Goals	Value to Select	Integral Protection Diode
Prioritize model fidelity by precisely specifying reverse-mode charge dynamics.	Protection diode with charge dynamics	The Commutation Diode block

You can also include a snubber circuit for each switching device. Snubber circuits contain a series-connected resistor and capacitor. They protect switching devices against high voltages that inductive loads produce when the device turns off the voltage supply to the load. Snubber circuits also prevent excessive rates of current change when a switching device turns on.

To include and configure a snubber circuit for each switching device, use the **Snubbers** parameters.

## Gate Control

To connect Simulink gate-control voltage signals to the gate ports of the internal switching devices:

- 1 Convert each voltage signal using a Simulink-PS Converter block.
- 2 Multiplex the converted gate signals into a single vector using a Two-Pulse Gate Multiplexer block.
- 3 Connect the vector signal to the **G** port.

## Ports

### Conserving

#### **G** — Switching device gate control

electrical | vector

Electrical conserving port associated with the gate terminals of the switching devices.

Data Types: double

#### **1+** — Positive DC voltage 1

electrical | scalar

Electrical conserving port associated with the positive terminal of the first DC voltage.

Data Types: double

### **1- — Negative DC voltage 1**

electrical | scalar

Electrical conserving port associated with the negative terminal of the first DC voltage.

Data Types: double

### **2+ — Positive DC voltage 2**

electrical | scalar

Electrical conserving port associated with the positive terminal of the second DC voltage.

Data Types: double

### **2- — Negative DC voltage 2**

electrical | scalar

Electrical conserving port associated with the negative terminal of the second DC voltage.

Data Types: double

## **Parameters**

### **Switching Devices**

This table shows how the visibility of **Switching Devices** parameters depends on the **Switching device** that you select. To learn how to read the table, see “Parameter Dependencies” on page A-2.

### Switching Devices Parameter Dependencies

Parameters and Options				
Switching device				
<i>Ideal Semiconductor Switch</i>	<i>GTO</i>	<i>IGBT</i>	<i>MOSFET</i>	<i>Thyristor</i>
On-state resistance	Forward voltage	Forward voltage	Drain-source on resistance	Forward voltage
Off-state conductance	On-state resistance	On-state resistance	Off-state conductance	On-state resistance
Threshold voltage	Off-state conductance	Off-state conductance	Threshold voltage	Off-state conductance
	Gate trigger voltage, Vgt	Threshold voltage		Gate trigger voltage, Vgt
	Gate turn-off voltage, Vgt_off			Gate turn-off voltage, Vgt_off
	Holding current			Holding current

#### Switching device — Switch type

`Ideal Semiconductor Switch` (default) | `GTO` | `IGBT` | `MOSFET` | `Thyristor`

Switching device type for the converter.

#### Dependencies

See the Switching Devices Parameter Dependencies table.

#### Forward voltage — Voltage

`0.8 Ohm` (default) | scalar

For the different switching device types, the **Forward voltage** is taken as:

- GTO — Minimum voltage required across the anode and cathode block ports for the gradient of the device I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**
- IGBT — Minimum voltage required across the collector and emitter block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On-state resistance**

- Thyristor — Minimum voltage required for the device to turn on

### Dependencies

See the Switching Devices Parameter Dependencies table.

### On-state resistance — Resistance

0.001 Ohm (default) | scalar

For the different switching device types, the **On-state resistance** is taken as:

- GTO — Rate of change of voltage versus current above the forward voltage
- Ideal semiconductor switch — Anode-cathode resistance when the device is on
- IGBT — Collector-emitter resistance when the device is on
- Thyristor — Anode-cathode resistance when the device is on

### Dependencies

See the Switching Devices Parameter Dependencies table.

### Drain-source on resistance — Resistance

0.001 Ohm (default) | scalar

Resistance between the drain and the source, which also depends on the gate-to-source voltage.

### Dependencies

See the Switching Devices Parameter Dependencies table.

### Off-state conductance — Conductance

1e-5 1/Ohm (default) | scalar

Conductance when the device is off. The value must be less than  $1/R$ , where  $R$  is the value of **On-state resistance**.

For the different switching device types, the **On-state resistance** is taken as:

- GTO — Anode-cathode conductance
- Ideal semiconductor switch — Anode-cathode conductance
- IGBT — Collector-emitter conductance

- MOSFET — Drain-source conductance
- Thyristor — Anode-cathode conductance

### **Dependencies**

See the Switching Devices Parameter Dependencies table.

### **Threshold voltage — Voltage threshold**

6 V (default) | scalar

Gate voltage threshold. The device turns on when the gate voltage is above this value. For the different switching device types, the device voltage of interest is:

- Ideal semiconductor switch — Gate-emitter voltage
- IGBT — Gate-cathode voltage
- MOSFET — Gate-source voltage

### **Dependencies**

See the Switching Devices Parameter Dependencies table.

### **Gate trigger voltage, $V_{gt}$ — Voltage threshold**

1 V (default) | scalar

Gate-cathode voltage threshold. The device turns on when the gate-cathode voltage is above this value.

### **Dependencies**

See the Switching Devices Parameter Dependencies table.

### **Gate turn-off voltage, $V_{gt\_off}$ — Voltage threshold**

-1 V (default) | scalar

Gate-cathode voltage threshold. The device turns off when the gate-cathode voltage is below this value.

### **Dependencies**

See the Switching Devices Parameter Dependencies table.

### **Holding current — Current threshold**

1 A (default) | scalar

Gate current threshold. The device stays on when the current is above this value, even when the gate-cathode voltage falls below the gate trigger voltage.

### Dependencies

See the Switching Devices Parameter Dependencies table.

## Protection Diode

The visibility of **Protection Diode** parameters depends on how you configure the protection diode **Model dynamics** and **Reverse recovery time parameterization** parameters. To learn how to read this table, see “Parameter Dependencies” on page A-2.

### Protection Diode Parameter Dependencies

Parameters and Options			
<b>Model dynamics</b>			
<i>Protection diode with no dynamics</i>	<i>Protection diode with charge dynamics</i>		
Forward voltage	Forward voltage		
On resistance	On resistance		
Off conductance	Off conductance		
	Junction capacitance		
	Peak reverse current, iRM		
	Initial forward current when measuring iRM		
	Rate of change of current when measuring iRM		
	<b>Reverse recovery time parameterization</b>		
	<i>Specify stretch factor</i>	<i>Specify reverse recovery time directly</i>	<i>Specify reverse recovery charge</i>
	Reverse recovery time stretch factor	Reverse recovery time, trr	Reverse recovery charge, Qrr

### Model dynamics – Diode model

Protection diode with no dynamics (default) | Protection diode with charge dynamics

Diode type. The options are:

- **Diode with no dynamics** — Select this option to prioritize simulation speed using the Diode block.
- **Diode with charge dynamics** — Select this option to prioritize model fidelity in terms of reverse mode charge dynamics using the Commutation Diode block.

### **Dependencies**

See the Protection Diode Parameter Dependencies table.

### **Forward voltage — Voltage**

0.8 V (default) | scalar

Minimum voltage required across the positive and negative block ports for the gradient of the diode I-V characteristic to be  $1/R_{on}$ , where  $R_{on}$  is the value of **On resistance**.

### **On resistance — Resistance**

0.001 Ohm (default) | scalar

Rate of change of voltage versus current above the **Forward voltage**.

### **Off conductance — Conductance**

1e-5 1/Ohm (default) | scalar

Conductance of the reverse-biased diode.

### **Junction capacitance — Capacitance**

50 nF (default) | scalar

Diode junction capacitance.

### **Dependencies**

See the Protection Diode Parameter Dependencies table.

### **Peak reverse current, $i_{RM}$ — Current**

-235 A (default) | scalar less than 0

Peak reverse current measured by an external test circuit.

### **Dependencies**

See the Protection Diode Parameter Dependencies table.



**Initial forward current when measuring  $i_{RM}$  – Current**

300 A (default) | scalar greater than 0

Initial forward current when measuring peak reverse current. This value must be greater than zero.

**Dependencies**

See the Protection Diode Parameter Dependencies table.

**Rate of change of current when measuring  $i_{RM}$  – Current change rate**

-50 A/us (default) | scalar

Rate of change of current when measuring peak reverse current.

**Dependencies**

See the Protection Diode Parameter Dependencies table.

**Reverse recovery time parameterization – Recovery-time model**

Specify stretch factor (default) | Specify reverse recovery time directly  
| Specify reverse recovery charge

Model for parameterizing the recovery time. When you select **Specify stretch factor** or **Specify reverse recovery charge**, you can specify a value that the block uses to derive the reverse recovery time. For more information on these options, see “Alternatives to Specifying  $t_{rr}$  Directly” on page 1-339.

**Dependencies**

See the Protection Diode Parameter Dependencies table.

**Reverse recovery time stretch factor – Stretch factor**

3 (default) | scalar greater than 1

Value that the block uses to calculate **Reverse recovery time,  $t_{rr}$** . Specifying the stretch factor is an easier way to parameterize the reverse recovery time than specifying the reverse recovery charge. The larger the value of the stretch factor, the longer it takes for the reverse recovery current to dissipate.

**Dependencies**

See the Protection Diode Parameter Dependencies table.

## Reverse recovery time, $t_{rr}$ — Time

15 us (default) | scalar

Interval between the time when the current initially goes to zero (when the diode turns off) and the time when the current falls to less than 10 percent of the peak reverse current.

The value of the **Reverse recovery time,  $t_{rr}$**  parameter must be greater than the value of the **Peak reverse current,  $i_{RM}$**  parameter divided by the value of the **Rate of change of current when measuring  $i_{RM}$**  parameter.

### Dependencies

See the Protection Diode Parameter Dependencies table.

## Reverse recovery charge, $Q_{rr}$ — Charge

1500 s\*uA (default) | scalar

Value that the block uses to calculate **Reverse recovery time,  $t_{rr}$** . Use this parameter if the data sheet for your diode device specifies a value for the reverse recovery charge instead of a value for the reverse recovery time.

The reverse recovery charge is the total charge that continues to dissipate when the

diode turns off. The value must be less than  $-\frac{i_{RM}^2}{2a}$ ,

where:

- $i_{RM}$  is the value specified for **Peak reverse current,  $i_{RM}$** .
- $a$  is the value specified for **Rate of change of current when measuring  $i_{RM}$** .

### Dependencies

See the Protection Diode Parameter Dependencies table.

## Snubbers

The table summarizes the **Snubbers** parameter dependencies. To learn how to read the table, see “Parameter Dependencies” on page A-2.

## Snubbers Parameter Dependencies

Snubbers Parameter Dependencies	
<b>Snubber</b>	
None	RC Snubber
	Snubber resistance
	Snubber capacitance

### Snubber — Snubber model

None (default) | RC snubber

Switching device snubber.

#### Dependencies

See the Snubbers Parameter Dependencies table.

### Snubber resistance — Resistance

0.1 Ohm (default) | scalar

Resistance of the switching device snubber.

#### Dependencies

See the Snubbers Parameter Dependencies table.

### Snubber capacitance — Capacitance

1e-7 (default) | F | scalar

Capacitance of the switching device snubber.

#### Dependencies

See the Snubbers Parameter Dependencies table.

## References

- [1] Trzynadlowski, A. M. *Introduction to Modern Power Electronics, 2nd Edition*. Hoboken, NJ: John Wiley & Sons Inc., 2010.

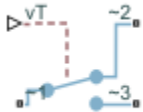
## **See Also**

[Average-Value Chopper](#) | [Four-Quadrant Chopper](#) | [One-Quadrant Chopper](#) | [Two-Pulse Gate Multiplexer](#)

**Introduced in R2018a**

# Two-Way Switch

Three-phase double-throw switch



## Library

Simscape / Electrical / Power Systems / Switches & Breakers

## Description

The Two-Way Switch block models a three-phase double-throw switch that uses an external signal to connect each phase of the port ~1 with the corresponding phase of either port ~2 or ~3.

The table shows how the external signal  $vT$  controls the block behavior.

Condition	Block Behavior	Resistance Parameter Used
$vT \leq \text{Threshold}$	Each phase of port ~1 is connected to the corresponding phase of port ~2 via internal resistance. Port ~3 is unconnected.	<b>Open conductance</b> (port ~1 to port ~3). <b>Closed resistance</b> (port ~1 to port ~2)
$vT > \text{Threshold}$	Each phase of port ~1 is connected to the corresponding phase of port ~3 via internal resistance. Port ~2 is unconnected.	<b>Open conductance</b> (port ~1 to port ~2). <b>Closed resistance</b> (port ~1 to port ~3)

## Parameters

### Closed resistance

Resistance between ports ~1 and ~3 when the switch is closed. The default value is 0.001 Ohm.

### Open conductance

Conductance between ports ~1 and ~2 when the switch is open. The default value is 1e-6 1/Ohm.

### Threshold

Threshold voltage for the control port vT. When the voltage is above the threshold, the switch is closed. The default value is 0.5 V.

## Ports

The block has the following ports:

~1

Expandable three-phase port

~2

Expandable three-phase port

~3

Expandable three-phase port

vT

Scalar control port, which is either a physical signal or an electrical port.

## See Also

[Single-Phase Switch](#) | [Single-Phase Two-Way Switch](#) | [Switch](#)

## Topics

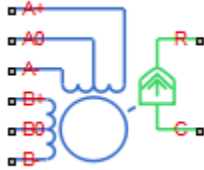
[“Expand and Collapse Three-Phase Ports on a Block”](#)

[“Switch Between Physical Signal and Electrical Ports”](#)

**Introduced in R2013b**

## Unipolar Stepper Motor

Model stepper motor with center taps on phase windings



## Library

Rotational Actuators

## Description

The Unipolar Stepper Motor block represents a stepper motor that has center taps on the two phase windings. The winding currents and mechanical output are defined by the following equations:

$$e_{A+} = -K_m \omega \sin(N_r \theta)$$

$$e_{A-} = K_m \omega \sin(N_r \theta)$$

$$e_{B+} = K_m \omega \cos(N_r \theta)$$

$$e_{B-} = -K_m \omega \cos(N_r \theta)$$

$$\frac{di_{A+}}{dt} = (v_{A+} - Ri_{A+} - e_{A+}) / L$$



$$\frac{di_{A-}}{dt} = (v_{A-} - Ri_{A-} + e_{A-}) / L$$

$$\frac{di_{B+}}{dt} = (v_{B+} - Ri_{B+} - e_{B+}) / L$$

$$\frac{di_{B-}}{dt} = (v_{B-} - Ri_{B-} + e_{B-}) / L$$

$$J \frac{d\omega}{dt} + B\omega = T_e$$

$$T_e = -K_m \left( i_{A+} - i_{A-} - \frac{e_{A+} - e_{A-}}{R_m} \right) \sin(N_r \theta) + K_m \left( i_{B+} - i_{B-} - \frac{e_{B+} - e_{B-}}{R_m} \right) \cos(N_r \theta) - T_d \sin(4N_r \theta)$$

$$\frac{d\theta}{dt} = \omega$$

where:

- $e_{A+}$  is the back emf induced across the A+ to A0 half-winding.
- $e_{A-}$  is the back emf induced across the A- to A0 half-winding.
- $e_{B+}$  is the back emf induced across the B+ to B0 half-winding.
- $e_{B-}$  is the back emf induced across the B- to B0 half-winding.
- $i_{A+}$  is the current flowing from the A+ port to the A0 center tap port.
- $i_{A-}$  is the current flowing from the A- port to the A0 center tap port.
- $i_{B+}$  is the current flowing from the B+ port to the B0 center tap port.
- $i_{B-}$  is the current flowing from the B- port to the B0 center tap port.
- $v_{A+}$  is the voltage at the A+ port relative to the A0 center tap port.
- $v_{A-}$  is the voltage at the A- port relative to the A0 center tap port.
- $v_{B+}$  is the voltage at the B+ port relative to the B0 center tap port.

- $v_B$  is the voltage at the B- port relative to the B0 center tap port.
- $K_m$  is the motor torque constant.
- $N_r$  is the number of teeth on each of the two rotor poles. The **Full step size** parameter is  $(\pi/2)/N_r$ .
- $R$  is the half-winding resistance. For example, it is the resistance between A+ and A0 ports.
- $L$  is the half-winding inductance. For example, it is the inductance between A+ and A0 ports.
- $R_m$  is the magnetizing resistance.
- $B$  is the rotational damping.
- $J$  is the inertia.
- $\omega$  is the rotor speed.
- $\theta$  is the rotor angle.
- $T_d$  is the detent torque amplitude.

If the initial rotor is zero or some multiple of  $(\pi/2)/N_r$ , the rotor is aligned with the A-phase winding. If a positive current flows from the A+ port to the A0 center tap port, then the stepper acts to stay aligned with the A-phase. Equivalently, a positive current flowing from the A0 center tap port to the A- port also acts on the rotor to stay aligned with the A-phase.

The Unipolar Stepper Motor block produces a positive torque acting from the mechanical C to R ports for either of the following sequences. Both sequences assume the rotor initial angle is zero or some multiple of  $(\pi/2)/N_r$ .

Sequence	Center taps connected to ground	Center taps connected to positive supply
1	Positive current from A+ to A0	Positive current from A0 to A-
2	Positive current from B+ to B0	Positive current from B0 to B-
3	Positive current from A- to A0	Positive current from A0 to A-
4	Positive current from B- to B0	Positive current from B0 to B-

## Averaged Mode

If you set the **Simulation mode** parameter to **Averaged**, both for a Unipolar Stepper Motor block and for the Unipolar Stepper Motor Driver block that controls it, then the

individual steps are not simulated. This can be a good way to speed up simulation. In Averaged mode, under non-slipping conditions, the motor and driver are represented by a second-order linear system that tracks the specified step rate. The demanded step rate is determined directly from voltage across A+ and A-. So, for example, a voltage of +10V across the A+ and A- terminals is interpreted as a step rate demand of ten steps per second. See the Unipolar Stepper Motor Driver reference page for more information on how to connect the Unipolar Stepper Motor Driver to your step angle controller.

Averaged mode includes a slip estimator to predict whether the stepper motor would have slipped if running in Stepping simulation mode. Slip is predicted if the motor torque exceeds the **Vector of maximum torque values** parameter value for longer than one step period, the step period being determined from the current step rate demand. Upon detecting slip, the simulation will proceed or stop with an error, according to the **Action on slipping** parameter value. If you choose the action that lets the simulation continue, note that simulation results may be incorrect: when slipping occurs, the torque generated by the motor will not generally be the maximum available torque; the maximum torque is only achieved if the stepper controller detects slip and adjusts the step rate command accordingly.

The dynamics of the equivalent second-order system are determined from the values that you specify for the **Approximate total load inertia** and **Maximum step rate command** parameters. It is important that you set as accurate values as possible for these parameters, so that the step rate command is tracked, and the block does not generate false slipping warnings or errors.

If you run the motor in Averaged mode with the optional thermal ports exposed (see “Thermal Ports” on page 1-2013), then heat is added to the thermal ports assuming that the windings are always powered, even when the step rate command is zero. The block makes adjustments for half stepping and for reduced torque (and winding currents) at higher speeds. For these adjustments to be correct, the **Vector of maximum torque** parameter values must be correct. For half stepping, at zero speed the heat generated by the block is the average of that generated when stopped at a half step and at a full step.

If you simulate or predict slip, MathWorks recommends that you do some validation runs comparing Stepping and Averaged modes before using the averaged model representation for simulation studies.

## Thermal Ports

The block has five optional thermal ports, one for each of the four half-windings and one for the rotor. These ports are hidden by default. To expose the thermal ports, right-click

the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal ports on the block icon, and adds the **Temperature Dependence** and **Thermal Port** tabs to the block dialog box. These tabs are described further on this reference page.

Use the thermal ports to simulate the effects of copper resistance and iron losses that convert electrical power to heat. For more information on using thermal ports in actuator blocks, see “Simulating Thermal Effects in Rotational and Translational Actuators”.

## Basic Assumptions and Limitations

The model is based on the following assumptions:

- This model neglects magnetic saturation effects and any magnetic coupling between phases.
- When you select the **Start simulation from steady state** check box in the Simscape Solver Configuration block, this block will not initialize an **Initial rotor angle** value between  $-\pi$  and  $\pi$ .
- All four half-windings are assumed to be identical, and therefore have the same resistance temperature coefficient, alpha, and the same thermal mass.
- To use Averaged mode, the Unipolar Stepper Motor block must be directly connected to a Unipolar Stepper Motor Driver block also running in Averaged mode.
- The Averaged mode is an approximation, and exact step tracking compared to the Stepping mode should not be expected.
- Slip detection in Averaged mode is approximate, and depends on a good estimate for load inertia and maximum step rate. Incorrect values may result in false slip detection.
- When simulating slip in Averaged mode, it is assumed that the stepper motor controller adjusts the step rate command so as to achieve maximum possible torque.

## Parameters

- “Electrical Torque Tab” on page 1-2015
- “Mechanical Tab” on page 1-2016
- “Temperature Dependence Tab” on page 1-2017
- “Thermal Port Tab” on page 1-2017

## Electrical Torque Tab

### Simulation mode

Select **Stepping** or **Averaged**. Use **Averaged** only if the block is connected directly to a Unipolar Stepper Motor Driver block also running in **Averaged** mode. The default value is **Stepping**.

### Half-winding resistance

Half of the resistance of the A and B phase windings as measured between the A+ and A-, and the B+ and B- ports. This parameter is visible only if **Simulation mode** is set to **Stepping**. The default value is 0.55  $\Omega$ .

### Half-winding inductance

Half of the inductance of the A and B phase windings as measured between the A+ and A-, and the B+ and B- ports. This parameter is visible only if **Simulation mode** is set to **Stepping**. The default value is 0.0015 H.

### Motor torque constant

Motor torque constant  $K_m$ . This parameter is visible only if **Simulation mode** is set to **Stepping**. The default value is 0.19 N\*m/A.

### Detent torque

The amplitude of the sinusoidal torque variation observed when rotating the shaft of the unpowered motor. This parameter is visible only if **Simulation mode** is set to **Stepping**. The default value is 0 N\*m.

### Magnetizing resistance

The total magnetizing resistance seen from each of the phase windings, for example across A+ and A0. This parameter is visible only if **Simulation mode** is set to **Stepping**. The value must be greater than zero. The default value is Inf, which implies that there are no iron losses.

### Vector of rotational speeds

Vector of rotational speeds at which to define maximum torque values, for slip prediction. This parameter is visible only if **Simulation mode** is set to **Averaged**. The default value is [0 1000 3000] rpm.

### Vector of maximum torque values

Vector of maximum torque values, to be used for slip prediction in conjunction with the **Vector of rotational speeds** parameter. The maximum torque values are often given on a datasheet, and should correspond to the supply voltage and stepping type (half step or full step) specified in the driver. This parameter is visible only if **Simulation mode** is set to **Averaged**. The default value is [2 2 1] N\*m.

## Action on slipping

Select the action for the block to perform during simulation upon detecting slip:

- none — Continue simulation, limiting the load torque according to the **Vector of maximum torque values**.
- warn — Continue simulation, limiting the load torque according to the **Vector of maximum torque values**, and generate a warning that the rotor is slipping.
- error — Stop the simulation and generate an error message that the rotor is slipping.

Note that if you choose an action that lets the simulation continue, simulation results may be incorrect: when slipping occurs, the torque generated by the motor will not generally be the maximum available torque; the maximum torque is only achieved if the stepper controller detects slip and adjusts the step rate command accordingly.

This parameter is visible only if **Simulation mode** is set to Averaged.

## Approximate total load inertia

The approximate total load inertia, including the rotor inertia. This value is used to help predict when slipping will occur due to rapid acceleration demands. This parameter is visible only if **Simulation mode** is set to Averaged. The default value is  $1e-4 \text{ kg}\cdot\text{m}^2$ .

## Maximum step rate command

The maximum step rate that your system will command. It is used to determine a suitable bandwidth for the second order system approximation to the stepper motor and driver. This parameter is visible only if **Simulation mode** is set to Averaged. The default value is 10 Hz.

## Full step size

Step size when changing the polarity of either the A or B phase current. The default value is  $1.8^\circ$ .

## Mechanical Tab

### Rotor inertia

Resistance of the rotor to change in motor motion. The default value is  $4.5e-05 \text{ kg}\cdot\text{m}^2$ . The value can be zero.

**Rotor damping**

Energy dissipated by the rotor. The default value is  $8e-04$  N\*m/(rad/s). The value can be zero.

**Initial rotor speed**

Speed of the rotor at the start of the simulation. The default value is 0 rpm.

**Initial rotor angle**

Angle of the rotor at the start of the simulation. The default value is 0 rad.

**Temperature Dependence Tab**

This tab appears only for blocks with exposed thermal ports. For more information, see “Thermal Ports” on page 1-2013.

**Resistance temperature coefficient**

Parameter  $\alpha$  in the equation defining resistance as a function of temperature, as described in “Thermal Model for Actuator Blocks”. It is assumed that all windings are made of the same material, and therefore have the same resistance temperature coefficient. The default value is for copper, and is  $0.00393$  1/K.

**Measurement temperature**

The temperature for which motor parameters are defined. The default value is 25 °C.

**Thermal Port Tab**

This tab appears only for blocks with exposed thermal ports. For more information, see “Thermal Ports” on page 1-2013.

**Half-winding thermal mass**

The thermal mass for half of either the A or B winding. The thermal mass is the energy required to raise the temperature by one degree. It is assumed that all four half-windings have the same thermal mass. The default value is 100 J/K.

**Half-winding initial temperatures, [T\_A+ T\_A- T\_B+ T\_B-]**

A 1 by 4 row vector defining the temperature of the four half-windings at the start of simulation. The default value is [ 25 25 25 25 ] °C.

**Rotor thermal mass**

The thermal mass of the rotor, that is, the energy required to raise the temperature of the rotor by one degree. The default value is 50 J/K.

### **Rotor initial temperature**

The temperature of the rotor at the start of simulation. The default value is 25 °C.

### **Percentage of magnetizing resistance associated with the rotor**

The percentage of the magnetizing resistance associated with the magnetic path through the rotor. It determines how much of the iron loss heating is attributed to the rotor thermal port HR, and how much is attributed to the four winding thermal ports. The default value is 90%.

## **Ports**

The block has the following ports:

A+

Top A-phase electrical connection

A0

A-phase center tap connection

A-

Lower A-phase electrical connection

B+

Top B-phase electrical connection.

B0

B-phase center tap connection

B-

Lower B-phase electrical connection

C

Mechanical rotational conserving port

R

Mechanical rotational conserving port

HA+

Thermal port for winding between A+ and A0. For more information, see “Thermal Ports” on page 1-2013.



HA-

Thermal port for winding between A- and A0. For more information, see “Thermal Ports” on page 1-2013.

HB+

Thermal port for winding between B+ and B0. For more information, see “Thermal Ports” on page 1-2013.

HB-

Thermal port for winding between B- and B0. For more information, see “Thermal Ports” on page 1-2013.

HR

Thermal port for rotor. For more information, see “Thermal Ports” on page 1-2013.

## Examples

See the Unipolar Stepper Motor with Control and Unipolar Stepper Motor Averaged Mode examples.

## References

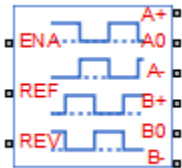
- [1] M. Bodson, J. N. Chiasson, R. T. Novotnak and R. B. Rekowski. “High-Performance Nonlinear Feedback Control of a Permanent Magnet Stepper Motor.” IEEE Transactions on Control Systems Technology, Vol. 1, No. 1, March 1993.
- [2] P. P. Acarnley. *Stepping Motors: A Guide to Modern Theory and Practice*. New York: Peregrinus, 1982.
- [3] S.E. Lyshevski. *Electromechanical Systems, Electric Machines, and Applied Mechatronics*. CRC, 1999.

## See Also

Stepper Motor | Unipolar Stepper Motor Driver

# Unipolar Stepper Motor Driver

Driver for unipolar stepper motor



## Library

Drivers

## Description

The Unipolar Stepper Motor Driver block represents a driver specifically configured for use with the Unipolar Stepper Motor block. It connects the two winding center-tap connections A0 and B0 to the positive supply with a voltage equal to the value you provide for the **Output voltage amplitude** parameter. The A+, A-, B+, and B- ports are grounded in the appropriate sequence to create the stepping motion. The block initiates a step each time the voltage at the ENA port rises above the **Enable threshold voltage** parameter value.

If the voltage at the REV port is less than or equal to the **Reverse threshold voltage** parameter value, pulse A leads pulse B by 90 degrees. If the voltage at the REV port is greater than the **Reverse threshold voltage** value, pulse B leads pulse A by 90 degrees and the motor direction is reversed.

At time zero, A- and B+ are grounded.

If you set the **Stepping mode** parameter to `Half stepping`, the Unipolar Stepper Motor Driver block can produce the output waveforms required for half stepping. In this mode, there is an intermediate state between the full steps, in which just one of the A or the B half-windings is powered. As a result, the step size is half of the stepper motor's full

step size. At half steps, windings that are not powered are short-circuited. This approximates the effect of a freewheeling diode connected across the windings.

## Averaged Mode

If you set the **Simulation mode** parameter to **Averaged**, both for a Unipolar Stepper Motor Driver block and for the Unipolar Stepper Motor block connected to it, then the individual steps are not simulated. This can be a good way to speed up simulation. The Averaged mode assumes that the external controller provides a step rate demand. This step rate demand is determined from the voltage applied between the ENA and REF ports on the Unipolar Stepper Motor Driver block, by multiplying this voltage by the value of the **Step rate sensitivity** parameter. The rotation direction is set by the REF port in the same way as for the Stepping mode.

Averaged mode needs to communicate the step rate demand and also output voltage amplitude information to the Unipolar Stepper Motor block. To do this, the step rate demand is applied as an equivalent voltage across the A+ and A- ports. Similarly the output voltage amplitude information is conveyed by applying a steady-state voltage across the B+ and B- ports with value equal to the **Output voltage amplitude** parameter.

## Basic Assumptions and Limitations

The model is based on the following assumptions:

- To use Averaged mode, the Unipolar Stepper Motor Driver block must be directly connected to a Unipolar Stepper Motor block also running in Averaged mode.
- When changing from Stepping to Averaged mode and back, you will need to modify your upstream blocks that provide the input voltages to the Unipolar Stepper Motor Driver. One way to achieve this easily is to use Simulink variant subsystems.

## Parameters

### Simulation mode

Select **Stepping** or **Averaged**. Use **Averaged** only if the block is connected directly to a Unipolar Stepper Motor block also running in Averaged mode. The default value is **Stepping**.

## Enable threshold voltage

When the voltage at the ENA port rises above this threshold, the Unipolar Stepper Motor Driver block initiates a step. This parameter is visible only if **Simulation mode** is set to **Stepping**. The default value is 2.5 V.

## Step rate sensitivity

This parameter converts the voltage presented across the ENA and REF ports into a step rate demand. This parameter is visible only if **Simulation mode** is set to **Averaged**. The default value is 10 steps-per-second per volt.

## Reverse threshold voltage

When the voltage at the REV port rises above this threshold, pulse B leads pulse A by 90 degrees, and the motor direction is reversed. The default value is 2.5 V.

## Output voltage amplitude

Amplitude of the output pulse trains. The default value is 10 V.

## Stepping mode

Select **Full stepping** or **Half stepping**. The default value is **Full stepping**.

## Ports

The block has the following ports:

A+

Top A-phase electrical connection

A0

A-phase center tap connection

A-

Lower A-phase electrical connection

B+

Top B-phase electrical connection

B0

B-phase center tap connection

B-

Lower B-phase electrical connection

ENA

Triggering input step voltage

REF

Input floating reference voltage

REV

Input voltage that controls motor direction

## Examples

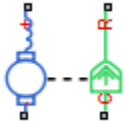
See the Unipolar Stepper Motor with Control and Unipolar Stepper Motor Averaged Mode examples.

## See Also

Controlled PWM Voltage | Unipolar Stepper Motor

## Universal Motor

Model electrical and torque characteristics of a universal (or series) motor

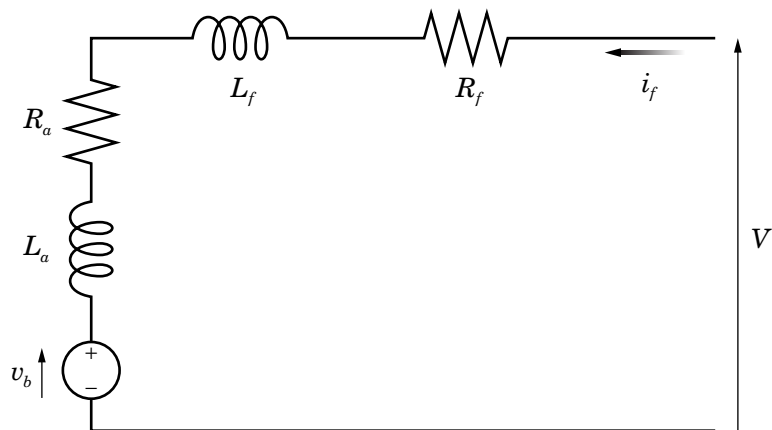


## Library

Rotational Actuators

## Description

The Universal Motor block represents the electrical and torque characteristics of a universal (or series) motor using the following equivalent circuit model.



Where:

- $R_a$  is the armature resistance.

- $L_a$  is the armature inductance.
- $R_f$  is the field winding resistance.
- $L_f$  is the field winding inductance.

When you set the **Model parameterization** parameter to **By equivalent circuit parameters**, you specify the equivalent circuit parameters for this model. The Universal Motor block computes the motor torque as follows:

- 1 The magnetic field in the motor induces the following back emf  $v_b$  in the armature:

$$v_b = L_{af} i_f \omega$$

where  $L_{af}$  is a constant of proportionality and  $\omega$  is the angular velocity.

- 2 The mechanical power is equal to the power reacted by the back emf:

$$P = v_b i_f = L_{af} i_f^2 \omega$$

- 3 The motor torque is:

$$T = P / \omega = L_{af} i_f^2$$

The torque-speed characteristic for the Universal Motor block model is related to the parameters in the preceding figure. When you set the **Model parameterization** parameter to **By DC rated power, rated speed & maximum torque** or **By DC rated power, rated speed & electrical power**, the block solves for the equivalent circuit parameters as follows:

- 1 For the steady-state torque-speed relationship when using a DC supply,  $L$  has no effect.
- 2 Sum the voltages around the loop:

$$V = (R_f + R_a) i_f + v_b = (R_f + R_a + L_{af} \omega) i_f$$

- 3 Solve the preceding equation for  $i_f$  and substitute this value into the equation for torque:

$$T = L_{af} \left( \frac{V}{R_f + R_a + L_{af} \omega} \right)^2$$

The block uses the rated speed and power to calculate the rated torque. The block uses the rated torque and rated speed values in the preceding equation plus the corresponding electrical power to determine values for  $R_f+R_a$  and  $L_{af}$ .

When you set the **Model parameterization** parameter to By AC rated power, rated speed, current & electrical power, then the block must include the inductive terms  $L_a$  and  $L_f$  in the model. This requires information about the RMS rated current and voltage for the total inductance.

The block models motor inertia  $J$  and damping  $B$  for all values of the **Model parameterization** parameter. The output torque is:

$$T_{load} = L_{af} \left( \frac{V}{R_f + R_a + L_{af} \omega} \right)^2 - J \dot{\omega} - B \omega$$

The block produces a positive torque acting from the mechanical C to R ports.

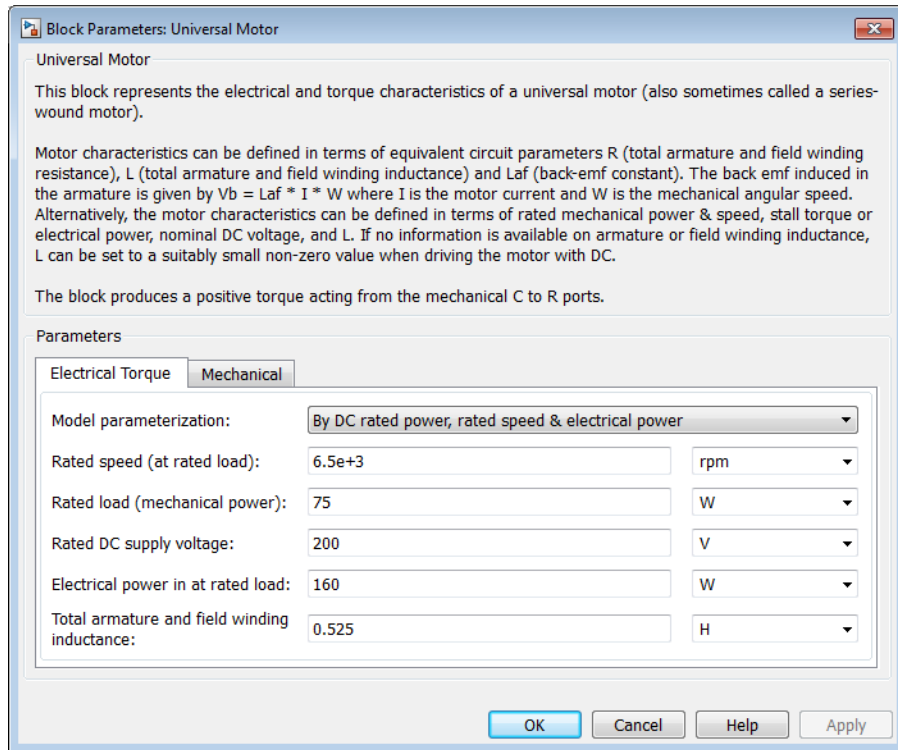
## Thermal Ports

The block has two optional thermal ports, one per winding, hidden by default. To expose the thermal ports, right-click the block in your model, and then from the context menu select **Simscape > Block choices > Show thermal port**. This action displays the thermal ports on the block icon, and adds the **Temperature Dependence** and **Thermal Port** tabs to the block dialog box. These tabs are described further on this reference page.

Use the thermal ports to simulate the effects of copper resistance losses that convert electrical power to heat. For more information on using thermal ports in actuator blocks, see “Simulating Thermal Effects in Rotational and Translational Actuators”.



## Parameters



- “Electrical Torque Tab” on page 1-2027
- “Mechanical Tab” on page 1-2029
- “Temperature Dependence Tab” on page 1-2030
- “Thermal Port Tab” on page 1-2030

## Electrical Torque Tab

### Model parameterization

Select one of the following methods for block parameterization:

- By equivalent circuit parameters — Provide electrical parameters for an equivalent circuit model of the motor.

- By DC rated power, rated speed & maximum torque — Provide DC power and speed parameters that the block converts to an equivalent circuit model of the motor. This is the default method.
- By DC rated power, rated speed & electrical power — Provide AC power and speed parameters that the block converts to an equivalent circuit model of the motor.
- By AC rated power, rated speed, current & electrical power — Provide AC power and speed parameters that the block converts to an equivalent circuit model of the motor.

### **Total armature and field winding resistance**

Total resistance of the armature and field winding. This parameter is only visible when you select **By equivalent circuit parameters** for the **Model parameterization** parameter. The default value is 132.8  $\Omega$ .

### **Rated speed (at rated load)**

Motor speed at the rated mechanical load. This parameter is only visible when you select **By DC rated power, rated speed & maximum torque**, **By DC rated power, rated speed & electrical power**, or **By AC rated power, rated speed, current & electrical power** for the **Model parameterization** parameter. The default value is 6.5e+03 rpm.

### **Rated load (mechanical power)**

The mechanical load for which the motor is rated to operate. This parameter is only visible when you select **By DC rated power, rated speed & maximum torque**, **By DC rated power, rated speed & electrical power**, or **By AC rated power, rated speed, current & electrical power** for the **Model parameterization** parameter. The default value is 75 W.

### **Rated DC supply voltage**

The DC voltage at which the motor is rated to operate. This parameter is only visible when you select **By DC rated power, rated speed & maximum torque** or **By DC rated power, rated speed & electrical power** for the **Model parameterization** parameter. The default value is 200 V.

### **Electrical power in at rated load**

The amount of electrical power the motor uses at the rated mechanical power. This parameter is only visible when you select **By DC rated power, rated speed & electrical power** or **By AC rated power, rated speed, current & electrical power** for the **Model parameterization** parameter. The default value is 160 W.

**Maximum (starting) torque**

Maximum torque the motor produces. This parameter is only visible when you select By DC rated power, rated speed & maximum torque for the **Model parameterization** parameter. The default value is 0.39 N\*m.

**Total armature and field winding inductance**

Total inductance of the armature and field winding. If you do not have information about this inductance, set the value of this parameter to a small, nonzero number. This parameter is only visible when you select By equivalent circuit parameters, By DC rated power, rated speed & maximum torque, or By DC rated power, rated speed & electrical power for the **Model parameterization** parameter. The default value is 0.525 H.

---

**Note** You can set the **Total armature and field winding inductance** value to zero, but this only makes sense if you are driving the motor with a DC source.

---

**RMS rated voltage**

RMS supply voltage when the motor operates on AC power. This parameter is only visible when you select By AC rated power, rated speed, current & electrical power for the **Model parameterization** parameter. The default value is 240 V.

**RMS current at rated load**

RMS current when the motor operates on AC power at the rated load. This parameter is only visible when you select By AC rated power, rated speed, current & electrical power for the **Model parameterization** parameter. The default value is 0.8 A.

**AC frequency**

Frequency of the AC supply voltage. This parameter is only visible when you select By AC rated power, rated speed, current & electrical power for the **Model parameterization** parameter. The default value is 50 Hz.

**Mechanical Tab****Rotor inertia**

Rotor inertia. The default value is  $2e-04$  kg\*m<sup>2</sup>. The value can be zero.

### **Rotor damping**

Rotor damping. The default value is  $1e-06$  N\*m/(rad/s). The value can be zero.

### **Initial rotor speed**

Speed of the rotor at the start of the simulation. The default value is 0 rpm.

## **Temperature Dependence Tab**

This tab appears only for blocks with exposed thermal ports. For more information, see “Thermal Ports” on page 1-2026.

### **Field to armature resistance ratio, $R_f/R_a$**

The ratio of the field to the armature resistance. This parameter is required only when showing the field and armature thermal ports. It is used to determine individual resistance values for the field and armature windings so that the thermal heat generated by the two resistors can be apportioned correctly. The default value is 1.

### **Resistance temperature coefficients, $[\alpha_f \alpha_a]$**

A 1 by 2 row vector defining the coefficient  $\alpha$  in the equation relating resistance to temperature, as described in “Thermal Model for Actuator Blocks”. The first element corresponds to the field winding, and the second to the armature. The default value is for copper, and is  $[ 0.00393 \ 0.00393 ]$  1/K.

### **Measurement temperature**

The temperature for which motor parameters are defined. The default value is 25 °C.

## **Thermal Port Tab**

This tab appears only for blocks with exposed thermal ports. For more information, see “Thermal Ports” on page 1-2026.

### **Thermal masses, $[M_f M_a]$**

A 1 by 2 row vector defining the thermal mass for the field and armature windings. The thermal mass is the energy required to raise the temperature by one degree. The default value is  $[ 100 \ 100 ]$  J/K.

### **Initial temperatures, $[T_f T_a]$**

A 1 by 2 row vector defining the temperature of the field and armature thermal ports at the start of simulation. The default value is  $[ 25 \ 25 ]$  °C.

## Ports

The block has the following ports:

+

Positive electrical port.

-

Negative electrical port.

C

Mechanical rotational conserving port.

R

Mechanical rotational conserving port.

Hf

Field winding thermal port. For more information, see “Thermal Ports” on page 1-2026.

Ha

Armature winding thermal port. For more information, see “Thermal Ports” on page 1-2026.

## References

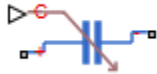
[1] Bolton, W. *Mechatronics: Electronic Control Systems in Mechanical and Electrical Engineering* 3rd edition, Pearson Education, 2004.

## See Also

DC Motor | Induction Motor | Servomotor | Shunt Motor

# Variable Capacitor

Model linear time-varying capacitor



## Library

Passive Devices

## Description

The Variable Capacitor block represents a linear time-varying capacitor. The block provides two options for the relationship between the current  $i$  through the capacitor and the voltage  $v$  across the device when the capacitance at port C is  $C$ . The **Equation** parameter determines which of the following equations the block uses:

•

$$i = C \frac{dv}{dt}$$

Use the preceding equation when the capacitance is defined as the local gradient of the charge-voltage curve for a given voltage:

$$C(v) = \frac{dQ(v)}{dv}$$

•

$$i = \frac{dC}{dt}v + C \frac{dv}{dt}$$

Use the preceding equation when the capacitance is defined as the ratio of the charge  $Q$  to the steady-state voltage:

$$C(v) = \frac{Q(v)}{v}$$

The block includes a resistor in series with the variable capacitor. You can use this resistor to represent the total ohmic connection resistance of the capacitor. You may need to use this resistor to prevent numerical issues for some circuit topologies, such as where a Variable Capacitor block is connected in parallel with another capacitor block that does not have a series resistance.

## Parameters

### Equation

Select one of the following options for block capacitance:

- $I = C \cdot dV/dt$  — This equation assumes the capacitance is defined as the local gradient of the charge-voltage curve for a given voltage. This option is the default.
- $I = C \cdot dV/dt + dC/dt \cdot V$  — This equation assumes the capacitance is defined as the ratio of the charge to the steady-state voltage.

### Minimum capacitance $C > 0$

The lower limit on the value of the signal at port C. This limit prevents the signal from reaching a value that has no physical meaning. The default value is  $1e-09$  F.

### Series resistance

The value of the resistance placed in series with the variable capacitor. The default value is  $1e-06$   $\Omega$ .

### Initial charge

The charge at the start of the simulation. This parameter is only visible when you select  $I = C \cdot dV/dt + dC/dt \cdot V$  for the **Equation** parameter. The default value is 0 C.

### Initial voltage

The output voltage at the start of the simulation. This parameter is only visible when you select  $I = C \cdot dV/dt$  for the **Equation** parameter. The default value is 0 V.

## Ports

The block has the following ports:

C

Capacitance physical signal port (C must be finite and greater than zero)

+

Positive electrical port

-

Negative electrical port

## **See Also**

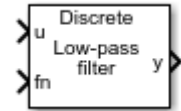
Variable Inductor | Variable Resistor



# Variable-Frequency Second-Order Filter

Variable-Frequency Second-Order Filter

**Library:** Simscape / Electrical / Power Systems / Control /  
General Control

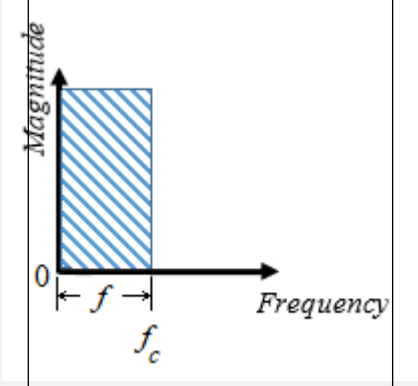
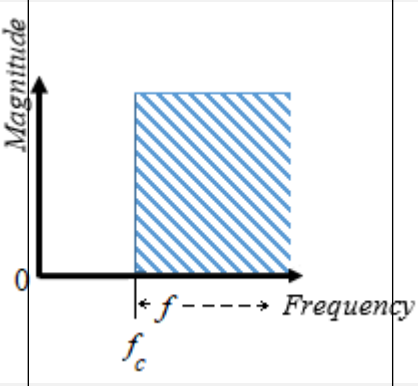


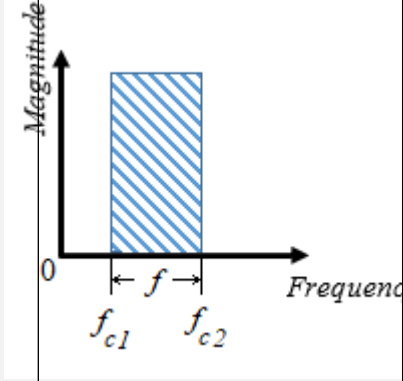
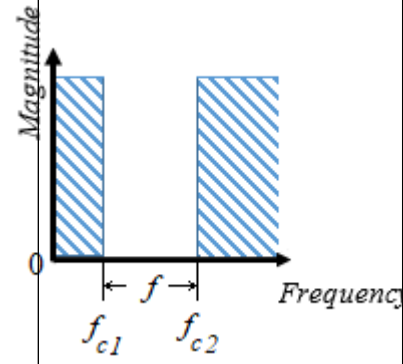
## Description

The Variable-Frequency Second-Order Filter block implements four different types of second-order filters, each with external frequency input.. Filters are useful for attenuating noise in measurement signals.

The block provides these filter types:

- Low pass — Allows signals,  $f$ , only in the range of frequencies below the cutoff frequency,  $f_c$ , to pass.
- High pass — Allows signals,  $f$ , only in the range of frequencies above the cutoff frequency,  $f_c$ , to pass.
- Band pass — Allows signals,  $f$ , only in the range of frequencies between two cutoff frequencies,  $f_{c1}$  and  $f_{c2}$ , to pass.
- Band stop — Prevents signals,  $f$ , only in the range of frequencies between two cutoff frequencies,  $f_{c1}$  and  $f_{c2}$ , from passing.

Filter Type	Frequency Range, $f$	
<p><b>Low-Pass</b></p>  <p>The graph shows a vertical axis labeled 'Magnitude' and a horizontal axis labeled 'Frequency'. The origin is marked '0'. A shaded rectangular region extends from the origin to a frequency value <math>f_c</math> on the horizontal axis. A double-headed arrow below the horizontal axis indicates the width of this region is <math>f</math>.</p>	$f < f_c$	
<p><b>High-Pass</b></p>  <p>The graph shows a vertical axis labeled 'Magnitude' and a horizontal axis labeled 'Frequency'. The origin is marked '0'. A shaded rectangular region starts at a frequency value <math>f_c</math> on the horizontal axis and extends to the right. A dashed arrow below the horizontal axis indicates the width of this region is <math>f</math>.</p>	$f > f_c$	

Filter Type	Frequency Range, $f$	
<b>Band-Pass</b>		$f_{c1} < f < f_{c2}$
<b>Band-Stop</b>		$f_{c1} < f < f_{c2}$

## Equations

The second order derivative state equation for the filter is:

$$\frac{d^2x}{dt^2} = u - 2\zeta\omega_n \frac{dx}{dt} - \omega_n^2 x$$

Where:

- $x$  is the filter internal state.

- $u$  is the filter input.
- $\omega_n$  is the filter natural frequency.
- $\zeta$  is the filter damping factor.

For each filter type, the table maps the block output,  $y(x)$ , as a function of the internal state of the filter, to the s-domain transfer function,  $G(s)$ .

<b>Filter Type</b>	<b>Output, <math>y(x)</math></b>	<b>Transfer Function, <math>G(s)</math></b>
<b>Low-Pass</b>	$\omega_n^2 x$	$\frac{\omega_n^2}{s^2 + 2\zeta\omega_n s + \omega_n^2}$
<b>High-Pass</b>	$\frac{d^2 x}{dt^2}$	$\frac{s^2}{s^2 + 2\zeta\omega_n s + \omega_n^2}$
<b>Band-Pass</b>	$2\zeta\omega_n \frac{dx}{dt}$	$\frac{2\zeta\omega_n s}{s^2 + 2\zeta\omega_n s + \omega_n^2}$
<b>Band-Stop</b>	$\frac{d^2 x}{dt^2} + x$	$\frac{s^2 + \omega_n^2}{s^2 + 2\zeta\omega_n s + \omega_n^2}$

For Initialization:

$$\dot{x}(0) = \left. \frac{dx}{dt} \right|_{t=0}$$

$$u(0) = u_1(0) + u_2(0)$$

$$u_1(0) = A_0 e^{j\varphi_0}$$

$$u_2(0) = b_0 e^{j\frac{\pi}{2}}$$

Where:

- $x(0)$  is the initial state of the filter.
- $u(0)$  is the initial input to the filter.
- $u_1(0)$  is the AC component of the steady-state initial input.
- $A_0$  is the initial amplitude.
- $\varphi_0$  is the initial phase.
- $u_2(0)$  is the DC component of the steady-state initial input.
- $b_0$  is the initial bias.

In the  $s$ -domain  $s = j\omega_0$ . Therefore, for the initial frequency,  $\omega_0$ :

$$\dot{x}(0) = \text{Im} \left( \frac{j\omega_0 u_1(0)}{-\omega_0^2 + j\omega_0 2\zeta\omega_n + \omega_n^2} \right)$$

$$x(0) = \text{Im} \left( \frac{\dot{x}(0)\omega_n^2}{j\omega_0} + u_2(0) \right)$$

## Ports

### Input

#### **u** — Filter input

scalar

Filter input.

Data Types: single | double

**fn — Natural frequency**

scalar

Natural frequency.

Data Types: single | double

## Output

**y — Filtered output**

scalar

Filtered output.

Data Types: single | double

## Parameters

### Main

**Filter type — Filter type**

Low-pass (default) | High-pass | Band-pass | Band-stop

Type of second-order filter.

**Initial natural frequency (Hz) — Initial natural frequency**

60 (default) | positive scalar

Natural frequency, in Hz, at the start of simulation.

### Initial Conditions

**Damping factor — Damping factor**

0.707 (default) | nonnegative scalar

Damping factor of the filter.

**Sample time (-1 for inherited) — Block sample time**

-1 (default) | 0 | positive scalar

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For inherited discrete-time operation, specify  $-1$ . For discrete-time operation, specify a positive integer. For continuous-time operation, specify  $0$ .

If this block is in a masked subsystem, or other variant subsystem that allows you to switch between continuous operation and discrete operation, promote the sample time parameter. Promoting the sample time parameter ensures correct switching between the continuous and discrete implementations of the block. For more information, see “Promote Parameter to Mask” (Simulink).

**Initial amplitude – Initial amplitude**

$0$  (default) | nonnegative scalar

Amplitude at the start of simulation.

**Initial phase (rad) – Initial phase**

$0$  (default) | nonnegative scalar

Phase, in rad, at the start of simulation.

**Initial frequency (Hz) – Initial frequency**

$0$  (default) | scalar

Frequency, in Hz, at the start of simulation.

**Initial bias – Initial bias**

$0$  (default) | nonnegative scalar

Bias at the start of simulation.

**References**

- [1] Agarwal, A. and Lang, J. H. *Foundations of Analog and Digital Electronic Circuits*. New York: Elsevier, 2005.

## **See Also**

Low-Pass Filter (Discrete or Continuous) | SM PSS1A | Second-Order Filter | Second-Order Low-Pass Filter (Discrete or Continuous) | Washout (Discrete or Continuous)

**Introduced in R2018b**



# Variable Inductor

Model linear time-varying inductor



## Library

Passive Devices

## Description

The Variable Inductor block represents a linear time-varying inductor. The block provides two options for the relationship between the voltage  $v$  across the device and the current through the inductor  $i$  when the inductance at port L is  $L$ . The **Equation** parameter determines which of the following equations the block uses:

•

$$v = \frac{dL}{dt} i + L \frac{di}{dt}$$

Use the preceding equation when the inductance is defined as the ratio of the magnetic flux  $\Phi$  to the steady-state current:

$$L(i) = \frac{\Phi(i)}{i}$$

•

$$v = L \frac{di}{dt}$$

Use the preceding equation when the inductance is defined as the local gradient of the flux-current curve for a given current:

$$L(i) = \frac{d\Phi(i)}{di}$$

The block includes a conductance in parallel with the variable inductor. You can use the conductor to represent the total insulation conductance of the inductor. You may need to use the conductor to prevent numerical issues for some circuit topologies, such as where a Variable Inductor block is connected in series with another inductor block that does not have a parallel conductance.

## Parameters

### Equation

Select one of the following options for block inductance:

- $V = L \cdot dI/dt + dL/dt \cdot I$  — This equation assumes the inductance is defined as the ratio of the magnetic flux to the steady-state current. This option is the default.
- $V = L \cdot dI/dt$  — This equation assumes the inductance is defined as the local gradient of the flux-current curve for a given current.

### Minimum inductance $L > 0$

The lower limit on the value of the signal at port L. This limit prevents the signal from reaching a value that has no physical meaning. The default value is  $1e-06$  H.

### Parallel conductance

The value of the conductance placed in parallel with the variable inductor. The default value is  $1e-09$  1/Ω.

### Initial magnetic flux

The magnetic flux at the start of the simulation. This parameter is only visible when you select  $V = L \cdot dI/dt + dL/dt \cdot I$  for the **Equation** parameter. The default value is 0 Wb.

### Initial current

The output current at the start of the simulation. This parameter is only visible when you select  $V = L \cdot dI/dt$  for the **Equation** parameter. The default value is 0 A.

## Ports

The block has the following ports:

L

Inductance physical signal port (L must be finite and greater than zero)

+

Positive electrical port

-

Negative electrical port

## See Also

[Variable Capacitor](#) | [Variable Resistor](#)

# Varistor

Voltage-dependent resistor

**Library:** Simscape / Electrical / Electronics and Mechatronics /  
Passive Devices  
Simscape / Electrical / Power Systems / Passive  
Devices / Fundamental Components



## Description

The Varistor block represents a voltage-dependent resistor (VDR). This component is also commonly known as a metal-oxide varistor (MOV). The block exhibits high resistance at low voltages and low resistance at high voltages.

You can protect parts of an electrical circuit from high-voltage surges by placing this block in parallel with them. When a surge occurs, the resistance of the varistor drops significantly, causing the current to be shunted through the varistor rather than through the circuit.

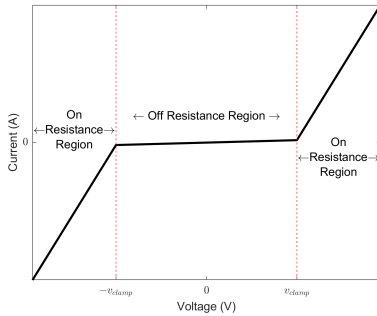
Use the **Parameterization** parameter to choose between two different behaviors for this block. The **Linear** option focuses on the on- and off-states of the varistor and uses a linear relationship between current and voltage in both regions. The **Power-law** option uses an exponential relationship between current and voltage in the initial on-state. This option also adds a third, linear region at higher voltages.

## Linear Parameterization

This parameterization option separates the voltage-current relationship into two linear regions:

- Off-region — resistance is high and current increases slowly with increasing voltage.
- On-region — resistance is low and current increases rapidly with increasing voltage.

This figure shows the voltage-current relationship across the on- and off-regions.



Use linear parameterization in one of these scenarios:

- You are modeling voltage surges close to the threshold voltage
- You expect your varistor to behave linearly in all regions

The voltage-current relationship for the linear varistor is:

$$i_{varistor} = \begin{cases} \frac{v_{varistor}}{R_{off}} & |v_{varistor}| < v_{clamp} \\ \frac{v_{varistor}}{R_{on}} + c_1 \operatorname{sgn}(v_{varistor}) & |v_{varistor}| \geq v_{clamp} \end{cases}.$$

where:

- $v_{varistor}$  and  $i_{varistor}$  are the varistor voltage and current, respectively.
- $v_{clamp}$  is the threshold voltage that separates the two regions of operation. Set this value using the **Clamping voltage** parameter.
- $R_{on}$  and  $R_{off}$  are the resistances in the on- and off-regions. Set these values using the **On resistance** and **Off resistance**, respectively.
- $c_1$  is a constant used to enforce current continuity between the two regions:

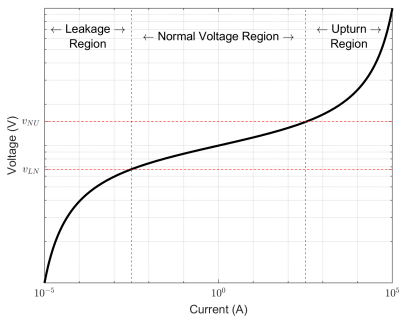
$$c_1 = v_{clamp} \left( \frac{1}{R_{off}} - \frac{1}{R_{on}} \right).$$

## Power-Law Parameterization

This parameterization option separates the voltage-current relationship into three regions:

- Leakage region — Resistance is high and current increases slowly with increasing voltage.
- Normal region — Resistance decreases exponentially with increasing voltage.
- Upturn region — Resistance is low and current increases rapidly with increasing voltage.

This figure shows the three regions of operation in log-log-scale.



Use power-law parameterization in one of these scenarios:

- You are modeling voltage surges across a large range of voltages
- You expect your varistor to behave exponentially in the first on-region

The voltage-current relationship for the power-law varistor is:

$$i_{\text{varistor}} = \begin{cases} \frac{v_{\text{varistor}}}{R_L} & |v_{\text{varistor}}| < v_{LN} \\ k(v_{\text{varistor}})^\alpha + c_1 & v_{LN} \leq |v_{\text{varistor}}| \leq v_{NU} \\ \frac{v_{\text{varistor}}}{R_U} + c_2 & |v_{\text{varistor}}| > v_{NU} \end{cases}$$

where:

- $v_{\text{varistor}}$  and  $i_{\text{varistor}}$  are the varistor voltage and current, respectively.
- $\alpha$  is the power-law exponent which determines the rate of current increase with voltage increase in the normal region. Set this value using the **Normal-mode power-law exponent** parameter.
- $v_{LN}$  and the  $v_{NU}$  are the threshold voltages corresponding to the leakage-normal and normal-upturn transition points. Set these values using the **Leakage to normal voltage transition** and **Normal to upturn voltage transition** parameters, respectively.
- $R_L$  and  $R_U$  are the resistances in the leakage- and upturn-regions. Set these values using the **Leakage-mode resistance** and **Upturn-mode resistance** parameters, respectively.
- $k$ ,  $c_1$ , and  $c_2$  are constants used to enforce current continuity between the regions:

$$k = \frac{1}{\alpha R_U v_{NU}^{\alpha-1}},$$

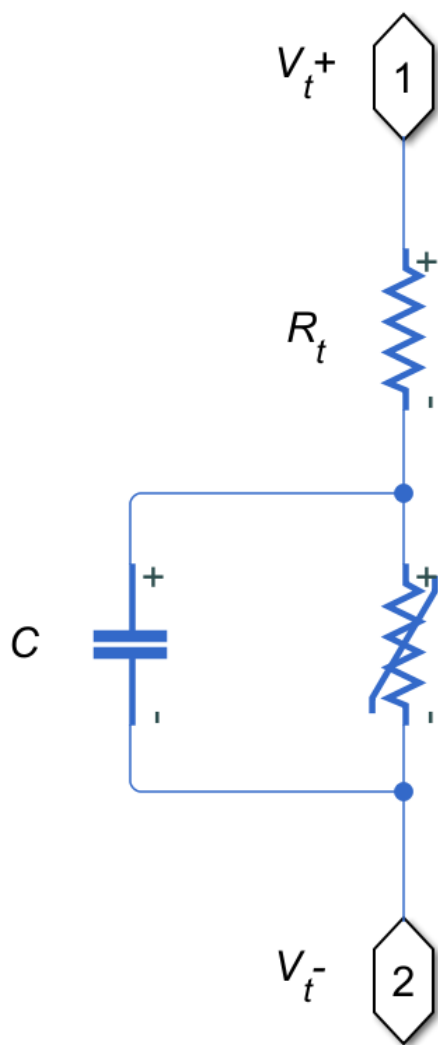
$$c_1 = \frac{v_{LN}}{R_L} - \frac{v_{LN}^\alpha}{\alpha R_U v_{NU}^{\alpha-1}},$$

and

$$c_2 = \frac{1}{\alpha R_U v_{NU}^{\alpha-1}} (v_{NU}^\alpha - v_{LN}^\alpha) - \frac{v_{NU}}{R_U} + \frac{v_{LN}}{R_L}.$$

## Equivalent Circuit

In addition to the varistor equations, you can also specify a constant terminal resistance  $R_t$  and device capacitance  $C$ . This figure shows the equivalent circuit for the varistor in either parameterization mode.





## Ports

### Conserving

#### + — Positive terminal

electrical

Electrical conserving port associated with the varistor positive terminal.

#### - — Negative terminal

electrical

Electrical conserving port associated with the varistor negative terminal.

## Parameters

### Parameterization — Varistor operation mode

Linear (default) | Power-law

Choose how the varistor resistance changes with increasing voltages:

- **Linear** — Two regions. The low-voltage region has high resistance and the high-voltage region has low resistance.
- **Power-law** — Three regions. The leakage region has high resistance. The normal region has exponentially decreasing resistance. The upturn region has low resistance.

### Clamping voltage — Threshold voltage

260 (default) | positive number

Transition point voltage,  $v_{clamp}$ , between the off- and on-states of the linear varistor.

### Dependencies

Enabled when the **Parameterization** parameter is set to **Linear**.

### Off resistance — Low-voltage resistance

3e8 (default) | positive number

Low-voltage resistance,  $R_{off}$ , of the varistor in the off-state.

**Dependencies**

Enabled when the **Parameterization** parameter is set to Linear.

**On resistance — High-voltage resistance**

1 (default) | positive number

High-voltage resistance,  $R_{on}$ , of the varistor in the on-state.

**Dependencies**

Enabled when the **Parameterization** parameter is set to Linear.

**Leakage to normal voltage transition — First threshold voltage**

130 (default) | positive number

Transition point voltage,  $v_{LN}$ , between the leakage and normal regions of the power-law varistor.

**Dependencies**

Enabled when the **Parameterization** parameter is set to Power-law.

**Normal to upturn voltage transition — Second threshold voltage**

300 (default) | positive number

Transition point voltage,  $v_{NU}$ , between the normal and upturn regions of the power-law varistor.

**Dependencies**

Enabled when the **Parameterization** parameter is set to Power-law.

**Leakage-mode resistance — Low-voltage resistance**

3e8 (default) | positive number

Low-voltage resistance,  $R_L$ , of the varistor in the leakage region.

**Dependencies**

Enabled when the **Parameterization** parameter is set to Power-law.

**Normal-mode power-law exponent — Mid-voltage resistance**

45 (default) | positive number

Exponent that determines rate of current increase with voltage increase of the varistor in the normal region.

**Dependencies**

Enabled when the **Parameterization** parameter is set to Power - Law.

**Upturn-mode resistance — High-voltage resistance**

0.07 (default) | positive number

High-voltage resistance,  $R_U$ , of the varistor in the upturn region.

**Dependencies**

Enabled when the **Parameterization** parameter is set to Power - Law.

**Terminal resistance — Terminal resistance**

100e-6 (default) | non-negative number

Small, constant resistance in series with the varistor. Set this value to zero to remove the resistance from the equivalent circuit.

**Capacitance — Parallel capacitance**

4.4 (default) | non-negative number

Capacitor in parallel with the varistor. Set this value to zero to remove the capacitor from the equivalent circuit.

## See Also

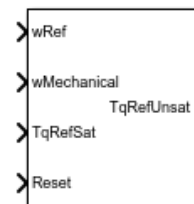
Resistor | Variable Resistor

**Introduced in R2018a**

## Velocity Controller

Discrete-time velocity controller

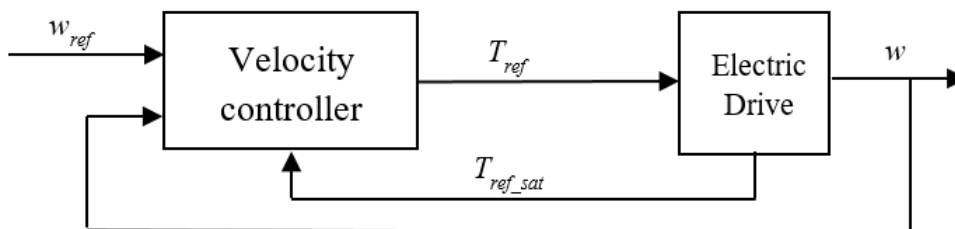
**Library:**       Simscape / Electrical / Power Systems / Control /  
                      General Machine Control



## Description

The Velocity Controller block implements a velocity controller in discrete-time.

You provide measured and reference rotor velocities ( $w$  and  $w_{ref}$ ) as inputs to the block. The block then outputs a reference torque  $T_{ref}$  for an electric drive.



To prevent windup in the integrator, feed the saturated reference torque  $T_{ref\_sat}$  from the electric drive back to the velocity controller.

## Equations

You can control the rotor angular velocity with discrete sample time  $T_s$  using one of three common approaches:

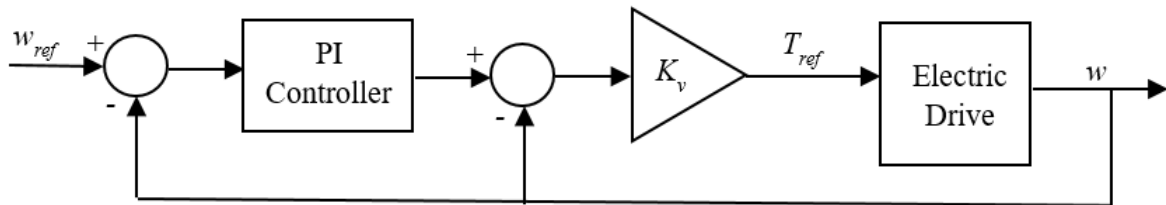
- Proportional-integral (PI) control, with proportional and integral gains  $K_{p\_w}$  and  $K_{i\_w}$ :

$$T_{ref} = \left( K_{p\_w} + K_{i\_w} \frac{T_s z}{z-1} \right) (w_{ref} - w)$$

- Proportional (P) control, with proportional gain  $K_{p\_w}$ :

$$T_{ref} = K_{p\_w} (w_{ref} - w)$$

- P-PI control characterized by a double velocity feedback loop as shown in the following figure:



Here, the PI Controller block is structured as in the PI control strategy, and  $K_v$  is the proportional gain for a P controller.

Using PI control results in a zero in the closed-loop transfer function, which can result in undesired overshoot in the closed-loop response. This zero can be canceled by introducing a zero-cancellation block in the feedforward path. The zero cancellation transfer function in discrete time is

$$G_{ZC\_w}(z) = \frac{\frac{T_s K_{i\_w}}{K_{p\_w}}}{z + \left( \frac{T_s - \frac{K_{p\_w}}{K_{i\_w}}}{\frac{K_{p\_w}}{K_{i\_w}}} \right)}$$

## Ports

### Input

#### **wRef — Desired velocity**

scalar

Desired or reference velocity, in rad/s.

Data Types: `single` | `double`

#### **wMechanical — Actual velocity**

scalar

Measured mechanical velocity, in rad/s.

Data Types: `single` | `double`

#### **TqRefSat — Saturated reference torque**

scalar

Saturated torque reference used for integral anti-windup gain, in N\*m.

Data Types: `single` | `double`

#### **Reset — Integral reset**

scalar

External reset signal (rising edge) for the integrator.

Data Types: `single` | `double`

### Output

#### **TqRefUnsat — Unsaturated desired torque**

scalar

Unsaturated reference torque, in N\*m.

Data Types: `single` | `double`

## Parameters

### Control type — Control model

PI control (default) | P control | P-PI control

Type of controller:

- `PI control` — Proportional-integral control using a single feedback loop
- `P control` — Proportional-integral control using a single feedback loop
- `P-PI control` — Proportional and proportional-integral control using a double feedback loop

### Dependencies

The **Control type** options affect the visibility or configurability of these parameters:

- **Controller integral gain**
- **P controller proportional gain**
- **Anti-windup gain**
- **Integral anti-windup gain**
- **Sample time (-1 for inherited)**
- **Enable zero cancellation**

### Controller proportional gain — Proportional gain

1 (default) | positive scalar

Proportional gain for the:

- PI controller
- P controller in the single-loop control model
- PI controller in the P-PI controller

### Controller integral gain — Integral gain

1 (default) | positive scalar

Integral gain for the PI or P-PI controller.

## Dependencies

This parameter is visible only when the **Control type** is set to `PI control` or `P-PI control`.

## **P controller proportional gain — P proportional gain**

1 (default) | positive scalar

Proportional gain for the P controller in the P-PI controller.

## Dependencies

This parameter is visible only when the **Control type** is set to `P-PI control`.

## **Integral anti-windup gain — PI anti-windup gain**

1 (default) | positive scalar

Anti-windup gain for the PI controller.

## Dependencies

This parameter is visible only when the **Control type** is set to `PI control` or `P-PI control`.

## **Sample time (-1 for inherited) — Block sample time**

-1 (default) | positive scalar

Time, in s, between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

If this block is inside a triggered subsystem, inherit the sample time by setting this parameter to -1. If this block is in a continuous variable-step model, specify the sample time explicitly using a positive scalar.

## Dependencies

This parameter is visible only when the **Control type** is set to `PI control` or `P-PI control`.

## **Discretization sample time — Sample time for discretization**

0.001 (default) | positive scalar



Time, in s, between consecutive discretizations. Discretization is required for zero cancellation.

### Dependencies

This parameter is only visible when all these conditions are met:

- **Control type** is set to `PI control` or `P-PI control`.
- **Sample time** is set to `-1`.
- **Enable zero cancellation** is selected .

### Enable zero cancellation — Feedforward zero-cancellation

off (default) | on

Option to use zero cancellation on the feedforward path.

### Dependencies

The **Enable zero cancellation** parameter is visible only when **Control type** is set to `PI control` or `P-PI control`.

The **Discretization sample time** parameter is only visible when **Enable zero cancellation** is selected .

## References

- [1] Naouar, M. W., A. A. Naassani, E. Monmasson, and I. Slama-Belkhdja. "FPGA-based predictive current controller for synchronous machine speed drive." *IEEE Transactions on Power Electronics*. Vol. 23, Number 4, 2008, pp. 2115-2126.

## See Also

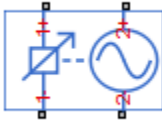
### Blocks

SM Current Controller | SM Current Reference Generator

**Introduced in R2017b**

## Voltage-Controlled Oscillator

Behavioral model of voltage-controlled oscillator



### Library

Integrated Circuits

### Description

The Voltage-Controlled Oscillator block provides a behavioral model of a voltage-controlled oscillator (VCO). The output voltage is defined by the following equations:

$$v_{\text{lim}} = \begin{cases} v_{\text{min}} & \text{for } v_{\text{in}} < v_{m\text{in}} \\ v_{\text{in}} & \text{for } v_{m\text{in}} \leq v_{\text{in}} \leq v_{\text{max}} \\ v_{\text{max}} & \text{for } v_{\text{in}} > v_{\text{max}} \end{cases}$$

$$\dot{\Phi} = 2\pi F(v_{\text{lim}})$$

$$v_{\text{out}} = A \sin(2\pi f_{\text{nom}} t + \Phi) - i_{\text{out}} R_{\text{out}}$$

where:

- $v_{\text{in}}$  is the voltage applied across the 1+ and 1- ports.
- $v_{\text{out}}$  is the voltage across the 2+ and 2- ports.
- $f_{\text{nom}}$  is the oscillator frequency when the input control voltage is  $v_{\text{nom}}$ .

- $F$  is a linear function of  $v_{lim}$  or a lookup table function of  $v_{lim}$ .
- $A$  is the output voltage peak amplitude.
- $t$  is simulation time.
- $i_{out}$  is the output current.
- $R_{out}$  is the output resistance.

If you choose **Linear** for the **Frequency dependence on input voltage** parameter, then the function  $F$  is given by:

$$F = f_{nom} + k(v_{lim} - v_{nom})$$

where  $k$  is the rate of change of frequency with input voltage.

If you choose **Tabulated** for the **Frequency dependence on input voltage** parameter, then the function  $F$  is defined by the vectors of input voltages and corresponding output frequency deviations from nominal that you supply. The values for  $v_{min}$  and  $v_{max}$  are the first and the last values of the input voltage vector.

You can model the time delay between a change in the input control voltage and the oscillator frequency. Do this by modeling a first-order dynamic between  $v_{lim}$  and the value passed to the function  $F$ .

## Parameters

- “Frequency Tab” on page 1-2061
- “Electrical Characteristics Tab” on page 1-2063
- “Dynamics Tab” on page 1-2063

## Frequency Tab

### Frequency dependence on input voltage

Select one of the following methods for block parameterization:

- **Linear** — Define a linear function by specifying the rate of change of frequency with input voltage. This is the default option.

- **Tabulated** — Provide the vectors of input voltages and corresponding output frequency deviations from nominal. The block determines the frequency deviation by table lookup based on these values.

## **Nominal frequency**

The oscillator frequency when the input control voltage is at the nominal value. The default value is 1000 Hz.

## **Input voltage corresponding to nominal frequency**

The input voltage corresponding to the oscillator nominal frequency. This parameter is visible only if you select **Linear** for the **Frequency dependence on input voltage** parameter. The default value is 0.5 V.

## **Rate of change of frequency with input voltage**

The linear coefficient defining the rate of change of frequency depending on input voltage. This parameter is visible only if you select **Linear** for the **Frequency dependence on input voltage** parameter. The default value is 1 Hz/V.

## **Minimum input voltage**

The minimum input voltage that affects VCO frequency. This parameter is visible only if you select **Linear** for the **Frequency dependence on input voltage** parameter. The default value is 0 V.

## **Maximum input voltage**

The maximum input voltage that affects VCO frequency. This parameter is visible only if you select **Linear** for the **Frequency dependence on input voltage** parameter. The default value is 1 V.

## **Input voltage vector**

The vector of voltages for the tabulated VCO frequency. This parameter is visible only if you select **Tabulated** for the **Frequency dependence on input voltage** parameter. The default value is [0 0.2 0.4 0.6 0.8 1] V.

## **Frequency deviation from nominal**

The corresponding vector of VCO frequencies relative to the nominal frequency. This parameter is visible only if you select **Tabulated** for the **Frequency dependence on input voltage** parameter. The default value is [-1000 -329 -51 162 342 500] Hz.

## Electrical Characteristics Tab

### Output voltage peak amplitude

The peak amplitude of the voltage across the 2+ and 2- terminals. The default value is 1 V.

### Input resistance

The resistance seen at the 1+ and 1- terminals. The default value is Inf Ohm.

### Output resistance

The value of the series output resistance. The default value is 0 Ohm.

## Dynamics Tab

### Dynamics

Select one of the following methods for specifying dynamics:

- **No dynamics** — Do not model the time delay between a change in the input control voltage and the oscillator frequency. This is the default option.
- **Model frequency tracking dynamics** — Model a first order dynamic between the input control voltage and the oscillator frequency.

### Frequency tracking time constant

Time constant for the first-order filter that delays the measured input control voltage, to model the lag between a change in VCO demanded frequency and the resulting VCO frequency. This parameter is visible only if you select **Model frequency tracking dynamics** for the **Dynamics** parameter. The default value is 0.001 s.

### Initial frequency

The initial VCO output frequency. This parameter is visible only if you select **Model frequency tracking dynamics** for the **Dynamics** parameter. The default value is 1000 Hz.

## Ports

The block has the following ports:

1+

Positive input voltage

1-

Negative input voltage

2+

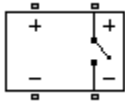
Positive output voltage

2-

Negative output voltage

# Voltage-Controlled Switch

Model voltage-controlled switch with hysteresis



## Library

SPICE-Compatible Components/Passive Devices

## Description

The Voltage-Controlled Switch block represents the electrical characteristics of a switch whose state is controlled by the voltage across the input ports (the controlling voltage):

- When the controlling voltage is greater than the sum of the **Threshold voltage, VT** and **Hysteresis voltage, VH** parameter values, the switch is closed and has a resistance equal to the **On resistance, RON** parameter value.
- When the controlling voltage is less than the **Threshold voltage, VT** parameter value minus the **Hysteresis voltage, VH** parameter value, the switch is open and has a resistance equal to the **Off resistance, ROFF** parameter value.
- When the controlling voltage is greater than or less than the **Threshold voltage, VT** parameter value by an amount less than or equal to the **Hysteresis voltage, VH** parameter value, the voltage is in the crossover region and the state of the switch remains unchanged.

## Basic Assumptions and Limitations

The block output resistance model is discontinuous during switching. The discontinuity might cause numerical issues. Try the following actions to resolve the issues:

- Set the **On resistance, RON** and **Off resistance, ROFF** parameter values to keep the ratio  $RON/ROFF$  as small as possible, and less than  $1e+12$ .

- Increase the **Hysteresis voltage, VH** parameter value to reduce switch chatter.
- Decrease the **Max step size** parameter value (in the Configuration Parameters block dialog box).

---

**Note** This increases the simulation time.

---

## Parameters

### Threshold voltage, VT

The voltage above which the block interprets the controlling voltage as HIGH. The default value is 0 V.

---

**Note** The controlling voltage must differ from the threshold voltage by at least the **Hysteresis voltage, VH** parameter value to change the state of the switch.

---

### Hysteresis voltage, VH

The amount by which the controlling voltage must exceed or fall below the **Threshold voltage, VT** parameter value to change the state of the switch. The default value is 0 V.

### On resistance, RON

The resistance of the switch when it is closed. The default value is 1  $\Omega$ .

### Off resistance, ROFF

The resistance of the switch when it is open. The default value is 1e+12  $\Omega$ .

### Initial switch state

Select one of the following options for the state of the switch at the start of the simulation:

- On — The switch is initially closed and its resistance value is equal to the **On resistance, RON** parameter value. This is the default option.
- Off — The switch is initially open and its resistance value is equal to the **Off resistance, ROFF** parameter value.



## Ports

The block has the following ports:

+

Positive electrical input and output ports.

-

Negative electrical input and output ports.

## Voltage Source

Simulate voltage source with DC, AC, and noise components



## Library

Sources

## Description

The Voltage Source block implements a voltage source with DC, AC, and noise components. The voltage across the + and - terminals is given by:

$$v = v_{DC} + v_{AC} \sin(2\pi ft + \phi) + v_N$$

where:

- $v_{DC}$  is the steady-state DC voltage component.
- $v_{AC}$  is the amplitude of the AC voltage component.
- $f$  is the frequency of the AC component.
- $\phi$  is the phase offset of the AC component.
- $v_N$  is the noise voltage.

You can configure your source as DC-only, AC-only, or a combination of both. By default, both AC and DC components are set to 0. Define the AC/DC voltage by specifying nonzero parameter values after placing the block in your model.

The noise component is also optional. If you set the **Noise mode** parameter to Enabled, then the added noise voltage is given by:

$$v_N = \sqrt{P_v/2} \frac{N(0,1)}{\sqrt{h}}$$

where:

- $P_v$  is the single-sided noise power spectral density for a 1 ohm load, in  $V^2/Hz$ .
- $N$  is a Gaussian random number with zero mean and standard deviation of one.
- $h$  is the sampling interval.

By default, the **Noise mode** parameter is set to **Disabled**, and the voltage source generates no thermal noise.

## Noise Options

The block generates Gaussian noise by using the Random Number source in the Simscape Foundation library. You can control the random number seed by setting the **Repeatability** parameter:

- **Not repeatable** — Every time you simulate your model, the block resets the random seed using the MATLAB random number generator:

```
seed = randi(2^32-1);
```

- **Repeatable** — The block automatically generates a seed value and stores it inside the block, to always start the simulation with the same random number. This auto-generated seed value is set when you add a Voltage Source block from the block library to the model. When you make a new copy of the Voltage Source block from an existing one in a model, a new seed value is generated. The block sets the value using the MATLAB random number generator command shown above.
- **Specify seed** — If you select this option, the additional **Seed** parameter lets you directly specify the random number seed value.

## Basic Assumptions and Limitations

Simulating with noise enabled slows down simulation. Choose the sample time ( $h$ ) so that noise is generated only at frequencies of interest, and not higher.

## Parameters

- “DC & AC Components Tab” on page 1-2070
- “Noise Tab” on page 1-2070

### DC & AC Components Tab

#### DC voltage

The DC component of the output voltage. The default value is 0 V. Enter a nonzero value to add a DC component to the voltage source.

#### AC voltage peak amplitude

Amplitude of the AC component of the output voltage. The default value is 0 V. Enter a nonzero value to add an AC component to the voltage source.

#### AC voltage phase shift

Phase offset of the AC component of the output voltage. The default value is 0 degrees.

#### AC voltage frequency

Frequency of the AC component of the output voltage. The default value is 60 Hz.

### Noise Tab

#### Noise mode

Select the noise option:

- **Disabled** — No noise is produced by the voltage source. This is the default.
- **Enabled** — The voltage source generates thermal noise, and the associated parameters become visible on the **Noise** tab.

#### Power spectral density

The single-sided spectrum noise power. Strictly-speaking, this is a density function for the square of the voltage, commonly thought of as a power into a 1 ohm load, and therefore the units are  $V^2/\text{Hz}$ . To avoid this unit ambiguity, some datasheets quote noise voltage as a noise density with units of  $V/\sqrt{\text{Hz}}$ . In this case, you should enter the square of the noise density quoted in the datasheet as the parameter value. The default value is 0  $V^2/\text{Hz}$ .

### Sample time

Defines the rate at which the noise source is sampled. Choose it to reflect the frequencies of interest in your model. Making the sample time too small will unnecessarily slow down your simulation. The default value is  $1e-3$  s.

### Repeatability

Select the noise control option:

- **Not repeatable** — The random sequence used for noise generation is not repeatable. This is the default.
- **Repeatable** — The random sequence used for noise generation is repeatable, with a system-generated seed.
- **Specify seed** — The random sequence used for noise generation is repeatable, and you control the seed by using the **Seed** parameter.

### Auto-generated seed used for repeatable option

Random number seed stored inside the block to make the random sequence repeatable. This parameter is visible only if you select **Repeatable** for the **Repeatability** parameter. The parameter value is automatically generated using the MATLAB random number generator command. You can modify this parameter value, but it gets overwritten by a new random value if you copy the block to another block in the model. Therefore, if you want to control the seed of the random sequence, use the **Specify seed** option for the **Repeatability** parameter and specify the desired seed value using the **Seed** parameter.

### Seed

Random number seed used by the noise random number generator. This parameter is visible only if you select **Specify seed** for the **Repeatability** parameter. The default value is 0.

## Ports

The block has the following ports:

+

Positive electrical port

-

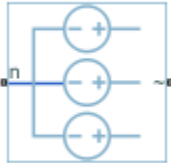
Negative electrical port

## **See Also**

Current Source | Resistor

# Voltage Source (Three-Phase)

Three-phase voltage source



## Library

Simscape / Electrical / Power Systems / Sources

## Description

The Voltage Source (Three-Phase) block models an ideal three-phase voltage source or a three-phase voltage source with harmonics. You specify the configuration using the Source representation parameter.

When you select **None** for the **Source Impedance** parameter, the Voltage Source block models an ideal three-phase voltage source that maintains sinusoidal voltage of the specified magnitude across its terminals, independently of the current flowing through the source.

The source has a wye configuration, and port **n** provides a connection to the center of the wye. Port **~** is an expandable three-phase port representing the three phases, *a*, *b*, and *c*. The current is positive if it flows from positive to the center of the wye, and the voltage across each phase is equal to the difference between the voltage at the positive terminal and the center of the wye,  $V(+)$  -  $V_n$ .

## Equations

The output voltage for the Voltage Source block is defined by these equations:

$$v = v_{DC} + v_{AC} \sin(2\pi ft + \phi) + v_N$$

$$v_a = V_0 \sin(2\pi ft + \phi)$$

$$v_N = \sqrt{P_v/2} \frac{N(0,1)}{\sqrt{h}}$$

$$v_c = V_0 \sin(2\pi ft + \phi + 120^\circ),$$

where:

- $V_0$  is the peak phase voltage.
- $v_{\text{line\_rms}}$  is the root-mean square (RMS) phase-to-phase voltage.
- $v_a, v_b, v_c$  are the respective phase voltages.
- $f$  is frequency.
- $\phi$  is the phase shift.
- $t$  is time.

When you specify the three-phase voltage source with harmonics representation, the output voltage for the Voltage Source block is defined by these equations:

$$V_0 = \frac{\sqrt{2}}{\sqrt{3}} v_{\text{line\_rms}} H_{\text{ratios}}$$

$$v_a = V_0 \sin((2\pi ft + \phi) H'_{\text{orders}})$$

$$v_b = V_0 \sin((2\pi ft + \phi - \theta) H'_{\text{orders}})$$

$$v_c = V_0 \sin((2\pi ft + \phi + \theta) H'_{\text{orders}}),$$

where:

- $V_0$  is a row-vector containing the peak voltage of the fundamental and harmonic sinusoids.



- $v_{line\_rms}$  is the RMS phase-to-phase voltage.
- $H_{ratios}$  is a row-vector of harmonic ratios. The first element is 1 to represent the fundamental.
- $H_{orders}$  is a row-vector of harmonic orders. The first element is 1 to represent the fundamental.
- $v_a, v_b, v_c$  are the respective phase voltages.
- $f$  is a column-vector of harmonic frequencies. The first element is the fundamental frequency.
- $\phi$  is a column-vector of harmonic phase shifts. The first element is the fundamental phase shift.
- $\theta$  is a column-vector of harmonic phase offsets. The first element is  $120^\circ$ .
- $t$  is the time.

When you select X/R ratio for the **Source Impedance** parameter, the equations for source impedance are:

$$R = \frac{v_{line\_rms}^2}{S_{sc}\sqrt{1+\phi^2}}$$

$$X = R\phi$$

$$L = \frac{X}{2\pi f},$$

where:

- $S_{sc}$  is the **Short-circuit power level** that you specify.
- $\phi$  is the **Source X/R ratio** that you specify.
- $R$  is the calculated source resistance.
- $X$  is the calculated source reactance.
- $L$  is the calculated source inductance.

## Parameters

### Main

#### Voltage (phase-to-phase RMS)

RMS phase-to-phase, or line, voltage. The default value is  $\sqrt{3} \cdot 100 / \sqrt{2}$ , or 122.4745, V.

#### Phase shift

Phase shift in angular units. The default value is 0 deg.

#### Frequency

Voltage frequency, specified in Hz or units directly convertible to Hz (where Hz is defined as 1/s). For example, kHz and MHz are valid units, but rad/s is not. The default value is 60 Hz.

#### Source Impedance

Choose a method for specifying source impedance. The default option is X/R Ratio. Selecting any other options enables other parameters. The options are:

- None
- X/R Ratio
- Series R
- Series L
- Series RL

#### Short-circuit power level

Selecting X/R Ratio for the **Source Impedance** parameter enables this parameter. The default value is  $1e6$  V\*A.

#### Source X/R ratio

Complex impedance, that is, the reactance-to-resistance ratio. Selecting X/R Ratio for the **Source Impedance** parameter enables this parameter. The default value is 15.

#### Source Resistance

Selecting Series R or Series RL for the **Source Impedance** parameter enables this parameter. The default value is 0.01 Ohm.

### Source Inductance

Selecting **Series L** or **Series RL** for the **Source Impedance** parameter enables this parameter. The default value is  $3.97e-4$  H.

## Harmonics

### Source Representation

Choose between **None** and **Generate** harmonics. The default value is **None**.

### Harmonic orders

A row-vector of additional integer harmonic orders at which harmonics are to be generated. This parameter is only visible when you set the **Source representation** parameter to **Generate** harmonics. The default value is [5, 7, 11, 13].

### Harmonic magnitude to peak magnitude ratios

A row-vector of ratios of harmonic magnitudes relative to the fundamental magnitude. This parameter is only visible when you set the **Source representation** parameter to **Generate** harmonics. The default value is [0.1, 0.1, 0.1, 0.1].

## Parasitics

### Source impedance parasitic parallel conductance

Selecting **X/R Ratio**, **Series L**, or **Series RL** for the **Source Impedance** parameter enables this parameter. The default value is  $0.1/0\text{hm}$ .

## Variables

Use the **Variables** tab to set the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** tab lists all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Ports

The block has the following ports:

~

Expandable three-phase port

n

Electrical conserving port associated with the center of the wye

## See Also

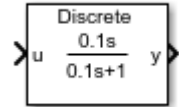
Battery | Controlled Current Source (Three-Phase) | Current Source (Three-Phase) | Supercapacitor

**Introduced in R2013b**

# Washout (Discrete or Continuous)

Discrete-time or continuous-time washout or high-pass filter

**Library:** Simscape / Electrical / Power Systems / Control / General Control



## Description

The Washout (Discrete or Continuous) block implements a washout filter in conformance with IEEE 421.5-2016<sup>[1]</sup>. The washout is also known as a high-pass filter.

You can switch between continuous and discrete implementations of the integrator using the **Sample time** parameter.

## Equations

To configure the washout block for continuous time, set the **Sample time** property to 0. This representation is equivalent to the continuous transfer function:

$$G(s) = \frac{T_s}{T_s + 1},$$

where  $T$  is the time constant. From the preceding transfer function, the washout defining equations are:

$$\begin{cases} \dot{x}(t) = \frac{1}{T}(-x(t) + u(t)) \\ y(t) = -x(t) + u(t) \end{cases} \quad x(0) = u_0, \quad y(0) = 0,$$

where:

- $u$  is the washout input.
- $x$  is the washout state.
- $y$  is the washout output.
- $t$  is the simulation time.
- $u_0$  is the initial input to the washout block.

To configure the washout block for discrete time, set the **Sample time** property to a positive, nonzero value, or to -1 to inherit the sample time from an upstream block. The discrete representation is equivalent to the transfer function:

$$G(z) = \frac{z-1}{z+T_s/T-1},$$

where  $T_s$  is the sample time. From the discrete transfer function, the washout defining equations are defined using the forward Euler method:

$$\begin{cases} x(n+1) = \left(1 - \frac{T_s}{T}\right)x(n) + \left(\frac{T_s}{T}\right)u(n) \\ y(n) = u(n) - x(n) \end{cases} \quad x(0) = u_0, y(0) = 0,$$

where:

- $u$  is the washout input.
- $x$  is the washout state.
- $y$  is the washout output.
- $n$  is the simulation time step.
- $u_0$  is the initial input to the washout block.

## Initial Conditions

The block sets the state initial condition to the initial input, making the initial output zero.

## Bypass Filter Dynamics

Set the time constant to a value smaller than or equal to the sample time to ignore the dynamics of the filter. When bypassed, the block feeds the input directly to the output:

$$T \leq T_s \rightarrow y = u.$$

In the continuous case, the sample time and time constant must both be zero.

## Ports

### Input

#### **u — Washout input**

vector

Washout input signal. The block uses the input initial value to determine the state initial value.

Data Types: single | double

### Output

#### **y — Washout output**

vector

Washout output signal.

Data Types: single | double

## Parameters

#### **Time constant — Washout time constant**

0.1 (default) | positive number

Washout time constant. Set this value less than the **Sample time** to bypass the dynamics of the filter.

#### **Sample time (-1 for inherited) — Block sample time**

-1 (default) | 0 | positive scalar

Time between consecutive block executions. During execution, the block produces outputs and, if appropriate, updates its internal state. For more information, see “What Is Sample Time?” (Simulink) and “Specify Sample Time” (Simulink).

For inherited discrete-time operation, specify -1. For discrete-time operation, specify a positive integer. For continuous-time operation, specify 0.

If this block is in a masked subsystem, or other variant subsystem that allows you to switch between continuous operation and discrete operation, promote the sample time parameter. Promoting the sample time parameter ensures correct switching between the continuous and discrete implementations of the block. For more information, see “Promote Parameter to Mask” (Simulink).

## References

[1] *IEEE Recommended Practice for Excitation System Models for Power System Stability Studies*. IEEE Std 421.5-2016. Piscataway, NJ: IEEE-SA, 2016.

## See Also

### Blocks

Filtered Derivative (Discrete or Continuous) | Integrator (Discrete or Continuous) | Integrator with Wrapped State (Discrete or Continuous) | Lead-Lag (Discrete or Continuous) | Low-Pass Filter (Discrete or Continuous)

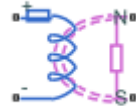
### Introduced in R2017b



# Winding

Electromagnetic converter with ohmic and magnetic flux losses

**Library:** Simscape / Electrical / Power Systems / Passive  
Devices / Transformers / Fundamental Components



## Description

The Winding block represents an electromagnetic converter with winding resistance and leakage reluctance. You can use this block as a base component for building custom transformers. For an ideal electromagnetic converter, see the Electromagnetic Converter.

When you apply a positive current across the electrical ports of the block, a positive magnetomotive force (MMF) is induced across the magnetic terminals.

$$\mathcal{F} = Ni$$

Where:

- $\mathcal{F}$  is the MMF across the magnetic terminals of the block
- $N$  is the number of winding turns
- $i$  is the current through the winding

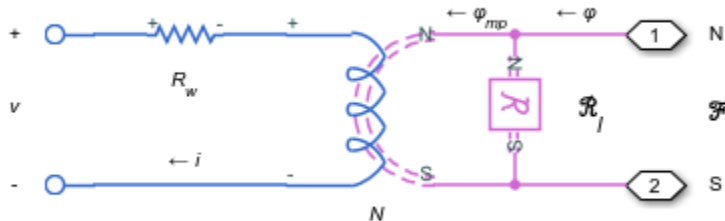
When you apply a positive time-varying flux across the magnetic terminals of the block, a negative voltage is induced across the electrical terminals of the block.

$$v = -N \frac{d\phi}{dt} + \frac{N^2}{R_l} \frac{di}{dt} + R_w i$$

Where:

- $\varphi$  is the flux through the magnetic terminals of the block
- $i$  is the current through the electrical terminals of the block
- $\mathcal{R}_l$  is the leakage reluctance
- $R_w$  is the winding resistance
- $\mathcal{F}$  is the magnetomotive force across the magnetic terminals of the block
- $v$  is the voltage drop across the electrical terminals of the block

This figure shows the equivalent circuit for the block.



In the diagram,  $\varphi_{mp}$  corresponds to the main-path flux, or the flux through the main winding. You can set the initial condition for this flux in the block's **Variables** tab.

## Ports

### Conserving

**+ — Positive terminal**  
electrical

Electrical conserving port associated with the positive terminal of the block.

**- — Negative terminal**  
electrical

Electrical conserving port associated with the negative terminal of the block.

**N — North terminal**

magnetic

Magnetic conserving port associated with the north terminal of the block.

**S — South terminal**

magnetic

Magnetic conserving port associated with the south terminal of the block.

## Parameters

**Number of winding turns — Winding turn count**

10 (default) | positive number

Number of wire turns on the transformer winding.

**Winding resistance — Series resistance**

1e-3 Ohm (default) | positive number

Power loss in the winding.

**Leakage reluctance — Parallel reluctance**

1e5 1/H (default) | positive number

Magnetic flux loss in the winding.

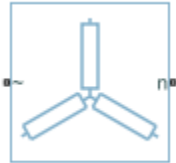
## See Also

Delta-Delta Transformer | Delta1-Delta1-Wye Transformer | Delta11-Delta11-Wye Transformer | Wye-Delta1 Transformer | Wye-Delta1-Wye Transformer | Wye-Delta11 Transformer | Wye-Delta11-Wye Transformer | Wye-Wye Transformer | Zigzag-Delta1-Wye Transformer | Zigzag-Delta11-Wye Transformer

**Introduced in R2018a**

## Wye-Connected Load

Three-phase load wired in wye configuration



## Library

Simscape / Electrical / Power Systems / Passive Devices

## Description

The Wye-Connected Load block models a three-phase load wired in a wye configuration. Each limb of the load can include any combination of a resistor (R), capacitor (C), and inductor (L), connected in series or in parallel.

You can specify values for the R, L, and C components directly in terms of resistance, inductance, and capacitance, or by rated powers at a rated voltage and frequency.

- If you parameterize the block directly in terms of R, L, and C values, then for initialization, provide a three-element row vector of initial voltages for a capacitor, and a three-element row vector of initial currents for an inductor.
- If you parameterize the block in terms of rated powers, then specify initial conditions in terms of an initial voltage, initial voltage phase, and initial frequency. For example, if the load is connected directly to a three-phase voltage source, then the initial conditions are identical to the source values for RMS line voltage, frequency, and phase shift. To specify zero initial-voltage magnitude, set the initial voltage to 0.

For certain combinations of R, L, and C, for some circuit topologies, specify parasitic resistance or conductance values that help the simulation to converge numerically. These parasitic terms ensure that an inductor has a small parallel resistive path and that a

capacitor has a small series resistance. When you parameterize the block in terms of rated powers, the rated power values do not account for these small parasitic terms. The rated powers represent only the R, L, and C values of the load itself.

## Parameters

- “Main Tab” on page 1-2087
- “Parasitics Tab” on page 1-2088
- “Initial Conditions Tab” on page 1-2089

### Main Tab

#### Parameterization

Select one of these values:

- **Specify by rated power** — Specify values for the R, L, and C components by rated powers at a rated voltage and frequency. This is the default.
- **Specify component values directly** — Specify values for the R, L, and C components directly in terms of resistance, inductance, and capacitance.

Switching the **Parameterization** value resets the **Component structure** value. Select the component parameterization option first, and then the component structure. If you later switch the **Parameterization** value, check the **Component structure** value and reselect it, if necessary.

#### Component structure

Select the desired combination of a resistor (R), capacitor (C), and inductor (L), connected in series or in parallel. The default is R, resistor.

#### Rated voltage

Voltage for which load powers are specified. This parameter is visible only when you specify values by rated power. The default value is  $2.4e4$  V.

#### Real power

Total real power dissipated by three-phase load when supplied at the rated voltage. This parameter is visible only when you specify values by rated power and select a component structure that includes a resistor. The value must be greater than 0. The default value is 1000 W.

### **Rated electrical frequency**

Frequency for which reactive load powers are specified. This parameter is visible only when you specify values by rated power. The default value is 60 Hz.

### **Inductive reactive power**

Total inductive reactive power taken by the three-phase load when supplied at the rated voltage. This parameter is visible only when you specify values by rated power and select a component structure that includes an inductor. The value must be greater than 0. The default value is 100 V\*A.

### **Capacitive reactive power**

Total capacitive reactive power taken by the three-phase load when supplied at the rated voltage. This parameter is visible only when you specify values by rated power and select a component structure that includes a capacitor. The value must be less than 0. The default value is -100 V\*A.

### **Resistance**

Resistance of each of the load limbs. This parameter is visible only when you specify component values directly and select a component structure that includes a resistor. The default value is 1 Ohm.

### **Inductance**

Inductance of each of the load limbs. This parameter is visible only when you specify component values directly and select a component structure that includes an inductor. The default value is 0.001 H.

### **Capacitance**

Capacitance in each of the load limbs. This parameter is visible only when you specify component values directly and select a component structure that includes a capacitor. The default value is 1e-6 F.

## **Parasitics Tab**

### **Parasitic series resistance**

Represents small parasitic effects. The parameter value corresponds to the series resistance value added to all instances of capacitors in the load. The default value is 1e-6 Ohm.

### **Parasitic parallel conductance**

Represents small parasitic effects. The parameter value corresponds to the parallel conductance value added across all instances of inductors in the load. The default value is 1e-6 1/Ohm.

## Initial Conditions Tab

### Terminal voltage magnitude

Expected initial RMS line voltage at the load. This parameter is visible only when you specify values by rated power. The default value is  $2.4 \times 10^4$  V.

### Terminal voltage angle

Expected initial phase of the voltage at the load. This parameter is visible only when you specify values by rated power. The default value is 0 deg.

### Frequency

Expected initial frequency at the load. This parameter is visible only when you specify values by rated power. The default value is 60 Hz.

### Initial inductor current [ Ia Ib Ic ]

Initial current in the a, b, and c phase inductors, respectively. This parameter is visible only when you specify component values directly and select a component structure that includes an inductor. The default value is [0 0 0] A.

### Initial capacitor voltage [ Va Vb Vc ]

Initial voltage across the a, b, and c phase capacitors, respectively. This parameter is visible only when you specify component values directly and select a component structure that includes a capacitor. The default value is [0 0 0] V.

## Block Parameterization

The following two tables list the block parameters for each **Component structure**, based on the selected **Parameterization** option:

- Specify by rated power
- Specify component values directly

**Specify by Rated Power**

<b>Component Structure</b>	<b>Main Tab</b>	<b>Parasitics Tab</b>	<b>Initial Conditions Tab</b>
R	Rated voltage Real power	None	None
L	Rated voltage Rated electrical frequency Inductive reactive power	Parasitic parallel conductance	Terminal voltage magnitude Terminal voltage angle Frequency
C	Rated voltage Rated electrical frequency Capacitive reactive power	Parasitic series resistance	Terminal voltage magnitude Terminal voltage angle Frequency
Series RL	Rated voltage Rated electrical frequency Real power Inductive reactive power	Parasitic parallel conductance	Terminal voltage magnitude Terminal voltage angle Frequency
Series RC	Rated voltage Rated electrical frequency Real power Capacitive reactive power	None	Terminal voltage magnitude Terminal voltage angle Frequency



<b>Component Structure</b>	<b>Main Tab</b>	<b>Parasitics Tab</b>	<b>Initial Conditions Tab</b>
Series LC	Rated voltage Rated electrical frequency Inductive reactive power Capacitive reactive power	Parasitic parallel conductance	Terminal voltage magnitude Terminal voltage angle Frequency
Series RLC	Rated voltage Rated electrical frequency Real power Inductive reactive power Capacitive reactive power	Parasitic parallel conductance	Terminal voltage magnitude Terminal voltage angle Frequency
Parallel RL	Rated voltage Rated electrical frequency Real power Inductive reactive power	None	Terminal voltage magnitude Terminal voltage angle Frequency

<b>Component Structure</b>	<b>Main Tab</b>	<b>Parasitics Tab</b>	<b>Initial Conditions Tab</b>
Parallel RC	Rated voltage Rated electrical frequency Real power Capacitive reactive power	Parasitic series resistance	Terminal voltage magnitude Terminal voltage angle Frequency
Parallel LC	Rated voltage Rated electrical frequency Inductive reactive power Capacitive reactive power	Parasitic series resistance	Terminal voltage magnitude Terminal voltage angle Frequency
Parallel RLC	Rated voltage Rated electrical frequency Real power Inductive reactive power Capacitive reactive power	Parasitic series resistance	Terminal voltage magnitude Terminal voltage angle Frequency

**Specify Component Values Directly**

<b>Component Structure</b>	<b>Main Tab</b>	<b>Parasitics Tab</b>	<b>Initial Conditions Tab</b>
R	Resistance	None	None
L	Inductance	Parasitic parallel conductance	Initial inductor current [ Ia Ib Ic ]
C	Capacitance	Parasitic series resistance	Initial capacitor voltage [ Va Vb Vc ]
Series RL	Resistance Inductance	Parasitic parallel conductance	Initial inductor current [ Ia Ib Ic ]
Series RC	Resistance Capacitance	None	Initial capacitor voltage [ Va Vb Vc ]
Series LC	Inductance Capacitance	Parasitic parallel conductance	Initial inductor current [ Ia Ib Ic ] Initial capacitor voltage [ Va Vb Vc ]
Series RLC	Resistance Inductance Capacitance	Parasitic parallel conductance	Initial inductor current [ Ia Ib Ic ] Initial capacitor voltage [ Va Vb Vc ]
Parallel RL	Resistance Inductance	None	Initial inductor current [ Ia Ib Ic ]
Parallel RC	Resistance Capacitance	Parasitic series resistance	Initial capacitor voltage [ Va Vb Vc ]
Parallel LC	Inductance Capacitance	Parasitic series resistance	Initial inductor current [ Ia Ib Ic ] Initial capacitor voltage [ Va Vb Vc ]

<b>Component Structure</b>	<b>Main Tab</b>	<b>Parasitics Tab</b>	<b>Initial Conditions Tab</b>
Parallel RLC	Resistance Inductance Capacitance	Parasitic series resistance	Initial inductor current [ Ia Ib Ic ] Initial capacitor voltage [ Va Vb Vc ]

## Ports

The block has the following ports:

~

Expandable three-phase port

n

Electrical conserving port associated with the neutral phase

## See Also

Delta-Connected Load | RLC

## Topics

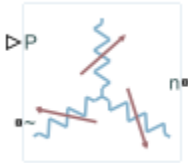
“Three-Phase Asynchronous Wind Turbine Generator”

“Expand and Collapse Three-Phase Ports on a Block”

**Introduced in R2013b**

# Wye-Connected Variable Load

Three-phase variable load wired in wye configuration



## Library

Simscape / Electrical / Power Systems / Passive Devices / Time Varying

## Description

The Wye-Connected Variable Load block models a three-phase variable load wired in a wye configuration. Each limb of the load contains a resistor. The block calculates the resistance required to draw the real power of the physical signal input  $P$  at the rated voltage that you specify. Therefore, the block can represent a real load.

To ensure that the resistance is always greater than zero, you specify the minimum real power that the load consumes. The minimum real power must be greater than zero.

## Electrical Defining Equations

The resistance is defined by

$$R = \frac{V_{Rated}^2}{P},$$

where:

- $R$  is the per-phase series resistance.
- $V_{Rated}$  is the RMS, rated line-line voltage.
- $P$  is the three-phase real power required.

## Parameters

### Main Tab

#### Rated voltage

RMS, rated line-line voltage for the resistance equation. The default value is 24e3 V.

#### Minimum real power

Minimum real power that the three-phase load dissipates when supplied at the rated voltage. The value must be greater than 0. The default value is 1e3 W.

## Ports

The block has the following ports:

P

Physical signal input port for real power

~

Expandable three-phase port

n

Electrical conserving port associated with the neutral phase

## See Also

Wye-Connected Load | Wye-Connected Variable Load (lagging)

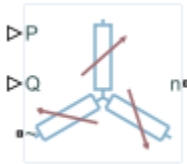
## Topics

“Expand and Collapse Three-Phase Ports on a Block”

**Introduced in R2014b**

## Wye-Connected Variable Load (lagging)

Three-phase variable, lagging load wired in wye configuration



### Library

Simscape / Electrical / Power Systems / Passive Devices / Time Varying

### Description

The Wye-Connected Variable Load (lagging) block models a three-phase variable, lagging load wired in a wye configuration. Each limb of the load contains a resistor (R) and an inductor (L) connected in series. The block calculates the resistance and inductance required to draw the real and reactive powers of the physical signal inputs P and Q at the rated voltage and rated frequency that you specify. Therefore, the block can represent a real and lagging reactive load.

To ensure that the resistance and inductance are always greater than zero, you specify the minimum real power and the reactive power that the load consumes. The minimum real power and the reactive power must be greater than zero.

### Electrical Defining Equations

The per-phase series resistance and inductance are defined by

$$R = \frac{PV_{Rated}^2}{P^2 + Q^2}$$



and

$$L = \frac{QV_{Rated}^2}{2\pi F_{Rated} (P^2 + Q^2)},$$

where:

- $R$  is the per-phase series resistance.
- $L$  is the per-phase series inductance.
- $V_{Rated}$  is the RMS, rated line-line voltage.
- $F_{Rated}$  is the nominal AC electrical frequency.
- $P$  is the three-phase real power required.
- $Q$  is the three-phase lagging reactive power required.

The inductance is defined as the ratio of the magnetic flux,  $\phi$ , to the steady-state current:

$$L(i) = \frac{\phi(i)}{i}.$$

Therefore the current-voltage relationship for the inductor is:

$$v = \frac{dL}{dt}i + L\frac{di}{dt}.$$

## Parameters

- “Main Tab” on page 1-2099
- “Parasitics” on page 1-2100
- “Variables Tab” on page 1-2100

### Main Tab

#### Rated voltage

RMS, rated line-line voltage for the resistance equation. The default value is 24e3 V.

### **Rated electrical frequency**

Nominal AC electrical frequency for the inductance equation. The default value is 60 Hz.

### **Minimum real power**

Minimum real power that the three-phase load dissipates when supplied at the rated voltage. The value must be greater than 0. The default value is 1e3 W.

### **Minimum reactive power**

Minimum reactive power that the three-phase load dissipates when supplied at the rated voltage. The value must be greater than 0. The default value is 1e3 V\*A.

## **Parasitics**

### **Parasitic parallel conductance**

Conductance that the block adds, in parallel, to the series RL. The default value is 1e-6 1/Ohm.

## **Variables Tab**

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## **Ports**

The block has the following ports:

P

Physical signal input port for real power

Q

Physical signal input port for reactive power

~

Expandable three-phase port

n

Electrical conserving port associated with the neutral phase.

## See Also

Wye-Connected Load | Wye-Connected Variable Load

## Topics

“Expand and Collapse Three-Phase Ports on a Block”

**Introduced in R2014b**

## Wye-Delta1 Transformer

Linear nonideal wye-delta1 transformer with three-limb core



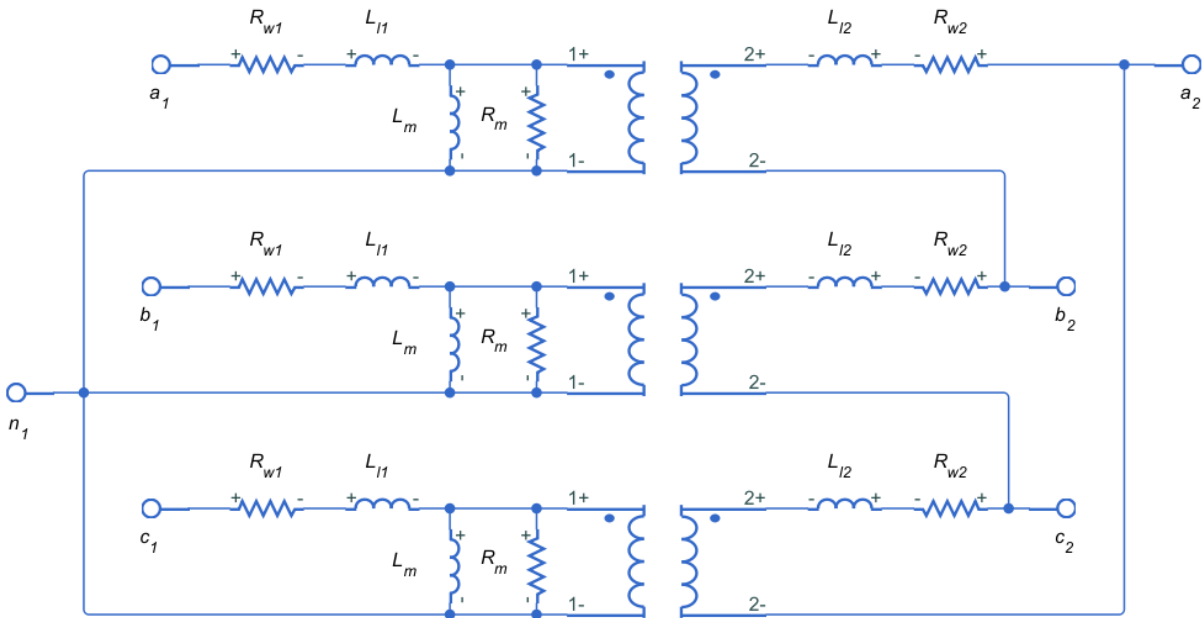
### Library

Simscape / Electrical / Power Systems / Passive Devices / Transformers

### Description

The Wye-Delta1 Transformer block models a linear, nonideal transformer with a three-limb core, in which the primary windings are configured in a wye connection and the secondary windings are configured in a delta connection. The delta voltages lag the wye voltages by 30 degrees, hence the name 1 o'clock delta. The block includes linear winding leakage and linear core magnetization effects.

The figure shows the equivalent circuit diagram for the wye-delta1 transformer.



- $R_{w1}$  is the primary winding resistance.
- $L_{l1}$  is the primary leakage reactance.
- $R_{w2}$  is the secondary winding resistance.
- $L_{l2}$  is the secondary leakage reactance.
- $R_m$  is the shunt magnetizing resistance.
- $L_m$  is the shunt magnetizing reactance.

## Display Options

You can display the transformer per-unit base values in the MATLAB command window using the block context menu. To display the values, right-click the block and select **Electrical > Display Base Values**.

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Parameters

- “Main Tab” on page 1-2104
- “Impedances Tab” on page 1-2104

## Main Tab

### Rated apparent power

Apparent power flowing through the transformer when operating at rated capacity. The default value is  $100e6 \text{ V}\cdot\text{A}$ .

### Primary rated voltage

RMS line voltage applied to the primary winding under normal operating conditions. The default value is  $4160 \text{ V}$ .

### Secondary rated voltage

RMS line voltage applied to the secondary winding under normal operating conditions. The default value is  $24e3 \text{ V}$ .

### Rated electrical frequency

Rated or nominal frequency of the AC network to which the transformer is connected. The default value is  $60 \text{ Hz}$ .

## Impedances Tab

Parameters in this tab are expressed in per unit (pu). For more information, see “Per-Unit System of Units”.

**Primary winding resistance (pu)**

Power loss in the primary winding. The default value is 0.01.

**Primary leakage reactance (pu)**

Magnetic flux loss in the primary winding. The default value is 0.001.

**Secondary winding resistance (pu)**

Power loss in the secondary winding. The default value is 0.01.

**Secondary leakage reactance (pu)**

Magnetic flux loss in the secondary winding. The default value is 0.001.

**Shunt magnetizing resistance (pu)**

Magnetic losses in transformer core. The default value is 500.

**Shunt magnetizing reactance (pu)**

Magnetic effects of the transformer core when operating in its linear region. The default value is 500.

## Ports

The block has the following ports:

~1

Expandable three-phase port for primary winding

~2

Expandable three-phase port for secondary winding

n1

Electrical conserving port associated with the primary winding neutral point

## See Also

Delta-Delta Transformer | Delta1-Delta1-Wye Transformer | Delta11-Delta11-Wye Transformer | Wye-Delta1-Wye Transformer | Wye-Delta11 Transformer | Wye-Delta11-Wye Transformer | Wye-Wye Transformer | Zigzag-Delta1-Wye Transformer | Zigzag-Delta11-Wye Transformer

## **Topics**

“Custom Zigzag Transformer”

“Push-Pull Buck Converter in Continuous Conduction Mode”

“Push-Pull Buck Converter in Discontinuous Conduction Mode”

**Introduced in R2013b**



# Wye-Delta11 Transformer

Linear nonideal wye-delta11 transformer with three-limb core



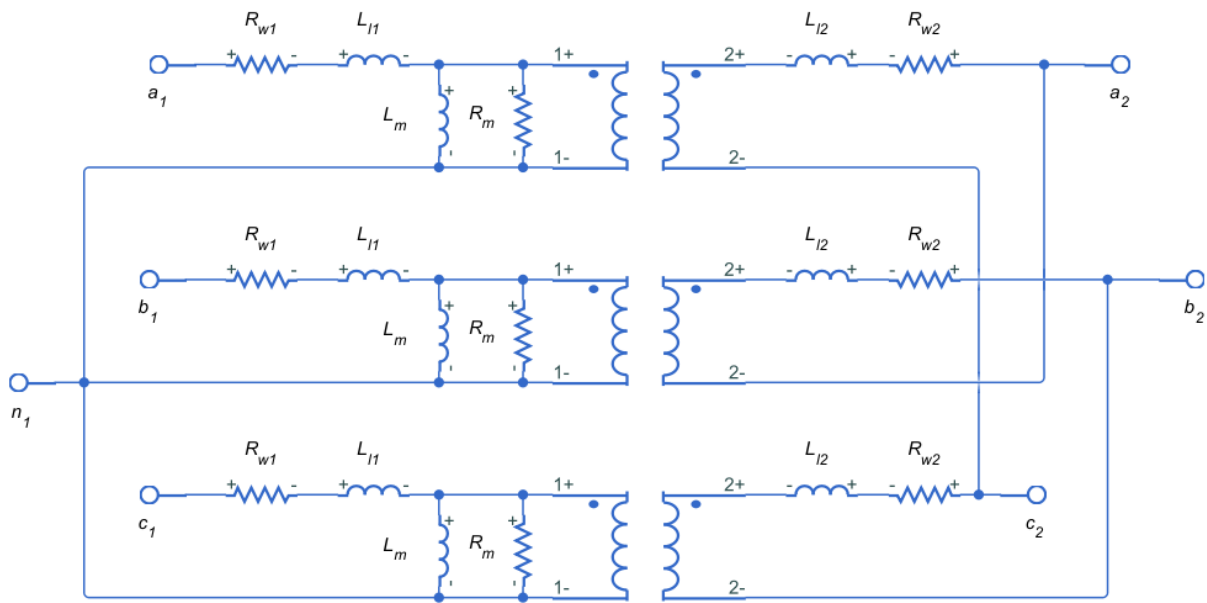
## Library

Simscape / Electrical / Power Systems / Passive Devices / Transformers

## Description

The Wye-Delta11 Transformer block models a linear, nonideal transformer with a three-limb core, in which the primary windings are configured in a wye connection and the secondary windings are configured in a delta connection. The delta voltages lead the wye voltages by 30 degrees, hence the name 11 o'clock delta. The block includes linear winding leakage and linear core magnetization effects.

The figure shows the equivalent circuit diagram for the wye-delta11 transformer.



- $R_{w1}$  is the primary winding resistance.
- $L_{l1}$  is the primary leakage reactance.
- $R_{w2}$  is the secondary winding resistance.
- $L_{l2}$  is the secondary leakage reactance.
- $R_m$  is the shunt magnetizing resistance.
- $L_m$  is the shunt magnetizing reactance.

## Display Options

You can display the transformer per-unit base values in the MATLAB command window using the block context menu. To display the values, right-click the block and select **Electrical > Display Base Values**.

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Parameters

- “Main Tab” on page 1-2109
- “Impedances Tab” on page 1-2109

## Main Tab

### Rated apparent power

Apparent power flowing through the transformer when operating at rated capacity.  
The default value is  $100\text{e}6 \text{ V}\cdot\text{A}$ .

### Primary rated voltage

RMS line voltage applied to the primary winding under normal operating conditions.  
The default value is  $4160 \text{ V}$ .

### Secondary rated voltage

RMS line voltage applied to the secondary winding under normal operating conditions. The default value is  $24\text{e}3 \text{ V}$ .

### Rated electrical frequency

Rated or nominal frequency of the AC network to which the transformer is connected.  
The default value is  $60 \text{ Hz}$ .

## Impedances Tab

Parameters in this tab are expressed in per-unit (pu). For more information, see “Per-Unit System of Units”.

**Primary winding resistance (pu)**

Power loss in the primary winding. The default value is 0.01.

**Primary leakage reactance (pu)**

Magnetic flux loss in the primary winding. The default value is 0.001.

**Secondary winding resistance (pu)**

Power loss in the secondary winding. The default value is 0.01.

**Secondary leakage reactance (pu)**

Magnetic flux loss in the secondary winding. The default value is 0.001.

**Shunt magnetizing resistance (pu)**

Magnetic losses in transformer core. The default value is 500.

**Shunt magnetizing reactance (pu)**

Magnetic effects of the transformer core when operating in its linear region. The default value is 500.

## Ports

The block has the following ports:

~1

Expandable three-phase port for primary winding

~2

Expandable three-phase port for secondary winding

n1

Electrical conserving port associated with the primary winding neutral point

## See Also

Delta-Delta Transformer | Delta1-Delta1-Wye Transformer | Delta11-Delta11-Wye Transformer | Wye-Delta1 Transformer | Wye-Delta1-Wye Transformer | Wye-Delta11-Wye Transformer | Wye-Wye Transformer | Zigzag-Delta1-Wye Transformer | Zigzag-Delta11-Wye Transformer

## **Topics**

“Custom Zigzag Transformer”

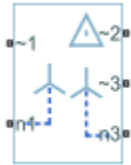
“Push-Pull Buck Converter in Continuous Conduction Mode”

“Push-Pull Buck Converter in Discontinuous Conduction Mode”

**Introduced in R2013b**

## Wye-Delta1-Wye Transformer

Linear nonideal wye-delta1-wye transformer with three-limb core



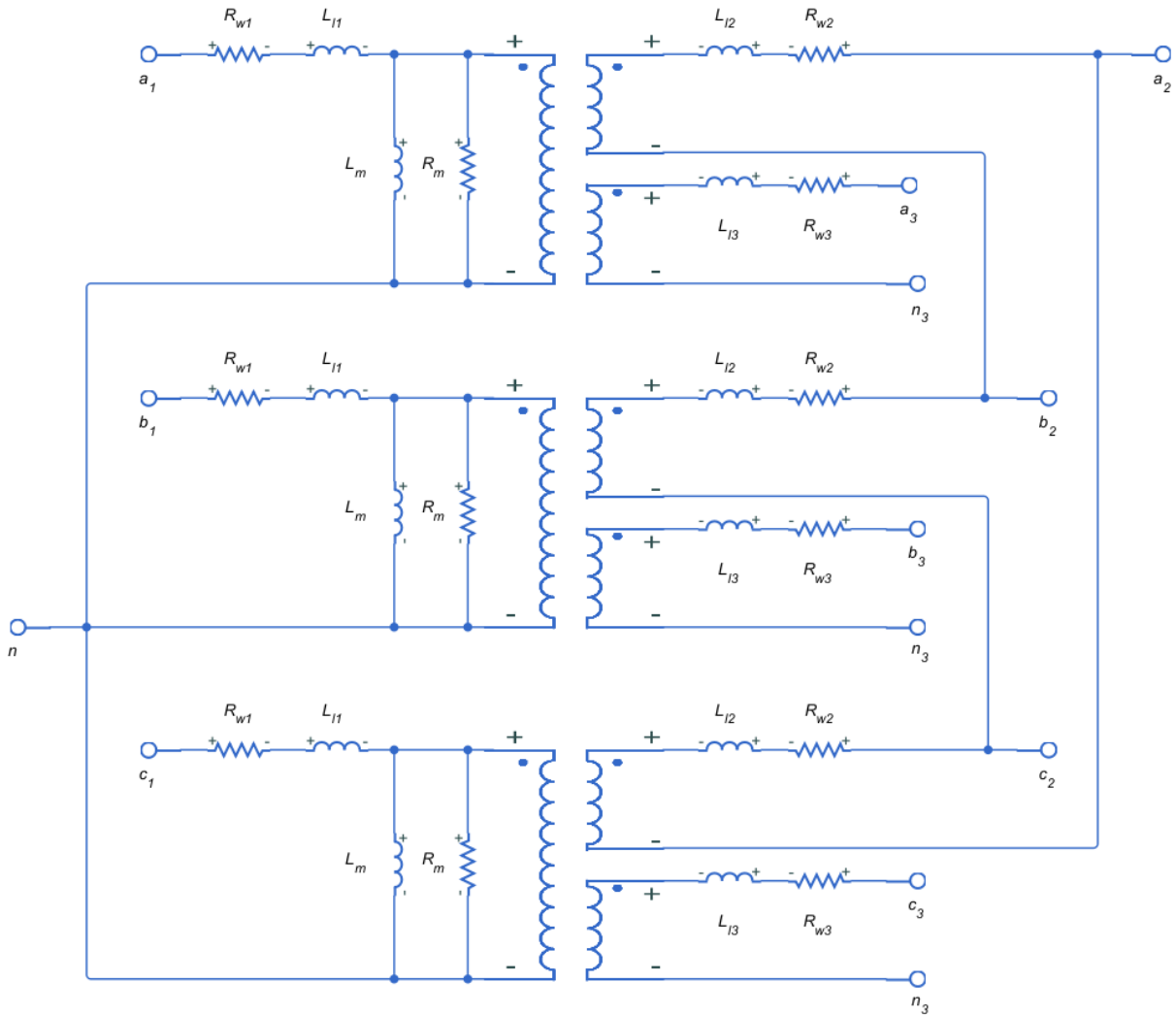
### Library

Simscape / Electrical / Power Systems / Passive Devices / Transformers

### Description

The Wye-Delta1-Wye Transformer block models a linear, nonideal transformer with a three-limb core, in which the primary windings are configured in a wye connection and there are delta secondary windings and wye secondary windings. The delta voltages lag the wye voltages by 30 degrees, hence the name 1 o'clock delta. The block includes linear winding leakage and linear core magnetization effects.

The figure shows the equivalent circuit diagram for the wye-delta1-wye transformer.



- $R_{w1}$  is the primary winding resistance.
- $L_{l1}$  is the primary leakage reactance.
- $R_{w2}$  is the delta secondary winding resistance.
- $L_{l2}$  is the delta secondary leakage reactance.

- $R_{w3}$  is the wye secondary winding resistance.
- $L_{l3}$  is the wye secondary leakage reactance.
- $R_m$  is the shunt magnetizing resistance.
- $L_m$  is the shunt magnetizing reactance.

## Display Options

You can display the transformer per-unit base values in the MATLAB command window using the block context menu. To display the values, right-click the block and select **Electrical > Display Base Values**.

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Parameters

- “Main Tab” on page 1-2114
- “Impedances Tab” on page 1-2115

### Main Tab

#### Rated apparent power

Apparent power flowing through the transformer when operating at rated capacity. The default value is  $100e6 \text{ V}\cdot\text{A}$ .

#### Primary rated voltage

RMS line voltage applied to the primary winding under normal operating conditions. The default value is  $4160 \text{ V}$ .



**Delta secondary rated voltage**

RMS line voltage applied to the delta secondary winding under normal operating conditions. The default value is 24e3 V.

**Wye secondary rated voltage**

RMS line voltage applied to the wye secondary winding under normal operating conditions. The default value is 24e3 V.

**Rated electrical frequency**

Rated or nominal frequency of the AC network to which the transformer is connected. The default value is 60 Hz.

**Impedances Tab**

Parameters in this tab are expressed in per-unit (pu). For more information, see “Per-Unit System of Units”.

**Primary winding resistance (pu)**

Power loss in the primary winding. The default value is 0.01.

**Primary leakage reactance (pu)**

Magnetic flux loss in the primary winding. The default value is 0.001.

**Delta secondary winding resistance (pu)**

Power loss in the delta secondary winding. The default value is 0.01.

**Delta secondary leakage reactance (pu)**

Magnetic flux loss in the delta secondary winding. The default value is 0.001.

**Wye secondary winding resistance (pu)**

Power loss in the wye secondary winding. The default value is 0.01.

**Wye secondary leakage reactance (pu)**

Magnetic flux loss in the wye secondary winding. The default value is 0.001.

**Shunt magnetizing resistance (pu)**

Magnetic losses in transformer core. The default value is 500.

**Shunt magnetizing reactance (pu)**

Magnetic effects of the transformer core when operating in its linear region. The default value is 500.

## Ports

The block has the following ports:

~1

Expandable three-phase port for primary winding

~2

Expandable three-phase port for delta secondary winding

~3

Expandable three-phase port for wye secondary winding

n1

Electrical conserving port associated with the primary winding neutral point

n3

Electrical conserving port associated with the wye secondary winding neutral point

## See Also

[Delta-Delta Transformer](#) | [Delta1-Delta1-Wye Transformer](#) | [Delta11-Delta11-Wye Transformer](#) | [Wye-Delta1 Transformer](#) | [Wye-Delta11 Transformer](#) | [Wye-Delta11-Wye Transformer](#) | [Wye-Wye Transformer](#) | [Zigzag-Delta1-Wye Transformer](#) | [Zigzag-Delta11-Wye Transformer](#)

## Topics

[“Custom Zigzag Transformer”](#)

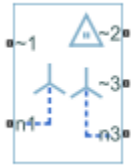
[“Push-Pull Buck Converter in Continuous Conduction Mode”](#)

[“Push-Pull Buck Converter in Discontinuous Conduction Mode”](#)

**Introduced in R2013b**

# Wye-Delta11-Wye Transformer

Linear nonideal wye-delta11-wye transformer with three-limb core



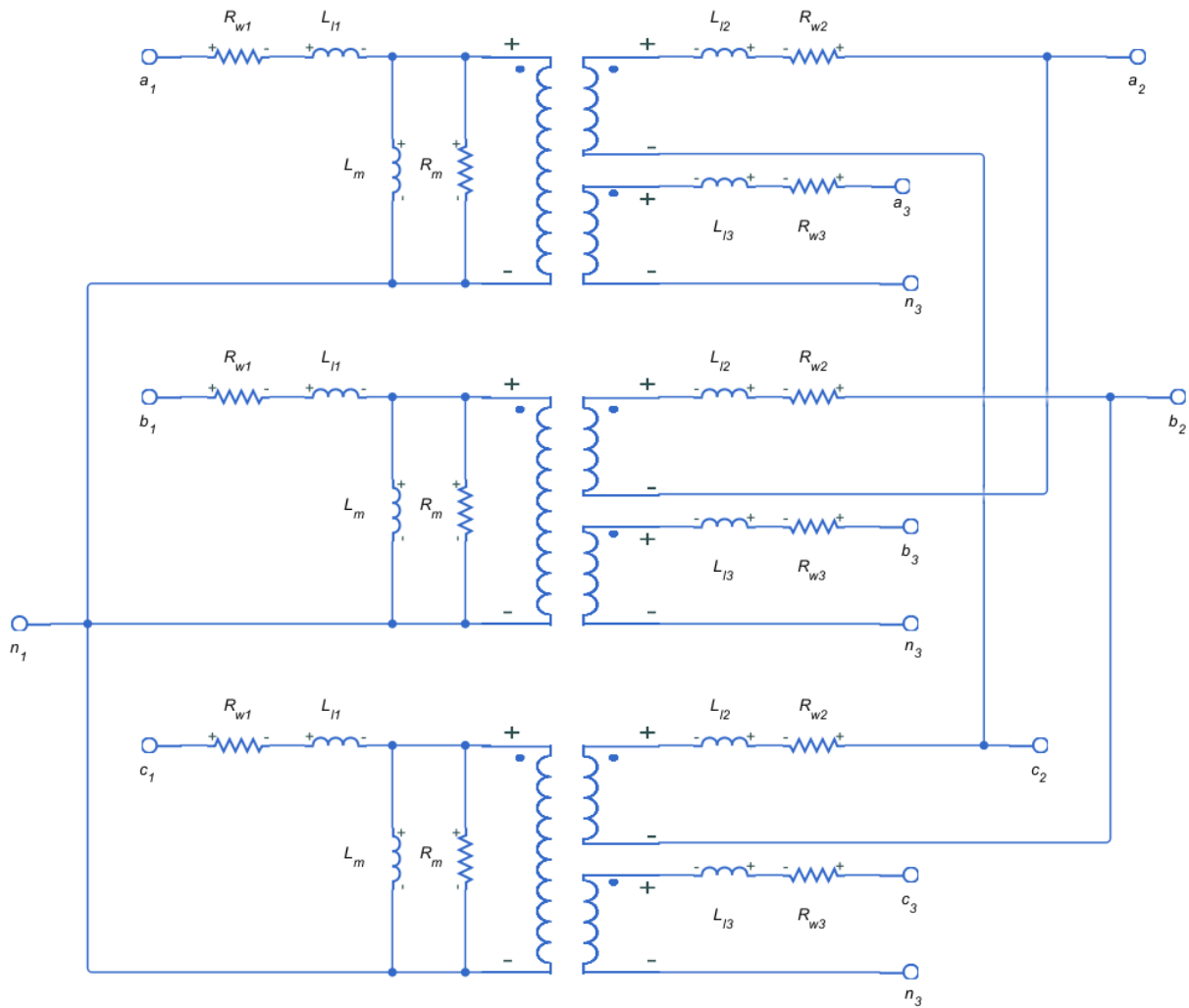
## Library

Simscape / Electrical / Power Systems / Passive Devices / Transformers

## Description

The Wye-Delta11-Wye Transformer block models a linear, nonideal transformer with a three-limb core, in which the primary windings are configured in a wye connection and there are delta secondary windings and wye secondary windings. The delta voltages lead the wye voltages by 30 degrees, hence the name 11 o'clock delta. The block includes linear winding leakage and linear core magnetization effects.

The figure shows the equivalent circuit diagram for the wye-delta11-wye transformer.



- $R_{w1}$  is the primary winding resistance.
- $L_{l1}$  is the primary leakage reactance.
- $R_{w2}$  is the delta secondary winding resistance.
- $L_{l2}$  is the delta secondary leakage reactance.

- $R_{w3}$  is the wye secondary winding resistance.
- $L_{l3}$  is the wye secondary leakage reactance.
- $R_m$  is the shunt magnetizing resistance.
- $L_m$  is the shunt magnetizing reactance.

## Display Options

You can display the transformer per-unit base values in the MATLAB command window using the block context menu. To display the values, right-click the block and select **Electrical > Display Base Values**.

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Parameters

- “Main Tab” on page 1-2119
- “Impedances Tab” on page 1-2120

### Main Tab

#### Rated apparent power

Apparent power flowing through the transformer when operating at rated capacity.  
The default value is  $100e6 \text{ V}\cdot\text{A}$ .

#### Primary rated voltage

RMS line voltage applied to the primary winding under normal operating conditions.  
The default value is  $4160 \text{ V}$ .

**Delta secondary rated voltage**

RMS line voltage applied to the delta secondary winding under normal operating conditions. The default value is 24e3 V.

**Wye secondary rated voltage**

RMS line voltage applied to the wye secondary winding under normal operating conditions. The default value is 24e3 V.

**Rated electrical frequency**

Rated or nominal frequency of the AC network to which the transformer is connected. The default value is 60 Hz.

**Impedances Tab**

Parameters in this tab are expressed in per-unit (pu). For more information, see “Per-Unit System of Units”.

**Primary winding resistance (pu)**

Power loss in the primary winding. The default value is 0.01.

**Primary leakage reactance (pu)**

Magnetic flux loss in the primary winding. The default value is 0.001.

**Delta secondary winding resistance (pu)**

Power loss in the delta secondary winding. The default value is 0.01.

**Delta secondary leakage reactance (pu)**

Magnetic flux loss in the delta secondary winding. The default value is 0.001.

**Wye secondary winding resistance (pu)**

Power loss in the wye secondary winding. The default value is 0.01.

**Wye secondary leakage reactance (pu)**

Magnetic flux loss in the wye secondary winding. The default value is 0.001.

**Shunt magnetizing resistance (pu)**

Magnetic losses in transformer core. The default value is 500.

**Shunt magnetizing reactance (pu)**

Magnetic effects of the transformer core when operating in its linear region. The default value is 500.

## Ports

The block has the following ports:

~1

Expandable three-phase port for primary winding

~2

Expandable three-phase port for delta secondary winding

~3

Expandable three-phase port for wye secondary winding

n1

Electrical conserving port associated with the primary winding neutral point

n3

Electrical conserving port associated with the wye secondary winding neutral point

## See Also

[Delta-Delta Transformer](#) | [Delta1-Delta1-Wye Transformer](#) | [Delta11-Delta11-Wye Transformer](#) | [Wye-Delta1 Transformer](#) | [Wye-Delta1-Wye Transformer](#) | [Wye-Delta11 Transformer](#) | [Wye-Wye Transformer](#) | [Zigzag-Delta1-Wye Transformer](#) | [Zigzag-Delta11-Wye Transformer](#)

## Topics

[“Custom Zigzag Transformer”](#)

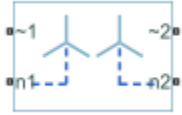
[“Push-Pull Buck Converter in Continuous Conduction Mode”](#)

[“Push-Pull Buck Converter in Discontinuous Conduction Mode”](#)

**Introduced in R2013b**

## Wye-Wye Transformer

Linear nonideal wye-wye transformer with three-limb core



### Library

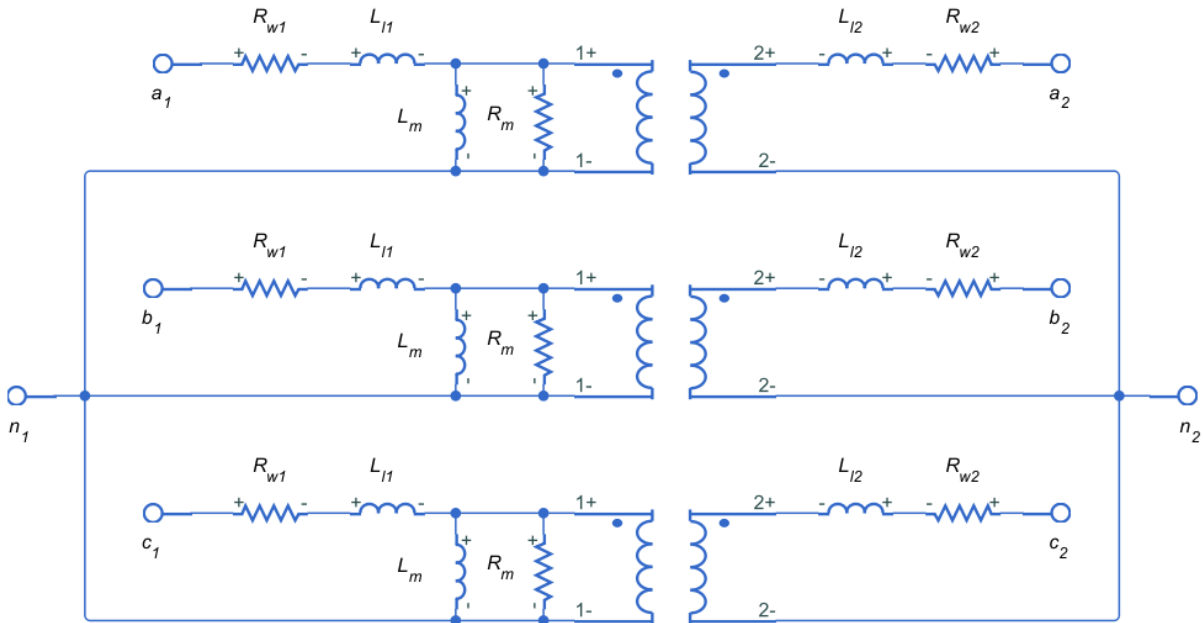
Simscape / Electrical / Power Systems / Passive Devices / Transformers

### Description

The Wye-Wye Transformer block models a linear, nonideal transformer with a three-limb core, in which both the primary and the secondary windings are configured in a wye connection. The block includes linear winding leakage and linear core magnetization effects.

The figure shows the equivalent circuit diagram for the wye-wye transformer.





- $R_{w1}$  is the primary winding resistance.
- $L_{l1}$  is the primary leakage reactance.
- $R_{w2}$  is the secondary winding resistance.
- $L_{l2}$  is the secondary leakage reactance.
- $R_m$  is the shunt magnetizing resistance.
- $L_m$  is the shunt magnetizing reactance.

## Display Options

You can display the transformer per-unit base values in the MATLAB command window using the block context menu. To display the values, right-click the block and select **Electrical > Display Base Values**.

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Parameters

- “Main Tab” on page 1-2124
- “Impedances Tab” on page 1-2124

## Main Tab

### Rated apparent power

Apparent power flowing through the transformer when operating at rated capacity. The default value is  $100e6 \text{ V}\cdot\text{A}$ .

### Primary rated voltage

RMS line voltage applied to the primary winding under normal operating conditions. The default value is  $4160 \text{ V}$ .

### Secondary rated voltage

RMS line voltage applied to the secondary winding under normal operating conditions. The default value is  $24e3 \text{ V}$ .

### Rated electrical frequency

Rated or nominal frequency of the AC network to which the transformer is connected. The default value is  $60 \text{ Hz}$ .

## Impedances Tab

Parameters in this tab are expressed in per-unit (pu). For more information, see “Per-Unit System of Units”

**Primary winding resistance (pu)**

Power loss in the primary winding. The default value is 0.01.

**Primary leakage reactance (pu)**

Magnetic flux loss in the primary winding. The default value is 0.001.

**Secondary winding resistance (pu)**

Power loss in the secondary winding. The default value is 0.01.

**Secondary leakage reactance (pu)**

Magnetic flux loss in the secondary winding. The default value is 0.001.

**Shunt magnetizing resistance (pu)**

Magnetic losses in transformer core. The default value is 500.

**Shunt magnetizing reactance (pu)**

Magnetic effects of the transformer core when operating in its linear region. The default value is 500.

## Ports

The block has the following ports:

~1

Expandable three-phase port for primary winding

~2

Expandable three-phase port for secondary winding

n1

Electrical conserving port associated with the primary winding neutral point

n2

Electrical conserving port associated with the secondary winding neutral point

## See Also

Delta-Delta Transformer | Delta1-Delta1-Wye Transformer | Delta11-Delta11-Wye Transformer | Wye-Delta1 Transformer | Wye-Delta1-Wye Transformer | Wye-Delta11 Transformer | Wye-Delta11-Wye Transformer | Zigzag-Delta1-Wye Transformer | Zigzag-Delta11-Wye Transformer

## **Topics**

“Custom Zigzag Transformer”

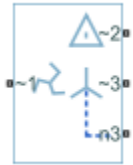
“Push-Pull Buck Converter in Continuous Conduction Mode”

“Push-Pull Buck Converter in Discontinuous Conduction Mode”

**Introduced in R2013b**

# Zigzag-Delta1-Wye Transformer

Linear nonideal zigzag-delta1-wye transformer with three-limb core



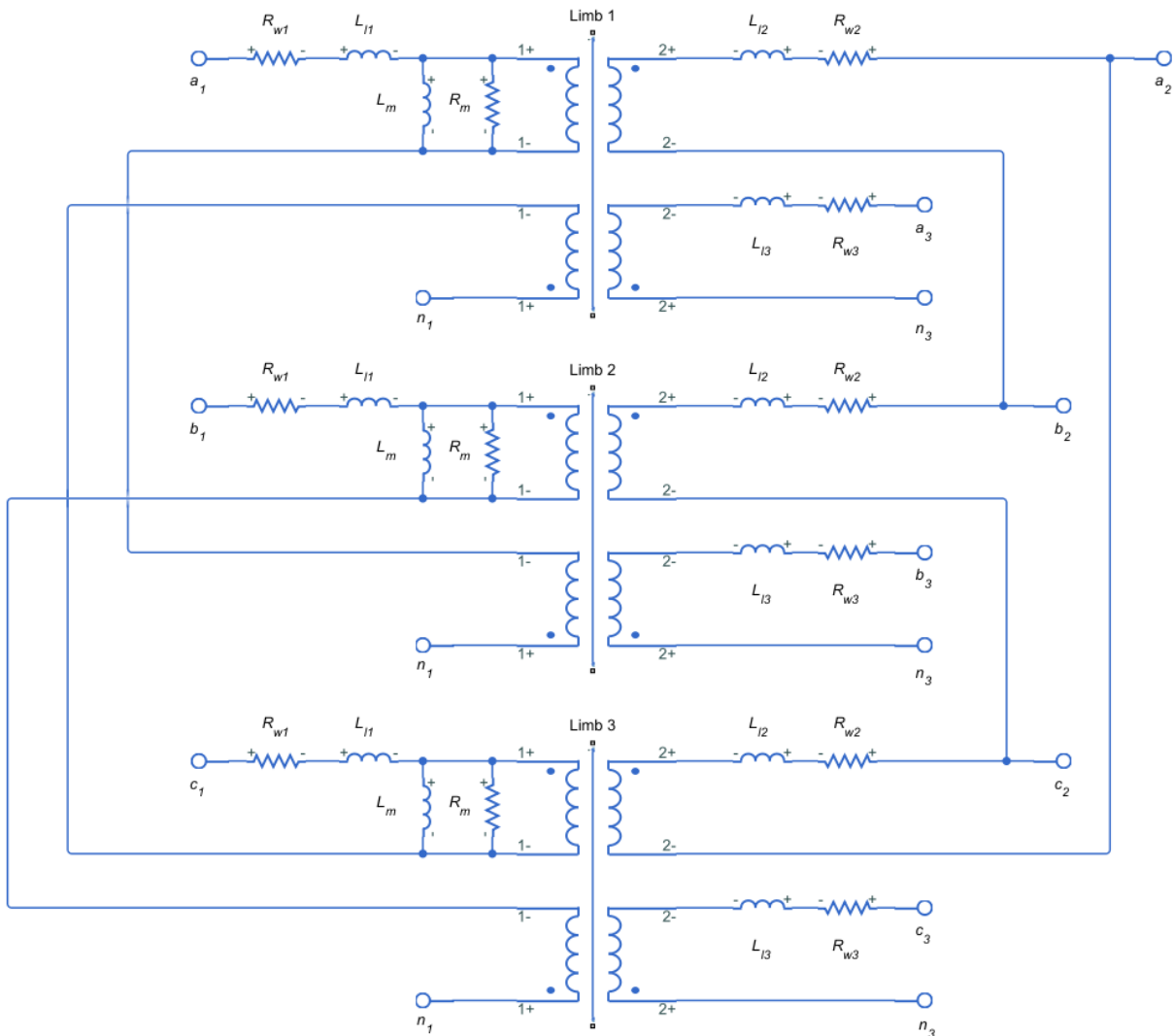
## Library

Simscape / Electrical / Power Systems / Passive Devices / Transformers

## Description

The Zigzag-Delta1-Wye Transformer block models a linear, nonideal transformer with a three-limb core, in which the primary windings are configured in a zigzag connection and there are delta secondary windings and wye secondary windings. You can specify the phase offset between the zigzag and wye windings via a block parameter. The delta voltages lag the wye voltages by 30 degrees, hence the name one o'clock delta. The block includes linear winding leakage and linear core magnetization effects.

The figure shows the equivalent circuit diagram for the zigzag-delta1-wye transformer.



- $R_{w1}$  is the primary winding resistance.
- $L_{l1}$  is the primary leakage reactance.
- $R_{w2}$  is the delta secondary winding resistance.

- $L_{12}$  is the delta secondary leakage reactance.
- $R_{w3}$  is the wye secondary winding resistance.
- $L_{13}$  is the wye secondary leakage reactance.
- $R_m$  is the shunt magnetizing resistance.
- $L_m$  is the shunt magnetizing reactance.

## Display Options

You can display the transformer per-unit base values in the MATLAB command window using the block context menu. To display the values, right-click the block and select **Electrical > Display Base Values**.

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Parameters

- “Main” on page 1-2129
- “Impedances” on page 1-2130

### Main

#### Rated apparent power

Apparent power flowing through the transformer when operating at rated capacity.  
The default value is  $100e6$  V\*A.

#### Primary rated voltage

RMS line voltage applied to the primary winding under normal operating conditions.  
The default value is  $24e3$  V.

**Delta secondary rated voltage**

RMS line voltage applied to the delta secondary winding under normal operating conditions. The default value is 4160 V.

**Wye secondary rated voltage**

RMS line voltage applied to the wye secondary winding under normal operating conditions. The default value is 4160 V.

**Rated electrical frequency**

Rated or nominal frequency of the AC network to which the transformer is connected. The default value is 60 Hz.

**Wye secondary phase shift**

The phase offset between the zigzag and wye secondary windings. The default value is -7.5 deg.

**Impedances**

Parameters in this tab are expressed in per unit (pu). For more information, see “Per-Unit System of Units”.

**Primary winding resistance (pu)**

Power loss in the primary winding. The default value is 0.01.

**Primary leakage reactance (pu)**

Magnetic flux loss in the primary winding. The default value is 0.001.

**Delta secondary winding resistance (pu)**

Power loss in the delta secondary winding. The default value is 0.01.

**Delta secondary leakage reactance (pu)**

Magnetic flux loss in the delta secondary winding. The default value is 0.001.

**Wye secondary winding resistance (pu)**

Power loss in the wye secondary winding. The default value is 0.01.

**Wye secondary leakage reactance (pu)**

Magnetic flux loss in the wye secondary winding. The default value is 0.001.

**Shunt magnetizing resistance (pu)**

Magnetic losses in transformer core. The default value is 500.



**Shunt magnetizing reactance (pu)**

Magnetic effects of the transformer core when operating in its linear region. The default value is 500.

**Ports**

The block has the following ports:

~1

Expandable three-phase port for primary winding

~2

Expandable three-phase port for delta secondary winding

~3

Expandable three-phase port for wye secondary winding

n3

Electrical conserving port associated with the wye secondary winding neutral point

**See Also**

[Delta-Delta Transformer](#) | [Delta1-Delta1-Wye Transformer](#) | [Delta11-Delta11-Wye Transformer](#) | [Wye-Delta1 Transformer](#) | [Wye-Delta1-Wye Transformer](#) | [Wye-Delta11 Transformer](#) | [Wye-Delta11-Wye Transformer](#) | [Wye-Wye Transformer](#) | [Zigzag-Delta11-Wye Transformer](#)

**Topics**

[“Custom Zigzag Transformer”](#)

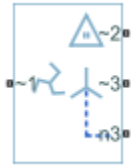
[“Push-Pull Buck Converter in Continuous Conduction Mode”](#)

[“Push-Pull Buck Converter in Discontinuous Conduction Mode”](#)

**Introduced in R2015a**

## Zigzag-Delta11-Wye Transformer

Linear nonideal zigzag-delta11-wye transformer with three-limb core



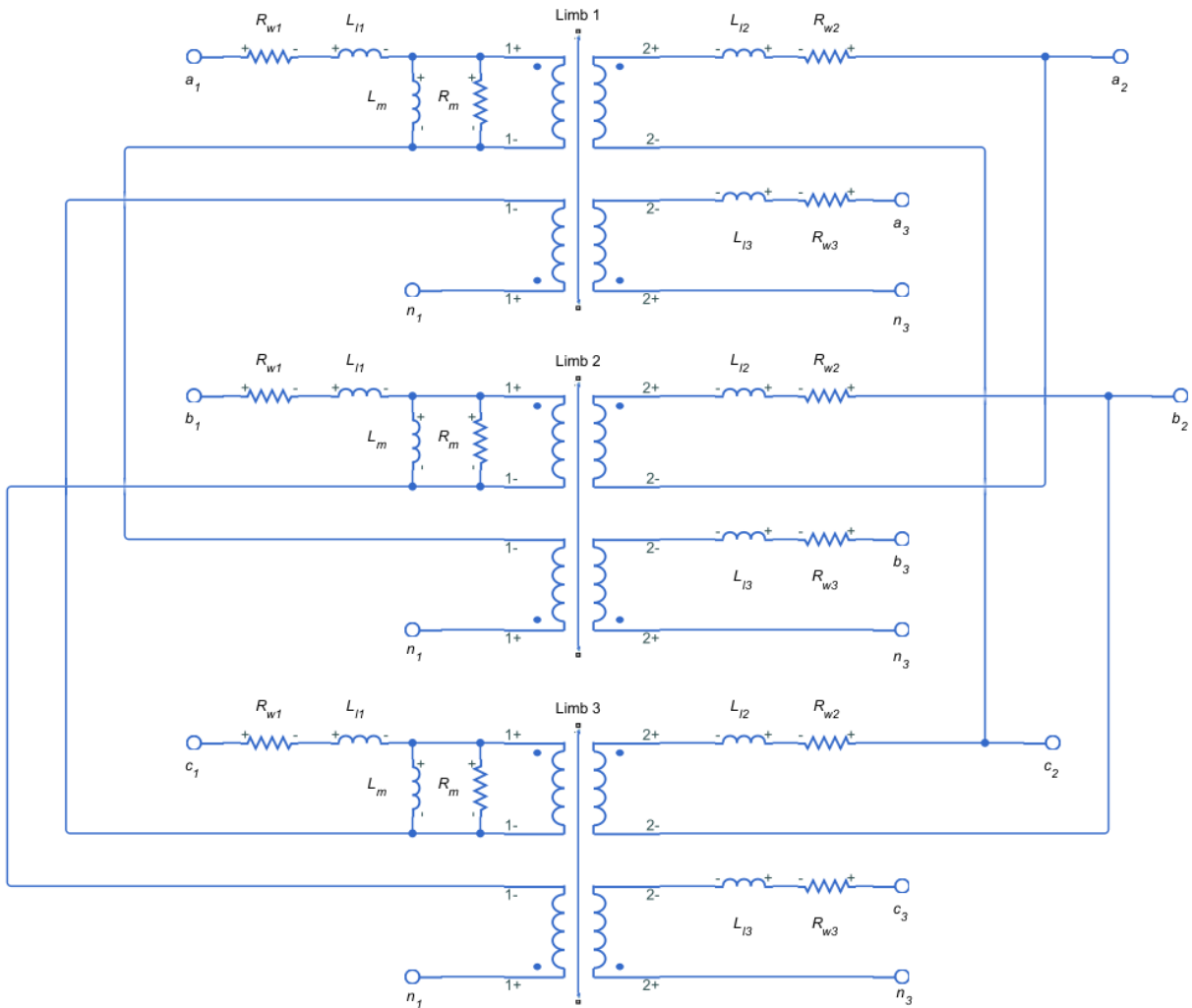
### Library

Simscape / Electrical / Power Systems / Passive Devices / Transformers

### Description

The Zigzag-Delta11-Wye Transformer block models a linear, nonideal transformer with a three-limb core, in which the primary windings are configured in a zigzag connection and there are delta secondary windings and wye secondary windings. You can specify the phase offset between the zigzag and wye windings via a block parameter. The delta voltages lead the wye voltages by 30 degrees, hence the name 11 o'clock delta. The block includes linear winding leakage and linear core magnetization effects.

The figure shows the equivalent circuit diagram for the zigzag-delta11-wye transformer.



- $R_{w1}$  is the primary winding resistance.
- $L_{l1}$  is the primary leakage reactance.
- $R_{w2}$  is the delta secondary winding resistance.
- $L_{l2}$  is the delta secondary leakage reactance.

- $R_{w3}$  is the wye secondary winding resistance.
- $L_{l3}$  is the wye secondary leakage reactance.
- $R_m$  is the shunt magnetizing resistance.
- $L_m$  is the shunt magnetizing reactance.

## Display Options

You can display the transformer per-unit base values in the MATLAB command window using the block context menu. To display the values, right-click the block and select **Electrical > Display Base Values**.

## Variables

Use the **Variables** settings to specify the priority and initial target values for the block variables before simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Unlike block parameters, variables do not have conditional visibility. The **Variables** settings include all the existing block variables. If a variable is not used in the set of equations corresponding to the selected block configuration, the values specified for this variable are ignored.

## Parameters

- “Main” on page 1-2134
- “Impedances” on page 1-2135

### Main

#### Rated apparent power

Apparent power flowing through the transformer when operating at rated capacity. The default value is  $100e6 \text{ V}\cdot\text{A}$ .

#### Primary rated voltage

RMS line voltage applied to the primary winding under normal operating conditions. The default value is  $24e3 \text{ V}$ .

**Delta secondary rated voltage**

RMS line voltage applied to the delta secondary winding under normal operating conditions. The default value is 4160 V.

**Wye secondary rated voltage**

RMS line voltage applied to the wye secondary winding under normal operating conditions. The default value is 4160 V.

**Rated electrical frequency**

Rated or nominal frequency of the AC network to which the transformer is connected. The default value is 60 Hz.

**Wye secondary phase shift**

The phase offset between the zigzag and wye secondary windings. The default value is -7.5 deg.

**Impedances**

Parameters in this tab are expressed in per-unit (pu). For more information, see “Per-Unit System of Units”.

**Primary winding resistance (pu)**

Power loss in the primary winding. The default value is 0.01.

**Primary leakage reactance (pu)**

Magnetic flux loss in the primary winding. The default value is 0.001.

**Delta secondary winding resistance (pu)**

Power loss in the delta secondary winding. The default value is 0.01.

**Delta secondary leakage reactance (pu)**

Magnetic flux loss in the delta secondary winding. The default value is 0.001.

**Wye secondary winding resistance (pu)**

Power loss in the wye secondary winding. The default value is 0.01.

**Wye secondary leakage reactance (pu)**

Magnetic flux loss in the wye secondary winding. The default value is 0.001.

**Shunt magnetizing resistance (pu)**

Magnetic losses in transformer core. The default value is 500.

### **Shunt magnetizing reactance (pu)**

Magnetic effects of the transformer core when operating in its linear region. The default value is 500.

## **Ports**

The block has the following ports:

~1

Expandable three-phase port for primary winding

~2

Expandable three-phase port for delta secondary winding

~3

Expandable three-phase port for wye secondary winding

n3

Electrical conserving port associated with the wye secondary winding neutral point

## **See Also**

Delta-Delta Transformer | Delta1-Delta1-Wye Transformer | Delta11-Delta11-Wye Transformer | Wye-Delta1 Transformer | Wye-Delta1-Wye Transformer | Wye-Delta11 Transformer | Wye-Delta11-Wye Transformer | Wye-Wye Transformer | Zigzag-Delta1-Wye Transformer

## **Topics**

“Custom Zigzag Transformer”

“Push-Pull Buck Converter in Continuous Conduction Mode”

“Push-Pull Buck Converter in Discontinuous Conduction Mode”

**Introduced in R2015a**

# Functions — Alphabetical List

---

# elec\_calculateFluxPartialDerivatives

Calculate flux partial derivatives for FEM-Parameterized PMSM block

## Syntax

```
[dFdA,dFdB,dFdC,dFdX] = elec_calculateFluxPartialDerivatives(A,B,C,X,F)
```

```
[dFdA,dFdB,dFdC,dFdX,D,Q] = elec_calculateFluxPartialDerivatives(A,B,C,X,F)
```

## Description

`[dFdA,dFdB,dFdC,dFdX] = elec_calculateFluxPartialDerivatives(A,B,C,X,F)` calculates the partial derivatives from flux linkage. For improved numerical performance, the FEM-Parameterized PMSM block works with flux linkage partial derivatives, rather than directly with flux linkage. If your finite-element motor design tool does not have an option to output partial derivatives, then you can use this function to calculate the partial derivatives from the flux linkage. The flux linkage  $F$  must be a four-dimensional matrix with the first three dimensions corresponding to the A, B, and C phase currents, and the fourth dimension corresponding to the rotor angle  $X$ . The function returns four-dimensional matrices for the four partial derivatives. Use this syntax in conjunction with the 4-D Data variant of the block.

`[dFdA,dFdB,dFdC,dFdX,D,Q] = elec_calculateFluxPartialDerivatives(A,B,C,X,F)` returns two additional output arguments corresponding to  $d$ -axis and  $q$ -axis currents, respectively. In this case, the four partial derivatives are three-dimensional, the first two dimensions corresponding to the  $d$ -axis and  $q$ -axis currents, and the third dimension corresponding to the rotor angle. Use this syntax in conjunction with the 3-D Data variant of the block.

## Examples



### Calculate 4-D flux linkage partial derivatives

Suppose  $F$  is a four-dimensional matrix containing flux linkage data, exported by your finite-element motor design tool. The matrix dimensions correspond to the three phase currents and the rotor angle, respectively. The data is cyclical in the fourth dimension, corresponding to the rotor angle.

Either directly import or recreate the current vectors. For example, if recreating a current vector with evenly spaced values between -250 and 250 A and 5 A increments, then:

```
iA = linspace(-250,250,5);
iB = iA;
iC = iA;
```

Import the rotor angle vector or recreate it based on the number of pole pairs.

```
X = pi/180*linspace(0,360/N,180/N+1);
```

Calculate the flux linkage partial derivatives.

```
[dFdA,dFdB,dFdC,dFdX] = elec_calculateFluxPartialDerivatives(iA,iB,iC,X,F);
```

The function returns four 4-D matrices for the flux linkage partial derivatives. The four matrices correspond to the three phase currents and the rotor angle, respectively. The matrix dimensions also correspond to the three phase currents and the rotor angle.

### Calculate 3-D flux linkage partial derivatives

Suppose  $F$  is a four-dimensional matrix containing flux linkage data, exported by your finite-element motor design tool. The matrix dimensions correspond to the three phase currents and the rotor angle, respectively. The data is cyclical in the fourth dimension, corresponding to the rotor angle.

Either directly import or recreate the current vectors. For example, if recreating a current vector with evenly spaced values between -250 and 250 A and 5 A increments, then:

```
iA = linspace(-250,250,5);
iB = iA;
iC = iA;
```

Import the rotor angle vector or recreate it based on the number of pole pairs.

```
X = pi/180*linspace(0,360/N,180/N+1);
```

Calculate the flux linkage partial derivatives.

```
[dFdA,dFdB,dFdC,dFdX,iD,iQ] = elec_calculateFluxPartialDerivatives(iA,iB,iC,X,F);
```

The function returns four 3-D matrices for the flux linkage partial derivatives and two vectors for the *d*-axis and *q*-axis current values. The four matrices correspond to the three phase currents and the rotor angle, respectively. The matrix dimensions correspond to the *d*-axis and *q*-axis currents and the rotor angle.

## Input Arguments

### **A — A-phase current, in amperes**

vector

A-phase current, in amperes, specified as a vector. The vector must be monotonically increasing and two-sided (contain both positive and negative values). Best practice is to include zero current as one of the points.

Data Types: double

### **B — B-phase current, in amperes**

vector

B-phase current, in amperes, specified as a vector. The vector must be monotonically increasing and two-sided (contain both positive and negative values). Best practice is to include zero current as one of the points.

Data Types: double

### **C — C-phase current, in amperes**

vector

C-phase current, in amperes, specified as a vector. The vector must be monotonically increasing and two-sided (contain both positive and negative values). Best practice is to include zero current as one of the points.

Data Types: double

### **X — Rotor angle, in radians**

vector

The rotor angle, in radians, specified as a vector. The values must be in the range from zero to  $2\pi/N$ , where  $N$  is the number of pole pairs.

Data Types: double

### **F — Flux linkage, in weber-turns**

four-dimensional matrix

The flux linkage, in weber-turns, specified as a four-dimensional matrix, with dimensions corresponding to the three phase currents and rotor angle. The data must be cyclic in the fourth (rotor angle) dimension, that is, for all  $i, j$ , and  $k$ ,  $F(i,j,k,0) = F(i,j,k,2\pi/N)$ , where  $N$  is the number of pole pairs.

Data Types: double

## **Output Arguments**

### **dFdA — Flux linkage partial derivative with respect to the A-phase current**

matrix

Flux linkage partial derivative with respect to the A-phase current, returned as a matrix. For syntax used with the 4-D Data variant of the block, the matrix is four-dimensional. For syntax used with the 3-D Data variant of the block, the matrix is three-dimensional, the first two dimensions corresponding to the  $d$ -axis and  $q$ -axis currents, and the third dimension corresponding to the rotor angle.

### **dFdB — Flux linkage partial derivative with respect to the B-phase current**

matrix

Flux linkage partial derivative with respect to the B-phase current, returned as a matrix. For syntax used with the 4-D Data variant of the block, the matrix is four-dimensional. For syntax used with the 3-D Data variant of the block, the matrix is three-dimensional, the first two dimensions corresponding to the  $d$ -axis and  $q$ -axis currents, and the third dimension corresponding to the rotor angle.

### **dFdC — Flux linkage partial derivative with respect to the C-phase current**

matrix

Flux linkage partial derivative with respect to the C-phase current, returned as a matrix. For syntax used with the 4-D Data variant of the block, the matrix is four-dimensional. For syntax used with the 3-D Data variant of the block, the matrix is three-dimensional, the

first two dimensions corresponding to the  $d$ -axis and  $q$ -axis currents, and the third dimension corresponding to the rotor angle.

### **dFdX — Flux linkage partial derivative with respect to the rotor angle**

matrix

Flux linkage partial derivative with respect to the rotor angle, returned as a matrix. For syntax used with the 4-D Data variant of the block, the matrix is four-dimensional. For syntax used with the 3-D Data variant of the block, the matrix is three-dimensional, the first two dimensions corresponding to the  $d$ -axis and  $q$ -axis currents, and the third dimension corresponding to the rotor angle.

### **D — D-axis current, in amperes**

vector

$D$ -axis current, in amperes, returned as a vector. This is an optional output argument, to be used when you want to generate 3-D flux linkage partial derivatives. The vector defines the  $d$ -axis current values at which the partial derivatives are determined.

### **Q — Q-axis current, in amperes**

vector

$Q$ -axis current, in amperes, returned as a vector. This is an optional output argument, to be used when you want to generate 3-D flux linkage partial derivatives. The vector defines the  $q$ -axis current values at which the partial derivatives are determined.

## Algorithms

The function calculates partial derivatives using Akima splines, the same method that is used for smooth interpolation in the Simscape language `tablelookup` function. For more information, see “Smooth Interpolation Algorithm” (Simscape). Akima splines are suitable for estimating partial derivatives due to their smooth nature and tendency not to introduce local gradient reversals.

## See Also

`elec_generateIdealPMSMfluxData`

**Introduced in R2017a**

# elec\_generateIdealPMSMfluxData

Generate tabulated flux linkage data for ideal PMSM

## Syntax

```
[F,T,dFdA,dFdB,dFdC,dFdX] = elec_generateIdealPMSMfluxData(PM,Ld,Lq,
L0,A,B,C,X)
[F,T,dFdA,dFdB,dFdC,dFdX] = elec_generateIdealPMSMfluxData(PM,Ld,Lq,
L0,D,Q,X)
```

## Description

[F,T,dFdA,dFdB,dFdC,dFdX] = elec\_generateIdealPMSMfluxData(PM,Ld,Lq,L0,A,B,C,X) generates 4-D flux linkage data for an ideal permanent magnet synchronous motor (PMSM).

Use this function to create test data for the FEM-Parameterized PMSM block, either for validation purposes or to set up a model before the actual flux linkage data is available.

[F,T,dFdA,dFdB,dFdC,dFdX] = elec\_generateIdealPMSMfluxData(PM,Ld,Lq,L0,D,Q,X) generates 3-D flux linkage data for an ideal PMSM.

## Examples

### Generate 4-D Flux Linkage Data

Specify the motor parameters.

```
PM = 0.1;      % Permanent magnet flux
N = 6;        % Number of pole pairs
Ld = 0.0002;  % D-axis inductance
Lq = 0.0002;  % Q-axis inductance
L0 = 0.00018; % Zero-sequence inductance
Rs = 0.013;   % Stator resistance
```

Define the phase current vectors.

```
iA = linspace(-250,250,5);  
iB = iA;  
iC = iA;
```

Specify the rotor angle vector based on the number of pole pairs.

```
X = pi/180*linspace(0,360/N,180/N+1);
```

Tabulate flux linkage partial derivatives and torque in terms of A-,B-,C-currents and rotor angle.

```
[F,T,dFdA,dFdB,dFdC,dFdX] = elec_generateIdealPMSMfluxData(PM,Ld,Lq,L0,iA,iB,iC,X);
```

The function returns a 4-D flux linkage matrix  $F$ , a 4-D torque matrix  $T$ , and four 4-D matrices for the flux linkage partial derivatives. The four partial derivative matrices correspond to the three phase currents and the rotor angle, respectively. The matrix dimensions correspond to the three phase currents and the rotor angle.

### Generate 3-D Flux Linkage Data

Specify the motor parameters.

```
PM = 0.1;      % Permanent magnet flux  
N = 6;        % Number of pole pairs  
Ld = 0.0002;  % D-axis inductance  
Lq = 0.0002;  % Q-axis inductance  
L0 = 0.00018; % Zero-sequence inductance  
Rs = 0.013;   % Stator resistance
```

Define the  $d$ -axis and  $q$ -axis current vectors.

```
iD = linspace(-250,250,5);  
iQ = iD;
```

Specify the rotor angle vector based on the number of pole pairs.

```
X = pi/180*linspace(0,360/N,180/N+1);
```

Tabulate flux linkage partial derivatives and torque in terms of in terms of  $d$ -axis and  $q$ -axis currents and rotor angle.

```
[F,T,dFdA,dFdB,dFdC,dFdX] = elec_generateIdealPMSMfluxData(PM,Ld,Lq,L0,iD,iQ,X);
```

The function returns a 3-D flux linkage matrix  $F$ , a 3-D torque matrix  $T$ , and four 3-D matrices for the flux linkage partial derivatives. The four partial derivative matrices correspond to the three phase currents and the rotor angle, respectively. The matrix dimensions correspond to the  $d$ -axis and  $q$ -axis currents and the rotor angle.

## Input Arguments

### **PM — Peak permanent magnet flux linkage, in weber-turns**

scalar

Peak permanent magnet flux linkage, in weber-turns, specified as a scalar.

Data Types: double

### **Ld — D-axis inductance, in henries**

scalar

$D$ -axis inductance, in henries, specified as a scalar.

Data Types: double

### **Lq — Q-axis inductance, in henries**

scalar

$Q$ -axis inductance, in henries, specified as a scalar.

Data Types: double

### **L0 — Zero-sequence inductance, in henries**

scalar

Zero-sequence inductance, in henries, specified as a scalar.

Data Types: double

### **A — A-phase current, in amperes**

vector

A-phase current, in amperes, specified as a vector. The vector must be monotonically increasing and two-sided (contain both positive and negative values). Best practice is to include zero current as one of the points. Use this input argument to generate 4-D flux linkage data.

Data Types: double

### **B — B-phase current, in amperes**

vector

B-phase current, in amperes, specified as a vector. The vector must be monotonically increasing and two-sided (contain both positive and negative values). Best practice is to include zero current as one of the points. Use this input argument to generate 4-D flux linkage data.

Data Types: double

### **C — C-phase current, in amperes**

vector

C-phase current, in amperes, specified as a vector. The vector must be monotonically increasing and two-sided (contain both positive and negative values). Best practice is to include zero current as one of the points. Use this input argument to generate 4-D flux linkage data.

Data Types: double

### **D — D-axis current, in amperes**

vector

D-axis current, in amperes, specified as a vector. The vector must be monotonically increasing and two-sided (contain both positive and negative values). Best practice is to include zero current as one of the points. Use this input argument to generate 3-D flux linkage data.

Data Types: double

### **Q — Q-axis current, in amperes**

vector

Q-axis current, in amperes, specified as a vector. The vector must be monotonically increasing and two-sided (contain both positive and negative values). Best practice is to include zero current as one of the points. Use this input argument to generate 3-D flux linkage data.

Data Types: double

### **X — Rotor angle, in radians**

vector



The rotor angle, in radians, specified as a vector. The values must be in the range from zero to  $2\pi/N$ , where  $N$  is the number of pole pairs.

Data Types: double

## Output Arguments

### **F — Flux linkage**

matrix

The flux linkage, in weber-turns, returned as a matrix. The matrix can be four-dimensional or three-dimensional, depending on the syntax used to call the function. In a four-dimensional matrix, the first three dimensions correspond to the phase currents, and the fourth dimension corresponds to the rotor angle. In a three-dimensional matrix, the first two dimensions correspond to the  $d$ -axis and  $q$ -axis currents, and the third dimension corresponds to the rotor angle.

### **T — Torque**

matrix

Torque, in N\*m, returned as a matrix. The matrix can be four-dimensional or three-dimensional, depending on the syntax used to call the function. In a four-dimensional matrix, the first three dimensions correspond to the phase currents, and the fourth dimension corresponds to the rotor angle. In a three-dimensional matrix, the first two dimensions correspond to the  $d$ -axis and  $q$ -axis currents, and the third dimension corresponds to the rotor angle.

### **dFdA — Flux linkage partial derivative with respect to the A-phase current**

matrix

Flux linkage partial derivative with respect to the A-phase current, returned as a matrix. The matrix can be four-dimensional or three-dimensional, depending on the syntax used to call the function. In a four-dimensional matrix, the first three dimensions correspond to the phase currents, and the fourth dimension corresponds to the rotor angle. In a three-dimensional matrix, the first two dimensions correspond to the  $d$ -axis and  $q$ -axis currents, and the third dimension corresponds to the rotor angle.

### **dFdB — Flux linkage partial derivative with respect to the B-phase current**

matrix

Flux linkage partial derivative with respect to the B-phase current, returned as a matrix. The matrix can be four-dimensional or three-dimensional, depending on the syntax used

to call the function. In a four-dimensional matrix, the first three dimensions correspond to the phase currents, and the fourth dimension corresponds to the rotor angle. In a three-dimensional matrix, the first two dimensions correspond to the  $d$ -axis and  $q$ -axis currents, and the third dimension corresponds to the rotor angle.

**dFdC — Flux linkage partial derivative with respect to the C-phase current**

matrix

Flux linkage partial derivative with respect to the C-phase current, returned as a matrix. The matrix can be four-dimensional or three-dimensional, depending on the syntax used to call the function. In a four-dimensional matrix, the first three dimensions correspond to the phase currents, and the fourth dimension corresponds to the rotor angle. In a three-dimensional matrix, the first two dimensions correspond to the  $d$ -axis and  $q$ -axis currents, and the third dimension corresponds to the rotor angle.

**dFdX — Flux linkage partial derivative with respect to the rotor angle**

matrix

Flux linkage partial derivative with respect to the rotor angle, returned as a matrix. The matrix can be four-dimensional or three-dimensional, depending on the syntax used to call the function. In a four-dimensional matrix, the first three dimensions correspond to the phase currents, and the fourth dimension corresponds to the rotor angle. In a three-dimensional matrix, the first two dimensions correspond to the  $d$ -axis and  $q$ -axis currents, and the third dimension corresponds to the rotor angle.

## Algorithms

The flux linking each winding has contributions from the permanent magnet plus the three windings. Therefore, the total flux is given by [1] on page 2-13:

$$\begin{bmatrix} \psi_a \\ \psi_b \\ \psi_c \end{bmatrix} = \begin{bmatrix} L_{aa} & L_{ab} & L_{ac} \\ L_{ba} & L_{bb} & L_{bc} \\ L_{ca} & L_{cb} & L_{cc} \end{bmatrix} \begin{bmatrix} i_a \\ i_b \\ i_c \end{bmatrix} + \begin{bmatrix} \psi_{am} \\ \psi_{bm} \\ \psi_{cm} \end{bmatrix}$$

$$\begin{aligned}
L_{aa} &= L_s + L_m \cos(2\theta_r) \\
L_{bb} &= L_s + L_m \cos(2(\theta_r - 2\pi/3)) \\
L_{cc} &= L_s + L_m \cos(2(\theta_r + 2\pi/3)) \\
L_{ab} &= L_{ba} = -M_s - L_m \cos(\theta_r + \pi/6) \\
L_{bc} &= L_{cb} = -M_s - L_m \cos(\theta_r + \pi/6 - 2\pi/3) \\
L_{ca} &= L_{ac} = -M_s - L_m \cos(\theta_r + \pi/6 + 2\pi/3) \\
\psi_{am} &= \psi_m \cos \theta_e \\
\psi_{bm} &= \psi_m \cos(\theta_e - 2\pi/3) \\
\psi_{cm} &= \psi_m \cos(\theta_e + 2\pi/3)
\end{aligned}$$

Here,  $\theta_e$  is the electrical angle, which is related to rotor angle  $\theta_r$  by  $\theta_e = N \cdot \theta_r$ . The function assumes that the permanent magnet flux linking the A-phase winding is at the maximum for  $\theta_e = 0$ .

The function output  $F$  corresponds to  $\psi_a$  tabulated as a function of A-phase current, B-phase current, C-phase current, and rotor angle.

$L_s$ ,  $L_m$ , and  $M_s$  are related to input arguments  $L_d$ ,  $L_q$ , and  $L_0$  by:

$$\begin{aligned}
L_s &= \frac{L_0}{3} + \frac{L_d}{3} + \frac{L_q}{3} \\
M_s &= \frac{L_d}{6} - \frac{L_0}{3} + \frac{L_q}{6} \\
L_m &= \frac{L_d}{3} - \frac{L_q}{3}
\end{aligned}$$

## References

- [1] Anderson, P.M. *Analysis of Faulted Power Systems*. 1st Edition. Wiley-IEEE Press, July 1995, p.187.

## See Also

elec\_calculateFluxPartialDerivatives

**Topics**

HEV PMSM Drive Test Harness

PMSM Iron Losses

**Introduced in R2017a**

## elec\_getEfficiency

Calculate efficiency as function of dissipated power losses

### Syntax

```
efficiency = elec_getEfficiency('loadIdentifier',node)
efficiency = elec_getEfficiency('loadIdentifier',node,startTime,
endTime)
[efficiency,lossesTable] = elec_getEfficiency('loadIdentifier',node)
```

### Description

`efficiency = elec_getEfficiency('loadIdentifier',node)` returns the efficiency of a circuit based on the data extracted from a Simscape logging node.

Before you call this function, you must have the simulation log variable in your current workspace. Create the simulation log variable by simulating the model with data logging turned on, or load a previously saved variable from a file. If `node` is the name of the simulation log variable, then the table contains the data for all semiconductor blocks in the model. If `node` is the name of a node in the simulation data tree, then the table contains the data only for the blocks within that node.

Checking efficiency allows you to determine if circuit components are operating within their requirements. All blocks in the Semiconductor Devices library, as well as some other blocks, have an internal variable called `power_dissipated`, which represents the instantaneous power dissipated by the block. This instantaneous dissipated power includes only the real power (not the reactive or apparent power) that the block dissipates. When you log simulation data, the time-value series for this variable represents the power dissipated by the block over time. You can view and plot this data using the Simscape Results Explorer. The `elec_getPowerLossTimeSeries` function also allows you to access this data.

The `elec_getEfficiency` function calculates the efficiency of the circuit based on the losses for blocks that have a `power_dissipated` variable and that you identify as a load block. The equation for efficiency is

$$Eff = 100 \cdot \frac{P_{load}}{P_{loss} + P_{load}},$$

where:

- $Eff$  is the efficiency of the circuit.
- $P_{load}$  is the output power, that is, the power dissipated by load blocks.
- $P_{loss}$  is the power dissipated by nonload blocks.

This equation assumes that all loss mechanisms are captured by blocks containing at least one `power_dissipated` variable. If the model contains any lossy blocks that do not have this variable, the efficiency calculation gives incorrect results.

Some blocks have more than one `power_dissipated` variable, depending on their configuration. For example, the N-Channel MOSFET block has separate `power_dissipated` logging nodes for the MOSFET, the gate resistor, and for the source and drain resistors if they have nonzero resistance values. The function sums all these losses to provide the total power loss for the block, averaged over simulation time. The function uses the loss data to calculate the efficiency of the circuit.

`efficiency = elec_getEfficiency('loadIdentifier',node,startTime, endTime)` returns the efficiency of a circuit based on the `power_dissipated` data extracted from a Simscape logging node within a time interval. `startTime` and `endTime` represent the start and end of the time interval for calculating the efficiency. If you omit these two input arguments, the function calculates the efficiency over the whole simulation time.

`[efficiency,lossesTable] = elec_getEfficiency('loadIdentifier',node)` returns the efficiency of a circuit and the power loss contributions of the nonload blocks in a circuit based on the data extracted from a Simscape logging node.

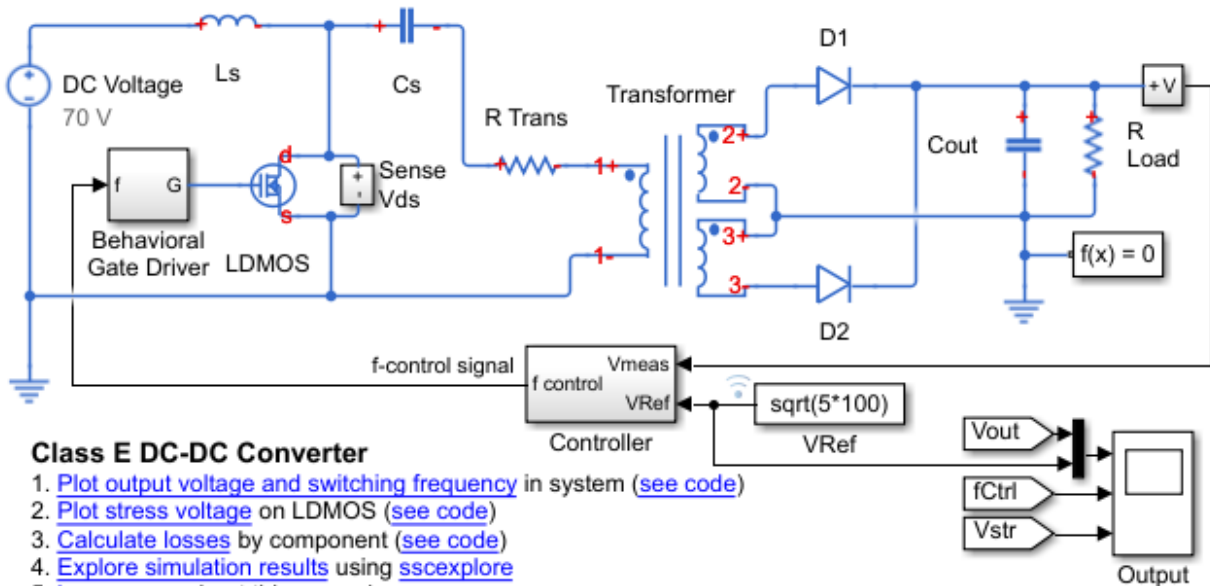
## Examples

### Calculate Efficiency for a Circuit

This example shows how to calculate efficiency based on the power dissipated by blocks in a circuit using the `elec_getEfficiency` function.

Open the model. At the MATLAB® command prompt, enter:

```
model = 'elec_converter_dcdc_class_e';
open_system(model)
```



The load in the model is represented by the R Load resistor. No other blocks with power\_dissipated variables contain 'Load' in their names. Therefore, you can use the string 'Load' as the 'loadIdentifier' argument.

If no string at least partially matches the names of all load blocks in your circuit, rename the load blocks using a schema that satisfies the matching criteria for the 'loadIdentifier' argument.

This example model has data logging enabled. Run the simulation and create the simulation log variable.

```
sim(model)
```

The simulation log variable simlog\_elec\_converter\_dcdc\_class\_e is saved in your current workspace.

Calculate efficiency and display the results.

```
efficiency = elec_getEfficiency('Load',simlog_elec_converter_dc_dc_class_e)
```

```
efficiency =
```

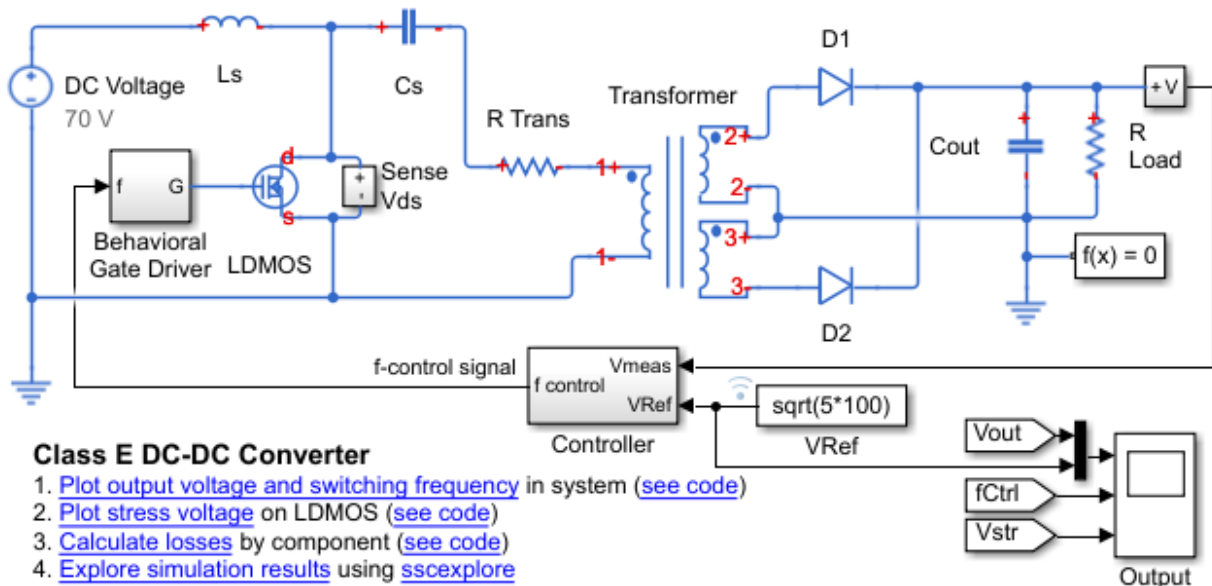
```
90.0326
```

### Calculate Efficiency of a Circuit for a Specific Time Period

This example shows how to calculate efficiency based on the power dissipated for a specific time period using the `elec_getEfficiency` function.

Open the model. At the MATLAB® command prompt, enter:

```
model = 'elec_converter_dc_dc_class_e';
open_system(model)
```





The load in the model is represented by the R Load resistor. No other blocks with `power_dissipated` variables contain 'Load' in their names. Therefore, you can use the string 'Load' as the 'loadIdentifier' argument.

If no string at least partially matches the names of all load blocks in your circuit, rename the load blocks using a schema that satisfies the matching criteria for the 'loadIdentifier' argument.

This example model has data logging enabled. Run the simulation and create the simulation log variable.

```
sim(model)
```

The simulation log variable `simlog_elec_converter_dcdc_class_e` is saved in your current workspace.

The model simulation time ( $t$ ) is  $1.25e-4$  seconds. Calculate efficiency for the interval when  $t$  is between  $1e-4$  and  $1.25e-4$  seconds.

```
efficiency = elec_getEfficiency('Load',simlog_elec_converter_dcdc_class_e,1e-4,1.25e-4)
```

```
efficiency =
```

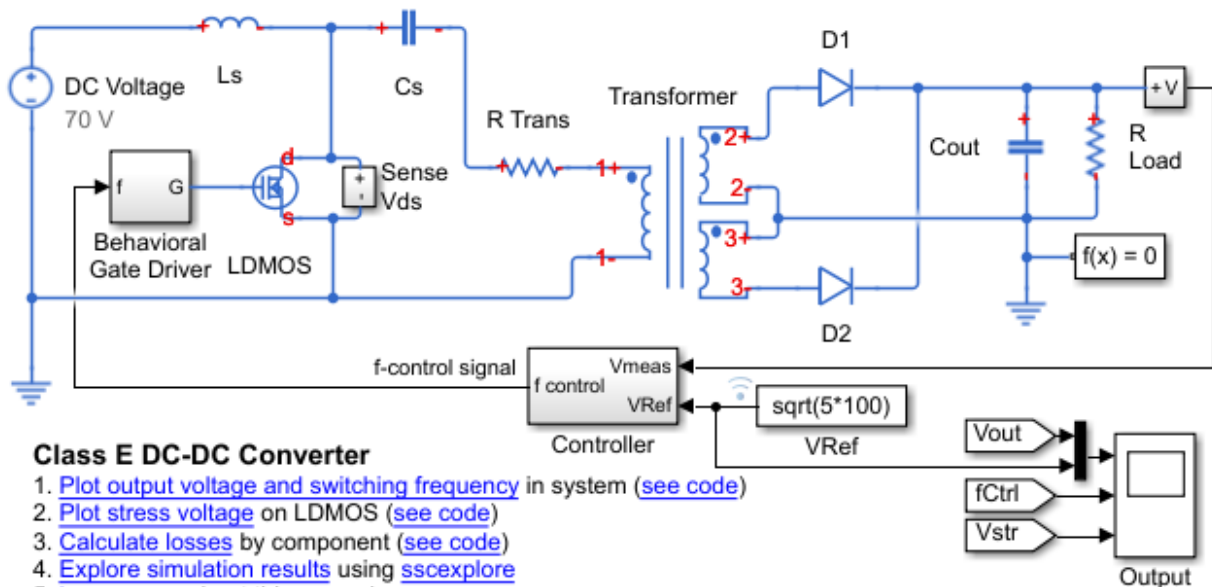
```
    90.4899
```

## Calculate Efficiency and Power-Loss Contributions

This example shows how using the `elec_getEfficiency` function allows you to calculate both the efficiency of the circuit and the power-loss contributions of the nonload blocks based on the power that they dissipate.

Open the model. At the MATLAB® command prompt, enter:

```
model = 'elec_converter_dcdc_class_e';  
open_system(model)
```



The load in the model is represented by the R Load resistor. No other blocks with power\_dissipated variables contain 'Load' in their names. Therefore, you can use the string 'Load' as the 'loadIdentifier' argument.

If no string at least partially matches the names of all load blocks in your circuit, rename the load blocks using a schema that satisfies the matching criteria for the 'loadIdentifier' argument.

This example model has data logging enabled. Run the simulation and create the simulation log variable.

```
sim(model)
```

The simulation log variable `simlog_elec_converter_dcdc_class_e` is saved in your current workspace.

Calculate the efficiency and power-loss contributions due to dissipated power.

```
[efficiency, lossesTable] = elec_getEfficiency('Load', simlog_elec_converter_dcdc_class_e)
```

```
efficiency =
```

```
90.0326
```

```
lossesTable =
```

```
7x2 table array
```

LoggingNode	Power
'elec_converter_dcdc_class_e.LDMOS'	3.6583
'elec_converter_dcdc_class_e.R_Trans.Resistor'	2.911
'elec_converter_dcdc_class_e.D2.diode'	1.9446
'elec_converter_dcdc_class_e.D1.diode'	1.837
'elec_converter_dcdc_class_e.Cs'	0.27391
'elec_converter_dcdc_class_e.Ls'	0.27097
'elec_converter_dcdc_class_e.Cout'	0.00044593

## Input Arguments

### 'loadIdentifier' — Identify load blocks in the circuit

case-sensitive string

String that is a complete or partial match for the names of load blocks in the circuit. For example, consider a circuit that contains the four semiconductor blocks shown in the table.

Block Name in the Model	IGBT	IGBT1_Load	Diode	Diode1	
<b>Block Type</b>	N-Channel IGBT	N-Channel IGBT	Diode	Diode	
<b>Block Role in the Model</b>	Source	Load	Load	Load	
<b>'loadIdentifier'</b>	'IGBT'	Yes	Yes	No	No
	'Diode'	No	No	Yes	Yes
	'Load'	No	Yes	No	No
	'1'	No	Yes	No	Yes
	'D'	No	No	Yes	Yes
	'd'	No	Yes	Yes	Yes

The `elec_getEfficiency` function returns data just for the three load blocks only when the `'loadIdentifier'` is `'d'`.

A load-block naming schema that gives you better control over the output of the `elec_getEfficiency` function is shown in this table.

Block Name in the Model	IGBT	IGBT1_Load	Diode_Load	Diode1_Load	
<b>Block Type</b>	N-Channel IGBT	N-Channel IGBT	Diode	Diode	
<b>Block Role in the Model</b>	Source	Load	Load	Load	
<b>'loadIdentifier'</b>	'IGBT'	Yes	Yes	No	No
	'Diode'	No	No	Yes	Yes
	'Load'	No	Yes	Yes	Yes

Example: `'Load'`

Data Types: `string`

**node** — Simulation log variable, or a specific node within the simulation log variable

Node object

Simulation log workspace variable, or a node within this variable, that contains the logged model simulation data, specified as a `Node` object. You specify the name of the simulation log variable by using the **Workspace variable name** parameter on the **Simscape** pane of the Configuration Parameters dialog box. To specify a node within the simulation log variable, provide the complete path to that node through the simulation data tree, starting with the top-level variable name.

If `node` is the name of the simulation log variable, then the table contains the data for all blocks in the model that contain `power_dissipated` variables. If `node` is the name of a node in the simulation data tree, then the table contains the data only for:

- Blocks or variables within that node
- Blocks or variables within subnodes at all levels of the hierarchy beneath that node

Example: `simlog.Cell1.MOS1`

**startTime** — Start of the time interval for calculating the efficiency

0 (default) | real number

Start of the time interval for calculating the efficiency, specified as a real number, in seconds. `startTime` must be greater than or equal to the simulation **Start time** and less than `endTime`.

Data Types: `double`

### **endTime** — End of the time interval for calculating the efficiency

simulation stop time (default) | real number

End of the time interval for calculating the efficiency, specified as a real number, in seconds. `endTime` must be greater than `startTime` and less than or equal to the simulation **Stop time**.

Data Types: `double`

## Output Arguments

### **efficiency** — Efficiency of the circuit

percentage

Efficiency of the circuit based on data extracted from a Simscape logging node.

### **lossesTable** — Dissipated power for each nonload blocks

table

Dissipated power losses for each nonload block, returned as a table. The first column lists logging nodes for all blocks that have at least one `power_dissipated` variable. The second column lists the corresponding losses in watts.

## Assumptions

- The output power equals the total power dissipated by blocks that you identify as load blocks.
- The input power equals the output power plus the total power dissipated by blocks that you do not identify as load blocks.
- The `power_dissipated` variables capture all loss contributions.

## **See Also**

`elec_getPowerLossSummary` | `elec_getPowerLossTimeSeries`

## **Topics**

“About Simulation Data Logging” (Simscape)

“About the Simscape Results Explorer” (Simscape)

**Introduced in R2017a**

## elec\_getNodeDvDtSummary

Calculate maximum absolute values of terminal voltage time derivatives (dv/dt) based on logged simulation data

### Syntax

```
summaryTable = elec_getNodeDvDtSummary(node, tau)
summaryTable = elec_getNodeDvDtSummary(node, tau, startTime, endTime)
```

### Description

`summaryTable = elec_getNodeDvDtSummary(node, tau)` calculates the maximum absolute values of rates-of-change of voltage variables for nodes that are based on the `foundation.electrical.electrical` domain, based on logged simulation data. The function returns the data for each terminal in a table. The data in the table appears in descending order according to the maximum magnitude of the rate-of-change of voltage variables with respect to the ground, over the whole simulation time. The table does not contain data for terminals that are held fixed.

Before you call this function, you must have the simulation log variable in your current workspace. Create the simulation log variable by simulating the model with data logging turned on, or load a previously saved variable from a file. If `node` is the name of the simulation log variable, then the table contains the data for all the blocks in the model that have nodes based on the `foundation.electrical.electrical` domain. If `node` is the name of a node in the simulation data tree, then the table contains the data only for the children of that node.

Examining rates-of-change of voltage variables in power electronics circuits is useful for determining the potential for unwanted conducted or radiated emissions. The rate-of-change data also helps you to identify switching devices that might be susceptible to parasitic turn-on. All nodes that are based on the `foundation.electrical.electrical` domain store the potential with respect to electrical ground as the variable `v`. When you log simulation data, the time-value series for this variable represents the trend of the potential over time. You can view and plot this data using the Simscape Results Explorer.

To evaluate the rates-of-change of voltage variables, the `elec_getNodeDvDtSummary` function employs finite difference approximation of the first derivative with respect to time. It performs 1-D data linear interpolation of voltage variables using a uniform grid with the time step, `tau`. The function then applies the central differencing scheme to the interpolated data.

---

**Tip** For small time steps, finite differencing may lead to inaccurate results. The time step `tau` should be small enough to capture waveforms, but not so small that the finite differencing error becomes large. For example, for power transistors with an expected limit of 50 V/ns for their voltage rate-of-change, a reasonable guess for `tau` is 1e-9 s.

---

`summaryTable = elec_getNodeDvDtSummary(node, tau, startTime, endTime)` calculates the maximum absolute values of rates-of-change of voltage variables within a time interval. `startTime` and `endTime` represent the start and end of the time interval for evaluating the maximum values. If you omit these two input arguments, the function evaluates the maximum absolute values of rates-of-change of voltage variables over the whole simulation time.

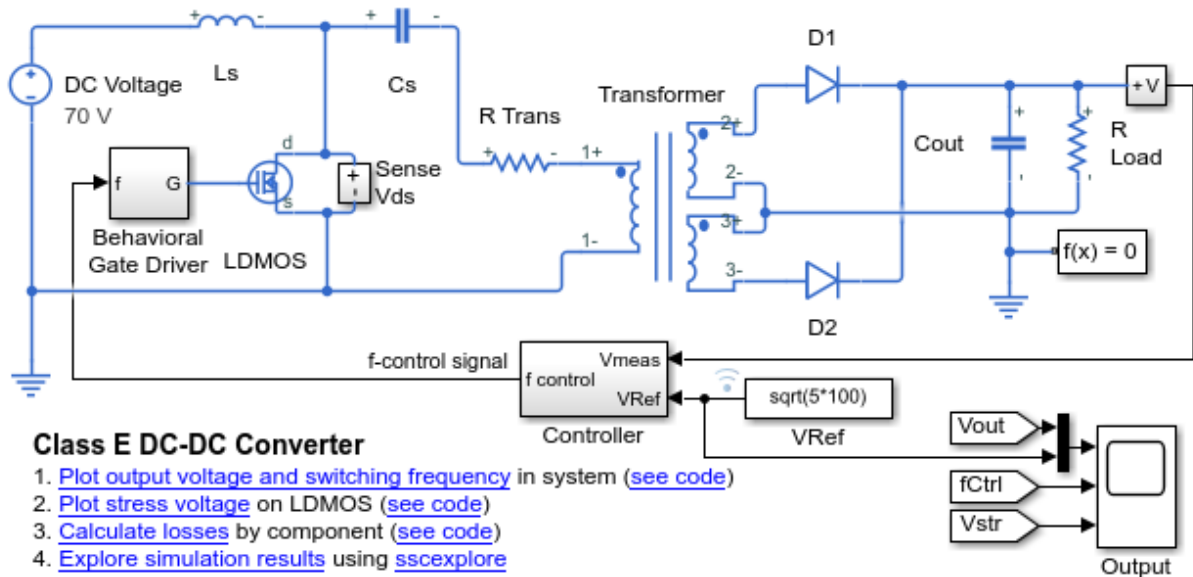
## Examples

### Calculate Maximum Voltage Derivatives by Block for the Whole Model

Open the Class E DC-DC Converter example model.

```
open_system('elec_converter_dcdc_class_e')
```





This example model has data logging enabled. Run the simulation to create the simulation log variable `simlog_elec_converter_dcdc_class_e` in your current workspace.

```
sim('elec_converter_dcdc_class_e');
```

Calculate the maximum absolute values of rates-of-change of voltage variables for the whole model with a time step of  $1e-9$  seconds, and display the results in a table.

```
summaryTable = elec_getNodeDvDtSummary(simlog_elec_converter_dcdc_class_e,1e-9)
```

```
summaryTable =
```

```
19x3 table
```

```
LoggingNode
```

```
"elec_converter_dcdc_class_e.R_Trans"
"elec_converter_dcdc_class_e.Transformer"
"elec_converter_dcdc_class_e.Cs"
```

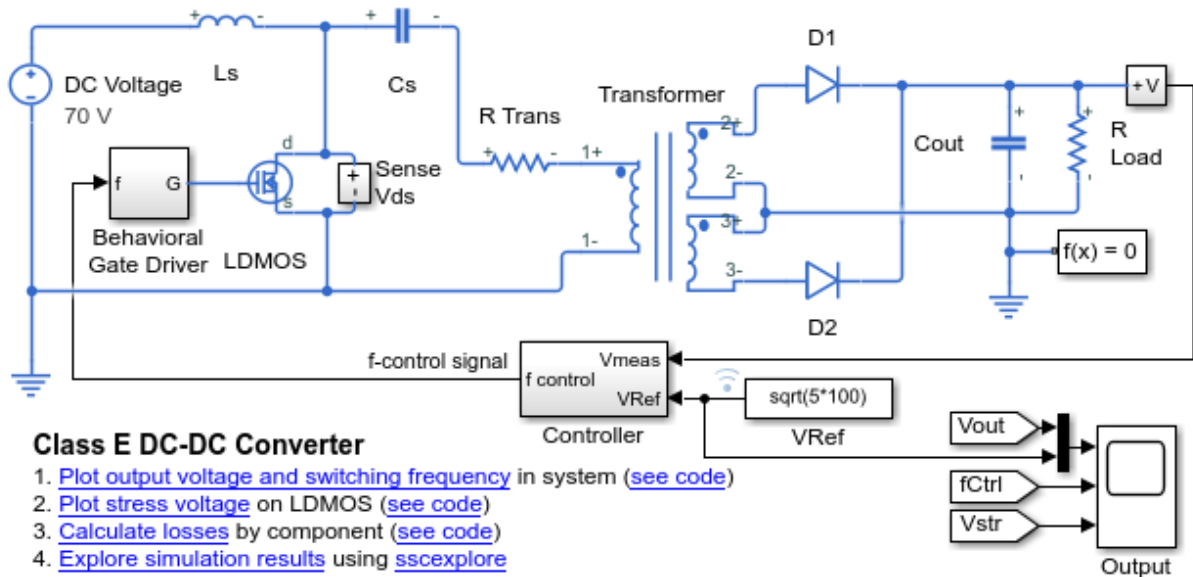
```
"elec_converter_dcdc_class_e.R_Trans"  
"elec_converter_dcdc_class_e.Cs"  
"elec_converter_dcdc_class_e.LDMOS"  
"elec_converter_dcdc_class_e.Ls"  
"elec_converter_dcdc_class_e.Sense_Vds.Voltage_Stress_Sensor"  
"elec_converter_dcdc_class_e.D2"  
"elec_converter_dcdc_class_e.Transformer"  
"elec_converter_dcdc_class_e.D1"  
"elec_converter_dcdc_class_e.Transformer"  
"elec_converter_dcdc_class_e.Behavioral_Gate_Driver.Controlled_Voltage_Source"  
"elec_converter_dcdc_class_e.LDMOS"  
"elec_converter_dcdc_class_e.Cout"  
"elec_converter_dcdc_class_e.D1"  
"elec_converter_dcdc_class_e.D2"  
"elec_converter_dcdc_class_e.R_Load"  
"elec_converter_dcdc_class_e.Sense_Vout.Voltage_Sensor"
```

The table shows the maximum absolute values over the whole simulation time of voltage rates-of-change for all the blocks in the model that have nodes based on the foundation.electrical.electrical domain.

### **Calculate Maximum Voltage Derivatives for One Block**

Open the Class E DC-DC Converter example model.

```
open_system('elec_converter_dcdc_class_e')
```



This example model has data logging enabled. Run the simulation to create the simulation log variable `simlog_elec_converter_dcdc_class_e` in your current workspace.

```
sim('elec_converter_dcdc_class_e');
```

Calculate the maximum absolute values of rates-of-change of voltage variables for the LDMOS block with a time step of  $1e-9$  seconds, and display the results in a table.

```
mosfetTable = elec_getNodeDvDtSummary(simlog_elec_converter_dcdc_class_e.LDMOS,1e-9)
```

```
mosfetTable =
```

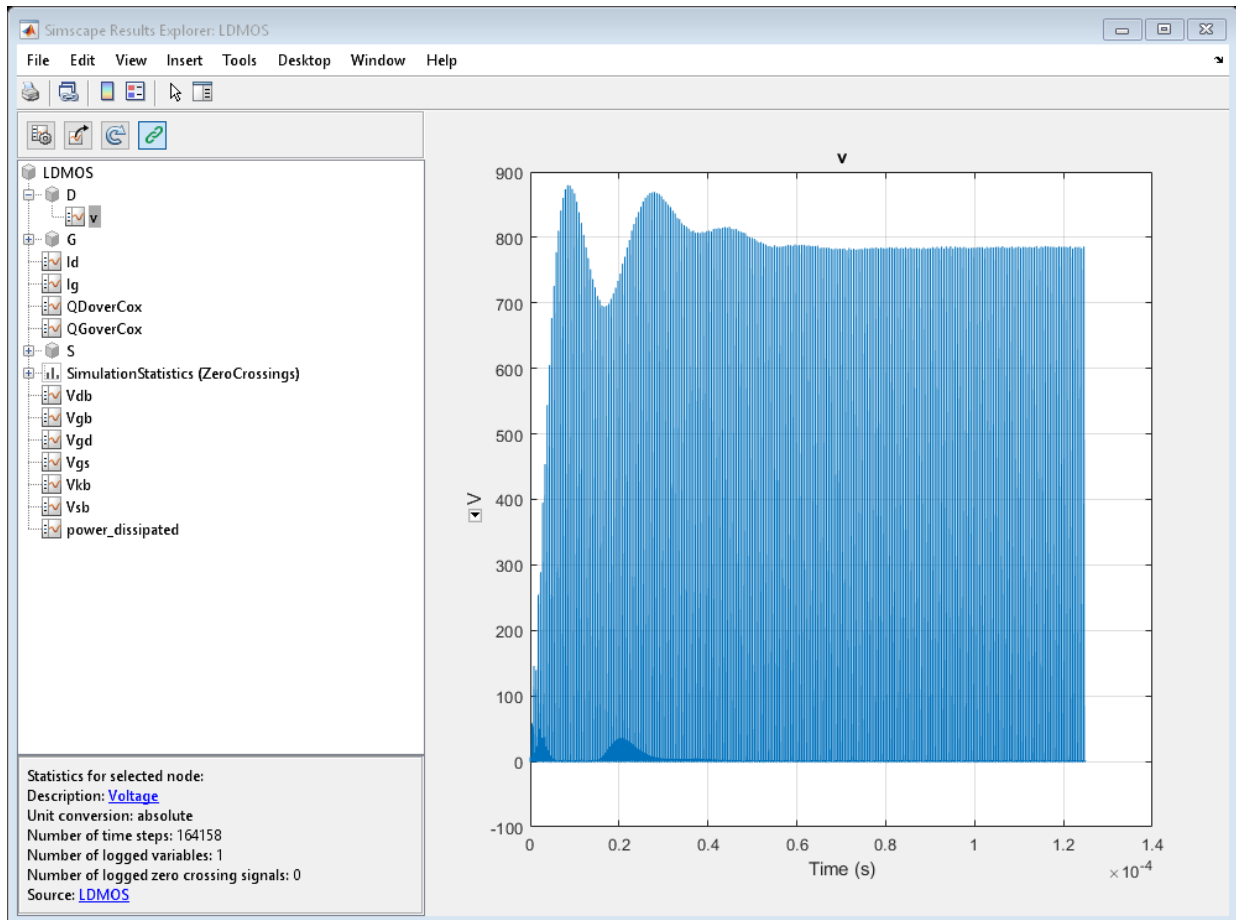
```
2x3 table
```

LoggingNode	Terminal	max_abs_dvdt
"LDMOS"	"D"	3.344e+10
"LDMOS"	"G"	1e+09

The table shows the maximum absolute values over the whole simulation time of voltage rates-of-change for the LDMOS block. The table does not list the S terminal because it is held fixed to the ground.

To explore the voltage data for the LDMOS block further, use the `sscexplore` function.

```
sscexplore(simlog_elec_converter_dcdc_class_e.LDMOS, 'D.v')
```

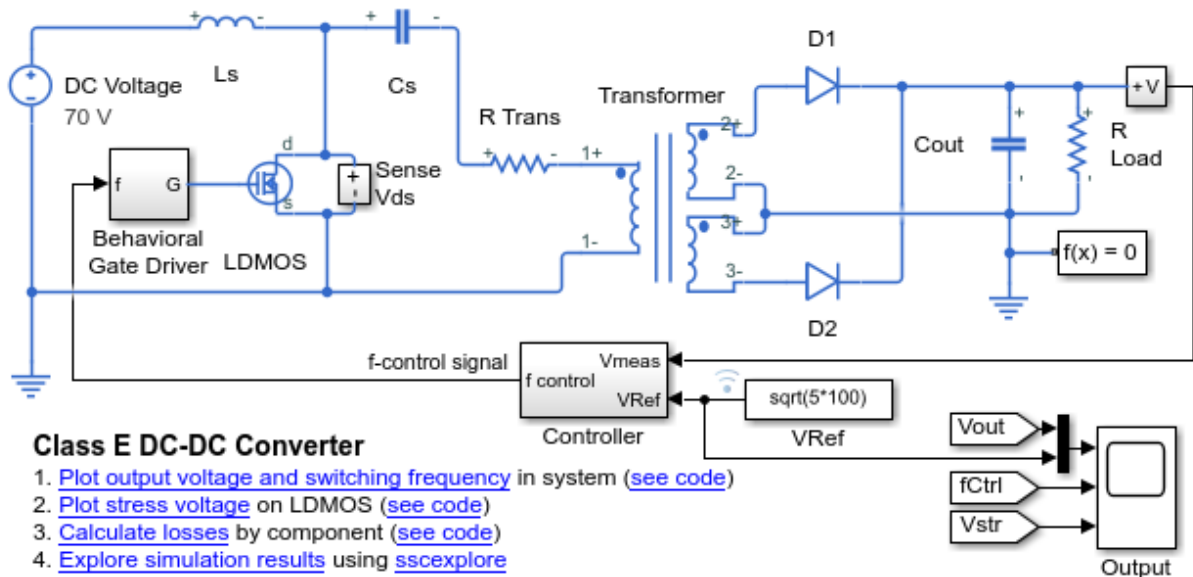


The block has a variable,  $v$ , for each of the D, G, and S terminals.

## Calculate Maximum Voltage Derivatives for a Specific Time Period

Open the Class E DC-DC Converter example model.

```
open_system('elec_converter_dcdc_class_e')
```



This example model has data logging enabled. Run the simulation to create the simulation log variable `simlog_elec_converter_dcdc_class_e` in your current workspace.

```
sim('elec_converter_dcdc_class_e');
```

The model simulation time is  $1.25e-4$  seconds. Calculate and display the maximum absolute values of rates-of-change of voltage variables for the LDMOS block during the second half of the simulation. Use a time step of  $1e-9$  seconds.

```
mosfetTable1 = elec_getNodeDvDtSummary(simlog_elec_converter_dcdc_class_e.LDMOS,1e-9,0
```

```
mosfetTable1 =
```

```
2x3 table
```

LoggingNode	Terminal	max_abs_dvdt
"LDMOS"	"D"	2.8464e+10
"LDMOS"	"G"	1e+09

The table shows the maximum absolute values of voltage rates-of-change for the LDMOS block during the second half of the simulation. The table does not list the S terminal because it is held fixed to the ground. The magnitude of the D terminal is lower than the magnitude for the whole simulation time because the initial high-magnitude spikes of the stress voltage are disregarded.

To see the voltage time derivative for the D terminal of the LDMOS block over the whole simulation time, use the `elec_getNodeDvDtTimeSeries` function.

## Input Arguments

**node** — Simulation log variable, or a specific node within the simulation log variable

Node object

Simulation log workspace variable, or a node within this variable, that contains the logged model simulation data, specified as a Node object. You specify the name of the simulation log variable by using the **Workspace variable name** parameter on the **Simscape** pane of the Configuration Parameters dialog box. To specify a node within the simulation log variable, provide the complete path to that node through the simulation data tree, starting with the top-level variable name.

Example: `simlog_elec_converter_dcdc_class_e.LDMOS`

**tau** — Time step for numerical differentiation

real number

Time step for numerical differentiation, specified as a real number, in seconds. tau determines the interpolation grid as `startTime:tau:endTime`.

Example: `1e-9`

Data Types: double

**startTime — Start of the time interval for evaluating the maximum voltage rates-of-change**

simulation start time (default) | real number

Start of the time interval for evaluating the maximum absolute values of rates-of-change of voltage variables, specified as a real number, in seconds. `startTime` must be greater than or equal to the simulation **Start time** and less than `endTime`.

Data Types: double

**endTime — End of the time interval for evaluating the maximum voltage rates-of-change**

simulation stop time (default) | real number

End of the time interval for evaluating the maximum absolute values of rates-of-change of voltage variables, specified as a real number, in seconds. `endTime` must be greater than `startTime` and less than or equal to the simulation **Stop time**.

Data Types: double

## Output Arguments

**summaryTable — Maximum absolute values of the voltage rates-of-change for each block**

table

Maximum absolute values of the voltage rates-of-change for each block, returned as a table. The first column lists all the logging nodes in node that are based on the `foundation.electrical.electrical` domain. The second column lists the terminal names. The third column lists the corresponding maximum absolute values of voltage rates-of-change, in volts per second. The table does not contain data for terminals that are held fixed.

## See Also

`elec_getNodeDvDtTimeSeries` | `sscexplore`

## Topics

“About Simulation Data Logging” (Simscape)

“About the Simscape Results Explorer” (Simscape)

**Introduced in R2018b**



# elec\_getNodeDvDtTimeSeries

Calculate rates-of-change of voltage variables

## Syntax

```
seriesTable = elec_getNodeDvDtTimeSeries(node,tau)
seriesTable = elec_getNodeDvDtTimeSeries(node,tau,startTime,endTime)
```

## Description

`seriesTable = elec_getNodeDvDtTimeSeries(node,tau)` calculates rates-of-change of voltage variables for nodes that are based on the `foundation.electrical.electrical` domain, based on logged simulation data. The function returns the data for each terminal in a table. The data in the table appears in descending order according to the maximum absolute value of the rate-of-change of voltage variables with respect to the ground, over the whole simulation time. The table does not contain data for terminals that are held fixed.

Before you call this function, you must have the simulation log variable in your current workspace. Create the simulation log variable by simulating the model with data logging turned on, or load a previously saved variable from a file. If `node` is the name of the simulation log variable, then the table contains the data for all the blocks in the model that have nodes based on the `foundation.electrical.electrical` domain. If `node` is the name of a node in the simulation data tree, then the table contains the data only for the children of that node.

Examining rates-of-change of voltage variables in power electronics circuits is useful for determining the potential for unwanted conducted or radiated emissions. The rate-of-change data also helps you to identify unwanted turn-on of switching devices. All nodes that are based on the `foundation.electrical.electrical` domain store the potential with respect to electrical ground as the variable `v`. When you log simulation data, the time-value series for this variable represents the trend of the potential over time. You can view and plot this data using the Simscape Results Explorer.

To evaluate the rates-of-change of voltage variables, the `elec_getNodeDvDtTimeSeries` function employs finite difference approximation of the

first derivative with respect to time. It performs 1-D data linear interpolation of voltage variables using a uniform grid with the time step, `tau`. The function then applies the central differencing scheme to the interpolated data.

---

**Tip** For small time steps, finite differencing may lead to inaccurate results. The time step `tau` should be small enough to capture waveforms, but not so small that the finite differencing error becomes large. For example, for power transistors with an expected limit of 50 V/ns for their voltage rate-of-change, a reasonable guess for `tau` is 1e-9 s.

---

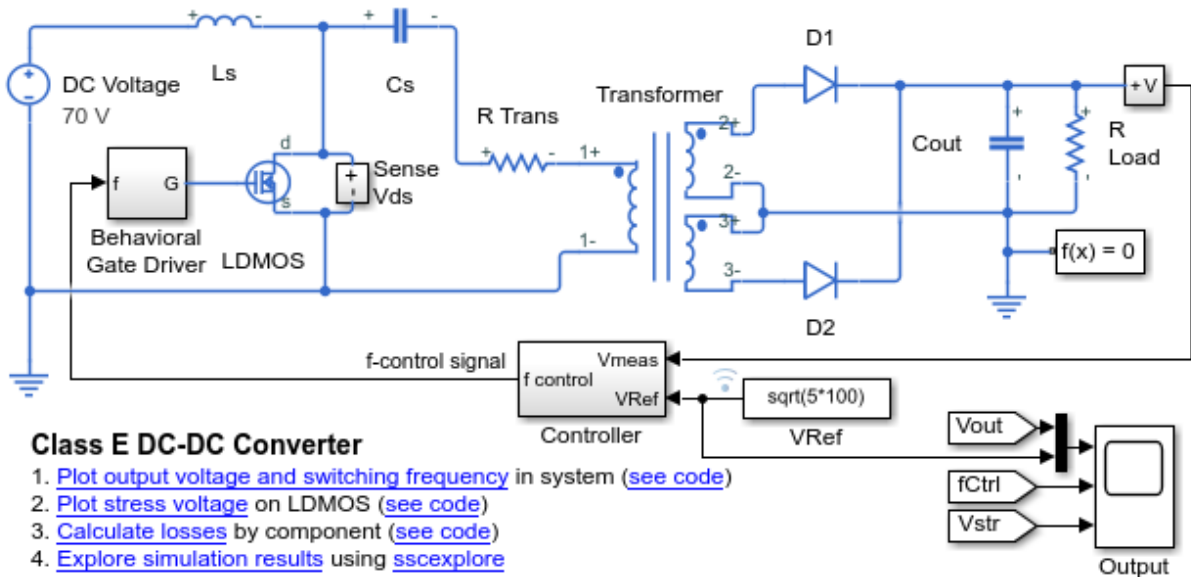
`seriesTable = elec_getNodeDvDtTimeSeries(node, tau, startTime, endTime)` calculates rates-of-change of voltage variables within a time interval. `startTime` and `endTime` represent the start and end of the time interval for evaluating the derivatives of the voltage variables with respect to time. If you omit these two input arguments, the function evaluates rates-of-change of voltage variables over the whole simulation time.

## Examples

### Calculate Voltage Derivatives by Block for the Whole Model

Open the Class E DC-DC Converter example model.

```
open_system('elec_converter_dcdc_class_e')
```



This example model has data logging enabled. Run the simulation to create the simulation log variable `simlog_elec_converter_dcdc_class_e` in your current workspace.

```
sim('elec_converter_dcdc_class_e');
```

Calculate rates-of-change of voltage variables for the whole model with a time step of  $1e-9$  seconds, and return the time series data in a table.

```
seriesTable = elec_getNodeDvDtTimeSeries(simlog_elec_converter_dcdc_class_e,1e-9)
```

```
seriesTable =
```

```
19x4 table
```

```
LoggingNode
```

```
"elec_converter_dcdc_class_e.R_Trans"
"elec_converter_dcdc_class_e.Transformer"
"elec_converter_dcdc_class_e.Cs"
```

```
"elec_converter_dcdc_class_e.R_Trans"  
"elec_converter_dcdc_class_e.Cs"  
"elec_converter_dcdc_class_e.LDMOS"  
"elec_converter_dcdc_class_e.Ls"  
"elec_converter_dcdc_class_e.Sense_Vds.Voltage_Stress_Sensor"  
"elec_converter_dcdc_class_e.D2"  
"elec_converter_dcdc_class_e.Transformer"  
"elec_converter_dcdc_class_e.D1"  
"elec_converter_dcdc_class_e.Transformer"  
"elec_converter_dcdc_class_e.Behavioral_Gate_Driver.Controlled_Voltage_Source"  
"elec_converter_dcdc_class_e.LDMOS"  
"elec_converter_dcdc_class_e.Cout"  
"elec_converter_dcdc_class_e.D1"  
"elec_converter_dcdc_class_e.D2"  
"elec_converter_dcdc_class_e.R_Load"  
"elec_converter_dcdc_class_e.Sense_Vout.Voltage_Sensor"
```

The table contains time series data of voltage variables and their first derivatives over the whole simulation time for all the blocks in the model that have nodes based on the foundation.electrical.electrical domain.

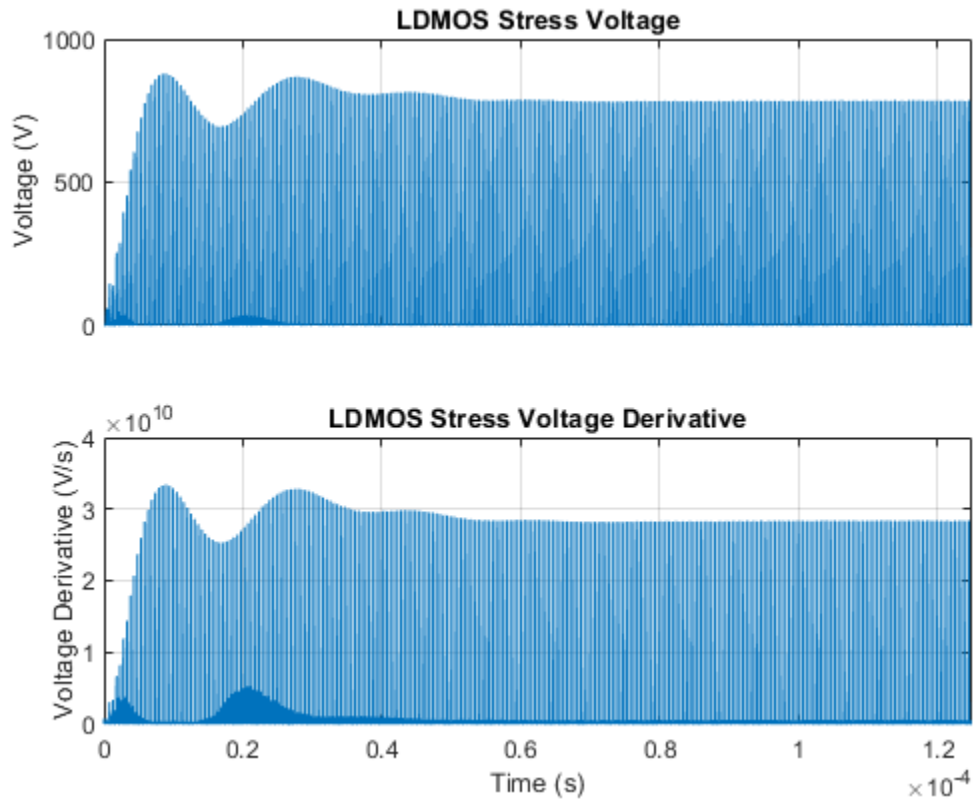
View the time series data. From the workspace, open the `seriesTable` table, then open the two `1x125001 double numeric` arrays for the `elec_converter_dcdc_class_e.LDMOS.D`.

The first array contains the voltage data. The second array contains the voltage derivative data.

Plot the data.

```
time = 0:1e-9:1.25e-4;  
vOut = seriesTable.Voltage{6};  
dvdtOut = seriesTable.dvdt{6};  
  
ax1 = subplot(2,1,1);  
plot(time,vOut),grid;  
ylabel('Voltage (V)');  
axis([0 1.25e-4 0 1000]);  
ax1.XTickLabel = {};  
ax1.Title.String = 'LDMOS Stress Voltage';  
  
ax2 = subplot(2,1,2);  
plot(time,dvdtOut),grid;
```

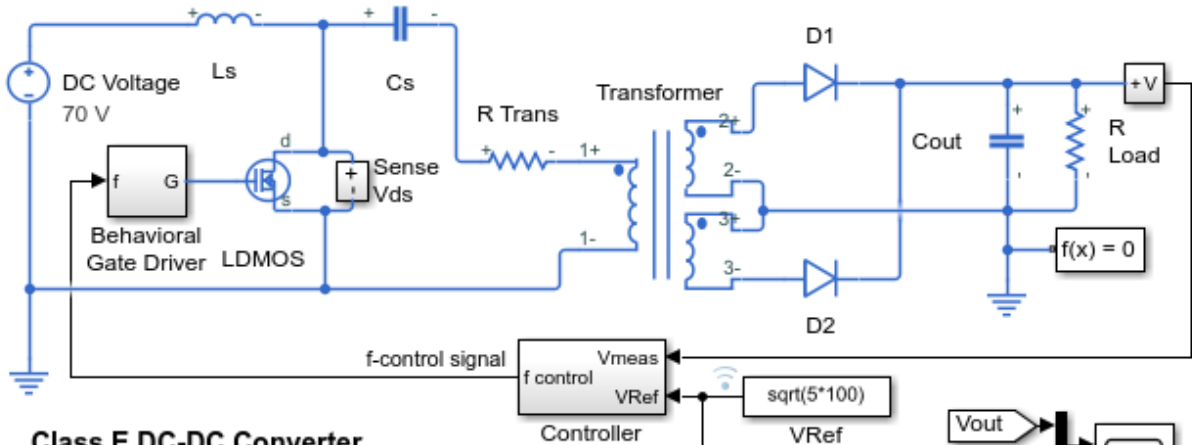
```
ylabel('Voltage Derivative (V/s)');  
xlabel('Time (s)');  
axis([0 1.25e-4 0 4e10]);  
ax2.Title.String = 'LDMOS Stress Voltage Derivative';
```



### Calculate Voltage Derivatives for One Block

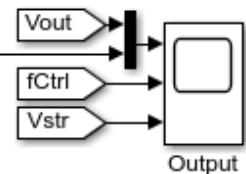
Open the Class E DC-DC Converter example model.

```
open_system('elec_converter_dcdc_class_e')
```



### Class E DC-DC Converter

1. [Plot output voltage and switching frequency](#) in system ([see code](#))
2. [Plot stress voltage](#) on LDMOS ([see code](#))
3. [Calculate losses](#) by component ([see code](#))
4. [Explore simulation results](#) using [sscexplore](#)
5. [Learn more](#) about this example



This example model has data logging enabled. Run the simulation to create the simulation log variable `simlog_elec_converter_dcdc_class_e` in your current workspace.

```
sim('elec_converter_dcdc_class_e');
```

Calculate rates-of-change of voltage variables for the LDMOS block with a time step of  $1e-9$  seconds, and return the time series data in a table.

```
mosfetTable = elec_getNodeDvDtTimeSeries(simlog_elec_converter_dcdc_class_e.LDMOS,1e-9)
```

```
mosfetTable =
```

2x4 table

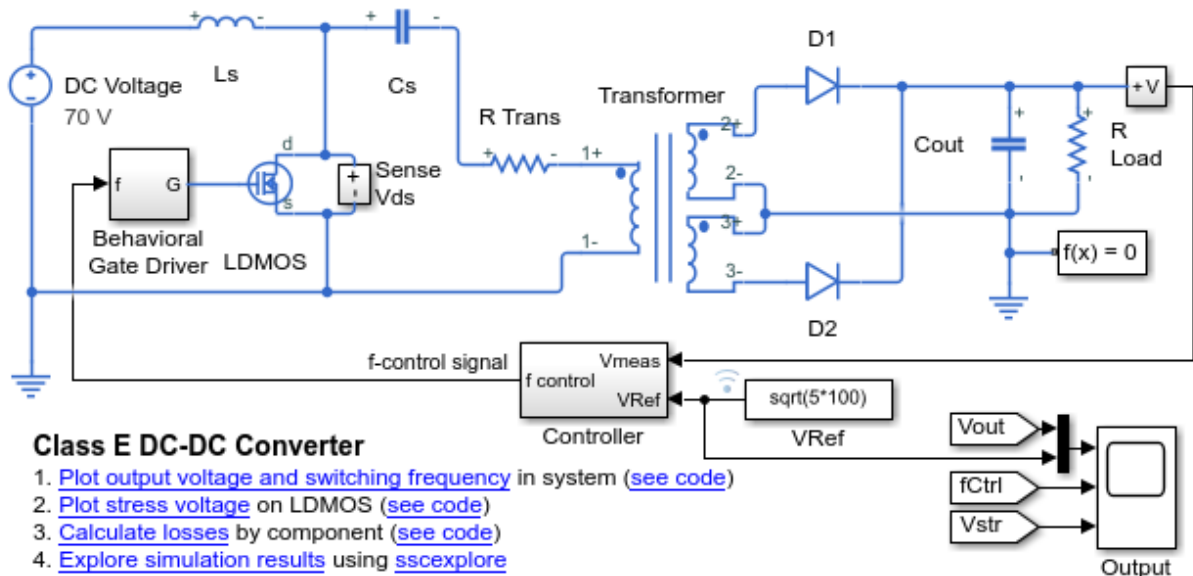
LoggingNode	Terminal	Voltage	dvdt
"LDMOS"	"D"	[1x125001 double]	[1x125001 double]
"LDMOS"	"G"	[1x125001 double]	[1x125001 double]

The table contains time series data of voltage variables and their first derivatives over the whole simulation time for the LDMOS block. The table does not list the S terminal because it is held fixed to the ground.

### Calculate Voltage Derivatives for a Specific Time Period

Open the Class E DC-DC Converter example model.

```
open_system('elec_converter_dcdc_class_e')
```



This example model has data logging enabled. Run the simulation to create the simulation log variable `simlog_elec_converter_dcdc_class_e` in your current workspace.

```
sim('elec_converter_dcdc_class_e');
```

The model simulation time is  $1.25e-4$  seconds. Calculate and display rates-of-change of voltage variables for the Transformer block during the last  $0.25e-4$  seconds of the simulation. Use a time step of  $1e-9$  seconds.

```
transformerTable = elec_getNodeDvDtTimeSeries(simlog_elec_converter_dc_dc_class_e.Trans
```

```
transformerTable =
```

```
3x4 table
```

LoggingNode	Terminal	Voltage	dvdt
"Transformer"	"p1"	[1x25001 double]	[1x25001 double]
"Transformer"	"n3"	[1x25001 double]	[1x25001 double]
"Transformer"	"p2"	[1x25001 double]	[1x25001 double]

The table contains time series data of voltage variables, and their first derivatives, for the Transformer block over the last 0.25e-4 seconds of the simulation. The table does not list terminals that are held fixed to the ground.

View the time series data. From the workspace, open the `transformerTable` table, then open the two `1x25001 double` numeric arrays for the `Transformer.p1`.

The first array contains the voltage data. The second array contains the voltage derivative data.

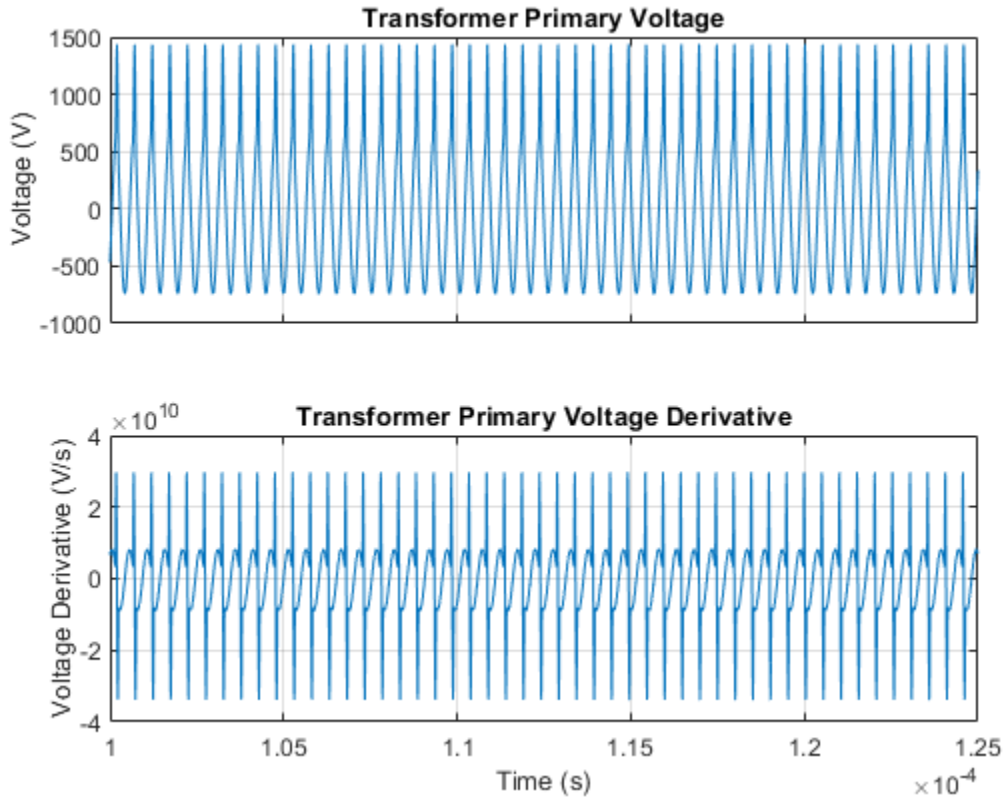
Plot the data.

```
time = 1e-4:1e-9:1.25e-4;
vOut = transformerTable.Voltage{1};
dvdtOut = transformerTable.dvdt{1};

ax1 = subplot(2,1,1);
plot(time,vOut),grid;
ylabel('Voltage (V)');
ax1.YLim = [-1000 1500];
ax1.XTickLabel = {};
ax1.Title.String = 'Transformer Primary Voltage';

ax2 = subplot(2,1,2);
plot(time,dvdtOut),grid;
ylabel('Voltage Derivative (V/s)');
xlabel('Time (s)');
ax2.Title.String = 'Transformer Primary Voltage Derivative';
```





## Input Arguments

**node** — Simulation log variable, or a specific node within the simulation log variable

Node object

Simulation log workspace variable, or a node within this variable, that contains the logged model simulation data, specified as a Node object. You specify the name of the simulation log variable by using the **Workspace variable name** parameter on the **Simscape** pane of the Configuration Parameters dialog box. To specify a node within the simulation log variable, provide the complete path to that node through the simulation data tree, starting with the top-level variable name.

Example: `simlog_elec_converter_dcdc_class_e.LDMOS`

### **tau — Time step for numerical differentiation**

real number

Time step for numerical differentiation, specified as a real number, in seconds. `tau` determines the interpolation grid as `startTime:tau:endTime`.

Example: `1e-9`

Data Types: `double`

### **startTime — Start of the time interval for evaluating rates-of-change of voltage variables**

simulation start time (default) | real number

Start of the time interval for evaluating rates-of-change of voltage variables, specified as a real number, in seconds. `startTime` must be greater than or equal to the simulation **Start time** and less than `endTime`.

Data Types: `double`

### **endTime — End of the time interval for evaluating rates-of-change of voltage variables**

simulation stop time (default) | real number

End of the time interval for evaluating rates-of-change of voltage variables, specified as a real number, in seconds. `endTime` must be greater than `startTime` and less than or equal to the simulation **Stop time**.

Data Types: `double`

## Output Arguments

### **seriesTable — Time series of the voltage rates-of-change for each block**

table

Time series of the voltage rates-of-change for each block, returned as a table. The first column lists all the logging nodes in node that are based on the `foundation.electrical.electrical` domain. The second column lists the terminal names. The third column lists the corresponding interpolated voltage values, in volts. The fourth column lists the corresponding numerically differentiated values of voltage rates-

of-change, in volts per second. The table does not contain data for terminals that are held fixed.

## See Also

[elec\\_getNodeDvDtSummary](#) | [sscexplore](#)

## Topics

["About Simulation Data Logging" \(Simscape\)](#)

["About the Simscape Results Explorer" \(Simscape\)](#)

**Introduced in R2018b**

## **elec\_getPowerLossSummary**

Calculate dissipated power losses

### **Syntax**

```
lossesTable = elec_getPowerLossSummary(node)  
lossesTable = elec_getPowerLossSummary(node, startTime, endTime)
```

### **Description**

`lossesTable = elec_getPowerLossSummary(node)` calculates dissipated power losses for semiconductor blocks in a model, based on logged simulation data, and returns the data for each block in a table.

Before you call this function, you must have the simulation log variable in your current workspace. Create the simulation log variable by simulating the model with data logging turned on, or load a previously saved variable from a file. If `node` is the name of the simulation log variable, then the table contains the data for all semiconductor blocks in the model. If `node` is the name of a node in the simulation data tree, then the table contains the data only for the blocks within that node.

Checking dissipated power is useful for verifying that circuit components are operating within their working envelopes. All blocks in the Semiconductor Devices library, as well as some other blocks, have an internal variable called `power_dissipated`, which represents the instantaneous power dissipated by the block. When you log simulation data, the time-value series for this variable represents the power dissipated by the block over time. You can view and plot this data using the Simscape Results Explorer.

The `elec_getPowerLossSummary` function calculates average losses for each block that has a `power_dissipated` variable. Some blocks have more than one `power_dissipated` variable, depending on their configuration. For example, the N-Channel MOSFET block has separate `power_dissipated` logging nodes for the MOSFET, the gate resistor, and for the source and drain resistors if they have nonzero resistance values. The function sums all these losses and provides the power loss value for the whole block, averaged over simulation time.

`lossesTable = elec_getPowerLossSummary(node, startTime, endTime)` calculates dissipated power losses within a time interval. `startTime` and `endTime` represent the start and end of the time interval for averaging the power losses. If you omit these two input arguments, the function averages the power losses over the whole simulation time.

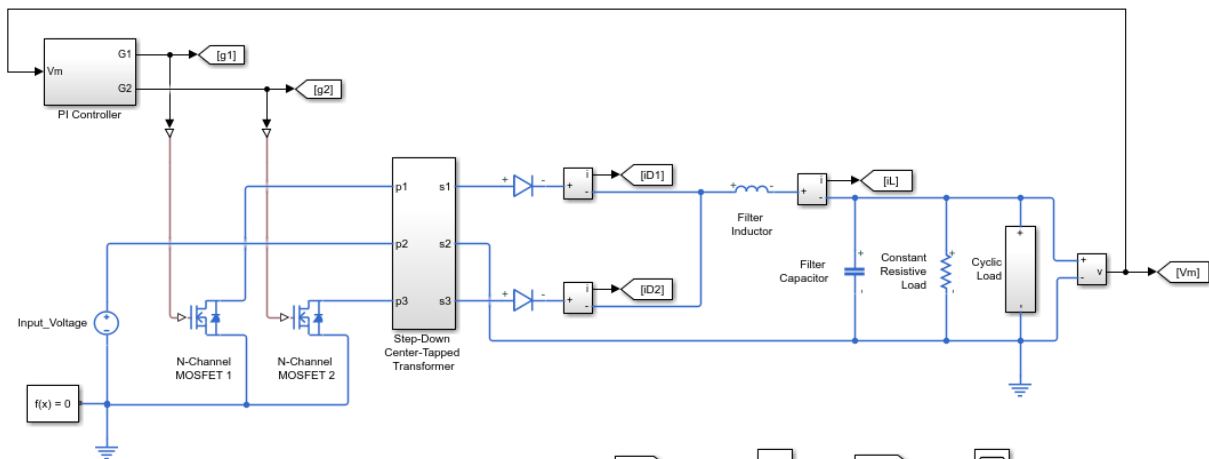
## Examples

### Calculate Average Power Losses for Components of a Block

You can calculate average power losses for the individual components of a block in your model.

1. Open the Push-Pull Buck Converter in Continuous Conduction Mode example model. At the MATLAB® command prompt, enter

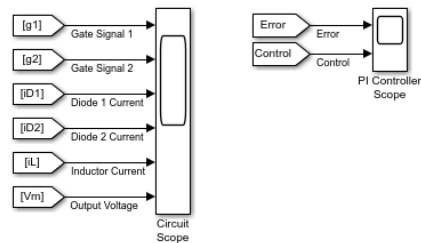
```
model = 'pe_push_pull_converter_ccm';  
open_system(model)
```



### Push-Pull Buck Converter in Continuous Conduction Mode

This example shows how to control the output voltage of a push-pull buck converter. The current flowing through the inductor is never zero, therefore the DC-DC converter operates in Continuous Conduction Mode (CCM). To convert and maintain the nominal output voltage, the PI Controller subsystem uses a simple integral control. During startup, the reference voltage is ramped up to the desired output voltage.

1. [Edit the parameterization script.](#)
2. [Explore simulation results using `sscexplore`.](#)
3. [Learn more](#) about this example.



The model has data logging enabled.

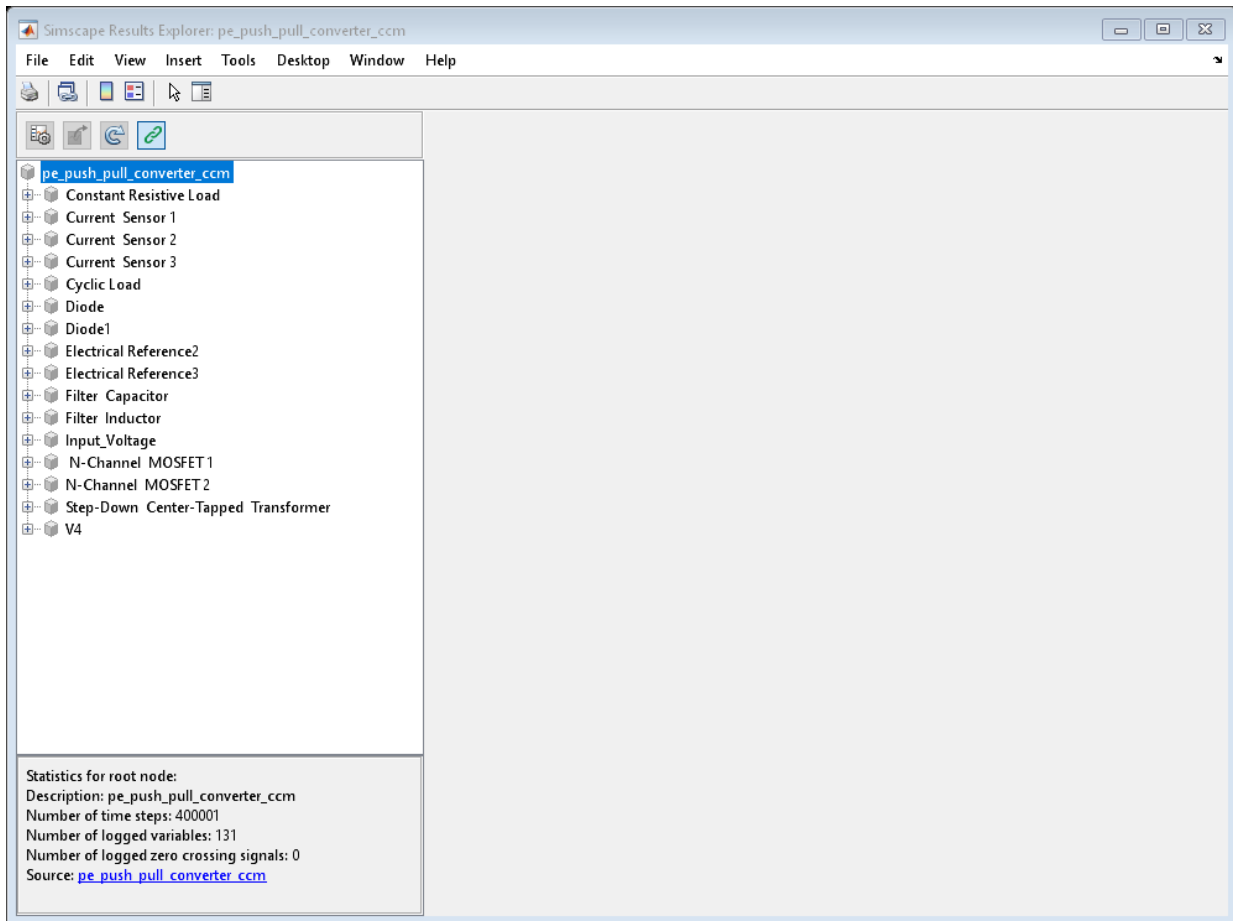
2. Add a diode component in the N-Channel MOSFET 1 block using the MATLAB® command prompt:

```
set_param('pe_push_pull_converter_ccm/ N-Channel MOSFET 1', 'diode_param', '2')
```

Alternatively, you can add the component in the Simulink® Editor:

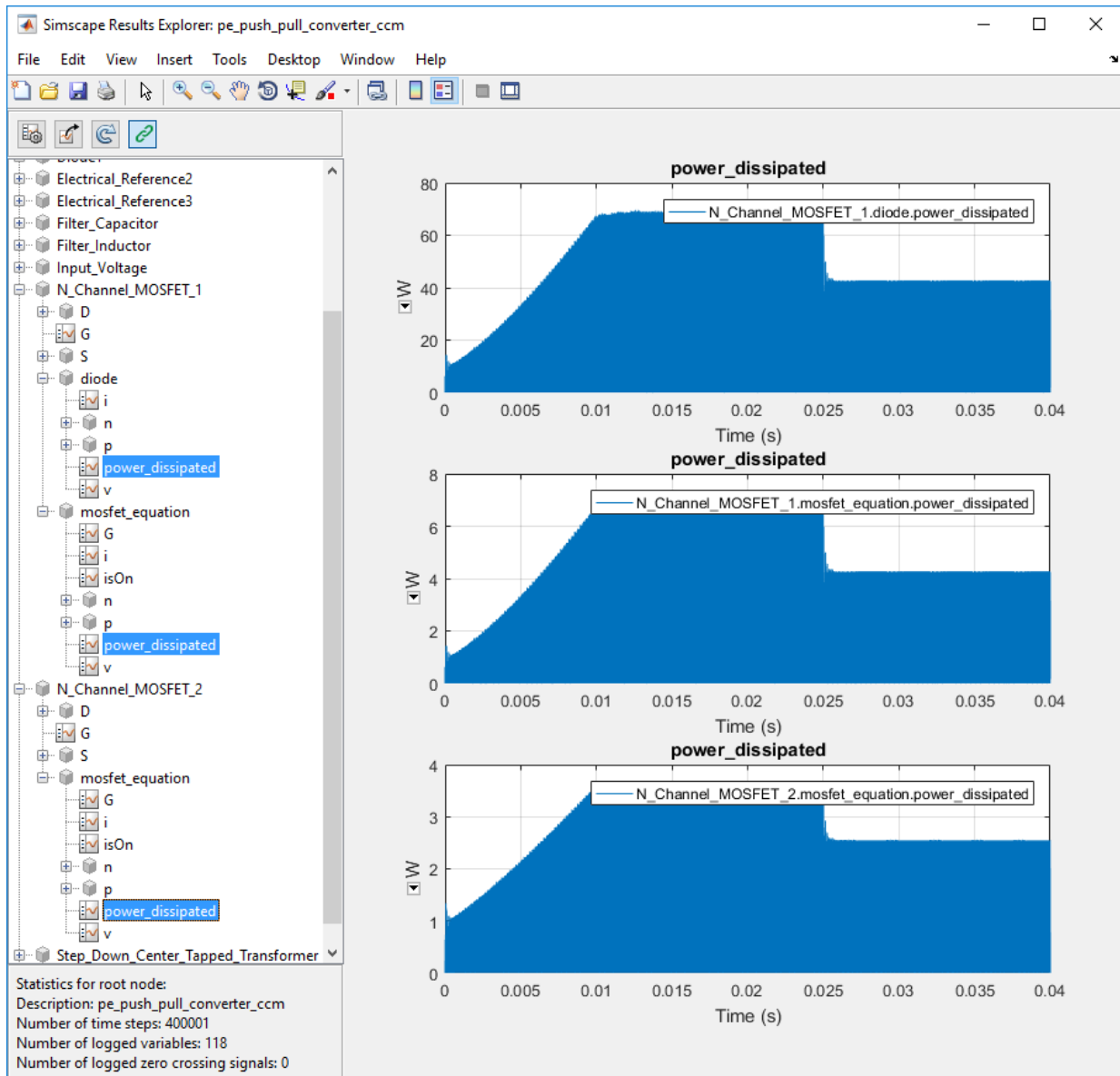
- a. Open the Property Inspector pane. In the model window, in the menu bar, click **View > Property Inspector**
  - b. Click the N-Channel MOSFET1 block to access the block parameters.
  - c. In the Property Inspector pane, expand the Integral Diode setting and change the value for the **Integral protection** from None to Protection diode with no dynamics.
3. Run the simulation, create a simulation log variable, and open the simlog in the Simscape Results Explorer using the `sscexplore` function.

```
sim(model)
sscxplore(simlog_pe_push_pull_converter_ccm)
```



4. View the power loss data for the two N-Channel MOSFET blocks, expand these nodes and CTRL + click the power\_dissipated nodes:

- N\_Channel\_MOSFET\_1 > diode > power\_dissipated
- N\_Channel\_MOSFET\_1 > mosfet\_equation > power\_dissipated
- N\_Channel\_MOSFET\_2 > mosfet\_equation > power\_dissipated





The N-Channel MOSFET 2 block has only one `power_dissipated` variable. The N-Channel MOSFET 1 block has one `power_dissipated` variable for each of the two components (MOSFET and diode) that the block contains.

5. Calculate power losses for both components of the N-Channel MOSFET 1 block and display the results in a table

```
tabulatedLosses = pe_getPowerLossSummary(simlog_pe_push_pull_converter_ccm.N_Channel_M
```

```
tabulatedLosses =
```

```
1x2 table
```

LoggingNode	Power
'N_Channel_MOSFET_1'	2.6075

The table shows the combined dissipated power losses for both the diode and the MOSFET components of the N-Channel MOSFET 1 block, averaged over the total simulation time.

6. Calculate power losses for only the diode component of the NChannel MOSFET 1 block and display the results in a table.

```
tabulatedLosses = pe_getPowerLossSummary(simlog_pe_push_pull_converter_ccm.N_Channel_M
```

```
tabulatedLosses =
```

```
1x2 table
```

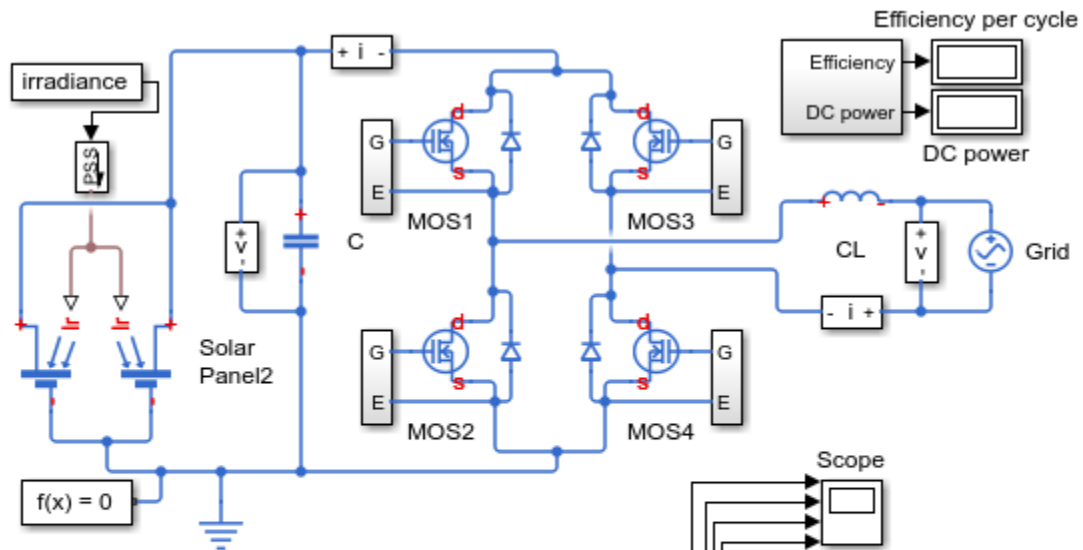
LoggingNode	Power
'diode'	2.3669

The table shows dissipated power losses only for the diode component of the block, averaged over the total simulation time.

### Calculate Power Losses for One Block

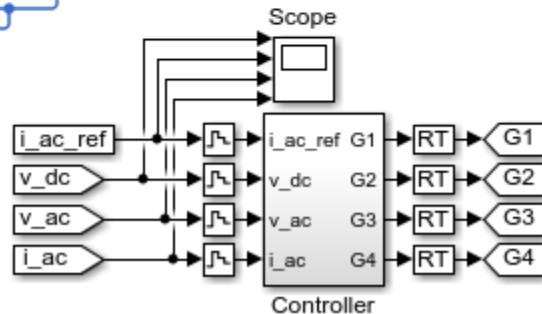
Open the Solar Power Converter example model.

`elec_solar_converter`



#### Solar Power Converter

1. [Plot power dissipated](#) by MOSFETs ([see code](#))
2. [Calculate losses](#) by component ([see code](#))
3. [Explore simulation results](#) using [sscexplore](#)
4. [Learn more](#) about this example



This example model has data logging enabled. Run the simulation to create the simulation log variable `simlog_elec_solar_converter` in your current workspace.

```
sim('elec_solar_converter');
```

Calculate power losses for the MOS1 block.

```
mosfetLosses = elec_getPowerLossSummary(simlog_elec_solar_converter.MOS1)
```

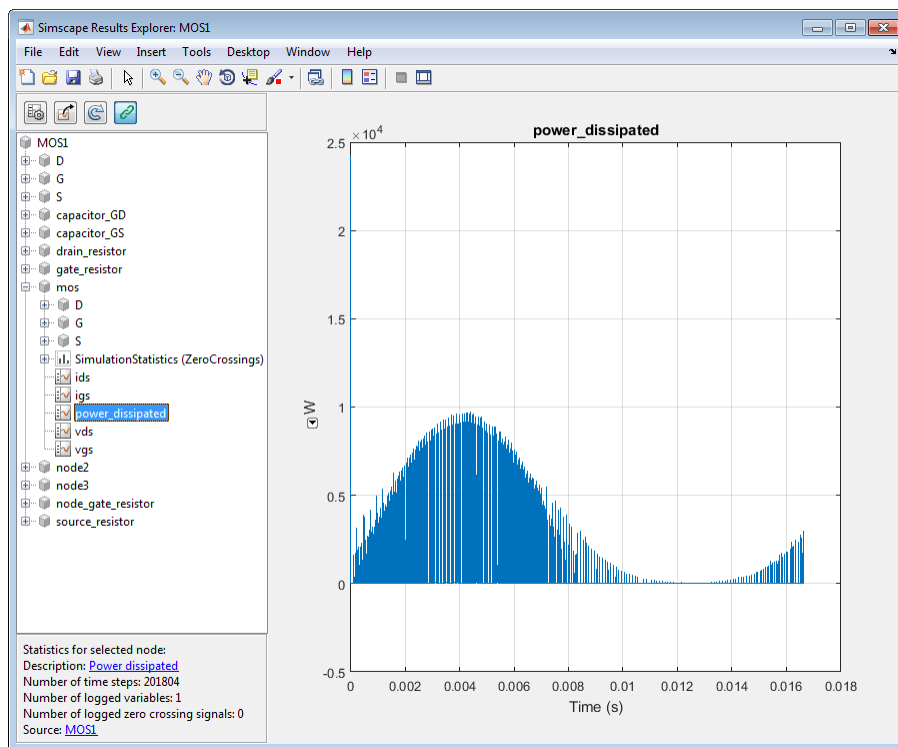
```
mosfetLosses =
```

LoggingNode	Power
'MOS1'	15.316

The table shows dissipated power losses for the MOS1 block, averaged over the whole simulation time.

Use the `sscexplore` function to further explore the power loss data for the MOSFET block.

```
sscexplore(simlog_elec_solar_converter.MOS1)
```



The block has several `power_dissipated` logging nodes: under `drain_resistor`, under `gate_resistor`, under `mos`, and under `source_resistor`. The 15.316 value

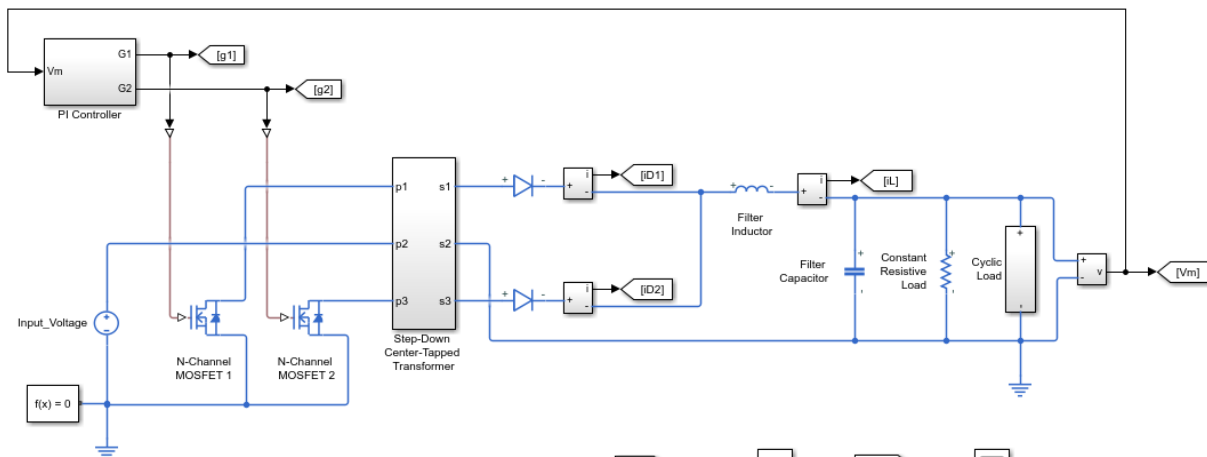
calculated by the `elec_getPowerLossSummary` function is a sum of all these losses, averaged over the simulation time.

### Calculate Average Power Losses for Components of a Block

You can calculate average power losses for the individual components of a block in your model.

1. Open the Push-Pull Buck Converter in Continuous Conduction Mode example model. At the MATLAB® command prompt, enter

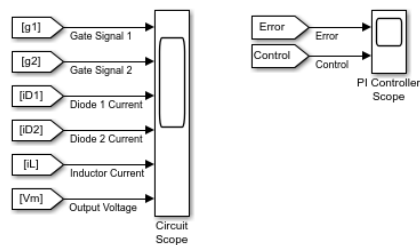
```
model = 'pe_push_pull_converter_ccm';
open_system(model)
```



#### Push-Pull Buck Converter in Continuous Conduction Mode

This example shows how to control the output voltage of a push-pull buck converter. The current flowing through the inductor is never zero, therefore the DC-DC converter operates in Continuous Conduction Mode (CCM). To convert and maintain the nominal output voltage, the PI Controller subsystem uses a simple integral control. During startup, the reference voltage is ramped up to the desired output voltage.

1. [Edit the parameterization script.](#)
2. [Explore simulation results](#) using `sscexplore`.
3. [Learn more](#) about this example.



The model has data logging enabled.

2. Add a diode component in the N-Channel MOSFET 1 block using the MATLAB® command prompt:

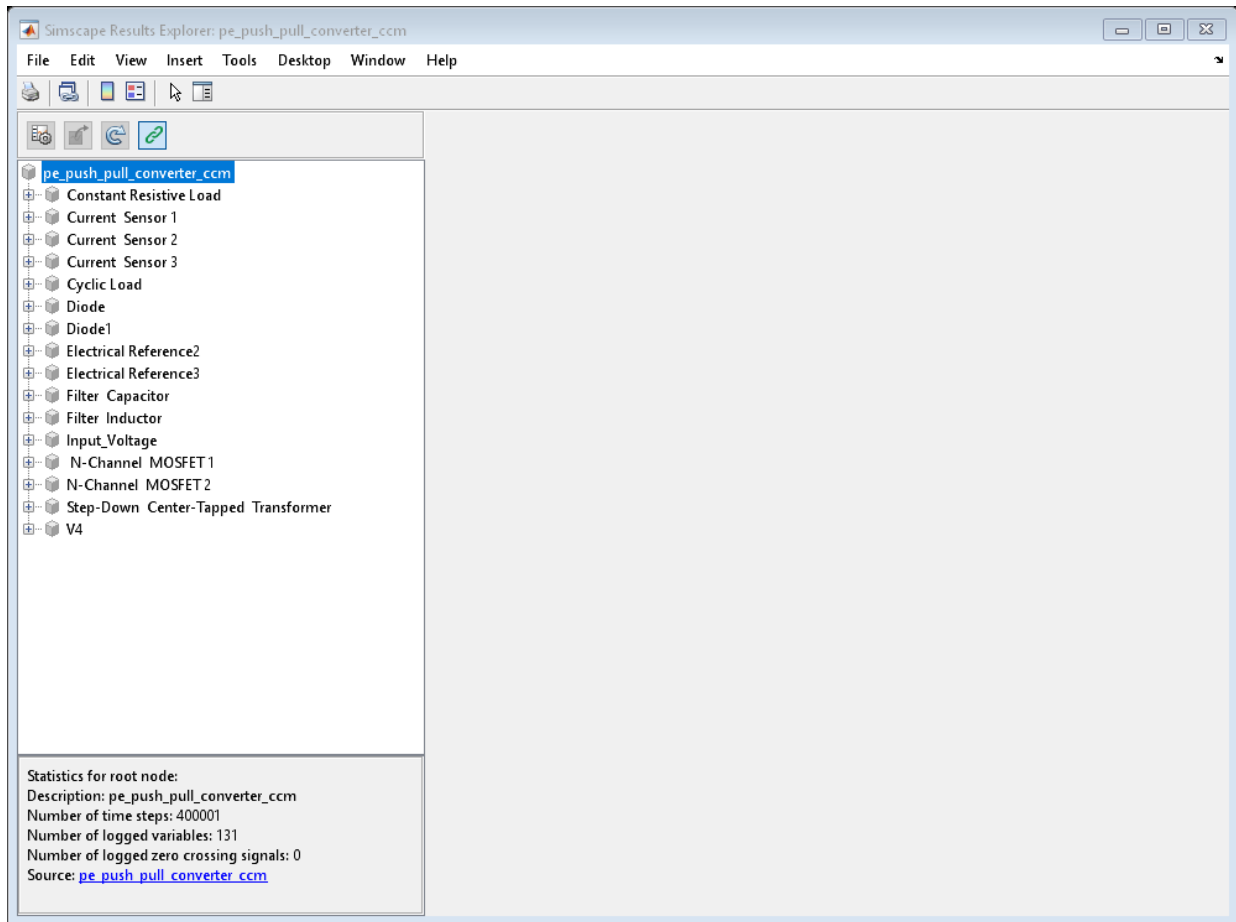
```
set_param('pe_push_pull_converter_ccm/ N-Channel MOSFET 1', 'diode_param', '2')
```

Alternatively, you can add the component in the Simulink® Editor:

- a. Open the Property Inspector pane. In the model window, in the menu bar, click **View > Property Inspector**
- b. Click the N-Channel MOSFET1 block to access the block parameters.
- c. In the Property Inspector pane, expand the Integral Diode setting and change the value for the **Integral protection** from None to Protection diode with no dynamics.

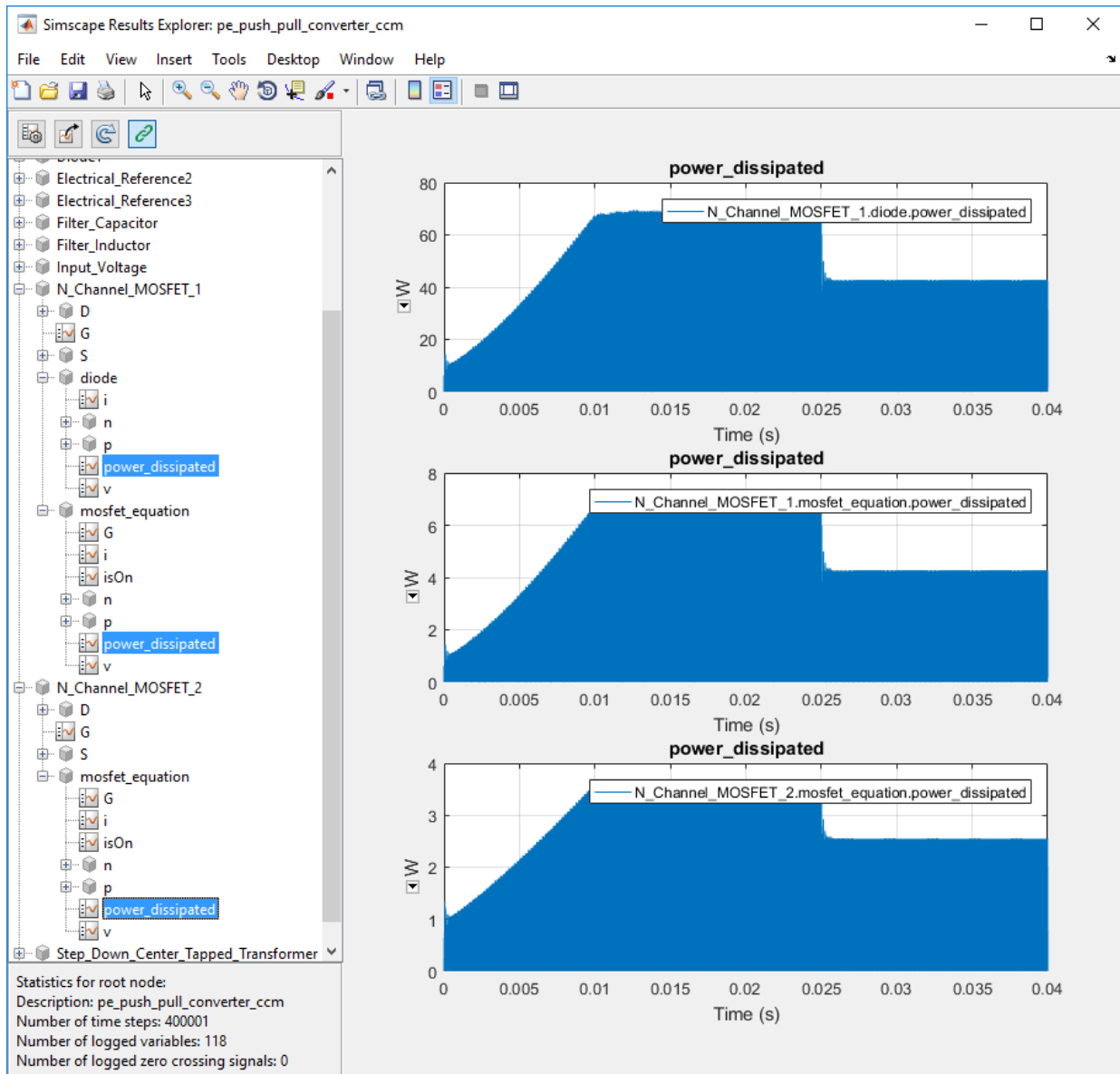
3. Run the simulation, create a simulation log variable, and open the simlog in the Simscape Results Explorer using the `sscexplore` function.

```
sim(model)  
sscexplore(simlog_pe_push_pull_converter_ccm)
```



4. View the power loss data for the two N-Channel MOSFET blocks, expand these nodes and CTRL + click the power\_dissipated nodes:

- N\_Channel\_MOSFET\_1 > diode > power\_dissipated
- N\_Channel\_MOSFET\_1 > mosfet\_equation > power\_dissipated
- N\_Channel\_MOSFET\_2 > mosfet\_equation > power\_dissipated



The N-Channel MOSFET 2 block has only one `power_dissipated` variable. The N-Channel MOSFET 1 block has one `power_dissipated` variable for each of the two components (MOSFET and diode) that the block contains.

5. Calculate power losses for both components of the N-Channel MOSFET 1 block and display the results in a table

```
tabulatedLosses = pe_getPowerLossSummary(simlog_pe_push_pull_converter_ccm.N_Channel_M
```

```
tabulatedLosses =
```

```
1x2 table
```

LoggingNode	Power
'N_Channel_MOSFET_1'	2.6075

The table shows the combined dissipated power losses for both the diode and the MOSFET components of the N-Channel MOSFET 1 block, averaged over the total simulation time.

6. Calculate power losses for only the diode component of the NChannel MOSFET 1 block and display the results in a table.

```
tabulatedLosses = pe_getPowerLossSummary(simlog_pe_push_pull_converter_ccm.N_Channel_M
```

```
tabulatedLosses =
```

```
1x2 table
```

LoggingNode	Power
'diode'	2.3669



The table shows dissipated power losses only for the diode component of the block, averaged over the total simulation time.

## Input Arguments

**node** — Simulation log variable, or a specific node within the simulation log variable

Node object

Simulation log workspace variable, or a node within this variable, that contains the logged model simulation data, specified as a Node object. You specify the name of the simulation log variable by using the **Workspace variable name** parameter on the **Simscape** pane of the Configuration Parameters dialog box. To specify a node within the simulation log variable, provide the complete path to that node through the simulation data tree, starting with the top-level variable name.

Example: `simlog.Cell1.MOS1`

**startTime** — Start of the time interval for averaging dissipated power losses

real number

Start of the time interval for averaging dissipated power losses, specified as a real number, in seconds. `startTime` must be greater than or equal to the simulation **Start time** and less than `endTime`.

Data Types: double

**endTime** — End of the time interval for averaging dissipated power losses

real number

End of the time interval for averaging dissipated power losses, specified as a real number, in seconds. `endTime` must be greater than `startTime` and less than or equal to the simulation **Stop time**.

Data Types: double

## Output Arguments

**lossesTable** — Dissipated power losses for each block

table

Dissipated power losses for each block, returned as a table. The first column lists logging nodes for all blocks that have at least one `power_dissipated` variable. The second column lists the corresponding losses in watts.

### See Also

`elec_getEfficiency` | `elec_getPowerLossTimeSeries` | `sscexplore`

### Topics

“About Simulation Data Logging” (Simscape)

“About the Simscape Results Explorer” (Simscape)

**Introduced in R2015a**

# elec\_getPowerLossTimeSeries

Calculate dissipated power losses and return time series data

## Syntax

```
lossesCell = elec_getPowerLossTimeSeries(node)
lossesCell = elec_getPowerLossTimeSeries(node,startTime,endTime)
lossesCell = elec_getPowerLossTimeSeries(node,startTime,endTime,
intervalWidth)
```

## Description

`lossesCell = elec_getPowerLossTimeSeries(node)` calculates dissipated power losses for blocks in a model, based on logged simulation data, and returns the time series data for each block.

Before you call this function, you must have the simulation log variable in your current workspace. Create the simulation log variable by simulating the model with data logging turned on, or load a previously saved variable from a file.

The `elec_getPowerLossTimeSeries` function calculates dissipated power losses for each block that has a `power_dissipated` variable. All blocks in the Semiconductor Devices library, as well as some other blocks, have an internal variable called `power_dissipated`, which represents the instantaneous power dissipated by the block. Some blocks have more than one `power_dissipated` variable, depending on their configuration. For example, the N-Channel MOSFET block has separate `power_dissipated` logging nodes for the MOSFET, the gate resistor, and for the source and drain resistors if they have nonzero resistance values. The function sums all these losses and provides the power loss value for all of the blocks as functions of time.

If `node` is the name of the simulation log variable, then the table contains the data for all the blocks in the model that dissipate power (that is, contain at least one `power_dissipated` variable). If `node` is the name of a node in the simulation data tree, then the table contains the data only for the blocks within that node.

`lossesCell = elec_getPowerLossTimeSeries(node,startTime,endTime)` calculates dissipated power losses and returns the time series data for time steps from

startTime to endTime. If startTime is equal to endTime, the interval is effectively zero and the function returns the instantaneous power for the time step that occurs at that moment. If you omit these two input arguments, the function returns data over the whole simulation time.

lossesCell = elec\_getPowerLossTimeSeries(node, startTime, endTime, intervalWidth) calculates dissipated power losses and returns the time series data for time steps from startTime to endTime, averaged over the time intervalWidth. If you omit the intervalWidth, or set it to 0, the function returns the instantaneous data, without averaging. If you omit all three optional arguments, the function returns the instantaneous data over the whole simulation time.

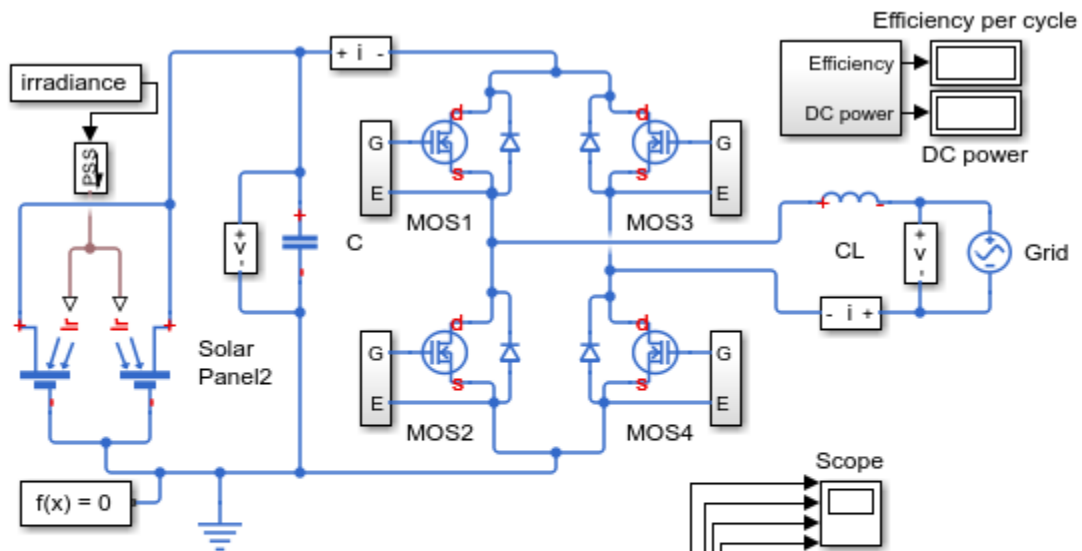
## Examples

### Calculate Dissipated Power Losses for the Entire Simulation Time

This example shows how to calculate instantaneous losses based on the power dissipated and return the time series data for all time steps in the entire simulation time using the elec\_getPowerLossTimeSeries function.

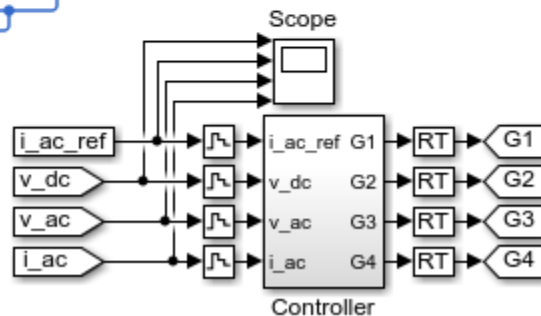
Open the model. At the MATLAB® command prompt, enter:

```
model = 'elec_solar_converter';  
open_system(model)
```



### Solar Power Converter

1. [Plot power dissipated](#) by MOSFETs ([see code](#))
2. [Calculate losses](#) by component ([see code](#))
3. [Explore simulation results](#) using [sscexplore](#)
4. [Learn more](#) about this example



This example model has data logging enabled. Run the simulation and create the simulation log variable.

```
sim(model)
```

The simulation log variable `simlog_elec_solar_converter` is saved in your current workspace.

Calculate dissipated power losses and return the time series data in cell array.

```
lossesCell = elec_getPowerLossTimeSeries(simlog_elec_solar_converter)
```

```
lossesCell =
```

```
8x2 cell array
```

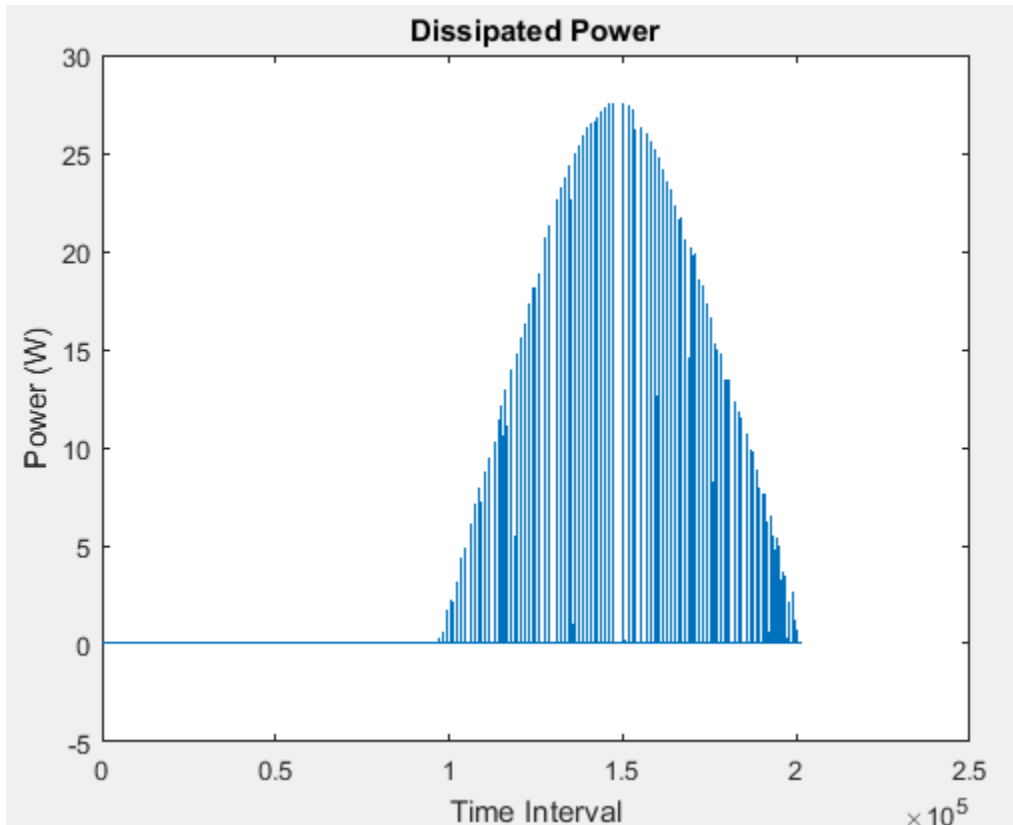
```
'elec_solar_converter.Diode1.diode' [201804×3 double]
'elec_solar_converter.MOS1' [201804×3 double]
'elec_solar_converter.MOS2' [201804×3 double]
'elec_solar_converter.MOS3' [201804×3 double]
'elec_solar_converter.MOS4' [201804×3 double]
'elec_solar_converter.Diode2.diode' [201804×3 double]
'elec_solar_converter.Diode3.diode' [201804×3 double]
'elec_solar_converter.Diode4.diode' [201804×3 double]
```

View the time series data. From the workspace, open the `lossesCell` cell array, then open the `201804×3 double` numeric array for the `elec_solar_converter.Diode1.diode`.

The first two columns contain the interval start and end time. The third column contains the power loss data.

Plot the data.

```
plot(lossesCell{1, 2}(:,end))
title('Dissipated Power')
xlabel('Time Interval')
ylabel('Power (W)')
```

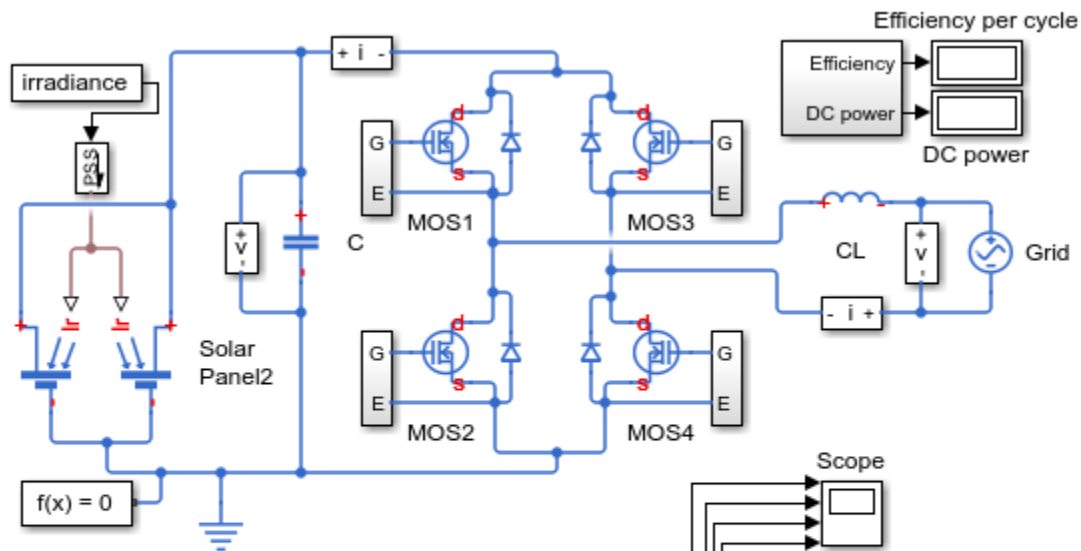


### Calculate Dissipated Power Losses for a Specific Time Period

This example shows how to calculate instantaneous losses based on the power dissipated and return the time series data for all time steps in a specific time period using the `elec_getPowerLossTimeSeries` function.

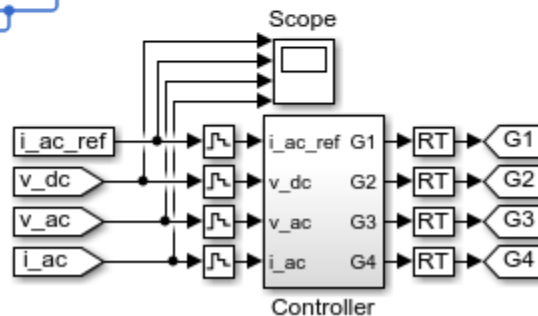
Open the model. At the MATLAB® command prompt, enter:

```
model = 'elec_solar_converter';  
open_system(model)
```



### Solar Power Converter

1. [Plot power dissipated](#) by MOSFETs ([see code](#))
2. [Calculate losses](#) by component ([see code](#))
3. [Explore simulation results](#) using [sscexplore](#)
4. [Learn more](#) about this example



This example model has data logging enabled. Run the simulation and create the simulation log variable.

```
sim(model)
```

The simulation log variable `simlog_elec_solar_converter` is saved in your current workspace.

The model simulation time ( $t$ ) is  $1/60$  seconds. Calculate dissipated power losses and return the time series data in cell array for the second half of the simulation cycle, when  $t$  is between  $1/120$  and  $1/60$  seconds..

```
lossesCell = elec_getPowerLossTimeSeries(simlog_elec_solar_converter,1/120,1/60)
```

```
lossesCell =
```



8×2 cell array

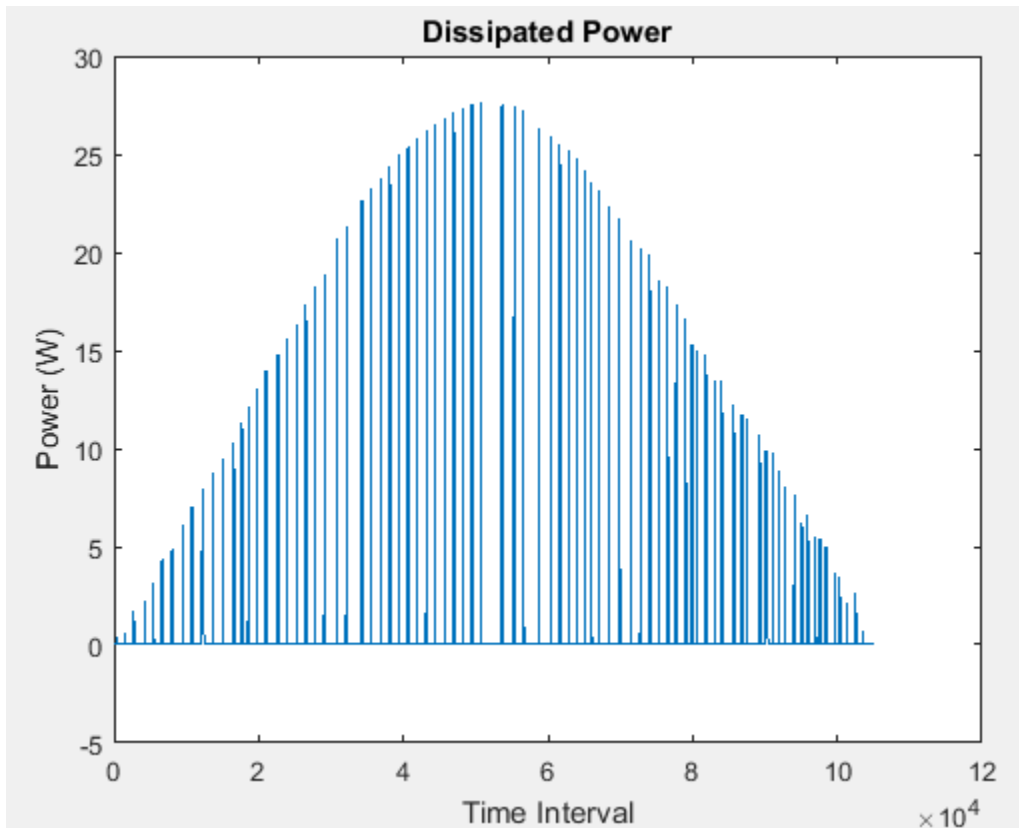
```
'elec_solar_converter.Diode1.diode' [105197×3 double]
'elec_solar_converter.MOS1' [105197×3 double]
'elec_solar_converter.MOS2' [105197×3 double]
'elec_solar_converter.MOS3' [105197×3 double]
'elec_solar_converter.MOS4' [105197×3 double]
'elec_solar_converter.Diode2.diode' [105197×3 double]
'elec_solar_converter.Diode3.diode' [105197×3 double]
'elec_solar_converter.Diode4.diode' [105197×3 double]
```

View the time series data. From the workspace, open the `lossesCell` cell array, then open the `105197×3 double` numeric array for the `elec_solar_converter.Diode1.diode`.

The first two columns contain the interval start and end time. The third column contains the power loss data.

Plot the data.

```
plot(lossesCell{1, 2}(:,end))
title('Dissipated Power')
xlabel('Time Interval')
ylabel('Power (W)')
```

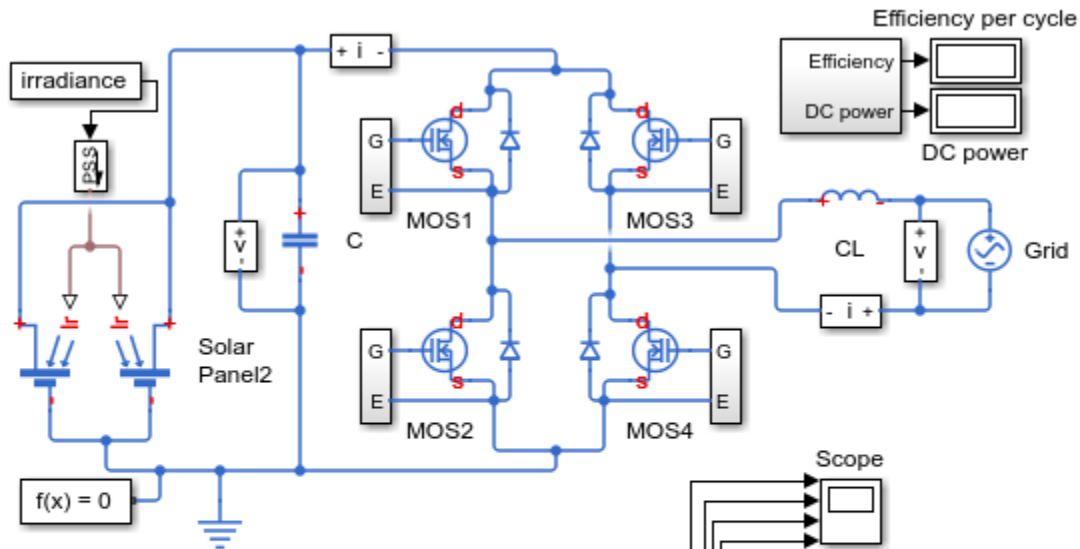


### Calculate Dissipated Power Losses Using Specific Interval Widths

This example shows how to calculate losses based on the power dissipated and return the time series data for a specific time period with averaging applied over intervals of a specified width.

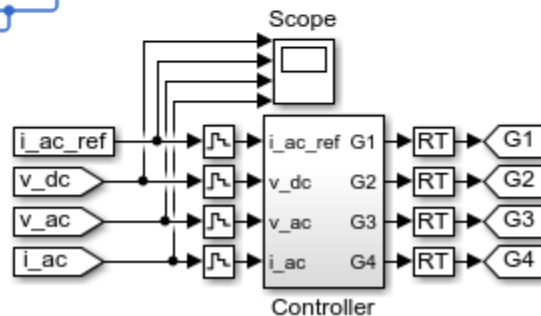
Open the model. At the MATLAB® command prompt, enter:

```
model = 'elec_solar_converter';  
open_system(model)
```



### Solar Power Converter

1. [Plot power dissipated](#) by MOSFETs ([see code](#))
2. [Calculate losses](#) by component ([see code](#))
3. [Explore simulation results](#) using [sscexplore](#)
4. [Learn more](#) about this example



This example model has data logging enabled. Run the simulation and create the simulation log variable.

```
sim(model)
```

The simulation log variable `simlog_elec_solar_converter` is saved in your current workspace.

The model simulation time ( $t$ ) is  $1/60$  seconds. Calculate the average dissipated power losses for  $1.1 \cdot 10^{-4}$  s intervals and return the time series data in cell array for the period when simulation time,  $t$ , is  $0.008$ - $0.017$  seconds.

```
lossesCell = elec_getPowerLossTimeSeries(simlog_elec_solar_converter,0.008,0.016,1.1e-4)
```

```
lossesCell =
```

8×2 cell array

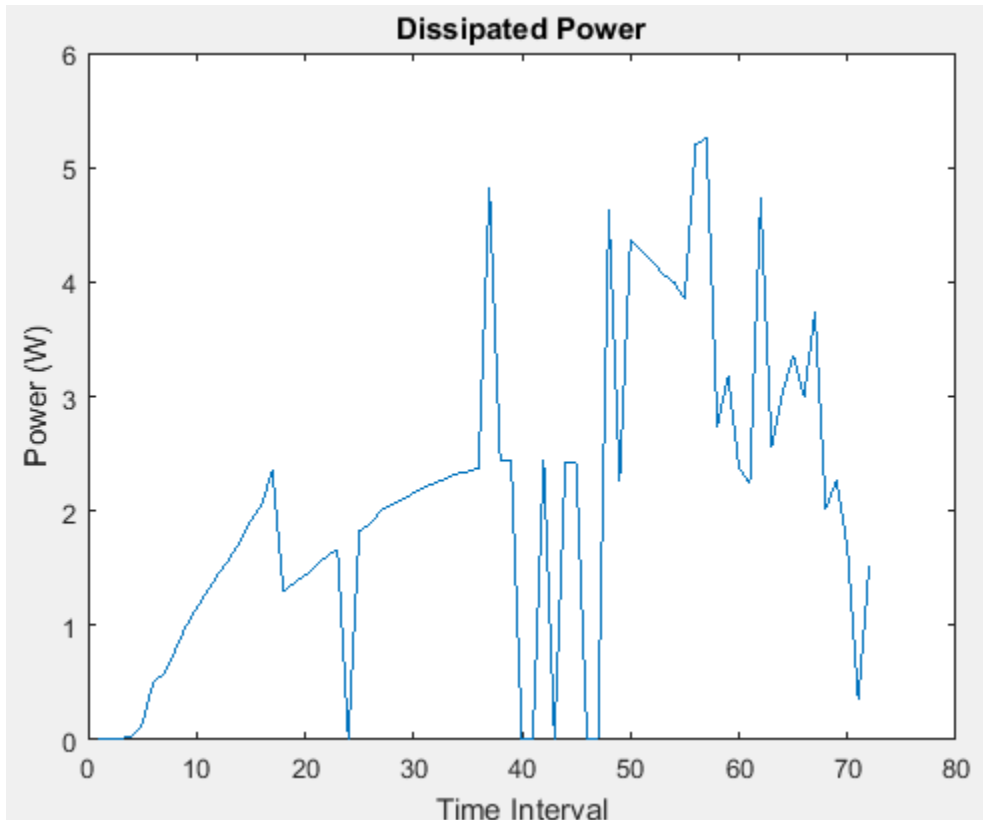
```
'elec_solar_converter.Diode1.diode' [72×3 double]
'elec_solar_converter.MOS1' [72×3 double]
'elec_solar_converter.MOS2' [72×3 double]
'elec_solar_converter.MOS3' [72×3 double]
'elec_solar_converter.MOS4' [72×3 double]
'elec_solar_converter.Diode2.diode' [72×3 double]
'elec_solar_converter.Diode3.diode' [72×3 double]
'elec_solar_converter.Diode4.diode' [72×3 double]
```

View the time series data. From the workspace, open the `lossesCell` cell array, then open the `105197×3 double` numeric array for the `elec_solar_converter.Diode1.diode`.

The first two columns contain the interval start and end time. The third column contains the power loss data. In this case, to use averaging intervals that are equal in width to  $1.1\text{e-}4$  seconds, the function adjusts the start time for the first interval from the specified value of 0.008 seconds to a value of 0.0081 seconds. There are 72 intervals of  $1.1\text{e-}4$  seconds.

Plot the data.

```
plot(lossesCell{1, 2}(:,end))
title('Dissipated Power')
xlabel('Time Interval')
ylabel('Power (W)')
```



## Input Arguments

**node** — Simulation log variable, or a specific node within the simulation log variable

Node object

Simulation log workspace variable, or a node within this variable, that contains the logged model simulation data, specified as a **Node** object. You specify the name of the simulation log variable by using the **Workspace variable name** parameter on the **Simscape** pane of the Configuration Parameters dialog box. To specify a node within the simulation log variable, provide the complete path to that node through the simulation data tree, starting with the top-level variable name.

If `node` is the name of the simulation log variable, then the table contains the data for all blocks in the model that contain `power_dissipated` variables. If `node` is the name of a node in the simulation data tree, then the table contains the data only for:

- Blocks or variables within that node
- Blocks or variables within subnodes at all levels of the hierarchy beneath that node

Example: `simlog.Cell1.MOS1`

### **startTime** — Start of the time interval for calculating the data

0 (default) | real number

Start of the time interval for calculating the power loss time series, specified as a real number, in seconds. `startTime` must be greater than or equal to the simulation **Start time** and less than `endTime`.

Data Types: double

### **endTime** — End of the time interval for calculating the data

simulation stop time (default) | real number

End of the time interval for calculating the power loss time series, specified as a real number, in seconds. `endTime` must be greater than `startTime` and less than or equal to the simulation **Stop time**.

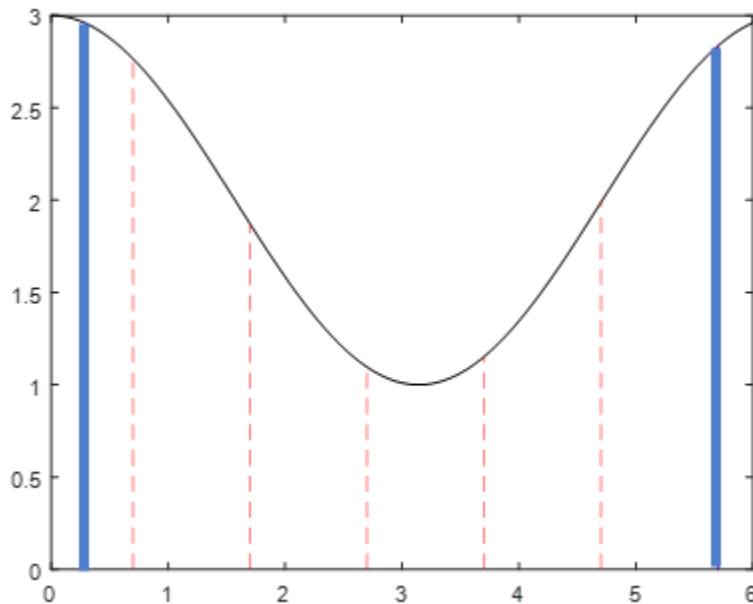
Data Types: double

### **intervalWidth** — Size of the time interval for calculating the average power dissipation

0 (default) | real number

Size of the time interval for calculating the average power dissipation, specified as a real number, in seconds. If specified, the function returns data for time steps from `startTime` to `endTime`, averaged over the `intervalWidth`. If you omit the `intervalWidth` argument, or set it to 0, the function returns the instantaneous data, without averaging. If all the optional arguments are omitted, the function returns the instantaneous data over the whole simulation time.

If the time between the specified `startTime` and `endTime` is not an integer multiple of `intervalWidth`, the function adjusts the start time. The figure shows how the function adjusts the start time to ensure that width of each time interval that the dissipated power is averaged over is equal to the specified `intervalWidth`.



The black line is an example of the instantaneous *power\_dissipated* variables summed over all elements in an individual block. The simulation runs for 6 seconds. The `startTime` and `endTime` are indicated by the solid blue lines. The `intervalWidth` is set to 1 second. There are five intervals as indicated by the red dashed lines. The right-most edge of the last interval coincides with `endTime`. The left-most edge of the first interval is always greater than or equal to `startTime`. The edge is equal to `startTime` only if  $(\text{endTime} - \text{startTime}) / \text{intervalWidth}$  is an integer. The output in this case consists of five values for the averaged power dissipation, one point for each time period. The function outputs the actual start and stop times in the tabulated output data.

Example:  $1.1 \times 10^{-3}$

Data Types: double

## Output Arguments

**lossesCell** — Time series of the dissipated power losses for each block  
cell array

Cell array that contains the names of the blocks in the nodes that contain *power\_dissipated* variables and, for each block, a three-column array:

- Column one contains the interval start time.
- Column two contains the interval end time.
- Column three contains the dissipated power for the time interval.

If the interval width is 0 seconds, that is, the start time is equal to the end time, then the dissipated power is the instantaneous power loss. If the interval is greater than 0 seconds, the dissipated power is the average power loss for the time of the interval.

## See Also

`elec_getEfficiency` | `elec_getPowerLossSummary`

## Topics

“About Simulation Data Logging” (Simscape)

“About the Simscape Results Explorer” (Simscape)

**Introduced in R2017a**



# subcircuit2ssc

Convert SPICE subcircuit to custom Simscape components

## Syntax

```
subcircuit2ssc(filename,target)
subcircuit2ssc(__,subcircuit1,...,subcircuitN)
subcircuitArray = subcircuit2ssc(__)
[subcircuitArray,unsupportedCommands] = subcircuit2ssc(__)
```

## Description

`subcircuit2ssc(filename,target)` reads the SPICE netlist specified by `filename` and converts every subcircuit into one or more Simscape files in the folder specified by `target`.

The function lists SPICE commands not supported by the conversion process in the comments of the corresponding Simscape files. After conversion, review the generated Simscape files and make manual edits for any unsupported items. You can also obtain a list of unsupported commands by using an optional output argument, described below.

For a detailed explanation of supported conversions, see “Converting a SPICE Netlist to Simscape Blocks”.

`subcircuit2ssc(__,subcircuit1,...,subcircuitN)` converts only the subcircuits with the specified names.

`subcircuitArray = subcircuit2ssc(__)` returns an array of objects containing the subcircuit information.

`[subcircuitArray,unsupportedCommands] = subcircuit2ssc(__)` returns an array of objects containing the subcircuit information and a `struct` array containing the subcircuit names and SPICE commands found in the converted subcircuits that are not supported by the conversion process.

## Examples

### Create Custom Simscape Blocks from a SPICE Netlist

Create a SPICE netlist named `rcsubcircuit.cir` that contains a simple RC subcircuit.

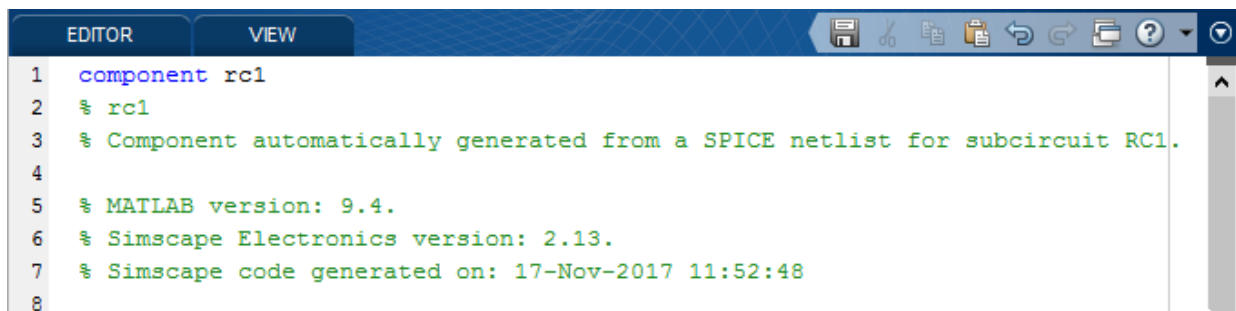
```
RCSUBCIRCUIT.CIR - RC SUBCIRCUIT
*
.SUBCKT RC1      1
*
R1      1      2      1k
C1      2      0      0.32mF
*
.ENDS
*
.END
```

Convert all SPICE subcircuits in `rcsubcircuit.cir` to equivalent Simscape files and place them in a package directory called `mylibrary`.

```
subcircuit2ssc('rcsubcircuit.cir','+mylibrary');
```

Netlist converted. Review files and make manual edits for any unsupported items before building the Simscape library located at: `+mylibrary`.

Check the comments at the beginning of the generated component file `rc1.ssc` in the `mylibrary` package to verify that no manual conversion is required.



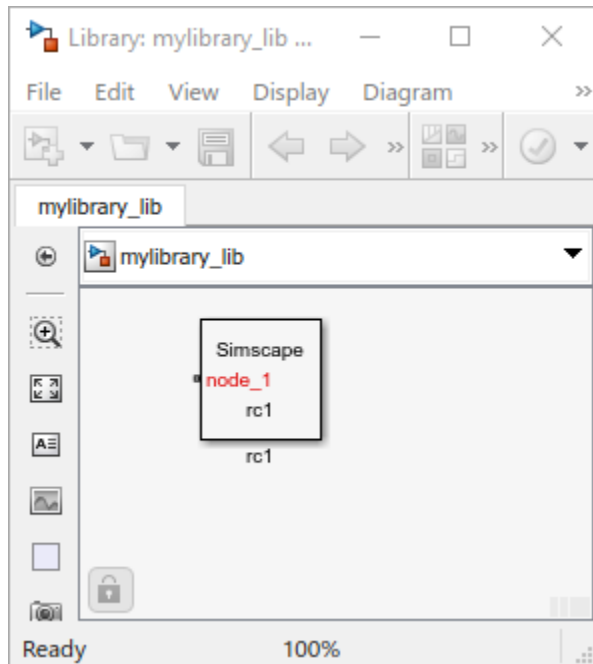
```
1 component rc1
2 % rc1
3 % Component automatically generated from a SPICE netlist for subcircuit RC1.
4
5 % MATLAB version: 9.4.
6 % Simscape Electronics version: 2.13.
7 % Simscape code generated on: 17-Nov-2017 11:52:48
8
```

Generate the Simscape library using `ssc_build`.

```
ssc_build mylibrary;
```

Generating Simulink library 'mylibrary\_lib' in the current directory

Open the generated library mylibrary\_lib.slx to access the RC component as a Simscape block.



### Make Manual Changes to a Converted Simscape Component File

Create a SPICE netlist named mutualinductor.cir that contains a simple mutual inductor.

```
MUTUALINDUCTOR.CIR - MUTUAL INDUCTOR SUBCIRCUIT
*
.SUBCKT MutualInductor      p1 n1 p2 n2
*
L1 p1 n1 1u
L2 p2 n2 5u
K1 L1 L2 0.6
*
```

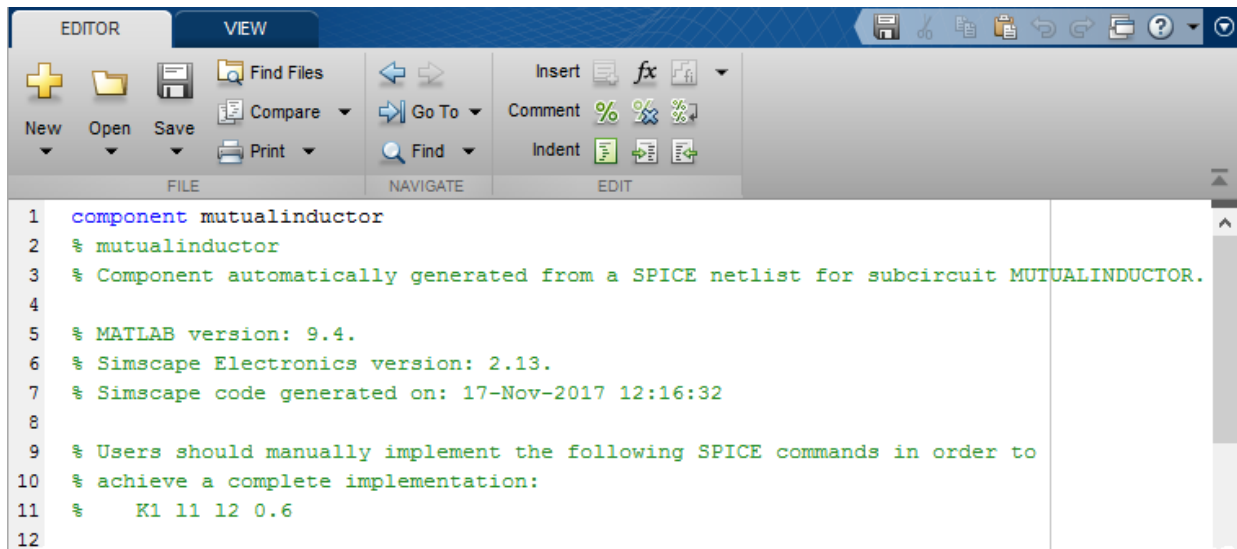
```
.ENDS  
*  
.END
```

Convert all SPICE subcircuits in `mutualinductor.cir` to equivalent Simscape files and place them in a package directory called `mylibrary`.

```
subcircuit2ssc('mutualinductor.cir','+mylibrary');
```

Netlist converted. Review files and make manual edits for any unsupported items before building the Simscape library located at: `+mylibrary`.

Check the comments at the beginning of the generated component file `mutualinductor.ssc` in the `mylibrary` package to identify required manual conversions.



```
1 component mutualinductor  
2 % mutualinductor  
3 % Component automatically generated from a SPICE netlist for subcircuit MUTUALINDUCTOR.  
4  
5 % MATLAB version: 9.4.  
6 % Simscape Electronics version: 2.13.  
7 % Simscape code generated on: 17-Nov-2017 12:16:32  
8  
9 % Users should manually implement the following SPICE commands in order to  
10 % achieve a complete implementation:  
11 % K1 11 12 0.6  
12
```

The comments suggest that the mutual inductance `K1` must be manually converted.

In the components section of the component file, replace the two inductors with a mutual inductor:

```
components(ExternalAccess=observe)  
M = foundation.electrical.elements.mutual_inductor(...
```

```
L1={ (1*1e-06), 'H'}, ...  
L2={ (5*1e-06), 'H'}, ...  
k={0.6, '1'});  
end
```

In the connections section of the component file, replace the connections to the two inductor nodes with connections to the mutual inductor nodes:

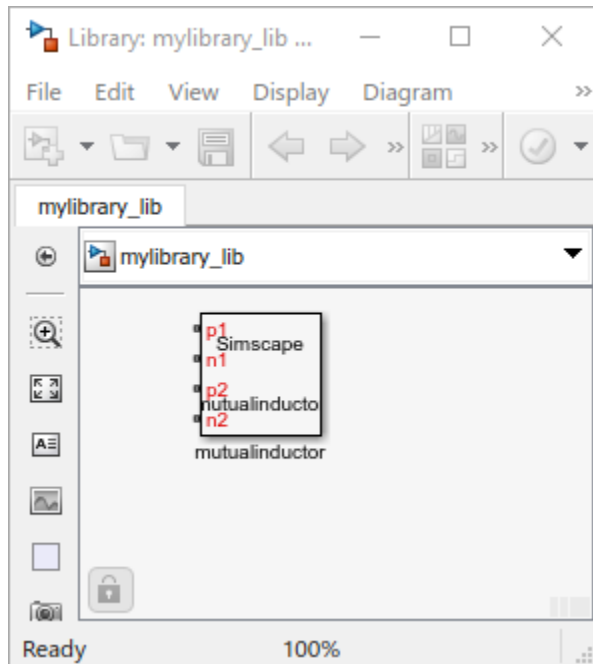
```
connections  
    connect(M.p1,p1);  
    connect(M.n1,n1);  
    connect(M.p2,p2);  
    connect(M.n2,n2);  
end
```

Generate the Simscape library using `ssc_build`.

```
ssc_build mylibrary;
```

Generating Simulink library 'mylibrary\_lib' in the current directory

Open the generated library `mylibrary_lib.slx` to access the mutual inductor component as a Simscape block.



## Input Arguments

### **filename** — SPICE network file

character vector (default) | string scalar

Name of the SPICE network file to read. This file must be on the path.

Example: 'SpiceSubcircuits.cir'

Data Types: char | string

### **target** — Location of generated Simscape component files

character vector (default) | string scalar

Name of the folder where the Simscape language files are generated. To allow the building of custom block libraries, specify a package directory with the '+' precursor. If the specified folder does not exist, the function creates it in the current folder.

Example: '+SimscapeSubcircuits'

Data Types: char | string

**subcircuit1, ..., subcircuitN — SPICE subcircuit names**

character vectors (default) | string scalars

Names of the SPICE subcircuits to convert to Simscape language files.

Example: 'Subcircuit1','Subcircuit2'

Data Types: char | string

## Output Arguments

**subcircuitArray — Array of objects containing the subcircuit information**

array

Array of objects containing the subcircuit information.

**unsupportedCommands — List of unsupported commands**

struct array

A struct array containing the subcircuit names and SPICE commands found in the converted subcircuits that are not supported by the conversion process.

## Limitations

- The netlist must be written in Cadence® PSpice format and be syntactically correct. The conversion assistant does not check for proper PSpice syntax.
- Only a subset of the PSpice netlist language is supported. However, unsupported PSpice commands are identified at the top of the corresponding Simscape component file to facilitate manual conversion.
- To build generated Simscape components into Simscape blocks, parameter values must conform to Simscape constraints. For example, capacitance of a fundamental capacitor and inductance of a fundamental inductor must be nonzero.

## **See Also**

### **Topics**

*“Converting a SPICE Netlist to Simscape Blocks”*

*“Building Custom Block Libraries” (Simscape)*

*“Composite Components” (Simscape)*

**Introduced in R2018b**



## pe\_calculateThdPercent

Compute the total harmonic distortion (THD) percentage

### Syntax

```
[thdPercent] = pe_calculateThdPercent(harmonicOrder,  
harmonicMagnitude)
```

### Description

[thdPercent] = pe\_calculateThdPercent(harmonicOrder, harmonicMagnitude) calculates the total harmonic distortion (THD) percentage using these equations:

$$M = \frac{\text{harmonic magnitude}}{\sqrt{2}},$$

and

$$\%THD = 100 \frac{\sqrt{\sum_{i=2}^n M_i^2}}{M_1},$$

where:

- $M_i$  is the root mean square (RMS) value of the harmonic magnitude corresponding to the  $i^{\text{th}}$  harmonic order.
- $M$  is  $V_{RMS}$  or  $I_{RMS}$  as required.

You can use the `pe_getHarmonics` function to obtain the vectors of harmonic order and harmonic magnitude for a `simscape.logging.Node`.

## Examples

### Calculate THD percent

Calculate the THD from harmonic orders [1;5;7;11;13] and harmonic magnitudes [1.1756e+03;0.0437e+03;0.0221e+03;0.0173e+03;0.0127e+03].

```
harmonicOrder = [1;5;7;11;13];  
harmonicMagnitude = [1.1756e+03;0.0437e+03;0.0221e+03;0.0173e+03;...  
    0.0127e+03];  
thdPercent = pe_calculateThdPercent( harmonicOrder, harmonicMagnitude )  
  
thdPercent = 4.5480
```

## Input Arguments

### **harmonicOrder** — Harmonic orders

vector

Harmonic orders from 0 up to and including number of harmonics, specified as a vector.

Example: [1;5;7;11;13]

Data Types: single | double | int8 | int16 | int32 | int64 | uint8 | uint16 |  
uint32 | uint64

### **harmonicMagnitude** — Harmonic magnitudes

vector

Harmonic magnitudes from the 0<sup>th</sup> harmonic up to and including the number of harmonics included in the analysis, specified as a vector.

Example: [1.1756e+03;0.0437e+03;0.0221e+03;0.0173e+03;0.0127e+03]

Data Types: single | double | int8 | int16 | int32 | int64 | uint8 | uint16 |  
uint32 | uint64

## See Also

### Blocks

Spectrum Analyzer

### Functions

pe\_getHarmonics | pe\_plotHarmonics

### Objects

simscape.logging.Node

## Topics

“Perform an Online Harmonic Analysis Using the Simscape Spectrum Analyzer Block”

“Choose a Simscape Electrical Power Systems Function for an Offline Harmonic Analysis”

“Data Logging” (Simscape)

“Harmonic Analysis of a Three-Phase Rectifier”

### Introduced in R2014a

## pe\_getEfficiency

Calculate efficiency as a function of dissipated power losses

### Syntax

```
efficiency = pe_getEfficiency('loadIdentifier',node)
efficiency = pe_getEfficiency('loadIdentifier',node,startTime,
endTime)
[efficiency,lossesTable] = pe_getEfficiency('loadIdentifier',node)
```

### Description

`efficiency = pe_getEfficiency('loadIdentifier',node)` returns the efficiency of a circuit based on the data extracted from a Simscape logging node.

Before you call this function, generate or load the simulation log variable to your workspace. To generate the variable, simulate the model with simulation data logging enabled. For more information, see “About Simulation Data Logging” (Simscape). To load a previously saved variable from a file, right-click on the file and select **Load**.

Checking efficiency allows you to determine if circuit components are operating within their requirements. Blocks in the Semiconductor > Fundamental Components library and the Delta-Connected Load, Wye-Connected Load, and RLC blocks have an internal block variable called *power\_dissipated*. This variable represents the instantaneous dissipated power, which includes only the real power (not the reactive or apparent power) that the block dissipates. When you log simulation data, the time-value series for this variable represents the power dissipated by the block over time. You can view and plot this data using the Simscape Results Explorer. The `pe_getPowerLossTimeSeries` function also allows you to access this data.

The `pe_getEfficiency` function calculates the efficiency of the circuit based on the losses for blocks that have a *power\_dissipated* variable and that you identify as a load block. The equation for efficiency is

$$Eff = 100 * \frac{P_{load}}{P_{loss} + P_{load}},$$

where:

- $Eff$  is the efficiency of the circuit.
- $P_{load}$  is the output power, that is, the power dissipated by load blocks.
- $P_{loss}$  is the power dissipated by nonload blocks.

This equation assumes that all loss mechanisms are captured by blocks containing at least one *power\_dissipated* variable. If the model contains any lossy blocks that do not have this variable, the efficiency calculation gives incorrect results.

Some blocks have more than one *power\_dissipated* variable, depending on their configuration. For example, for the MOSFET block, both the diode node and the ideal\_switch node have a *power\_dissipated* logging node. The function sums the power losses for both nodes to provide the total power loss for the block, averaged over simulation time. The function uses the loss data to calculate the efficiency of the circuit.

The nonideal semiconductor blocks also have thermal variants. Thermal variants have thermal ports that allow you to model the heat that is generated due to switching events and conduction losses. If you use a thermal variant, the function calculates power losses and efficiencies based on the thermal parameters that you specify. Essentially, the power dissipated is equal to the heat generated.

If you use a variant without a thermal port, the function calculates power losses and efficiencies based on the electrical parameters that you specify, such as on-state resistance and off-state conductance.

`efficiency = pe_getEfficiency('loadIdentifier',node,startTime, endTime)` returns the efficiency of a circuit based on the *power\_dissipated* data extracted from a Simscape logging node within a time interval. `startTime` and `endTime` represent the start and end of the time interval for calculating the efficiency. If you omit these two input arguments, the function calculates the efficiency over the whole simulation time.

`[efficiency,lossesTable] = pe_getEfficiency('loadIdentifier',node)` returns the efficiency of a circuit and the power loss contributions of the nonload blocks in a circuit based on the data extracted from a Simscape logging node.

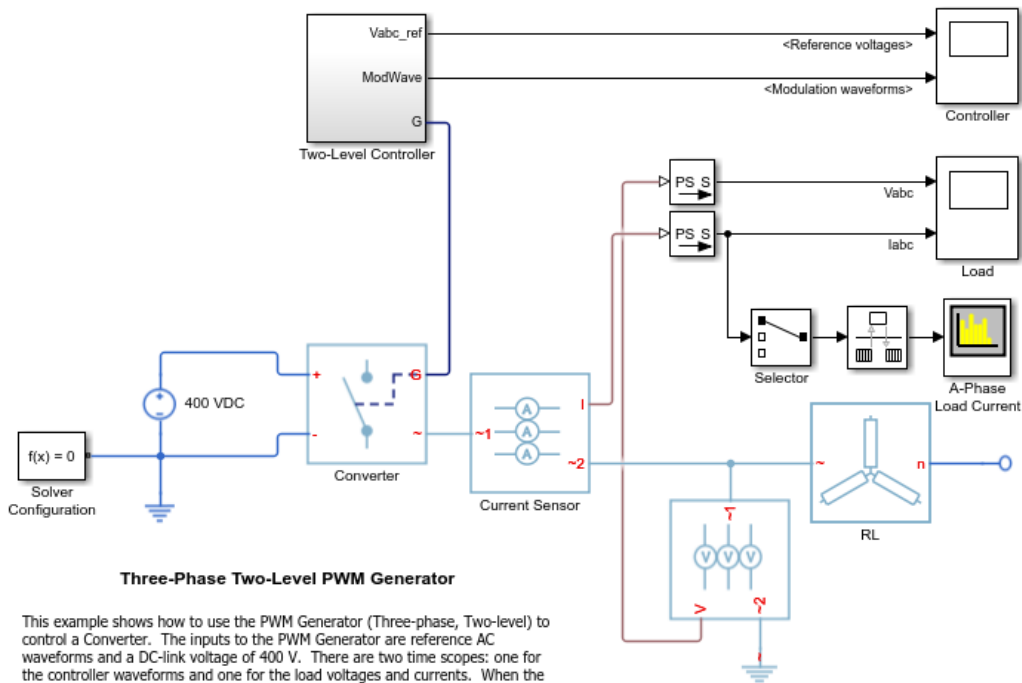
## Examples

### Calculate Efficiency for a Circuit

This example shows how to calculate efficiency based on the power dissipated by blocks in a circuit using the `pe_getEfficiency` function. Data logging is enabled locally, and the option to limit data points is off.

Open the model. At the MATLAB® command prompt, enter:

```
model = 'pe_pwm_two_level';
open_system(model)
```



Ensure that all blocks that have *power\_dissipated* variables are considered in the efficiency calculation. Enable data logging for the whole model.

```
set_param(model, 'SimscapeLogType', 'all')
```

Designate the load. Rename the Wye-Connected Load block from RL to RL\_Load.

```
set_param([model, '/RL'], 'Name', 'RL_Load')
```

Run the simulation and create the simulation log variable.

```
sim(model)
```

The simulation log variable *simlog\_pe\_pwm\_two\_level* is saved in the workspace.

Calculate the efficiency percentage.

```
efficiency = pe_getEfficiency('Load', simlog_pe_pwm_two_level)
```

```
efficiency =
```

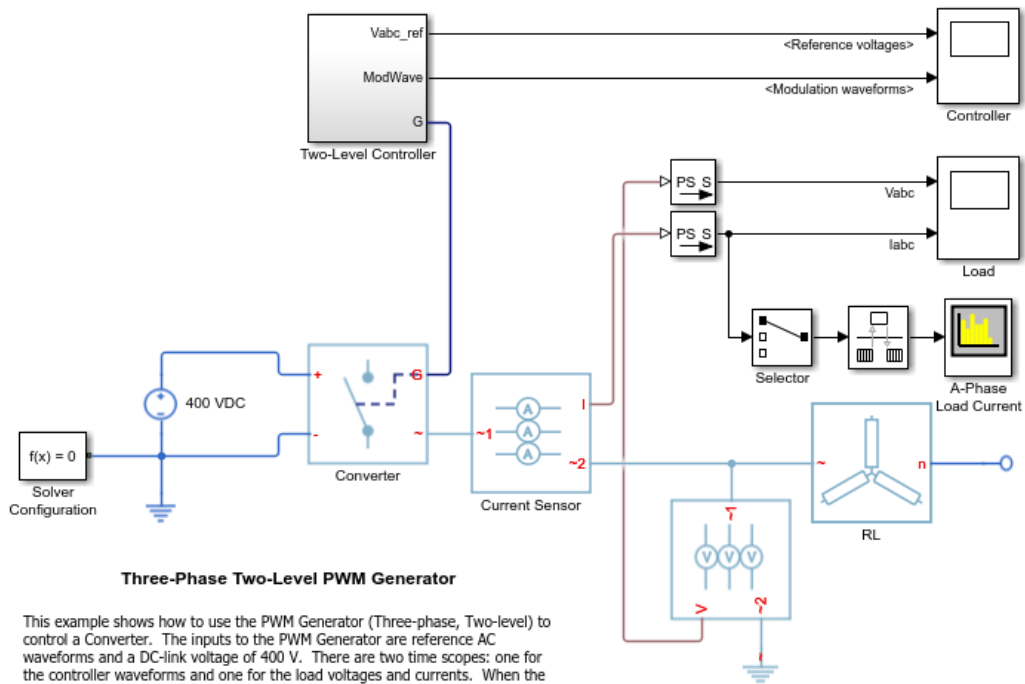
```
    99.1940
```

### **Calculate Efficiency of a Circuit for a Specific Time Period**

This example shows how to calculate efficiency based on the power dissipated for a specific time period using the `pe_getEfficiency` function. Data logging is enabled locally, and the option to limit data points is off.

Open the model. At the MATLAB® command prompt, enter:

```
model = 'pe_pwm_two_level';  
open_system(model)
```



This example shows how to use the PWM Generator (Three-phase, Two-level) to control a Converter. The inputs to the PWM Generator are reference AC waveforms and a DC-link voltage of 400 V. There are two time scopes: one for the controller waveforms and one for the load voltages and currents. When the model is executed, a Spectrum Analyzer opens and displays frequency data for the A-Phase Load Current.

Ensure that all blocks that have *power\_dissipated* variables are considered in the efficiency calculation. Enable data logging for the whole model.

```
set_param(model, 'SimscapeLogType', 'all')
```

Designate the load. Rename the Wye-Connected Load block from RL to RL\_Load.

```
set_param([model, '/RL'], 'Name', 'RL_Load')
```

Run the simulation and create the simulation log variable.

```
sim(model)
```

The simulation log variable *simlog\_pe\_pwm\_two\_level* is saved in the workspace.



The model simulation stop time is 0.2 seconds. Calculate efficiency for the interval when the simulation time,  $t$ , is between 0.00 and 0.005 seconds.

```
efficiency = pe_getEfficiency('Load',simlog_pe_pwm_two_level,0.000,0.005)
```

```
efficiency =
```

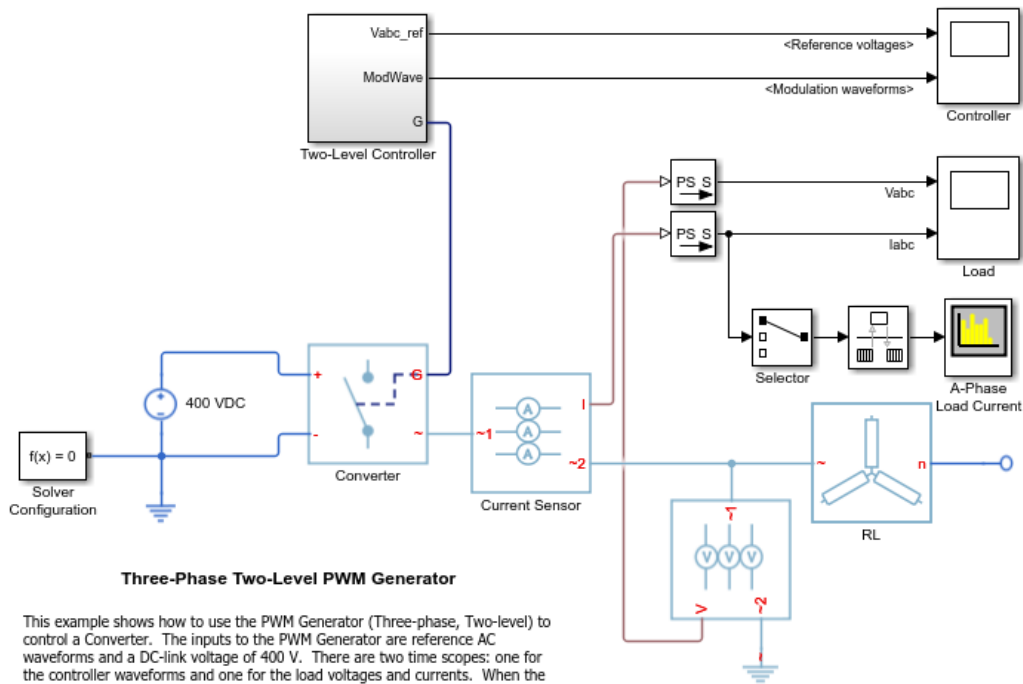
```
    99.1093
```

### Calculate Efficiency and Power-Loss Contributions

This example shows how using the `pe_getEfficiency` function allows you to calculate both the efficiency of the circuit and the power-loss contributions of the nonload blocks based on the power that they dissipate. Data logging is enabled locally, and the option to limit data points is off.

Open the model. At the MATLAB® command prompt, enter:

```
model = 'pe_pwm_two_level';  
open_system(model)
```



This example shows how to use the PWM Generator (Three-phase, Two-level) to control a Converter. The inputs to the PWM Generator are reference AC waveforms and a DC-link voltage of 400 V. There are two time scopes: one for the controller waveforms and one for the load voltages and currents. When the model is executed, a Spectrum Analyzer opens and displays frequency data for the A-Phase Load Current.

Ensure that all blocks that have *power\_dissipated* variables are considered in the efficiency calculation. Enable data logging for the whole model.

```
set_param(model, 'SimscapeLogType', 'all')
```

Designate the load. Rename the Wye-Connected Load block from RL to RL\_Load.

```
set_param([model, '/RL'], 'Name', 'RL_Load')
```

Run the simulation and create the simulation log variable.

```
sim(model)
```

The simulation log variable *simlog\_pe\_pwm\_two\_level* is saved in the workspace.

Calculate the efficiency and power-loss contributions due to dissipated power.

```
[efficiency,lossesTable] = pe_getEfficiency('Load',simlog_pe_pwm_two_level)
```

```
efficiency =
```

```
99.1940
```

```
lossesTable =
```

```
1×2 table
```

LoggingNode	Power
'pe_pwm_two_level.Converter'	268.73

## Input Arguments

### 'loadIdentifier' — Identify load blocks in the circuit

case-sensitive string

String that is a complete or partial match for the names of load blocks in the circuit. For example, consider a circuit that contains the blocks shown in the table.

Block Name in the Model		DC Impedance	AC Impedance	Y-Ld
<b>Block Type</b>		RLC	RLC	Wye-Connected Load
<b>Block Role in the Model</b>		Source Impedance	Load Impedance	Load
<b>'loadIdentifier'</b>	'd'	Yes	Yes	Yes
	'Load'	No	No	No
	'D'	Yes	No	No

The `pe_getEfficiency` function does not return the correct data for any of these 'loadIdentifier' values.

A load-block naming schema that gives you better control over the output of the `pe_getEfficiency` function is shown in this table.

Block Name in the Model		DC Impedance	AC Impedance_Load_1	Y-Load_2
Block Type		RLC	RLC	Wye-Connected Load
Block Role in the Model		Source Impedance	Load Impedance	Load
'loadIdentifier'	'1'	No	Yes	No
	'2'	No	No	Yes
	'Load'	No	Yes	Yes

Example: 'Load'

Data Types: string

**node — Simulation log variable, or a specific node within the simulation log variable**

Node object

Simulation log workspace variable, or a node within this variable, that contains the logged model simulation data, specified as a Node object. You specify the name of the simulation log variable by using the **Workspace variable name** parameter on the **Simscape** pane of the Configuration Parameters dialog box. To specify a node within the simulation log variable, provide the complete path to that node through the simulation data tree, starting with the top-level variable name.

If `node` is the name of the simulation log variable, then the table contains the data for all blocks in the model that contain `power_dissipated` variables. If `node` is the name of a node in the simulation data tree, then the table contains the data only for:

- Blocks or variables within that node
- Blocks or variables within subnodes at all levels of the hierarchy beneath that node

Example: `simlog_pe_pwm_two_level`

**startTime — Start of the time interval for calculating the efficiency**

0 (default) | real number

Start of the time interval for calculating the efficiency, specified as a real number, in seconds. `startTime` must be greater than or equal to the simulation **Start time** and less than `endTime`.

Data Types: `double`

**endTime** — End of the time interval for calculating the efficiency

simulation stop time (default) | real number

End of the time interval for calculating the efficiency, specified as a real number, in seconds. `endTime` must be greater than `startTime` and less than or equal to the simulation **Stop time**.

Data Types: `double`

## Output Arguments

**efficiency** — Efficiency of the circuit

percentage

Efficiency of the circuit based on data extracted from a Simscape logging node.

**lossesTable** — Dissipated power for each nonload blocks

table

Dissipated power losses for each nonload block, returned as a table. The first column lists logging nodes for all blocks that have at least one *power\_dissipated* variable. The second column lists the corresponding losses in watts.

## Assumptions

- The output power equals the total power dissipated by blocks that you identify as load blocks.
- The input power equals the output power plus the total power dissipated by blocks that you do not identify as load blocks.
- The *power\_dissipated* variables capture all loss contributions.

## **See Also**

`pe_getPowerLossSummary` | `pe_getPowerLossTimeSeries` | `sscexplore`

## **Topics**

“Perform a Power-Loss Analysis”

“Data Logging” (Simscape)

“About the Simscape Results Explorer” (Simscape)

**Introduced in R2017a**

## pe\_getHarmonics

Return harmonic orders, magnitudes, and fundamental frequency

### Syntax

```
[harmonicOrder,harmonicMagnitude,fundamentalFrequency] =...
pe_getHarmonics(loggingNode)
[harmonicOrder,harmonicMagnitude,fundamentalFrequency] =...
pe_getHarmonics(loggingNode,valueIdx)
[harmonicOrder,harmonicMagnitude,fundamentalFrequency] =...
pe_getHarmonics(loggingNode,valueIdx,tOfInterest)
[harmonicOrder,harmonicMagnitude,fundamentalFrequency] =...
pe_getHarmonics(loggingNode,valueIdx,tOfInterest,nPeriodOfInterest)
[harmonicOrder,harmonicMagnitude,fundamentalFrequency] =...
pe_getHarmonics(loggingNode,valueIdx,tOfInterest,
nPeriodOfInterest,...
offsetOfInterest)
[harmonicOrder,harmonicMagnitude,fundamentalFrequency] =...
pe_getHarmonics(loggingNode,valueIdx,tOfInterest,
nPeriodOfInterest,...
offsetOfInterest,nHarmonic)
```

### Description

[harmonicOrder,harmonicMagnitude,fundamentalFrequency] =...  
 pe\_getHarmonics(loggingNode) calculates the harmonic orders, magnitudes, and fundamental frequency of a `simscape.logging.Node` of an AC or periodic variable.

The function finds the points in the  $i^{\text{th}}$  signal (valueIdx) where the Simscape log crosses a threshold (offsetOfInterest). It uses the crossing points to find the required number of periods (nPeriodOfInterest) preceding the specified time (tOfInterest). Then it inputs the down-selected data to the Goertzel algorithm, which calculates the harmonic magnitudes up to and including the required number of harmonics (nHarmonic).

You enter the input arguments in a specific order. The Simscape logging node input argument is required. All other input arguments are optional and have default values. If

you are specifying a value for a subsequent optional input argument, enter `[]` to use the default value for an optional input argument.

You can use the `pe_plotHarmonics` function to obtain a bar chart from the same input arguments. You can use the outputs of this function as inputs to the `pe_calculateThdPercent` function to calculate the total harmonic distortion (THD) percentage.

```
[harmonicOrder,harmonicMagnitude,fundamentalFrequency] =...  
pe_getHarmonics(loggingNode,valueIdx) uses the index into value data.
```

```
[harmonicOrder,harmonicMagnitude,fundamentalFrequency] =...  
pe_getHarmonics(loggingNode,valueIdx,tOfInterest) uses the simulation time.
```

```
[harmonicOrder,harmonicMagnitude,fundamentalFrequency] =...  
pe_getHarmonics(loggingNode,valueIdx,tOfInterest,nPeriodOfInterest)  
uses the number of periods of fundamental frequency.
```

```
[harmonicOrder,harmonicMagnitude,fundamentalFrequency] =...  
pe_getHarmonics(loggingNode,valueIdx,tOfInterest,  
nPeriodOfInterest,...  
offsetOfInterest) uses the DC offset.
```

```
[harmonicOrder,harmonicMagnitude,fundamentalFrequency] =...  
pe_getHarmonics(loggingNode,valueIdx,tOfInterest,  
nPeriodOfInterest,...  
offsetOfInterest,nHarmonic) uses the number of harmonics.
```

## Examples

### Analyze Using Default Values

This set of function arguments uses the Simscape logging node `simlog.Load.V`, which contains data from a three-phase voltage. The function analyzes the default signal, which is the first, or a-phase, signal at the final simulation time. The function uses the default values of 12 for the number of periods of the signal, 0V for the signal bias, and 30 for the number of harmonics.

```
pe_getHarmonics(simLog.Load.V)
```



## Analyze Using Specified Values

This set of function arguments uses the Simscape logging node `simlog.Load.V`, which contains data from a three-phase voltage. The function analyzes the second, or b-phase, signal at a simulation time of 2.3 s. The function uses 10 periods of the signal, which has a bias of 1 V. The function analyzes 15 harmonics.

```
pe_getHarmonics(simlog.Load.V,2,2.3,10,1,15)
```

## Analyze Using Default and Specified Values

This set of function arguments uses the Simscape logging node `simlog.Load.V`, which contains data from a three-phase voltage. The function analyzes the first, or a-phase, signal at a simulation time of 2.3 s. The function uses 12 periods of the signal, which has a bias of 1 V. The function analyzes the default number, 30, of harmonics.

```
pe_getHarmonics(simlog.Load.V,[],2.3,[],1)
```

## Input Arguments

### LoggingNode — Simscape logging node

1-by-1 `simscape.logging.Node`

Simscape logging node, specified as a 1-by-1 `simscape.logging.Node`. You create a `simscape.logging.Node` by running a simulation with Simscape logging enabled. For information, see “Enable Data Logging for the Whole Model” (Simscape).

Example: `simlog.Load.V`

The Simscape logging node `simlog.Load.V` contains data from a three-phase voltage.

### valueIdx — Index into value data

1 (default) | scalar

Index into value data, specified as a scalar. Specifies the  $i^{\text{th}}$  variable of interest in the Simscape log.

Example: 2

Specify the b-phase, which is the second signal from a three-phase voltage.

Example: []

Use [] to specify the default value of 1. The a-phase, which is the first signal from a three-phase voltage, is the default signal of interest.

Data Types: `single` | `double` | `int8` | `int16` | `int32` | `int64` | `uint8` | `uint16` | `uint32` | `uint64`

### **t0fInterest** — Simulation time

final time in Simscape log (default) | scalar

Simulation time of interest for harmonic analysis, specified as a scalar.

Example: 2.3

Specify a 2.3 s simulation time.

Data Types: `single` | `double` | `int8` | `int16` | `int32` | `int64` | `uint8` | `uint16` | `uint32` | `uint64`

### **nPeriodOfInterest** — Number of periods

12 (default) | scalar

Number of periods of fundamental frequency to be included in harmonic analysis, specified as a scalar.

Example: 10

Specify 10 periods of the signal.

Data Types: `single` | `double` | `int8` | `int16` | `int32` | `int64` | `uint8` | `uint16` | `uint32` | `uint64`

### **offset0fInterest** — DC offset

0 (default) | scalar

DC offset in the input signal, specified as a scalar. The function uses this value to find the periods of interest.

Example: 1

Specify a bias of 1 V for the signal.

Data Types: `single` | `double` | `int8` | `int16` | `int32` | `int64` | `uint8` | `uint16` | `uint32` | `uint64`

### **nHarmonic** — Number of harmonics

30 (default) | scalar

Number of harmonics to include in analysis, specified as a scalar.

Example: 15

Specify that the number of harmonics to be analyzed is 15.

Data Types: `single` | `double` | `int8` | `int16` | `int32` | `int64` | `uint8` | `uint16` | `uint32` | `uint64`

## Output Arguments

### **harmonicOrder** — Harmonic order

vector

Harmonic orders from 0 up to and including the number of harmonics used in the analysis, returned as a vector.

### **harmonicMagnitude** — Harmonic magnitude

vector

Harmonic magnitudes from the 0<sup>th</sup> harmonic up to and including the number of harmonics used in the analysis, returned as a vector.

### **fundamentalFrequency** — Fundamental frequency

scalar

Fundamental frequency over the range of the down-selected input data, returned as a scalar.

## Limitations

- This function requires that you use a fixed-step solver for the Simscape Electrical Power Systems network that you are analyzing. To specify a fixed-step solver for the physical network, use one of the configuration combinations in the table.

Configuration Combination	Global Solver Configuration	Local Solver Configuration
Global variable-step with local fixed-step	Set <b>Type</b> to Variable-step	Enable the options to <b>Use local solver</b> and <b>Use fixed-cost runtime consistency iterations</b>
Global and local fixed-step	Set <b>Type</b> to Fixed-step	Enable the options to <b>Use local solver</b> and <b>Use fixed-cost runtime consistency iterations</b>
Global fixed-step	Set <b>Type</b> to Fixed-step	Clear the option to <b>Use local solver</b>

- This function uses threshold crossing points to determine the fundamental frequency of the data. If your input data is noisy or crosses the threshold more frequently than half of the fundamental period, filter it before you use this function to analyze it.
- This function requires a minimal number of periods. If the minimal number is not met, the function generates a warning message. To increase the number of periods, use one or both of these methods:
  - Increase the simulation time.
  - Increase the switching frequency.

## See Also

### Blocks

Spectrum Analyzer

### Functions

pe\_calculateThdPercent | pe\_plotHarmonics

### Objects

simscape.logging.Node

### Topics

“Perform an Online Harmonic Analysis Using the Simscape Spectrum Analyzer Block”

“Choose a Simscape Electrical Power Systems Function for an Offline Harmonic Analysis”  
“Data Logging” (Simscape)  
“Harmonic Analysis of a Three-Phase Rectifier”

**Introduced in R2014a**

# pe\_getPowerLossSummary

Calculate dissipated power losses

## Syntax

```
lossesTable = pe_getPowerLossSummary(node)
lossesTable = pe_getPowerLossSummary(node, startTime, endTime)
```

## Description

`lossesTable = pe_getPowerLossSummary(node)` calculates dissipated power losses for semiconductor blocks in a model, based on logged simulation data, and returns the data for each block in a table.

Before you call this function, generate or load the simulation log variable into your workspace. To generate the variable, simulate the model with simulation data logging enabled. For more information, see “About Simulation Data Logging” (Simscape). To load a previously saved variable from a file, right-click on the file and select **Load**.

Checking dissipated power allows you to determine if circuit components are operating within their efficiency requirements. Blocks in the **Semiconductor > Fundamental Components** library have an internal variable called *power\_dissipated*. This variable represents the instantaneous dissipated power, which includes only the real power (not the reactive or apparent power) that the block dissipates. When you log simulation data, the time-value series for this variable represents the power dissipated by the block over time. You can view and plot this data using the Simscape Results Explorer. The `pe_getPowerLossTimeSeries` function also allows you to access this data from a cell array.

The `pe_getPowerLossSummary` function calculates average losses for each block that has a *power\_dissipated* variable. Some blocks have more than one *power\_dissipated* variable, depending on their configuration. For example, for the MOSFET block, both the diode node and the `ideal_switch` node have a `power_dissipated` logging node. The function sums the power losses for both nodes to provide the total power loss for the block, averaged over simulation time.

The nonideal semiconductor blocks also have thermal variants. Thermal variants have thermal ports that allow you to model the heat that is generated due to switching events and conduction losses. If you use a thermal variant, the function calculates power losses based on the thermal parameters that you specify. Essentially, the power dissipated is equal to the heat generated.

If you use a variant without a thermal port, the function calculates power losses based on the electrical parameters that you specify, such as on-state resistance and off-state conductance.

`lossesTable = pe_getPowerLossSummary(node, startTime, endTime)` calculates dissipated power losses within a time interval. `startTime` and `endTime` represent the start and end of the time interval for averaging the power losses. If you omit these two input arguments, the function averages the power losses over the total simulation time.

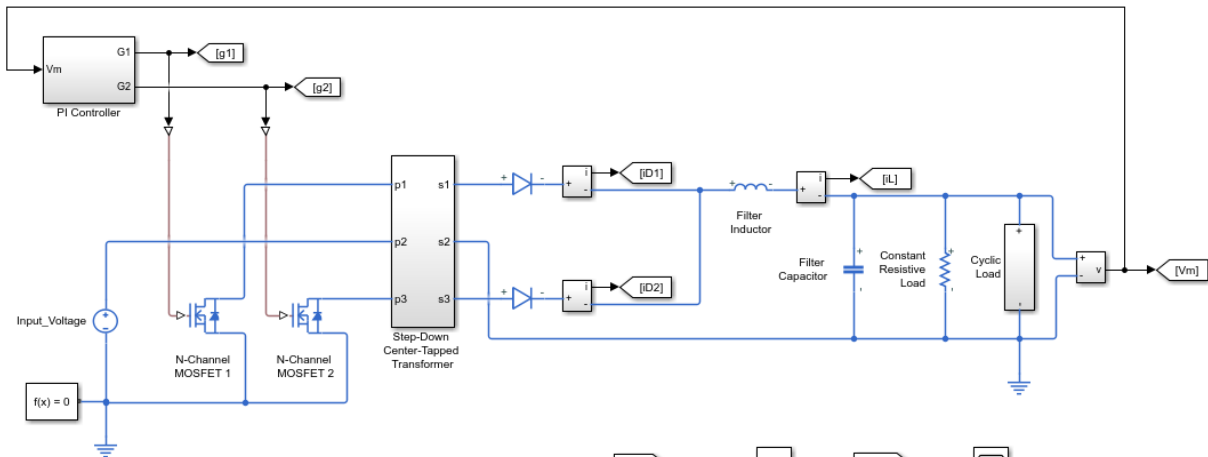
## Examples

### Calculate Average Power Losses by Block for the Whole Model

This example uses the Push-Pull Buck Converter in Continuous Conduction Mode model. Data logging is enabled for the whole model and the option to limit data points is off.

1. Open the example model. At the MATLAB® command prompt, enter

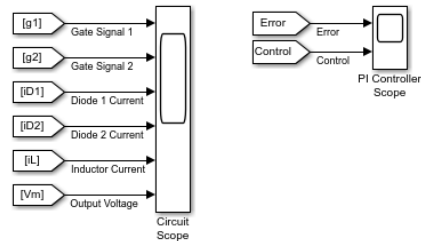
```
model = 'pe_push_pull_converter_ccm';  
open_system(model)
```



### Push-Pull Buck Converter in Continuous Conduction Mode

This example shows how to control the output voltage of a push-pull buck converter. The current flowing through the inductor is never zero, therefore the DC-DC converter operates in Continuous Conduction Mode (CCM). To convert and maintain the nominal output voltage, the PI Controller subsystem uses a simple integral control. During startup, the reference voltage is ramped up to the desired output voltage.

1. [Edit the parameterization script.](#)
2. [Explore simulation results using `sscexplore`.](#)
3. [Learn more](#) about this example.



2. Run the simulation and create the simulation log variable.

```
sim(model);
```

The simulation log variable `simlog_pe_push_pull_converter_ccm` is saved in your workspace.

3. Calculate average power losses for each semiconductor in the model and display the results in a table.

```
tabulatedLosses = pe_getPowerLossSummary(simlog_pe_push_pull_converter_ccm)
```

```
tabulatedLosses =
```

```
5x2 table
```

```
LoggingNode
```



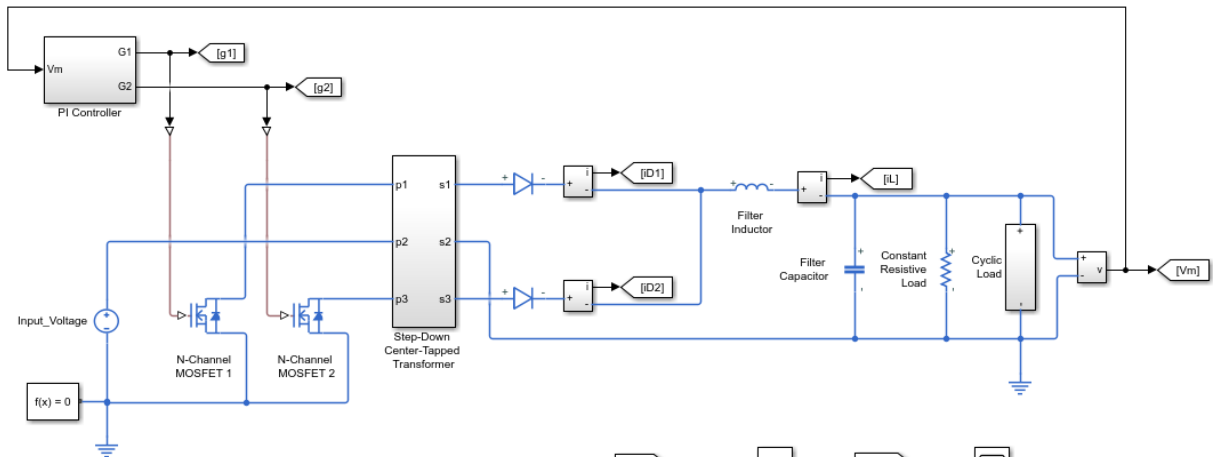
```
'pe_push_pull_converter_ccm.Diode1'  
'pe_push_pull_converter_ccm.Diode'  
'pe_push_pull_converter_ccm.N_Channel_MOSFET_1.mosfet_equation'  
'pe_push_pull_converter_ccm.N_Channel_MOSFET_2.mosfet_equation'  
'pe_push_pull_converter_ccm.Step_Down_Center_Tapped_Transformer.Eddy_Current'
```

The table shows dissipated power losses for each of the N-Channel MOSFET and Diode blocks, averaged over the entire simulation time.

### Calculate Average Power Losses for a Single Block

1. Open the Push-Pull Buck Converter in Continuous Conduction Mode example model. At the MATLAB® command prompt, enter

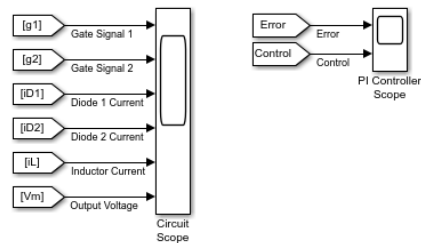
```
model='pe_push_pull_converter_ccm';  
open_system(model)
```



### Push-Pull Buck Converter in Continuous Conduction Mode

This example shows how to control the output voltage of a push-pull buck converter. The current flowing through the inductor is never zero, therefore the DC-DC converter operates in Continuous Conduction Mode (CCM). To convert and maintain the nominal output voltage, the PI Controller subsystem uses a simple integral control. During startup, the reference voltage is ramped up to the desired output voltage.

1. [Edit the parameterization script.](#)
2. [Explore simulation results using `sscexplore`.](#)
3. [Learn more](#) about this example.



The model has data logging enabled.

2. Run the simulation and create the simulation log variable.

```
sim(model)
```

The simulation log variable `simlog_pe_push_pull_converter_ccm` is saved in your workspace.

3. Calculate power losses for the N-Channel MOSFET 1 block and display the results in a table.

```
tabulatedLosses = pe_getPowerLossSummary(simlog_pe_push_pull_converter_ccm.N_Channel_MOSFET_1)
```

```
tabulatedLosses =
```

```
1x2 table
```

LoggingNode	Power
'N_Channel_MOSFET_1.mosfet_equation'	0.24497

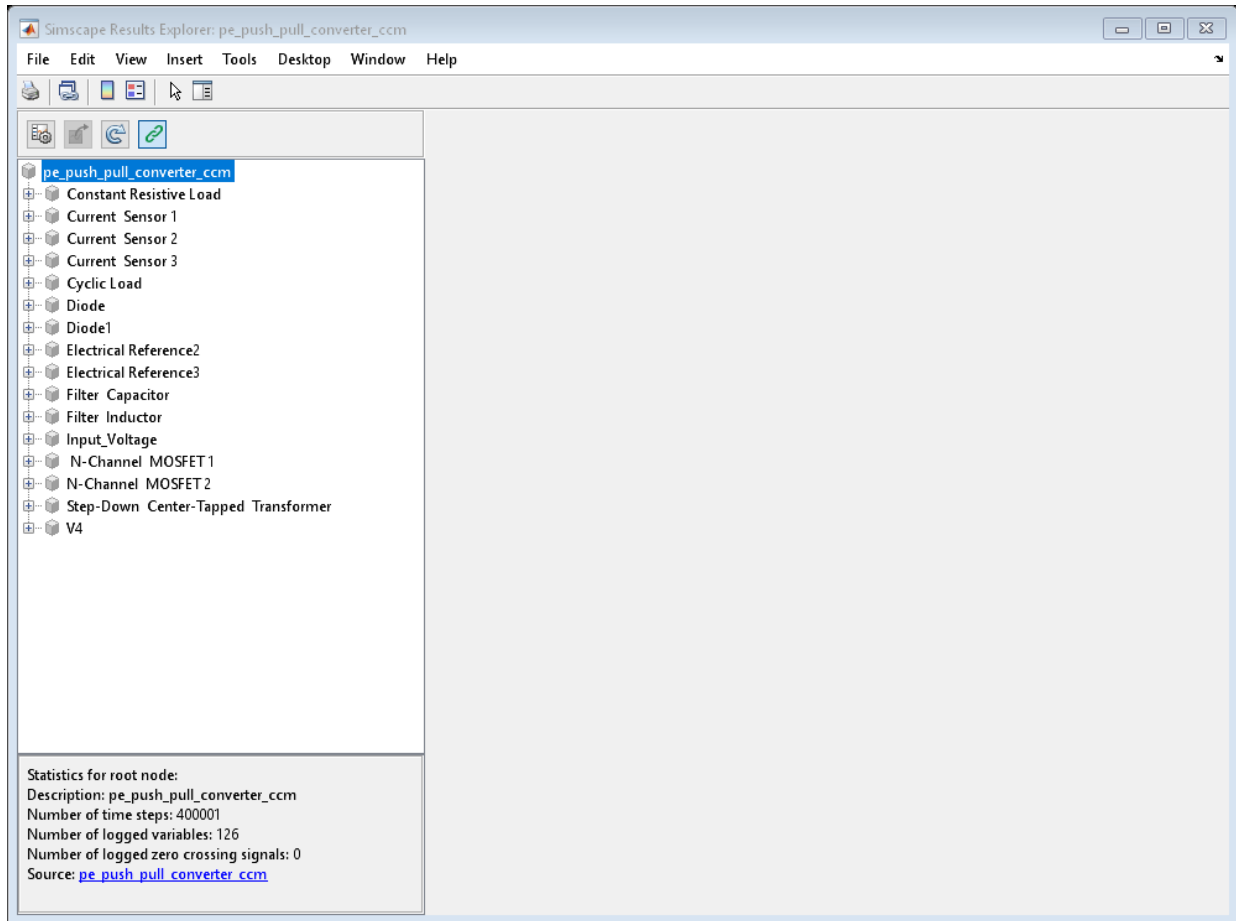
The table shows dissipated power losses for just the N-Channel MOSFET 1 block, averaged over the total simulation time.

4. Use the `sscexplore` function to explore the power loss data for the N-Channel MOSFET 1 block further.

a. Open the [Results Explorer](#).

```
sscexplore(simlog_pe_push_pull_converter_ccm)
```

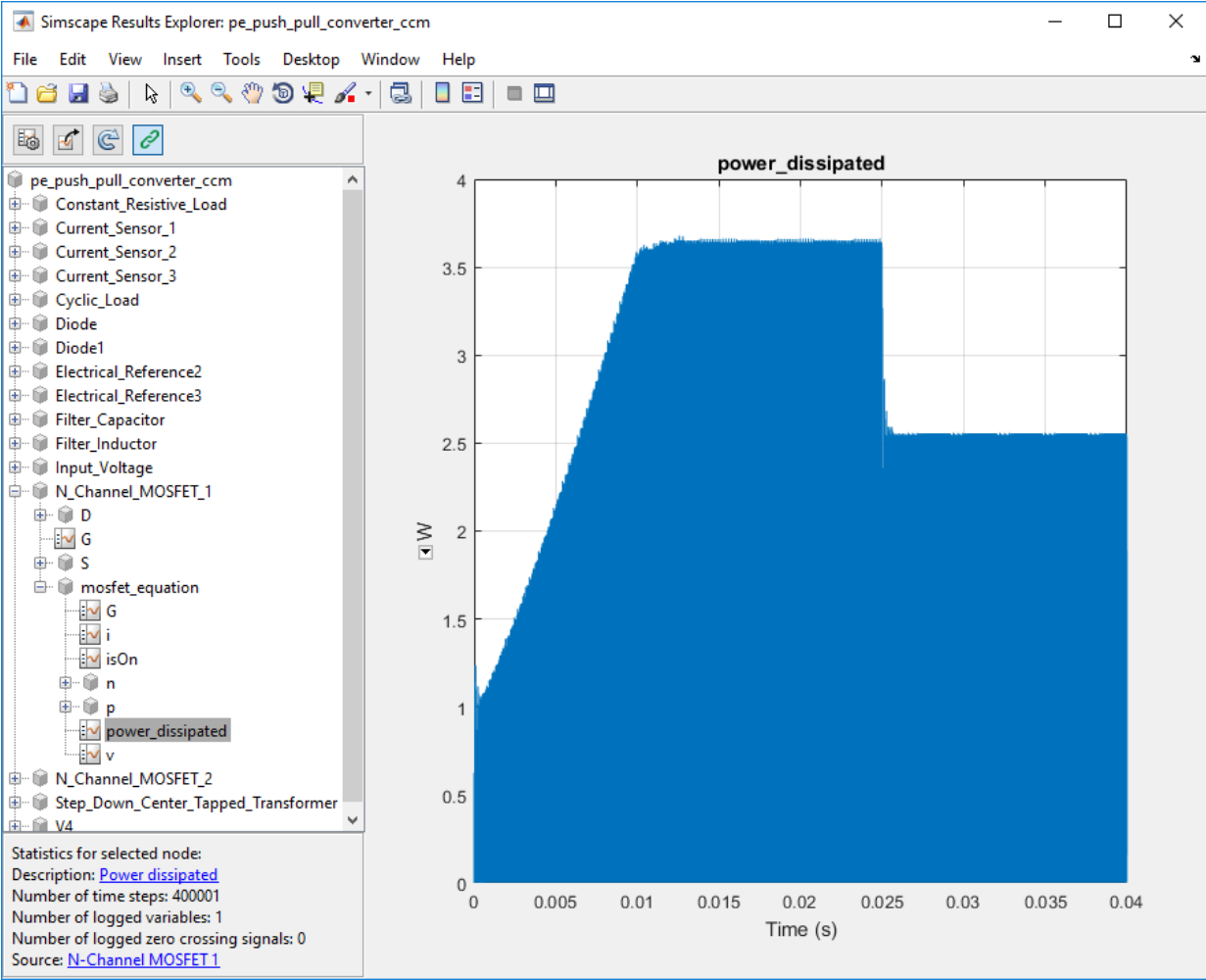
## 2 Functions — Alphabetical List



b. Expand these nodes:

- N\_Channel\_MOSFET\_1
- mosfet\_equation

c. Click the power\_dissipated node.



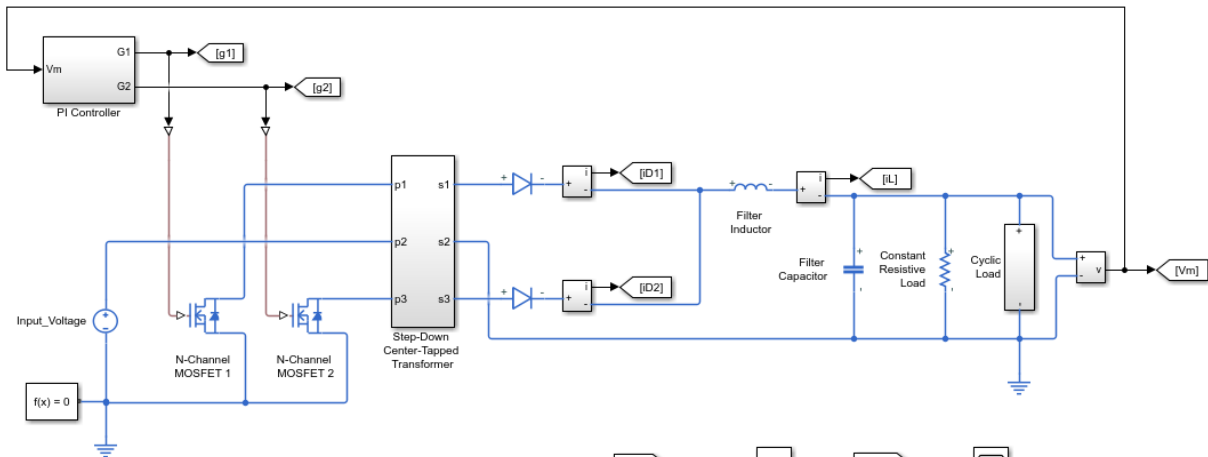
The N-Channel MOSFET 1 block has one `power_dissipated` variable for the `mosfet_equation` node. The `power_dissipated` figure shows the instantaneous power loss for the N-Channel MOSFET 1 block during the simulation.

## Calculate Average Power Losses for Components of a Block

You can calculate average power losses for the individual components of a block in your model.

1. Open the Push-Pull Buck Converter in Continuous Conduction Mode example model. At the MATLAB® command prompt, enter

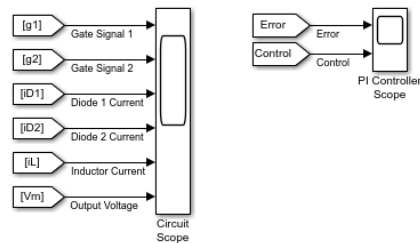
```
model = 'pe_push_pull_converter_ccm';
open_system(model)
```



### Push-Pull Buck Converter in Continuous Conduction Mode

This example shows how to control the output voltage of a push-pull buck converter. The current flowing through the inductor is never zero, therefore the DC-DC converter operates in Continuous Conduction Mode (CCM). To convert and maintain the nominal output voltage, the PI Controller subsystem uses a simple integral control. During startup, the reference voltage is ramped up to the desired output voltage.

1. [Edit the parameterization script.](#)
2. [Explore simulation results](#) using `sscexplore`.
3. [Learn more](#) about this example.



The model has data logging enabled.

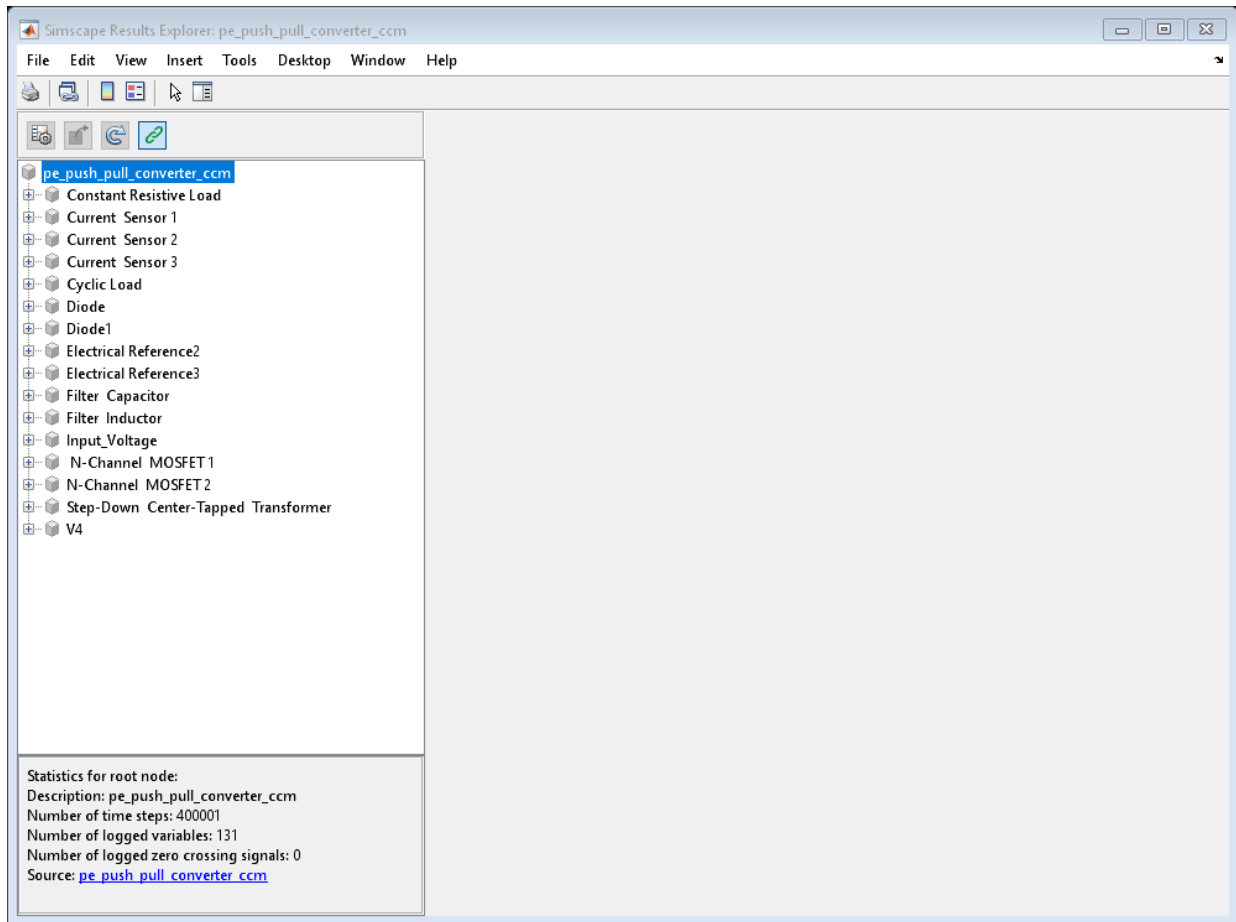
2. Add a diode component in the N-Channel MOSFET 1 block using the MATLAB® command prompt:

```
set_param('pe_push_pull_converter_ccm/ N-Channel MOSFET 1', 'diode_param', '2')
```

Alternatively, you can add the component in the Simulink® Editor:

- a. Open the Property Inspector pane. In the model window, in the menu bar, click **View > Property Inspector**
  - b. Click the N-Channel MOSFET1 block to access the block parameters.
  - c. In the Property Inspector pane, expand the Integral Diode setting and change the value for the **Integral protection** from None to Protection diode with no dynamics.
3. Run the simulation, create a simulation log variable, and open the simlog in the Simscape Results Explorer using the `sscexplore` function.

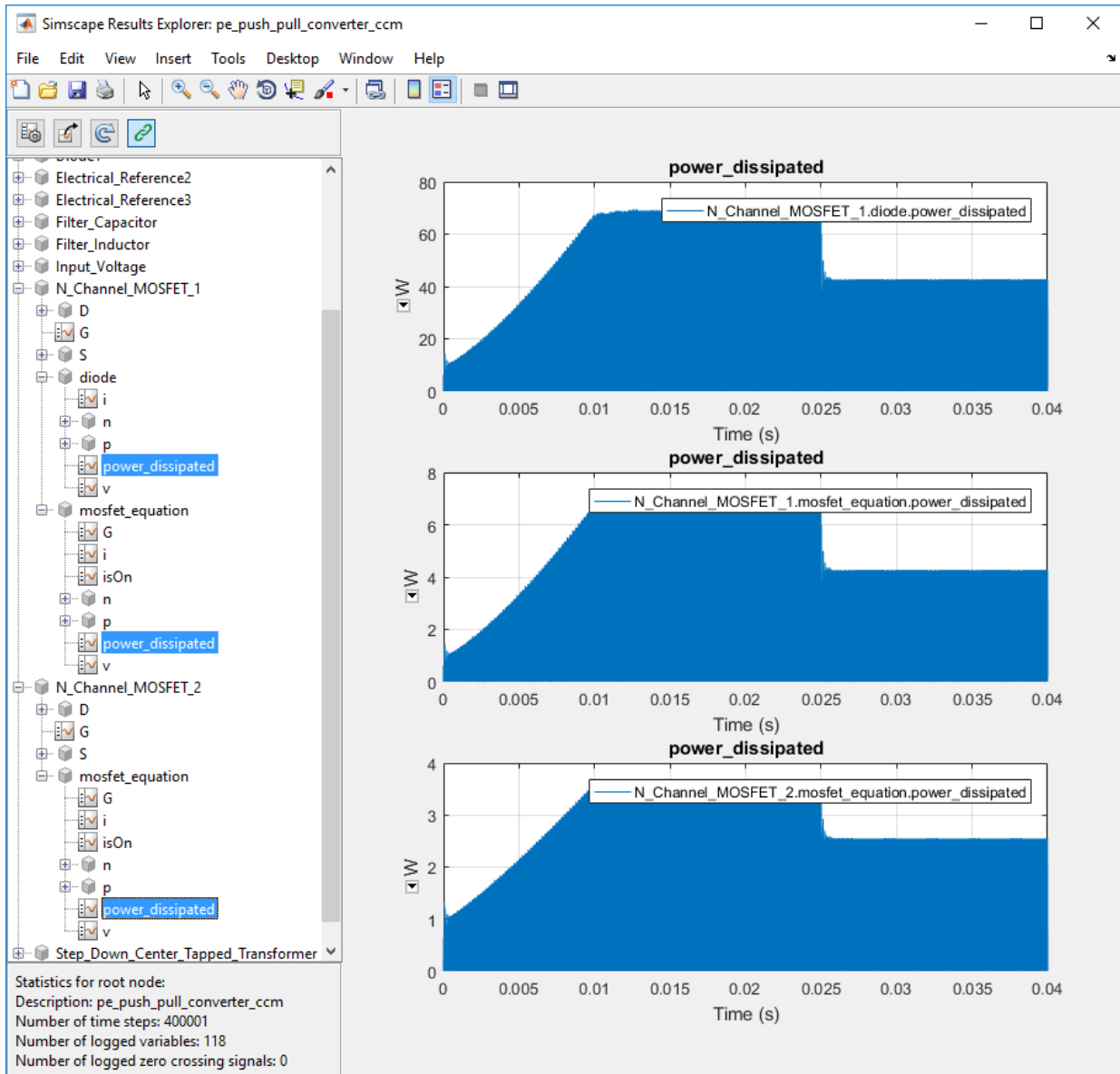
```
sim(model)
sscexplore(simlog_pe_push_pull_converter_ccm)
```



4. View the power loss data for the two N-Channel MOSFET blocks, expand these nodes and CTRL + click the power\_dissipated nodes:

- N\_Channel\_MOSFET\_1 > diode > power\_dissipated
- N\_Channel\_MOSFET\_1 > mosfet\_equation > power\_dissipated
- N\_Channel\_MOSFET\_2 > mosfet\_equation > power\_dissipated





The N-Channel MOSFET 2 block has only one `power_dissipated` variable. The N-Channel MOSFET 1 block has one `power_dissipated` variable for each of the two components (MOSFET and diode) that the block contains.

5. Calculate power losses for both components of the N-Channel MOSFET 1 block and display the results in a table

```
tabulatedLosses = pe_getPowerLossSummary(simlog_pe_push_pull_converter_ccm.N_Channel_MO
```

```
tabulatedLosses =
```

```
1x2 table
```

LoggingNode	Power
'N_Channel_MOSFET_1'	2.6075

The table shows the combined dissipated power losses for both the diode and the MOSFET components of the N-Channel MOSFET 1 block, averaged over the total simulation time.

6. Calculate power losses for only the diode component of the NChannel MOSFET 1 block and display the results in a table.

```
tabulatedLosses = pe_getPowerLossSummary(simlog_pe_push_pull_converter_ccm.N_Channel_MO
```

```
tabulatedLosses =
```

```
1x2 table
```

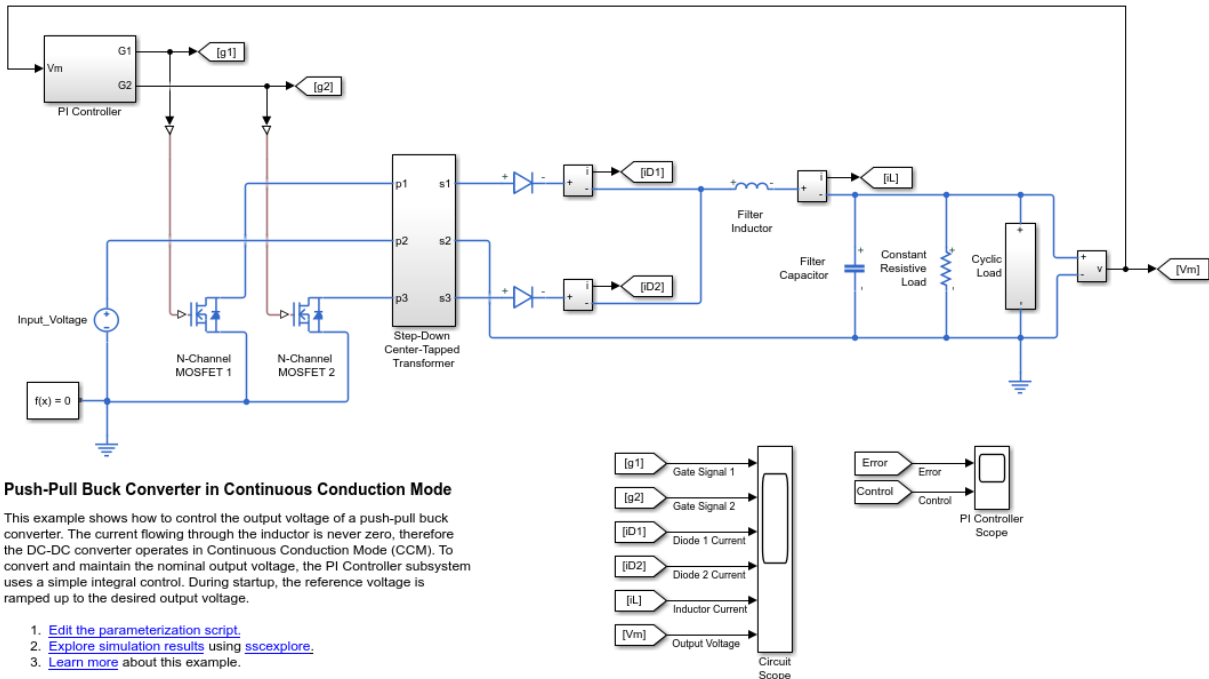
LoggingNode	Power
'diode'	2.3669

The table shows dissipated power losses only for the diode component of the block, averaged over the total simulation time.

## Calculate Average Power Losses for a Specific Time Period

1. Open the Push-Pull Buck Converter in Continuous Conduction Mode example model. At the MATLAB® command prompt, enter

```
pe_push_pull_converter_ccm
```



The model has data logging enabled.

2. Run the simulation and create the simulation log variable.

```
sim('pe_push_pull_converter_ccm');
```

The simulation log variable `simlog_pe_push_pull_converter_ccm` is saved in your workspace.

3. The model simulation time ( $t$ ) is 0.04 seconds. Calculate average power losses for the interval when  $t$  is 0.010–0.025 seconds

```
tabulatedLosses = pe_getPowerLossSummary(simlog_pe_push_pull_converter_ccm,0.010,0.025
```

```
tabulatedLosses =
```

```
5x2 table
```

```
LoggingNode
```

'pe_push_pull_converter_ccm.Diode'	0
'pe_push_pull_converter_ccm.Diode1'	0
'pe_push_pull_converter_ccm.N_Channel_MOSFET_2.mosfet_equation'	0
'pe_push_pull_converter_ccm.N_Channel_MOSFET_1.mosfet_equation'	0
'pe_push_pull_converter_ccm.Step_Down_Center_Tapped_Transformer.Eddy_Current'	0

The table shows dissipated power losses for each of the Diode and MOSFET blocks, averaged over the specified portion of simulation time.

## Input Arguments

**node** — Simulation log variable, or a specific node within the simulation log variable

Node object

Simulation log workspace variable, or a node within this variable, that contains the logged model simulation data, specified as a **Node** object. You specify the name of the simulation log variable by using the **Workspace variable name** parameter on the **Simscape** pane of the Configuration Parameters dialog box. To specify a node within the simulation log variable, provide the complete path to that node through the simulation data tree, starting with the top-level variable name.

If **node** is the name of the simulation log variable, then the table contains the data for all blocks in the model that contain *power\_dissipated* variables. If **node** is the name of a node in the simulation data tree, then the table contains the data only for:

- Blocks or variables within that node
- Blocks or variables within subnodes at all levels of the hierarchy beneath that node

Example: `simlog.Cell1.MOS1`

**startTime — Start of the time interval for averaging dissipated power losses**

real number

Start of the time interval for averaging dissipated power losses, specified as a real number, in seconds. `startTime` must be greater than or equal to the simulation **Start time** and less than `endTime`.

Data Types: double

**endTime — End of the time interval for averaging dissipated power losses**

real number

End of the time interval for averaging dissipated power losses, specified as a real number, in seconds. `endTime` must be greater than `startTime` and less than or equal to the simulation **Stop time**.

Data Types: double

## Output Arguments

**lossesTable — Dissipated power losses for each block**

table

Dissipated power losses for each block, returned as a table. The first column lists logging nodes for all blocks that have at least one `power_dissipated` variable. The second column lists the corresponding losses in watts.

## See Also

`pe_getEfficiency` | `pe_getPowerLossTimeSeries` | `sscexplore`

## Topics

“Perform a Power-Loss Analysis”

“Data Logging” (Simscape)

“About the Simscape Results Explorer” (Simscape)

**Introduced in R2017a**

# pe\_getPowerLossTimeSeries

Calculate dissipated power losses and return the time series data

## Syntax

```
lossesCell = pe_getPowerLossTimeSeries(node)
lossesCell = pe_getPowerLossTimeSeries(node, startTime, endTime)
lossesCell = pe_getPowerLossTimeSeries(node, startTime, endTime,
intervalWidth)
```

## Description

`lossesCell = pe_getPowerLossTimeSeries(node)` calculates dissipated power losses for blocks, based on logged Simscape simulation data, and returns the time series data for each block.

Before you call this function, generate or load the simulation log variable into your workspace. To generate the variable, simulate the model with simulation data logging enabled. For more information, see “About Simulation Data Logging” (Simscape). To load a previously saved variable from a file, right-click on the file and select **Load**.

Checking dissipated power allows you to determine if circuit components are operating within their efficiency requirements. Blocks in the **Semiconductor > Fundamental Components** library have an internal variable called *power\_dissipated*. This variable represents the instantaneous dissipated power, which includes only the real power (not the reactive or apparent power) that the block dissipates. When you log simulation data, the time-value series for this variable represents the power dissipated by the block over time. You can view and plot this data using the Simscape Results Explorer. The `pe_getPowerLossTimeSeries` function also allows you to access this data from a cell array.

The `pe_getPowerLossTimeSeries` function calculates losses for each block that has a *power\_dissipated* variable. Some blocks have more than one *power\_dissipated* variable, depending on their configuration. For example, for the MOSFET block, both the `diode` node and the `ideal_switch` node have a *power\_dissipated* logging node. The

function sums the power losses for both nodes to provide the total power loss for the block.

The nonideal semiconductor blocks also have thermal variants. Thermal variants have thermal ports that allow you to model the heat that is generated due to switching events and conduction losses. If you use a thermal variant, the function calculates power losses based on the thermal parameters that you specify. Essentially, the power dissipated is equal to the heat generated.

If you use a variant without a thermal port, the function calculates power losses based on the electrical parameters that you specify, such as on-state resistance and off-state conductance.

`lossesCell = pe_getPowerLossTimeSeries(node, startTime, endTime)` calculates dissipated power losses for blocks in a model, based on logged Simscape simulation data, and returns the time series data for each block for time steps from `startTime` to `endTime`. If `startTime` is equal to `endTime`, the interval is effectively zero and the function returns the instantaneous power for the time step that occurs at that moment.

`lossesCell = pe_getPowerLossTimeSeries(node, startTime, endTime, intervalWidth)` calculates dissipated power losses for blocks in a model, based on logged Simscape simulation data, and returns the time series data for each block for time steps from `startTime` to `endTime`, with averaging applied over intervals equal to `intervalWidth`. If `intervalWidth` is 0, the function returns the instantaneous power dissipation.

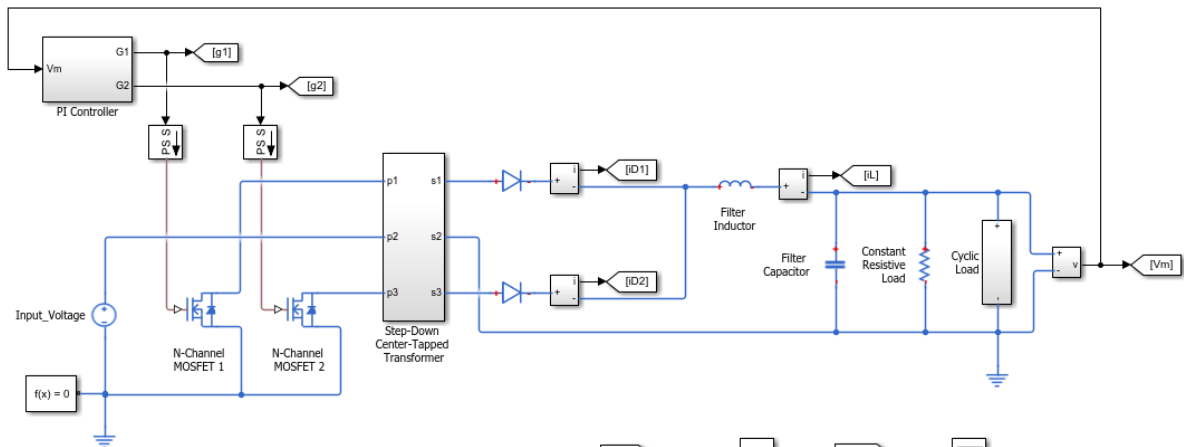
## Examples

### Calculate Dissipated Power Losses for the Entire Simulation Time

This example shows how to calculate instantaneous losses based on the power dissipated and return the time series data for all time steps in the entire simulation time using the `pe_getPowerLossTimeSeries` function. Data logging is enabled for the whole example model, and the option to limit data points is off.

Open the model. At the MATLAB® command prompt, enter:

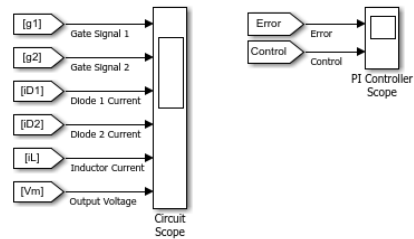
```
model = 'pe_push_pull_converter_ccm';  
open_system(model)
```



### Push-Pull Buck Converter in Continuous Conduction Mode

This example shows how to control the output voltage of a push-pull buck converter. The current flowing through the inductor is never zero, therefore the DC-DC converter operates in Continuous Conduction Mode (CCM). To convert and maintain the nominal output voltage, the PI Controller subsystem uses a simple integral control. During startup, the reference voltage is ramped up to the desired output voltage.

1. [Edit the parameterization script.](#)
2. [Explore simulation results](#) using [sscexplore](#).
3. [Learn more](#) about this example.



Run the simulation and create the simulation log variable.

```
sim(model)
```

The simulation log variable `simlog_pe_push_pull_converter_ccm` is saved in your workspace.

Calculate dissipated power losses and return the time series data in a cell array.

```
lossesCell = pe_getPowerLossTimeSeries(simlog_pe_push_pull_converter_ccm)
```

```
lossesCell =
```

```
4×2 cell array
```

```
'pe_push_pull_converter_ccm.N_Chann...' [400001×3 double]
'pe_push_pull_converter_ccm.N_Chann...' [400001×3 double]
'pe_push_pull_converter_ccm.Diode' [400001×3 double]
'pe_push_pull_converter_ccm.Diode1' [400001×3 double]
```

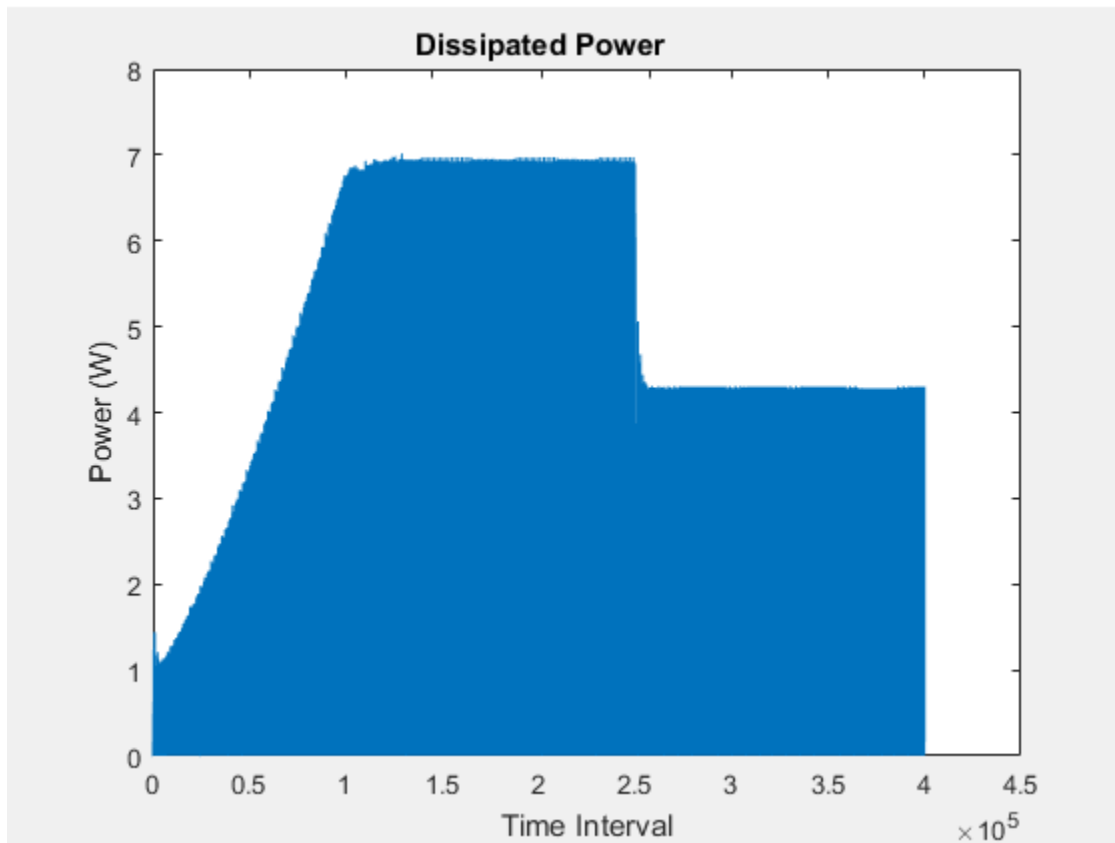


View the time series data. From the workspace, open the `lossesCell` cell array, then open the `400001x3` double numeric array for the `pe_push_pull_converter_ccm.N_Channel_MOSFET_1.mosfet_equation`.

The first two columns contain the interval start and end time. The third column contains the power loss data.

Plot the data.

```
plot(lossesCell{1, 2}(:,end))  
title('Dissipated Power')  
xlabel('Time Interval')  
ylabel('Power (W)')
```

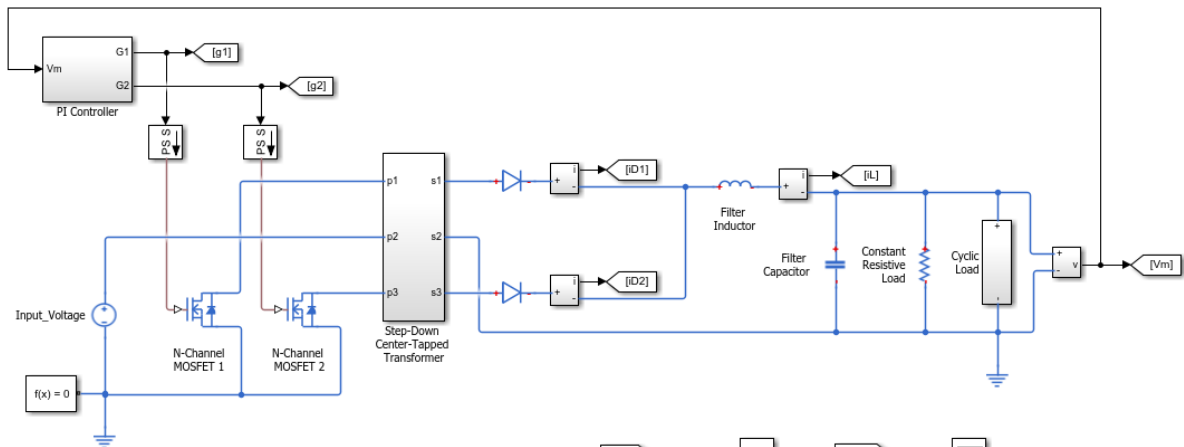


### Calculate Dissipated Power Losses for a Specific Time Period

This example shows how to calculate instantaneous losses based on the power dissipated and return the time series data for all time steps in a specific time period using the `pe_getPowerLossTimeSeries` function. Data logging is enabled for the whole example model, and the option to limit data points is off.

Open the model. At the MATLAB® command prompt, enter:

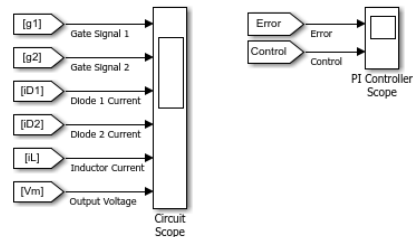
```
model = 'pe_push_pull_converter_ccm';  
open_system(model)
```



### Push-Pull Buck Converter in Continuous Conduction Mode

This example shows how to control the output voltage of a push-pull buck converter. The current flowing through the inductor is never zero, therefore the DC-DC converter operates in Continuous Conduction Mode (CCM). To convert and maintain the nominal output voltage, the PI Controller subsystem uses a simple integral control. During startup, the reference voltage is ramped up to the desired output voltage.

1. [Edit the parameterization script.](#)
2. [Explore simulation results using `sscexplore`.](#)
3. [Learn more](#) about this example.



Run the simulation and create the simulation log variable.

```
sim(model)
```

The simulation log variable `simlog_pe_push_pull_converter_ccm` is saved in your workspace.

The model simulation time ( $t$ ) is 0.04 seconds. Calculate dissipated power losses and return the time series data in a cell array for the interval when  $t$  is 0.010–0.025 seconds.

```
lossesCell = pe_getPowerLossTimeSeries(simlog_pe_push_pull_converter_ccm,0.010,0.025)
```

```
lossesCell =
```

```
4×2 cell array
```

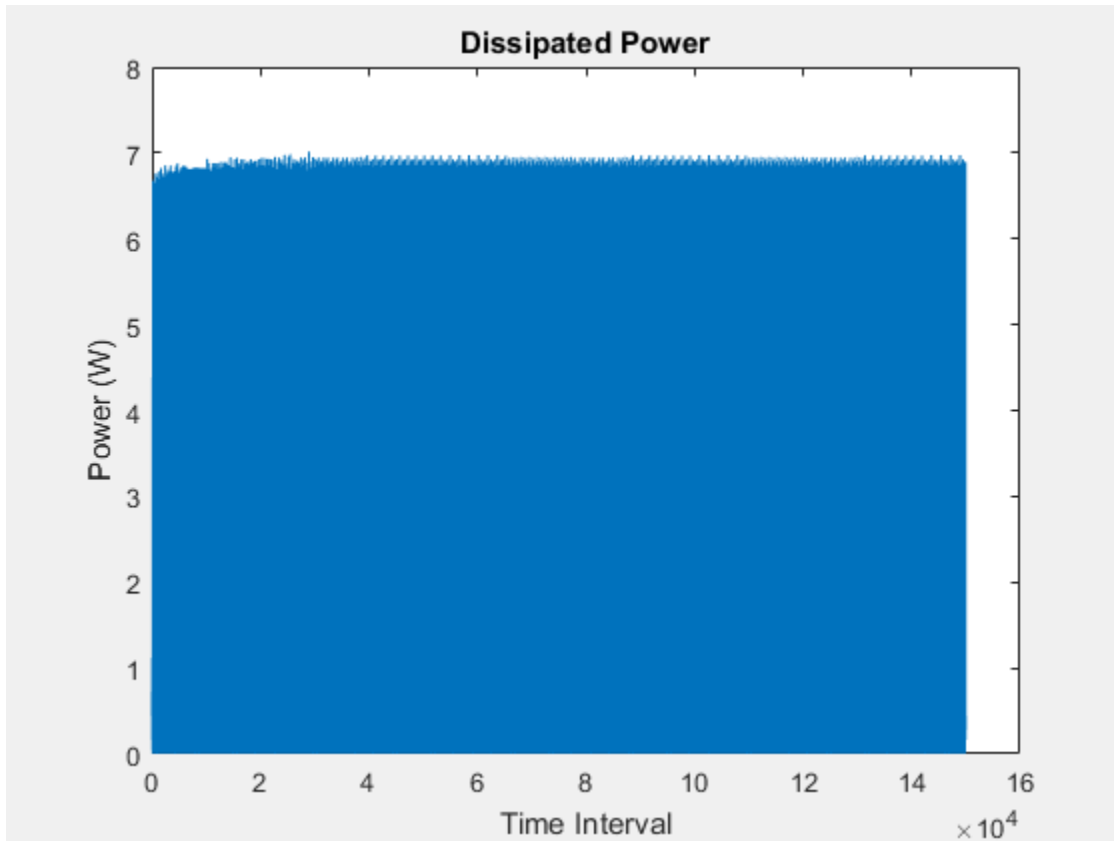
```
'pe_push_pull_converter_ccm.N_Chann...' [150002×3 double]
'pe_push_pull_converter_ccm.N_Chann...' [150002×3 double]
'pe_push_pull_converter_ccm.Diode' [150002×3 double]
'pe_push_pull_converter_ccm.Diode1' [150002×3 double]
```

View the time series data. From the workspace, open the `lossesCell` cell array, then open the 150002x3 double numeric array for the `pe_push_pull_converter_ccm.N_Channel_MOSFET_1.mosfet_equation`.

The first two columns contain the interval start and end time. The third column contains the power loss data.

Plot the data.

```
plot(lossesCell{1, 2}(:,end))  
title('Dissipated Power')  
xlabel('Time Interval')  
ylabel('Power (W)')
```

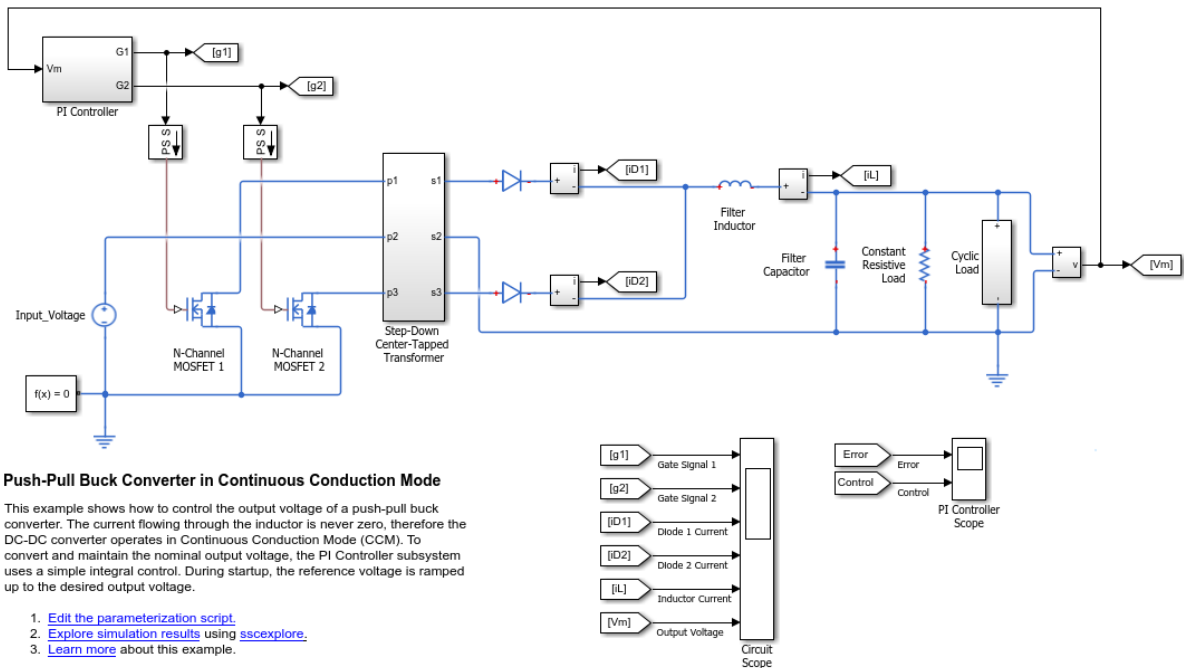


### Calculate Dissipated Power Losses Using Specific Interval Widths

This example shows how to calculate losses based on the power dissipated and return the time series data for a specific time period with averaging applied over intervals of a specified width. Data logging is enabled for the whole example model, and the option to limit data points is off.

Open the model. At the MATLAB® command prompt, enter:

```
model = 'pe_push_pull_converter_ccm';  
open_system(model)
```



Run the simulation and create the simulation log variable.

```
sim(model)
```

The simulation log variable `simlog_pe_push_pull_converter_ccm` is saved in your workspace.

The model simulation time,  $t$ , is 0.04 seconds. Calculate the average dissipated power losses for  $1.1e-4$  s intervals and return the time series data in a cell array for the period when simulation time,  $t$ , is 0.010-0.025 seconds.

```
lossesCell = pe_getPowerLossTimeSeries(simlog_pe_push_pull_converter_ccm,0.010,0.025,1.1e-4)
```

```
lossesCell =
```

```
4×2 cell array
```

```
'pe_push_pull_converter_ccm.N_...' [136×3 double]
'pe_push_pull_converter_ccm.N_...' [136×3 double]
```

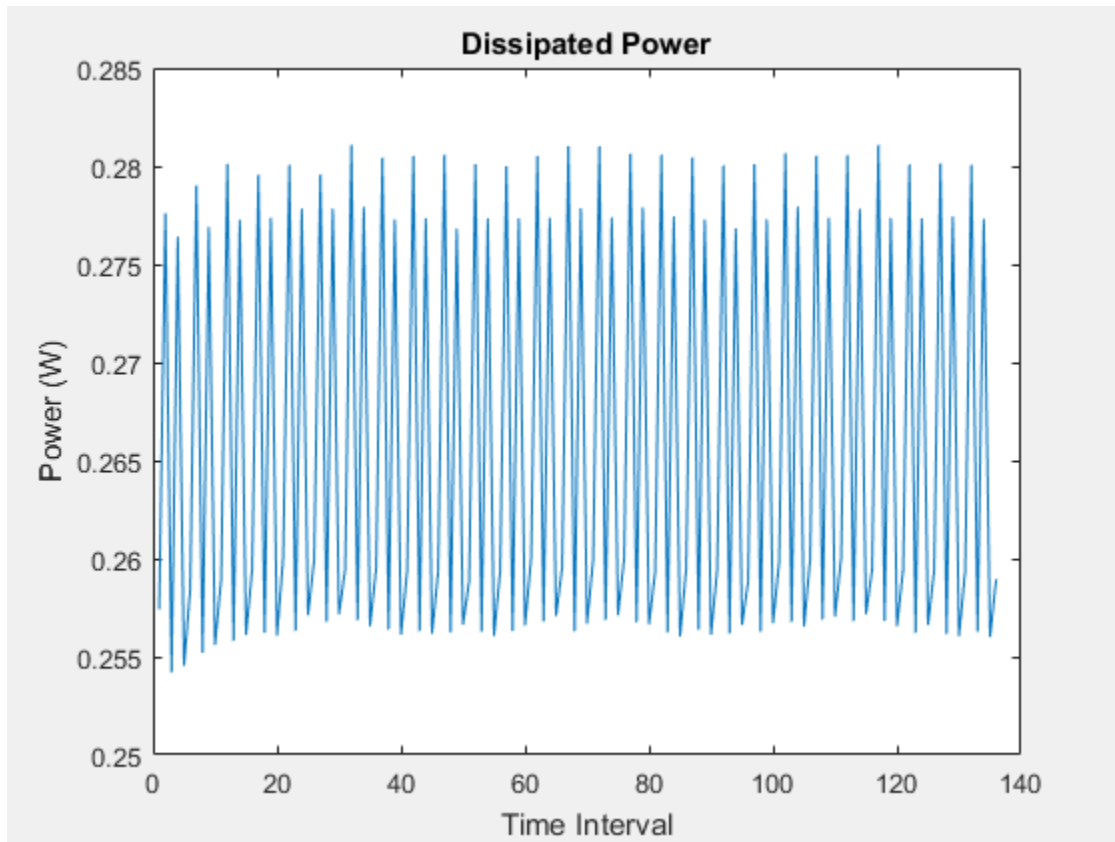
```
'pe_push_pull_converter_ccm.Di...' [136x3 double]
'pe_push_pull_converter_ccm.Di...' [136x3 double]
```

View the time series data. From the workspace, open the `lossesCell` cell array, then open the `136x3 double` numeric array for the `pe_push_pull_converter_ccm.N_Channel_MOSFET_1.mosfet_equation`.

The first two columns contain the interval start and end time. The third column contains the power loss data. In this case, to use averaging intervals that are equal in width to  $1.1\text{e-}4$  seconds, the function adjusts the start time for the first interval from the specified value of 0.010 seconds to a value of 0.01004 seconds. There are 136 intervals of  $1.1\text{e-}4$  seconds.

Plot the data.

```
plot(lossesCell{1, 2}(:,end))
title('Dissipated Power')
xlabel('Time Interval')
ylabel('Power (W)')
```



## Input Arguments

**node** — Simulation log variable, or a specific node within the simulation log variable

Node object

Simulation log workspace variable, or a node within this variable, that contains the logged model simulation data, specified as a Node object. You specify the name of the simulation log variable by using the **Workspace variable name** parameter on the **Simscape** pane of the Configuration Parameters dialog box. To specify a node within the



simulation log variable, provide the complete path to that node through the simulation data tree, starting with the top-level variable name.

If `node` is the name of the simulation log variable, then the table contains the data for all blocks in the model that contain *power\_dissipated* variables. If `node` is the name of a node in the simulation data tree, then the table contains the data only for:

- Blocks or variables within that node
- Blocks or variables within subnodes at all levels of the hierarchy beneath that node

Example: `simlog_pe_push_pull_converter_ccm`

### **startTime** — Start of the time interval for calculating the data

0 (default) | real number

Start of the time interval for calculating the power loss time series, specified as a real number, in seconds. `startTime` must be greater than or equal to the simulation **Start time** and less than `endTime`.

Data Types: double

### **endTime** — End of the time interval for calculating the data

simulation stop time (default) | real number

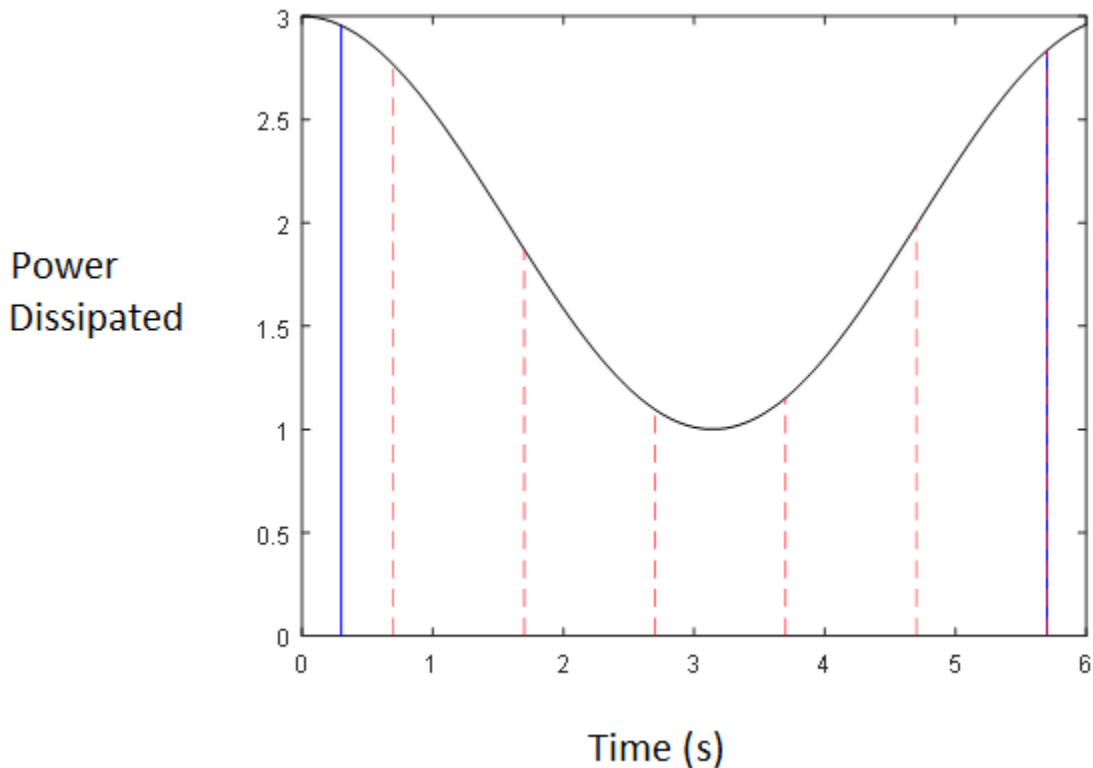
End of the time interval for calculating the power loss time series, specified as a real number, in seconds. `endTime` must be greater than `startTime` and less than or equal to the simulation **Stop time**.

Data Types: double

### **intervalWidth** — size of the interval in time for calculating the average power dissipation

0 (default)

If the time between the specified `startTime` and `endTime` is not an integer multiple of `intervalWidth`, the function adjusts the start time. The figure shows how the function adjusts the start time to ensure that width of each time interval that the dissipated power is averaged over is equal to the specified `intervalWidth`.



The black line is an example of the instantaneous *power\_dissipated* variables summed over all elements in an individual block. The simulation runs for 6 seconds. The `startTime` and `endTime` are indicated by the solid blue lines. The `intervalWidth` is set to 1 second. There are five intervals as indicated by the red dashed lines. The right-most edge of the last interval coincides with `endTime`. The left-most edge of the first interval is always greater than or equal to `startTime`. The edge is equal to `startTime` only if  $(\text{endTime} - \text{startTime}) / \text{intervalWidth}$  is an integer. The output in this case consists of five values for the averaged power dissipation, one point for each time period. The function outputs the actual start and stop times in the tabulated output data.

Example: `1.1e-3`

Data Types: `double`

## Output Arguments

### **lossesCell** — Time series of the dissipated power losses for each block

cell array

Cell array that contains the names of the blocks in the nodes that contain *power\_dissipated* variables and, for each block, a three-column array:

- Column one contains the interval start time.
- Column two contains the interval end time.
- Column three contains the dissipated power for the time interval.

If the interval width is 0 seconds, that is, the start time is equal to the end time, then the dissipated power is the instantaneous power loss. If the interval is greater than 0 seconds, the dissipated power is the average power loss for the time of the interval.

## See Also

[pe\\_getEfficiency](#) | [pe\\_getPowerLossSummary](#) | [sscxplore](#)

## Topics

“Perform a Power-Loss Analysis”

“Data Logging” (Simscape)

“About the Simscape Results Explorer” (Simscape)

**Introduced in R2017a**

## pe\_plotHarmonics

Plot percentage of fundamental magnitude versus harmonic order

### Syntax

```
pe_plotHarmonics(loggingNode)
pe_plotHarmonics(loggingNode,valueIdx)
pe_plotHarmonics(loggingNode,valueIdx,tOfInterest)
pe_plotHarmonics(loggingNode,valueIdx,tOfInterest,nPeriodOfInterest)
pe_plotHarmonics(loggingNode,valueIdx,tOfInterest,
nPeriodOfInterest,...
offsetOfInterest)
pe_plotHarmonics(loggingNode,valueIdx,tOfInterest,
nPeriodOfInterest,...
offsetOfInterest,nHarmonic)
```

### Description

`pe_plotHarmonics(loggingNode)` plots a bar chart of percentage of fundamental magnitude versus harmonic order of the `simscape.logging.Node` of an AC or periodic variable. The title of the bar chart includes the fundamental frequency, fundamental peak value, and total harmonic distortion (THD) percentage.

You enter the input arguments in a specific order. The Simscape logging node input argument is required. All other input arguments are optional and have default values. If you are specifying a value for a subsequent optional input argument, enter `[]` to use the default value for an optional input argument.

The `pe_plotHarmonics` function uses the `pe_getHarmonics` function to:

- Find the points in the  $i^{\text{th}}$  signal (`valueIdx`) where the Simscape log crosses a threshold (`offsetOfInterest`).
- Use the crossing points to find the required number of periods (`nPeriodOfInterest`) preceding the specified time (`tOfInterest`).

- Calculate the harmonic magnitudes, up to and including the required number of harmonics (nHarmonic).
- Input the down-selected data to the Goertzel algorithm, which calculates the harmonic magnitudes up to and including the required number of harmonics (nHarmonic).

---

**Note** The `pe_getHarmonics` function uses threshold crossing points to determine the fundamental frequency of the data. If your input data is noisy or crosses the threshold more frequently than half of the fundamental period, filter it before you use the `pe_plotHarmonics` function to plot it.

---

The `pe_plotHarmonics` function then inputs the harmonic orders and harmonic magnitudes to the `pe_calculateThdPercent` function to calculate the THD.

`pe_plotHarmonics(loggingNode,valueIdx)` uses the index into value data.

`pe_plotHarmonics(loggingNode,valueIdx,tOfInterest)` uses the simulation time.

`pe_plotHarmonics(loggingNode,valueIdx,tOfInterest,nPeriodOfInterest)` uses the number of periods of fundamental frequency.

`pe_plotHarmonics(loggingNode,valueIdx,tOfInterest,nPeriodOfInterest,...,offsetOfInterest)` uses the DC offset.

`pe_plotHarmonics(loggingNode,valueIdx,tOfInterest,nPeriodOfInterest,...,offsetOfInterest,nHarmonic)` uses the number of harmonics.

## Examples

### Plot Using Default Values

This set of function arguments uses the Simscape logging node `simlog.Load.V`, which contains data from a three-phase voltage. The function analyzes the default signal, which is the first, or a-phase, signal at the final simulation time. The function uses the default values of 12 for the number of periods of the signal, 0V for the signal bias, and 30 for the number of harmonics.

```
pe_plotHarmonics(simlog.Load.V)
```

### Plot Using Specified Values

This set of function arguments uses the Simscape logging node `simlog.Load.V`, which contains data from a three-phase voltage. The function analyzes the second, or b-phase, signal at a simulation time of 2.3 s. The function uses 10 periods of the signal, which has a bias of 1 V. The function analyzes 15 harmonics.

```
pe_plotHarmonics(simlog.Load.V,2,2.3,10,1,15)
```

### Plot Using Default and Specified Values

This set of function arguments uses the Simscape logging node `simlog.Load.V`, which contains data from a three-phase voltage. The function analyzes the first, or a-phase, signal at a simulation time of 2.3 s. The function uses the default number (12) of periods of the signal, which has a bias of 1 V. The function analyzes the default number (30) of harmonics.

```
pe_plotHarmonics(simlog.Load.V,[],2.3,[],1)
```

## Input Arguments

### loggingNode — Simscape logging node

1-by-1 `simscape.logging.Node`

Simscape logging node, specified as a 1-by-1 `simscape.logging.Node`. You create a `simscape.logging.Node` by running a simulation with Simscape logging enabled. To learn how to enable data logging, see “Enable Data Logging for the Whole Model” (Simscape).

Example: `simlog.Load.V`

The Simscape logging node `simlog.Load.V` contains data from a three-phase voltage.

### valueIdx — Index into value data

1 (default) | scalar

Index into value data, specified as a scalar. Specifies the  $i^{\text{th}}$  variable of interest in the Simscape log.

Example: 2

Specify the b-phase, which is the second signal from a three-phase voltage.

Example: []

Use [] to specify the default value of 1. The a-phase, which is the first signal from a three-phase voltage, is the default signal of interest.

Data Types: single | double | int8 | int16 | int32 | int64 | uint8 | uint16 | uint32 | uint64

### **tOfInterest — Simulation time**

final time in Simscape log (default) | scalar

Simulation time of interest for harmonic analysis, specified as a scalar.

Example: 2.3

Specify a 2.3s simulation time.

Data Types: single | double | int8 | int16 | int32 | int64 | uint8 | uint16 | uint32 | uint64

### **nPeriodOfInterest — Number of periods**

12 (default) | scalar

Number of periods of fundamental frequency to be included in harmonic analysis, specified as a scalar.

Example: 10

Specify 10 periods of the signal.

Data Types: single | double | int8 | int16 | int32 | int64 | uint8 | uint16 | uint32 | uint64

### **offsetOfInterest — DC offset**

0 (default) | scalar

DC offset in the input signal, specified as a scalar. The function uses this value to find the periods of interest.

Example: 1

Specify a bias of 1V for the signal.

Data Types: `single` | `double` | `int8` | `int16` | `int32` | `int64` | `uint8` | `uint16` | `uint32` | `uint64`

### **nHarmonic — Number of harmonics**

30 (default) | scalar

Number of harmonics to include in analysis, specified as a scalar.

Example: 15

Specify that the number of harmonics to be analyzed is 15.

Data Types: `single` | `double` | `int8` | `int16` | `int32` | `int64` | `uint8` | `uint16` | `uint32` | `uint64`

## See Also

### **Blocks**

Spectrum Analyzer

### **Functions**

`pe_calculateThdPercent` | `pe_getHarmonics`

### **Objects**

`simscape.logging.Node`

## Topics

“Perform an Online Harmonic Analysis Using the Simscape Spectrum Analyzer Block”

“Choose a Simscape Electrical Power Systems Function for an Offline Harmonic Analysis”

“Data Logging” (Simscape)

“Harmonic Analysis of a Three-Phase Rectifier”

### **Introduced in R2014a**



## Abbreviations and Naming Conventions in Simscape Electrical Power Systems Libraries

<b>B</b>	Susceptance.
<b>C</b>	Capacitance.
<b>composite three-phase port</b>	Three-phase electrical conserving port, i.e., a port that represents three electrical conserving ports with a single connection. You can use composite three-phase ports to build models corresponding to single-line diagrams of three-phase electrical systems. Instead of explicitly connecting each phase of the three-phase system between blocks, you connect all three phases using a single port.
<b>delta connection</b>	Three-phase winding configuration. Each of the three windings is connected between phases. Physically, the connection resembles the Greek capital letter $\Delta$ . For a delta connection, phase shifts can be specified in terms of the hours of a clock. An 11 o'clock delta connection represents a 30 degree phase advance. A 1 o'clock delta connection represents a 30 degree phase delay.
<b>expanded three-phase port</b>	Three separate electrical conserving ports that represent the individual phases of a three-phase system. You individually connect each phase of the three-phase system between blocks.
<b>F<sub>Rated</sub></b>	Rated electrical frequency of three-phase machine.
<b>G</b>	Conductance.
<b>i</b>	Instantaneous current.
<b>I</b>	RMS current.
<b>L</b>	Inductance.
<b>line voltage</b>	RMS value of the voltage measured between phases. In a balanced three-phase system with no harmonics, peak line voltage equals peak phase voltage multiplied by $\sqrt{3}$ . The RMS value equals peak line voltage divided by $\sqrt{2}$ .

	Standard abbreviations are $V_{ab}$ , $V_{ac}$ , $V_{bc}$ , etc. Line voltage is also known as rated voltage, rated RMS, name plate voltage, line-line voltage, and phase-phase voltage.
<b>nPolePairs</b>	Number of pole pairs for three-phase machine.
<b>phase voltage</b>	RMS value of the voltage measured between a phase and reference point. The reference point is usually a neutral or ground point. In a balanced three-phase system with no harmonics, peak phase voltage is equal to peak line voltage divided by $\sqrt{3}$ . The RMS value equals peak phase voltage divided by $\sqrt{2}$ . Standard abbreviations are $V_a$ , $V_b$ , and $V_c$ .
<b><math>P_{PerPhase}</math></b>	Real power per phase.
<b>psi</b>	Instantaneous peak magnetic flux linkage.
<b>Psi</b>	RMS magnetic flux linkage.
<b><math>Q_{PerPhase}</math></b>	Reactive power per phase.
<b>R</b>	Resistance.
<b><math>S_{Rated}</math></b>	Rated apparent power.
<b><math>S_{PerPhase}</math></b>	Apparent power per phase.
<b>torque</b>	Torque of three-phase machine.
<b>v</b>	Instantaneous voltage.
<b>V</b>	RMS voltage.
<b><math>V_a, V_b, V_c</math></b>	Phase voltages.
<b><math>V_{ab}, V_{ac}, V_{bc}, \dots</math></b>	Line voltages.
<b><math>V_{Rated}</math></b>	Rated voltage of three-phase machine.
<b><math>\omega_{Electrical}</math></b>	Electrical angular speed of three-phase machine.

<b><math>\omega</math></b> Mechanical	Mechanical angular speed of three-phase machine.
<b>winding voltage</b>	Voltage measured between both ends of a winding. For a wye connection, winding voltage equals phase voltage. For a delta connection, winding voltage equals line voltage.
<b>wye connection</b>	Three-phase winding configuration. Each of the three windings is connected between a phase and neutral point. Physically, the connection resembles the letter Y. A wye connection is also referred to as a star connection, or Y connection.
<b>X</b>	Reactance.
<b>Y</b>	Admittance.
<b>z</b>	Impedance.



# Parameter Dependencies

## Parameter Dependencies

A Simscape block parameter is considered *visible* when it appears in the Simulink Property Inspector, in a block dialog box, or as a block choice on the Simscape context menu. A block parameter is considered *configurable* or *enabled* when you can configure it by selecting an option, entering a value, or selecting or clearing a check box. Parameters that are visible but are not enabled are typically dimmed.

The visibility and configurability of some parameters depend on the options that you select for other parameters. Parameter dependencies are typically listed in the parameter description in the documentation for the block. For some blocks, the documentation also includes a parameter dependency table.

### Parameter Dependency Tables

Parameter dependency tables show how the visibility and configurability of some parameters depend on the options that you select for other parameters.

Parameter dependency tables indicate which parameter options affect the *visibility* of other parameters. For example, the next table shows the dependencies for the **Effects and Initial Conditions** parameters for the Pipe (2P) block from the **Simscape > Foundation Library > Two-Phase Fluid > Elements** library.

The row numbers in this table are for reference only. The first column of a parameter dependency table typically includes all parameters that are visible by default.

Parameters that affect the visibility or configurability of other parameters are shown in bold text. The options for the parameters that affect other parameters are shown in italic text in the following row of the table. Parameters that do not affect other parameters are shown in plain text.

**Effects and Initial Conditions Parameter Dependencies**

Row	Parameters and Options					
1	<b>Fluid Inertia</b>					
2	<i>Off</i>			<i>On</i>		
3	<b>Initial fluid regime</b>			<b>Initial fluid regime</b>		
4	<i>Subcooled liquid</i>	<i>Two-phase mixture</i>	<i>Superheated vapor</i>	<i>Subcooled liquid</i>	<i>Two-phase mixture</i>	<i>Superheated vapor</i>
5	Initial fluid pressure	Initial fluid pressure	Initial fluid pressure	Initial fluid pressure	Initial fluid pressure	Initial fluid pressure
6	Initial fluid temperature	Initial fluid vapor quality	Initial fluid temperature	Initial fluid temperature	Initial fluid vapor quality	Initial fluid temperature
7				Initial mass flow rate from port A to port B	Initial mass flow rate from port A to port B	Initial mass flow rate from port A to port B
8	Phase change time constant	Phase change time constant	Phase change time constant	Phase change time constant	Phase change time constant	Phase change time constant

The figure shows the **Effects and Initial Conditions** parameters that are visible by default on the block dialog box for the Pipe (2P) block.

Settings

Geometry Friction and Heat Transfer Effects and Initial Conditions

Fluid inertia:	Off	
Initial fluid regime:	Subcooled liquid	
Initial fluid pressure:	0.101325	MPa
Initial fluid temperature:	293.15	K
Phase change time constant:	0.1	s

If you change the **Fluid Inertia** parameter from Off to On, the **Initial mass flow rate from port A to port B** parameter becomes visible. The table shows this dependency in rows 1, 2, and 7.

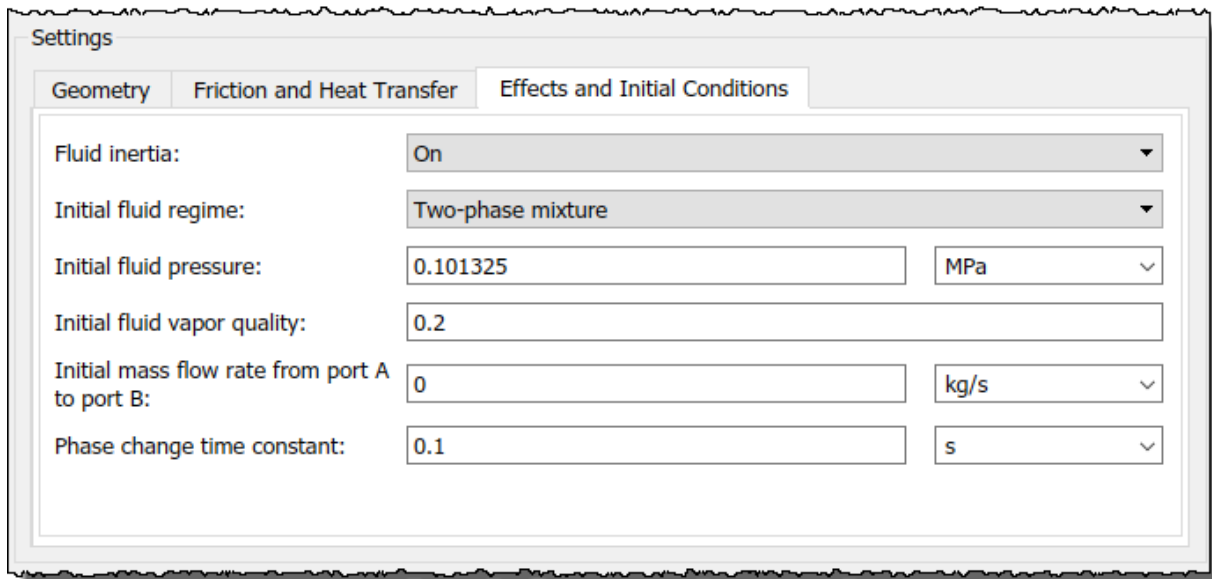


Settings

Geometry   Friction and Heat Transfer   **Effects and Initial Conditions**

Fluid inertia:	On	
Initial fluid regime:	Subcooled liquid	
Initial fluid pressure:	0.101325	MPa
Initial fluid temperature:	293.15	K
Initial mass flow rate from port A to port B:	0	kg/s
Phase change time constant:	0.1	s

If you then change the **Initial fluid regime** parameter from Subcooled liquid to Two-phase mixture, the **Initial fluid temperature** parameter is hidden and **Initial fluid vapor quality** becomes visible. The table shows these dependencies in rows 3, 4, and 6.



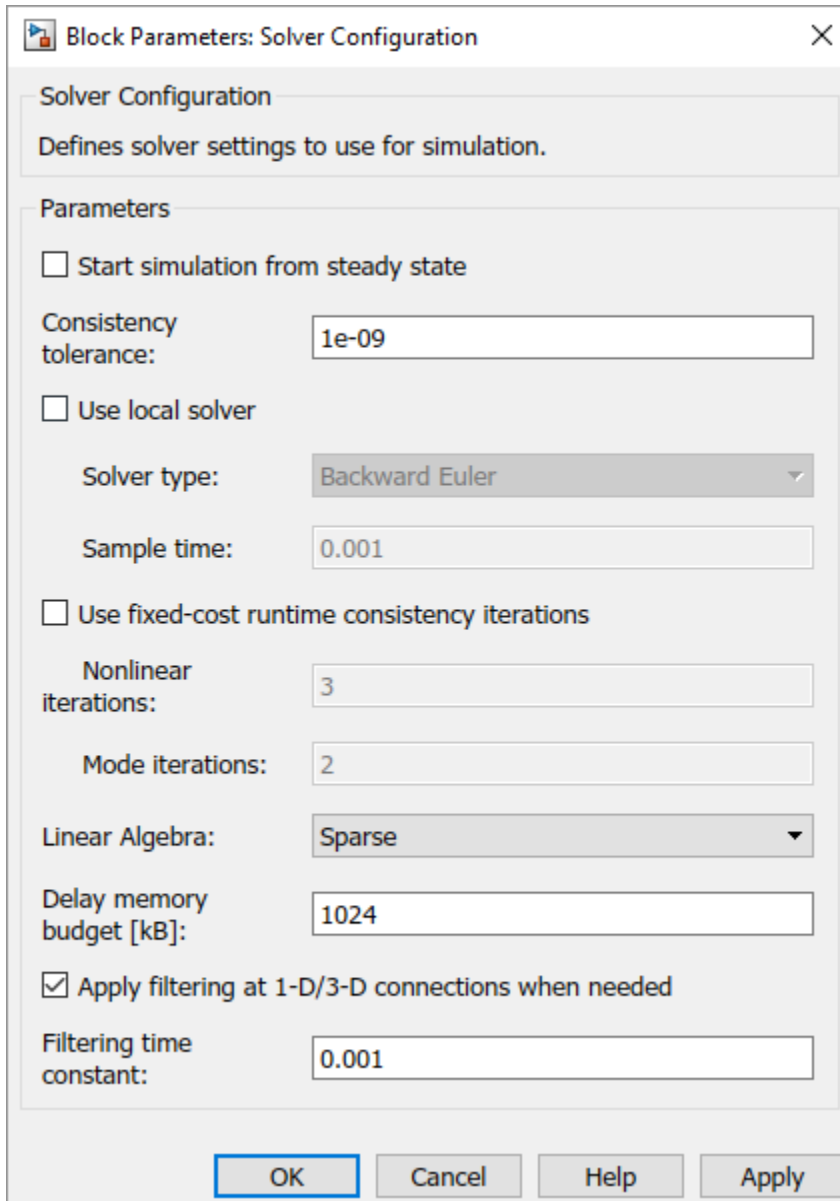
Parameter dependency tables also indicate which parameter options *enable* other parameters. For example, the next table shows the dependencies for the parameters for the Solver Configuration block from the **Simscape > Foundation Library > Utilities** library. The first column in the table contains row numbers, which are for reference only. The second column includes all parameters that are visible by default.

Parameters that affect the visibility and configurability of other parameters are shown in bold text. The options for the parameters that affect other parameters are shown as selected and cleared check boxes in the following row of the table. Parameters that are shown in plain text are enabled but do not affect other parameters. Parameters that are not enabled are not shown.

### Solver Configuration Block Parameter Dependencies

Row	Parameters and Options			
1	Start simulation from steady state			
2	Consistency tolerance			
3	<b>Use local Solver</b>			
4	<input type="checkbox"/>	<input checked="" type="checkbox"/>		
5			Solver type	
6			Sample time	
7	<b>Use fixed-cost runtime consistency iterations</b>		<b>Use fixed-cost runtime consistency iterations</b>	
8	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
9			Nonlinear iterations	Nonlinear iterations
10			Mode iterations	
11	Linear Algebra			
12	Equation formulation			
13	Delay memory budget [kB]			
14	<b>Apply filtering at 1-D/3-D connections when needed</b>			
15	<input checked="" type="checkbox"/>	<input type="checkbox"/>		
16	Filtering time constant			

The figure shows the parameters that are visible by default on the block dialog box for the Solver Configuration block. The dimmed parameters, for example, **Solver type**, are not enabled.



If you select the **Use local solver** check box, the **Use fixed-cost runtime consistency** check box becomes selected and these parameters become enabled.

- **Solver type**
- **Nonlinear iterations**
- **Sample time**

Selecting **Use local solver** does not enable the **Mode iterations** parameter. The **Mode iterations** parameter is only enabled when the **Use local solver** check box is cleared while **Use fixed-cost runtime consistency** check box is selected.

The table shows these dependencies in rows 3-10.

