SimElectronics®

Reference

R2013**b**

MATLAB®&SIMULINK®

MathWorks®

How to Contact MathWorks

`www.mathworks.com`	Web
`comp.soft-sys.matlab`	Newsgroup
`www.mathworks.com/contact_TS.html`	Technical Support
`suggest@mathworks.com`	Product enhancement suggestions
`bugs@mathworks.com`	Bug reports
`doc@mathworks.com`	Documentation error reports
`service@mathworks.com`	Order status, license renewals, passcodes
`info@mathworks.com`	Sales, pricing, and general information

508-647-7000 (Phone)

508-647-7001 (Fax)

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

For contact information about worldwide offices, see the MathWorks Web site.

SimElectronics® Reference

Revision History

April 2008	Online only	New for Version 1.0 (Release 2008a+)
October 2008	Online only	Revised for Version 1.1 (Release 2008b)
March 2009	Online only	Revised for Version 1.2 (Release 2009a)
September 2009	Online only	Revised for Version 1.3 (Release 2009b)
March 2010	Online only	Revised for Version 1.4 (Release 2010a)
September 2010	Online only	Revised for Version 1.5 (Release 2010b)
April 2011	Online only	Revised for Version 1.6 (Release 2011a)
September 2011	Online only	Revised for Version 2.0 (Release 2011b)
March 2012	Online only	Revised for Version 2.1 (Release 2012a)
September 2012	Online only	Revised for Version 2.2 (Release 2012b)
March 2013	Online only	Revised for Version 2.3 (Release 2013a)
September 2013	Online only	Revised for Version 2.4 (Release 2013b)

Contents

1 Blocks — Alphabetical List

Index

Blocks — Alphabetical List

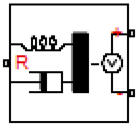
Purpose

Behavioral model of MEMS accelerometer

Library

Sensors

Description



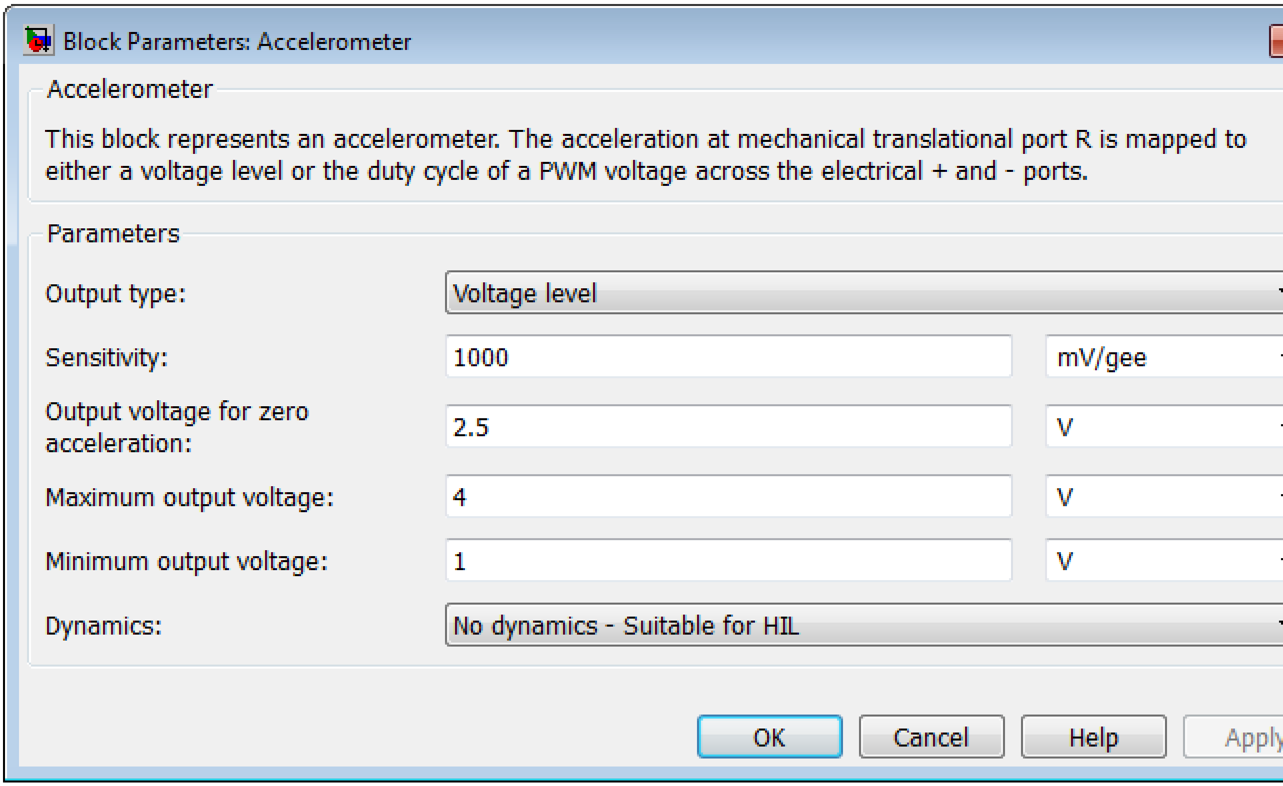
The Accelerometer block implements a behavioral model of a MicroElectroMechanical Systems (MEMS) accelerometer. For the default output type `Voltage level`, the accelerometer provides an output voltage that is proportional to the acceleration rate presented at the mechanical translational physical port 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.

Dialog Box and 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** and the **Initial angular rate** parameter values.

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.

Initial acceleration

Determines the initial condition for the lag by specifying the initial output for the sensor, expressed in units of acceleration. This parameter is only visible when you select `Model sensor bandwidth` for the **Dynamics** parameter. The default value is `0` gee.

Ports

The block has the following ports:

`R`

Mechanical translational port.

`+`

Positive electrical port.

`-`

Negative electrical port.

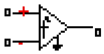
Purpose

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 *Vp* and *Vm*, respectively, the output voltage is:

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

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.

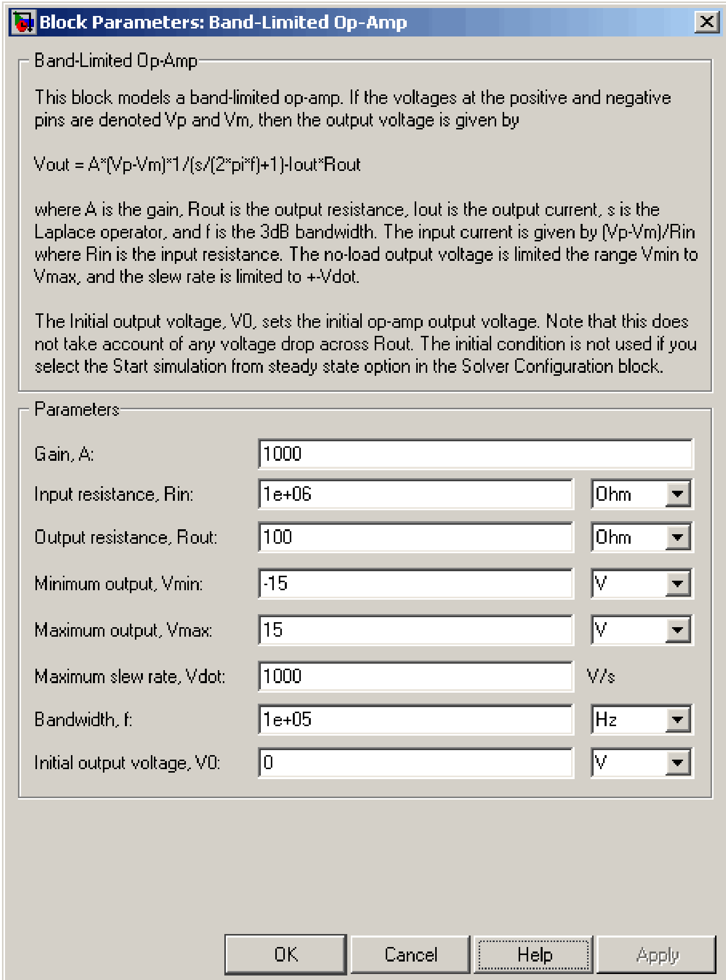
The input current is:

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

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.

Dialog Box and 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` Ω.

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` Ω.

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

Simscape Op-Amp, Finite-Gain Op-Amp, Fully Differential Op-Amp

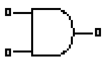
Purpose

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 1-12 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 Vcc (the **Supply voltage** parameter value), and output low is zero volts.

Basic Assumptions and Limitations

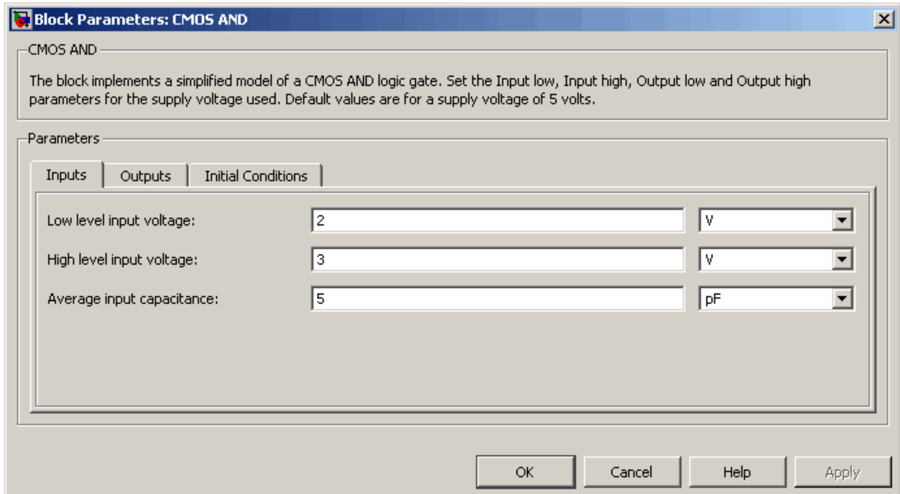
The block does not model the internal individual MOSFET devices that make up the gate (except for the final MOSFET pair if you select the `Quadratic` option for the **Output current-voltage relationship** parameter). This limitation has the following implications:

- 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.

Dialog Box and Parameters

Inputs Tab

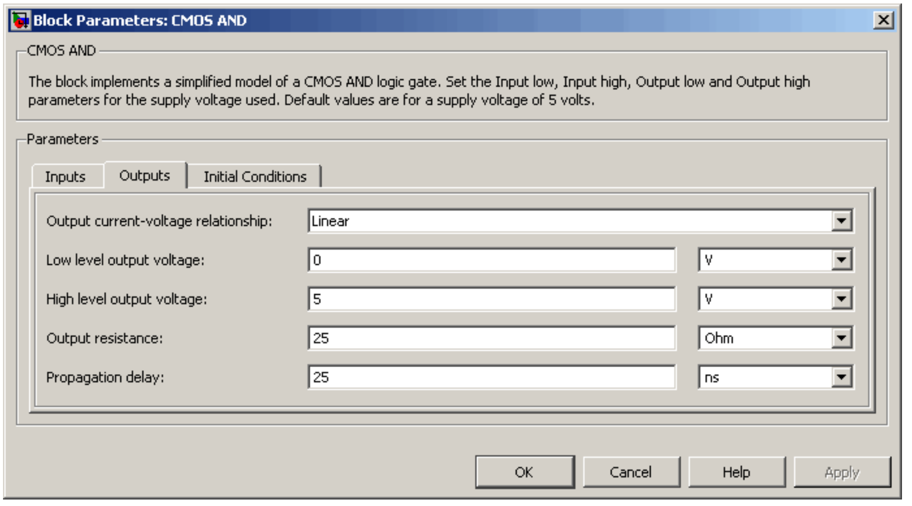


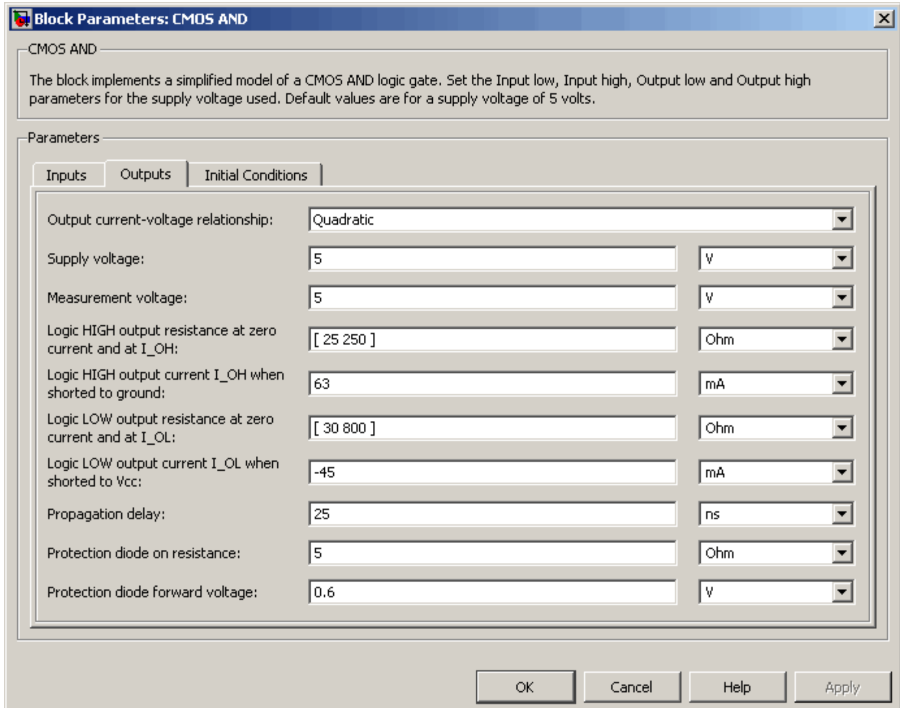
Low level input voltage
Voltage value below which the block interprets the input voltage as logic `LOW`. The default value is `2` V.

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


Block Parameters: CMOS AND
CMOS AND
The block implements a simplified model of a CMOS AND logic gate. Set the Input low, Input high, Output low and Output high parameters for the supply voltage used. Default values are for a supply voltage of 5 volts.
Parameters
Inputs
Outputs
Initial Conditions
Output current-voltage relationship:
Linear
Low level output voltage:
0
V
High level output voltage:
5
V
Output resistance:
25
Ohm
Propagation delay:
25
ns
OK
Cancel
Help
Apply



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` Ω. 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 ]` Ω. 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 ]` Ω. 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.

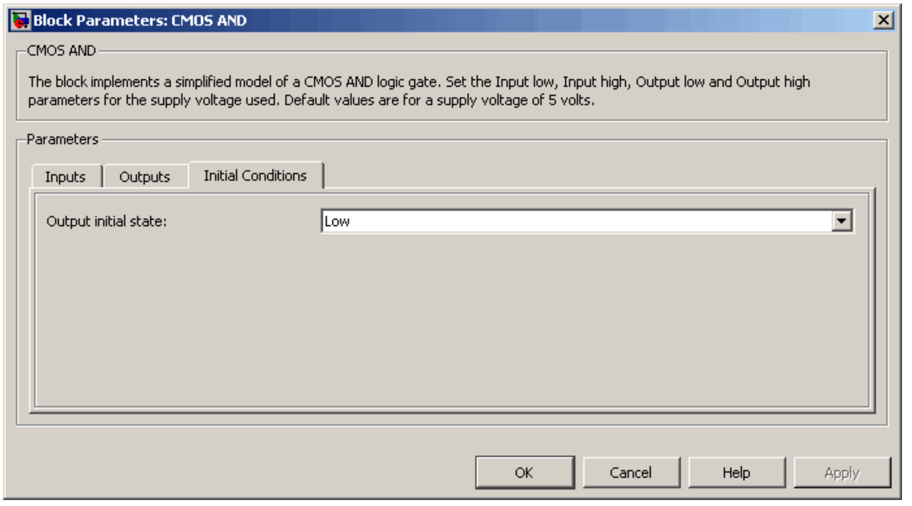
Protection diode on resistance

The gradient of the voltage-current relationship for the protection diodes when forward biased. The default value is `5` Ω. 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.

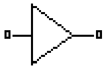
Purpose

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 1-20 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 Vcc (the **Supply voltage** parameter value), and output low is zero volts.

Basic Assumptions and Limitations

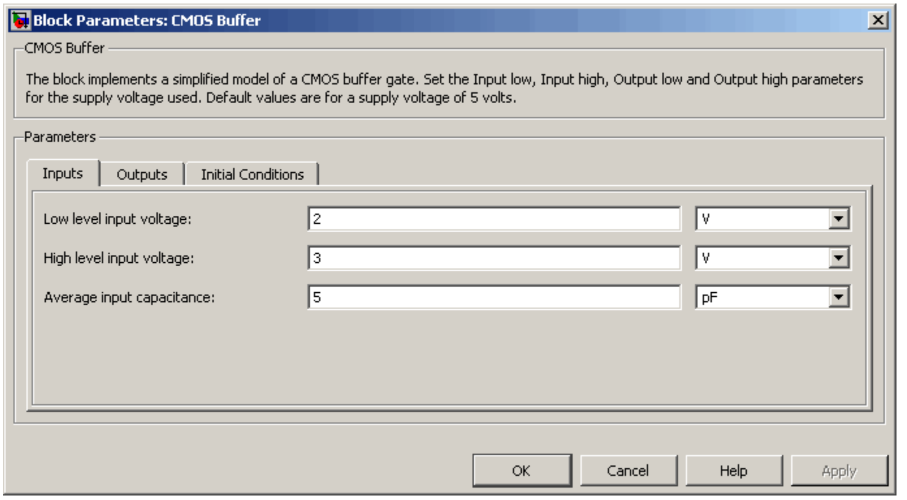
The block does not model the internal individual MOSFET devices that make up the gate (except for the final MOSFET pair if you select the `Quadratic` option for the **Output current-voltage relationship** parameter). This limitation has the following implications:

- 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.

Dialog Box and Parameters

Inputs Tab

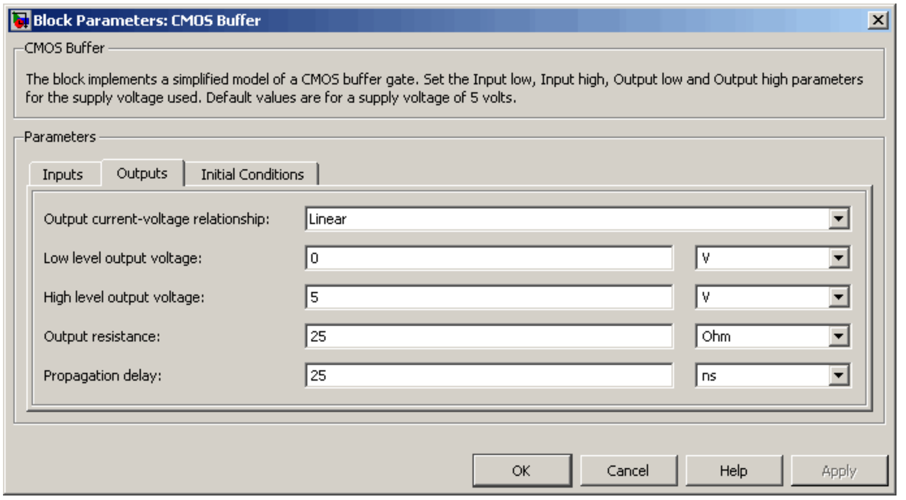


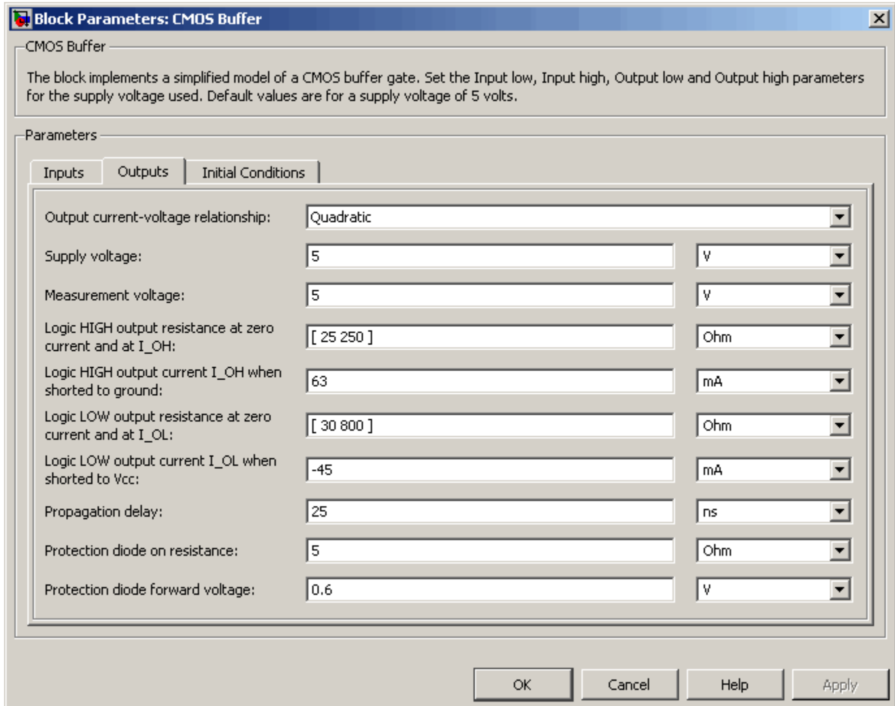
Low level input voltage
: Voltage value below which the block interprets the input voltage as logic `LOW`. The default value is `2` V.

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


Block Parameters: CMOS Buffer
CMOS Buffer
The block implements a simplified model of a CMOS buffer gate. Set the Input low, Input high, Output low and Output high parameters for the supply voltage used. Default values are for a supply voltage of 5 volts.
Parameters
Inputs
Outputs
Initial Conditions
Output current-voltage relationship:
Linear
Low level output voltage:
0
V
High level output voltage:
5
V
Output resistance:
25
Ohm
Propagation delay:
25
ns
OK
Cancel
Help
Apply



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` Ω. 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 ]` Ω. 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 ]` Ω. 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.

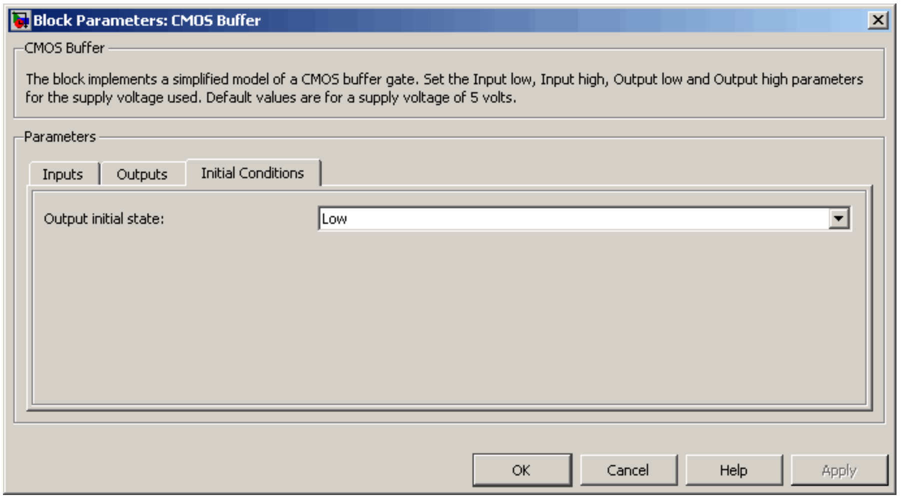
Protection diode on resistance

The gradient of the voltage-current relationship for the protection diodes when forward biased. The default value is `5` Ω. 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.

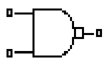
Purpose

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 1-28 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 Vcc (the **Supply voltage** parameter value), and output low is zero volts.

Basic Assumptions and Limitations

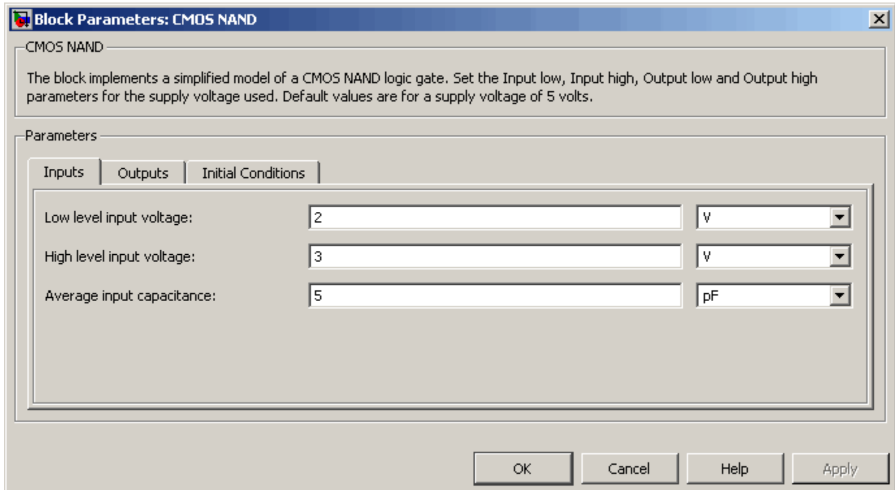
The block does not model the internal individual MOSFET devices that make up the gate (except for the final MOSFET pair if you select the `Quadratic` option for the **Output current-voltage relationship** parameter). This limitation has the following implications:

- 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.

Dialog Box and Parameters

Inputs Tab

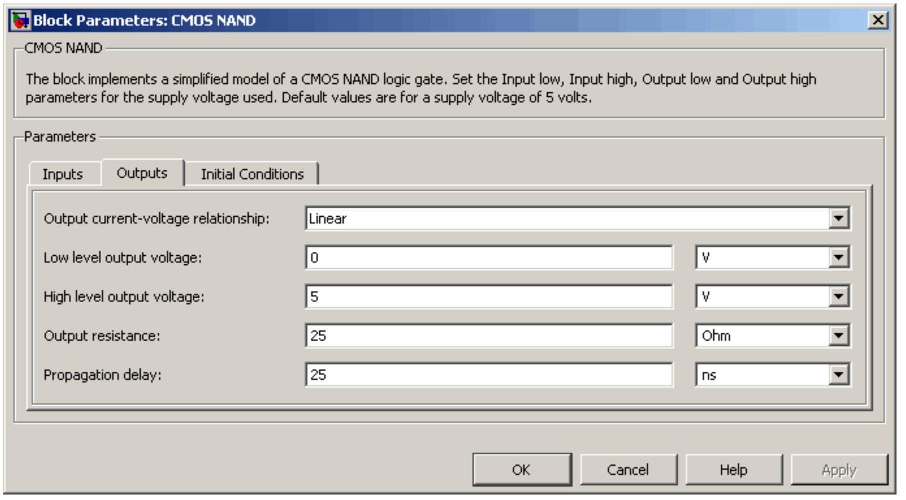


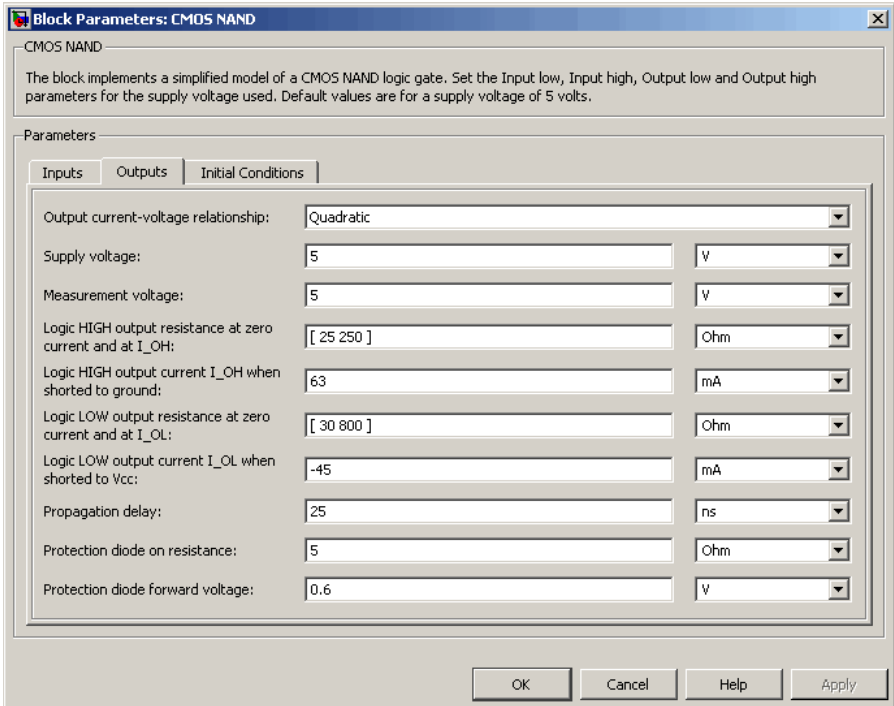
Low level input voltage
: Voltage value below which the block interprets the input voltage as logic `LOW`. The default value is `2` V.

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


Block Parameters: CMOS NAND
CMOS NAND
The block implements a simplified model of a CMOS NAND logic gate. Set the Input low, Input high, Output low and Output high parameters for the supply voltage used. Default values are for a supply voltage of 5 volts.
Parameters
Inputs
Outputs
Initial Conditions
Output current-voltage relationship:
Linear
Low level output voltage:
0
V
High level output voltage:
5
V
Output resistance:
25
Ohm
Propagation delay:
25
ns
OK
Cancel
Help
Apply



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` Ω. 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 ]` Ω. 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 ]` Ω. 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.

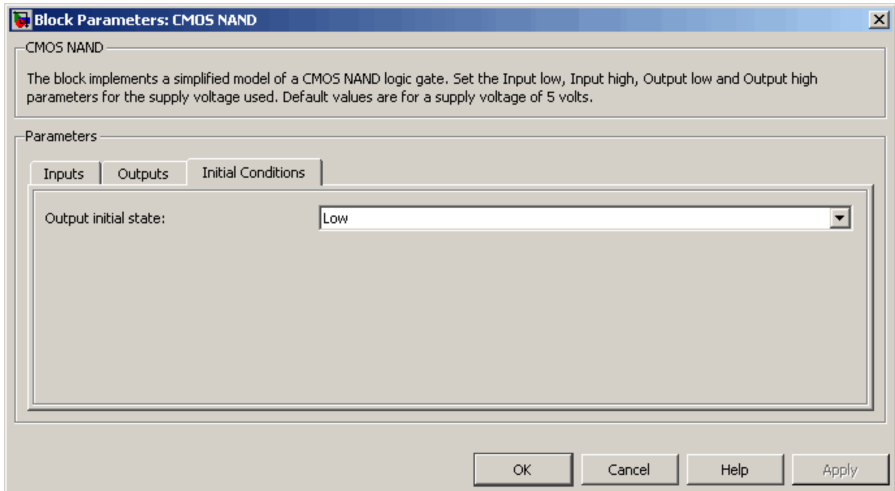
Protection diode on resistance

The gradient of the voltage-current relationship for the protection diodes when forward biased. The default value is `5` Ω. 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.

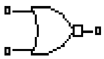
Purpose

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 1-36 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 Vcc (the **Supply voltage** parameter value), and output low is zero volts.

Basic Assumptions and Limitations

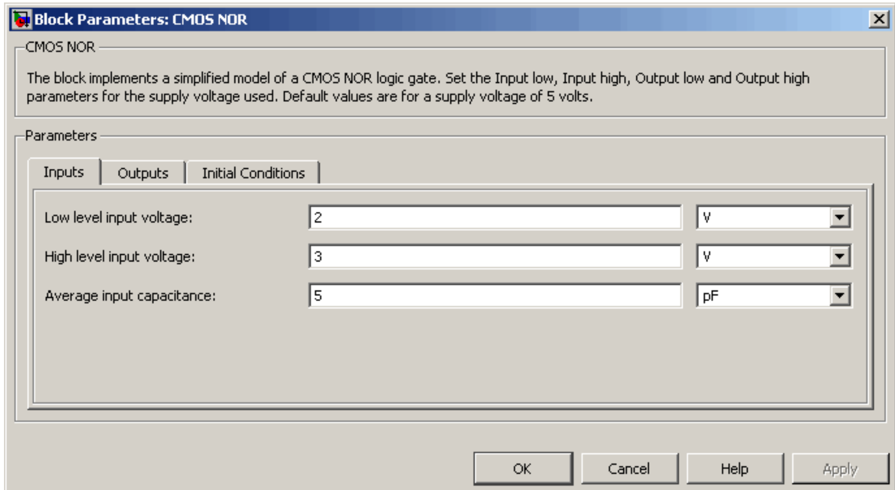
The block does not model the internal individual MOSFET devices that make up the gate (except for the final MOSFET pair if you select the `Quadratic` option for the **Output current-voltage relationship** parameter). This limitation has the following implications:

- 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.

Dialog Box and Parameters

Inputs Tab

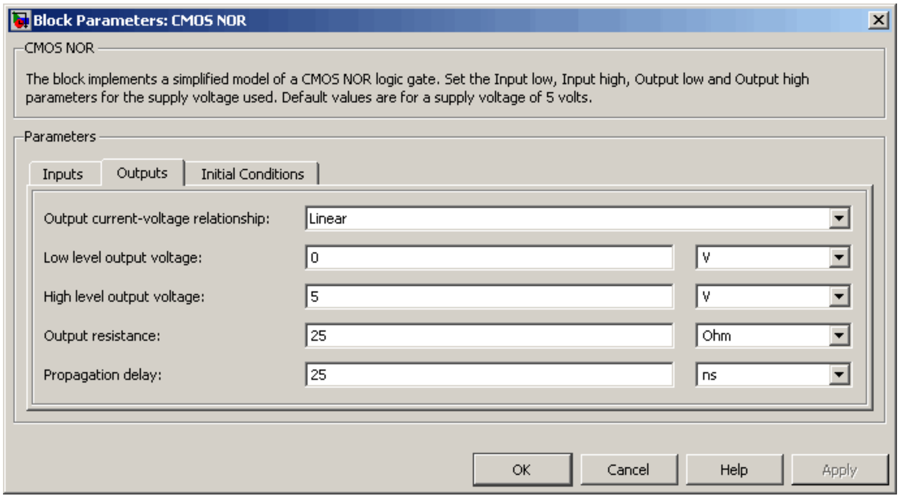


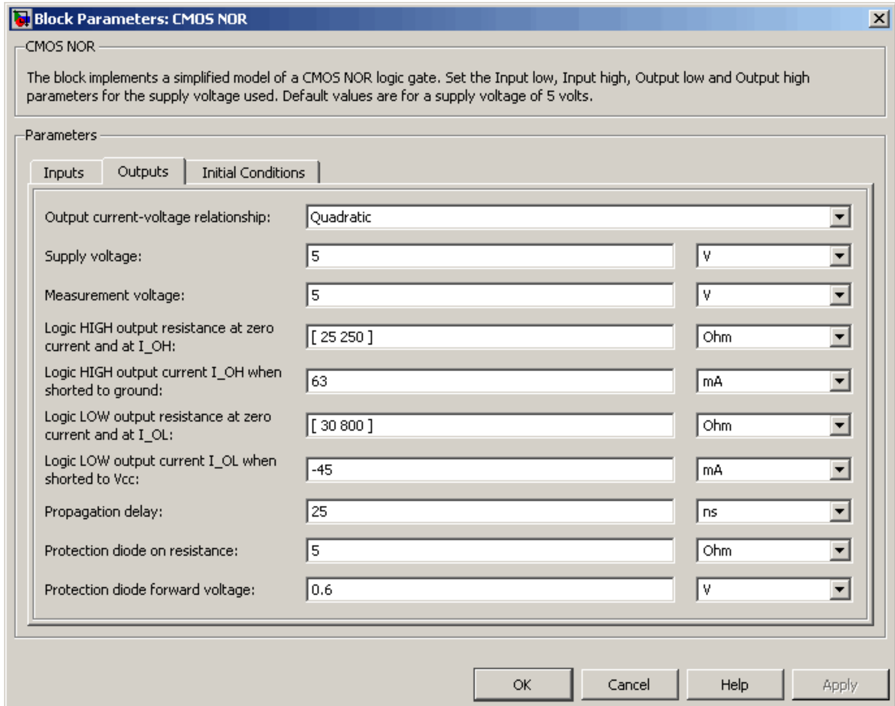
Low level input voltage
Voltage value below which the block interprets the input voltage as logic `LOW`. The default value is `2` V.

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


Block Parameters: CMOS NOR
CMOS NOR
The block implements a simplified model of a CMOS NOR logic gate. Set the Input low, Input high, Output low and Output high parameters for the supply voltage used. Default values are for a supply voltage of 5 volts.
Parameters
Inputs
Outputs
Initial Conditions
Output current-voltage relationship:
Linear
Low level output voltage:
0
V
High level output voltage:
5
V
Output resistance:
25
Ohm
Propagation delay:
25
ns
OK
Cancel
Help
Apply



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` Ω. 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 ]` Ω. 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 ]` Ω. 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.

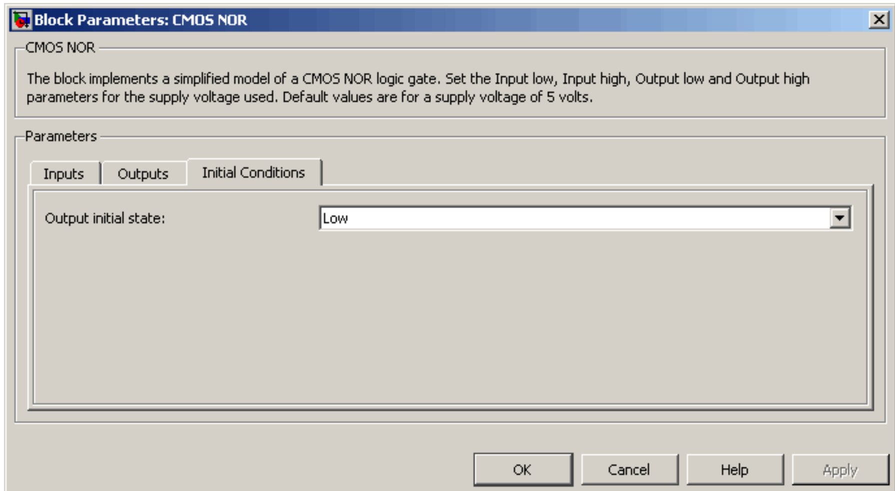
Protection diode on resistance

The gradient of the voltage-current relationship for the protection diodes when forward biased. The default value is `5` Ω. 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.

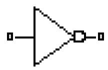
Purpose

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 1-44 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 Vcc (the **Supply voltage** parameter value), and output low is zero volts.

Basic Assumptions and Limitations

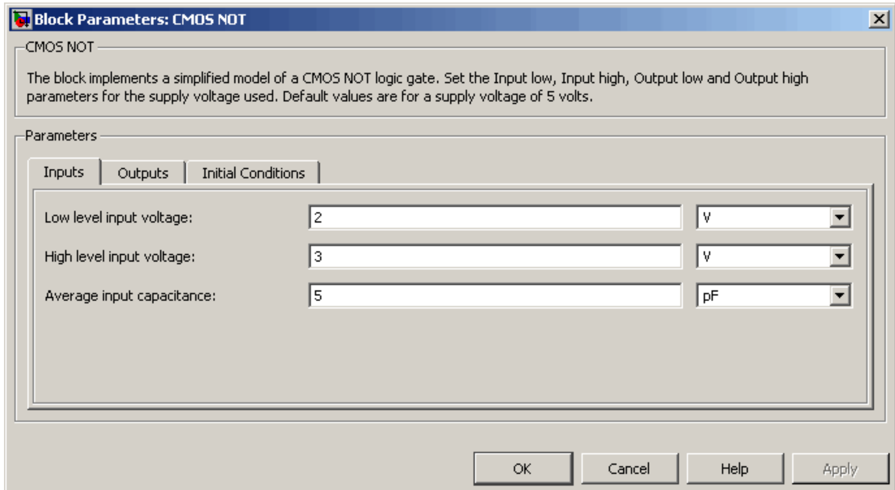
The block does not model the internal individual MOSFET devices that make up the gate (except for the final MOSFET pair if you select the `Quadratic` option for the **Output current-voltage relationship** parameter). This limitation has the following implications:

- 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.

Dialog Box and Parameters

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.

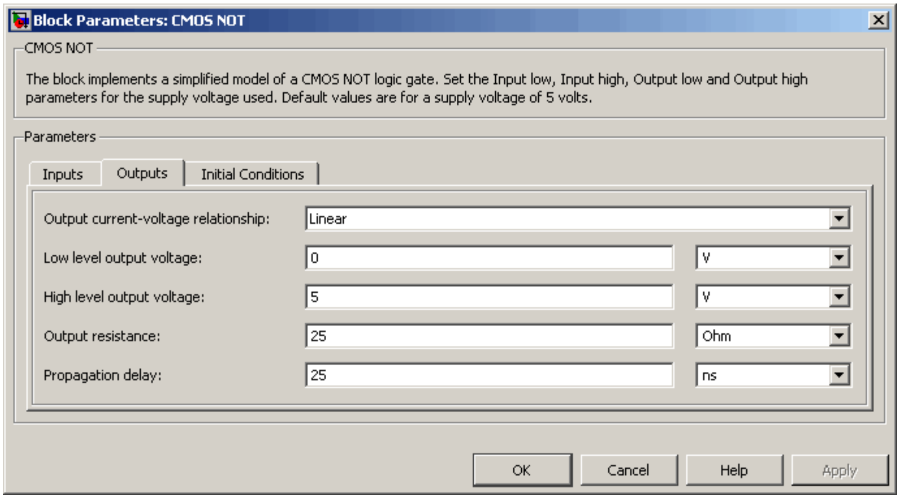
High level input voltage

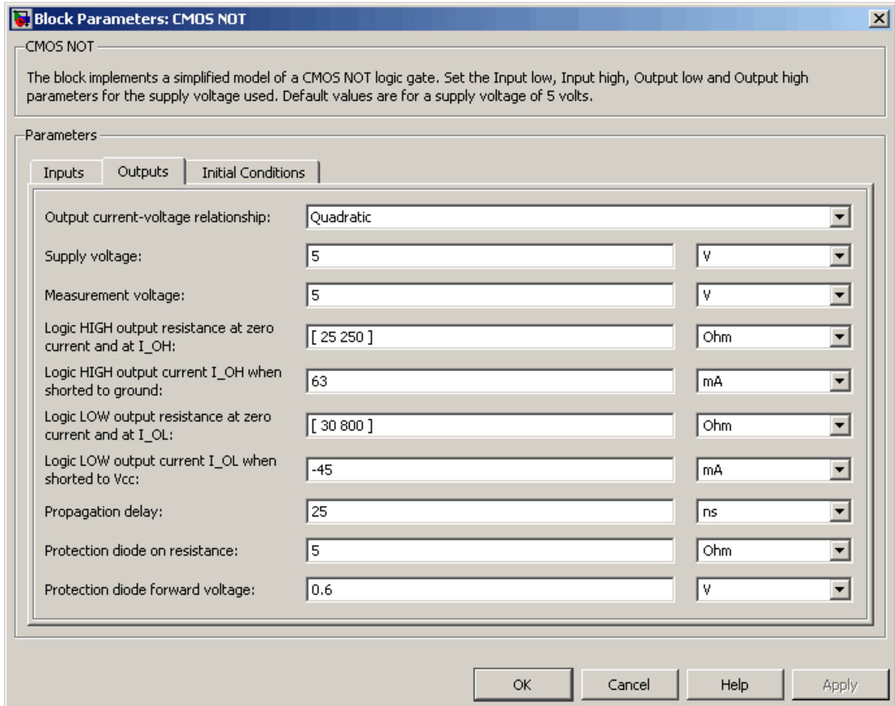
Voltage value above which the block interprets the input voltage as logic `HIGH`. The default value is `3` V.

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


Block Parameters: CMOS NOT
CMOS NOT
The block implements a simplified model of a CMOS NOT logic gate. Set the Input low, Input high, Output low and Output high parameters for the supply voltage used. Default values are for a supply voltage of 5 volts.
Parameters
Inputs
Outputs
Initial Conditions
Output current-voltage relationship:
Linear
Low level output voltage:
0
V
High level output voltage:
5
V
Output resistance:
25
Ohm
Propagation delay:
25
ns
OK
Cancel
Help
Apply



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` Ω. 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 ]` Ω. 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 ]` Ω. This parameter is available when you select the `Quadratic` option for the **Output current-voltage relationship** parameter.

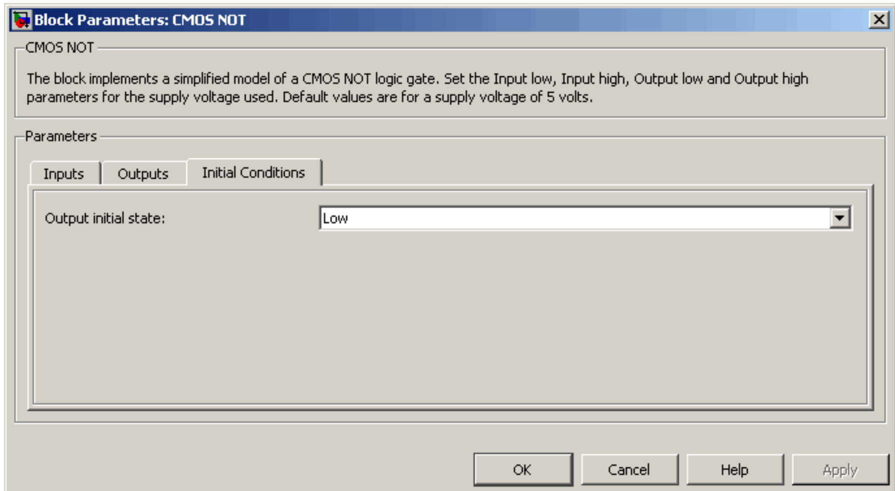
Logic LOW output current I_OL when shorted to Vcc
The resulting current when the gate is in the logic LOW state, but the load forces the output voltage to the supply voltage Vcc. The default value is `-45` mA. This parameter is available when you select the `Quadratic` option for the **Output current-voltage 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` Ω. 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.

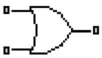
Purpose

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 1-52 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 Vcc (the **Supply voltage** parameter value), and output low is zero volts.

Basic Assumptions and Limitations

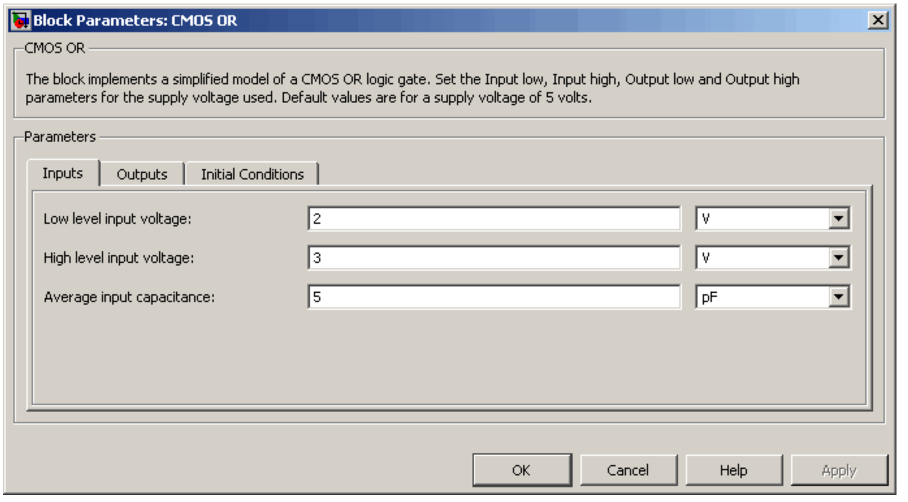
The block does not model the internal individual MOSFET devices that make up the gate (except for the final MOSFET pair if you select the `Quadratic` option for the **Output current-voltage relationship** parameter). This limitation has the following implications:

- 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.

Dialog Box and Parameters

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.

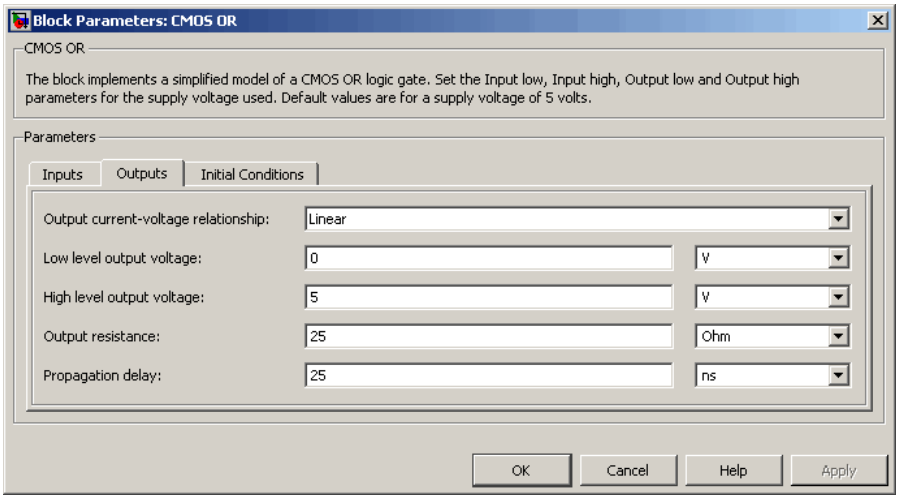
High level input voltage

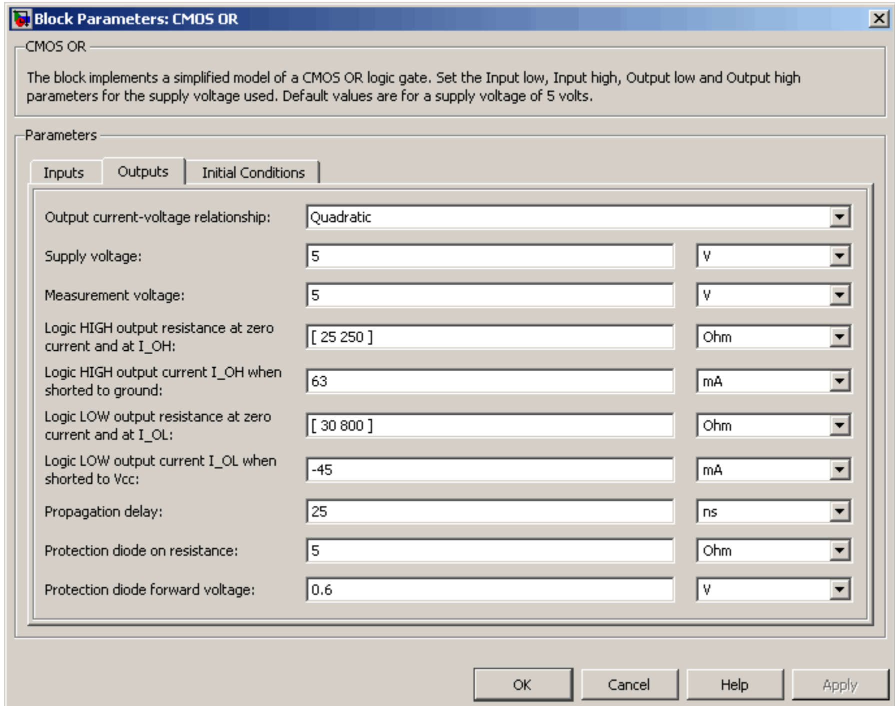
Voltage value above which the block interprets the input voltage as logic `HIGH`. The default value is `3` V.

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


Block Parameters: CMOS OR
CMOS OR
The block implements a simplified model of a CMOS OR logic gate. Set the Input low, Input high, Output low and Output high parameters for the supply voltage used. Default values are for a supply voltage of 5 volts.
Parameters
Inputs
Outputs
Initial Conditions
Output current-voltage relationship:
Linear
Low level output voltage:
0
V
High level output voltage:
5
V
Output resistance:
25
Ohm
Propagation delay:
25
ns
OK
Cancel
Help
Apply



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` Ω. 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 ]` Ω. 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 ]` Ω. This parameter is available when you select the `Quadratic` option for the **Output current-voltage relationship** parameter.

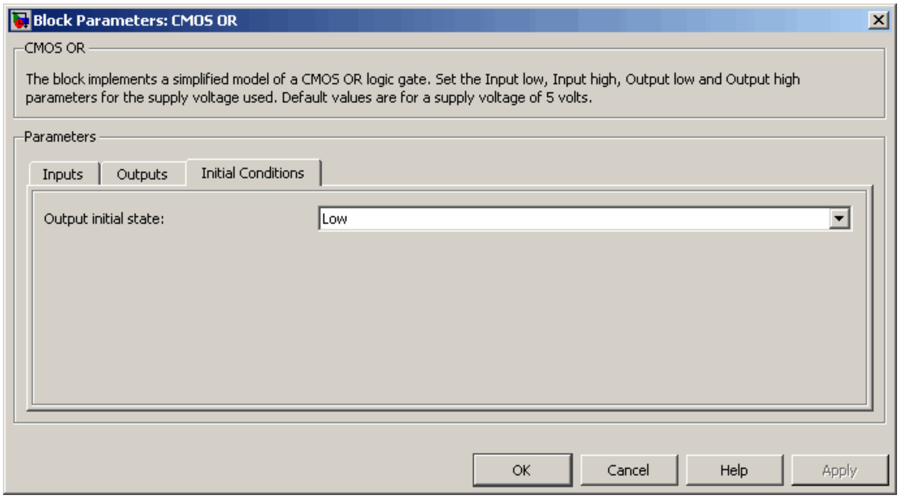
Logic LOW output current I_OL when shorted to Vcc
The resulting current when the gate is in the logic LOW state, but the load forces the output voltage to the supply voltage Vcc. The default value is `-45` mA. This parameter is available when you select the `Quadratic` option for the **Output current-voltage 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` Ω. 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.

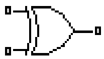
Purpose

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 1-60 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 Vcc (the **Supply voltage** parameter value), and output low is zero volts.

Basic Assumptions and Limitations

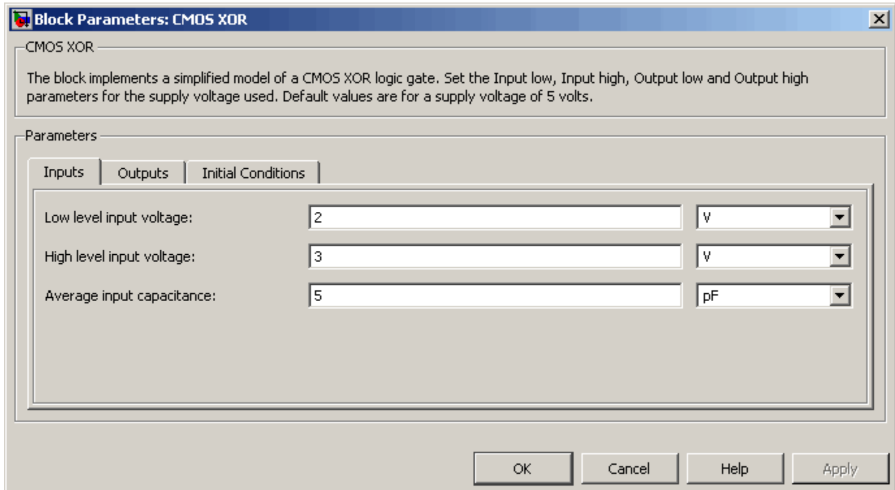
The block does not model the internal individual MOSFET devices that make up the gate (except for the final MOSFET pair if you select the `Quadratic` option for the **Output current-voltage relationship** parameter). This limitation has the following implications:

- 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.

Dialog Box and Parameters

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.

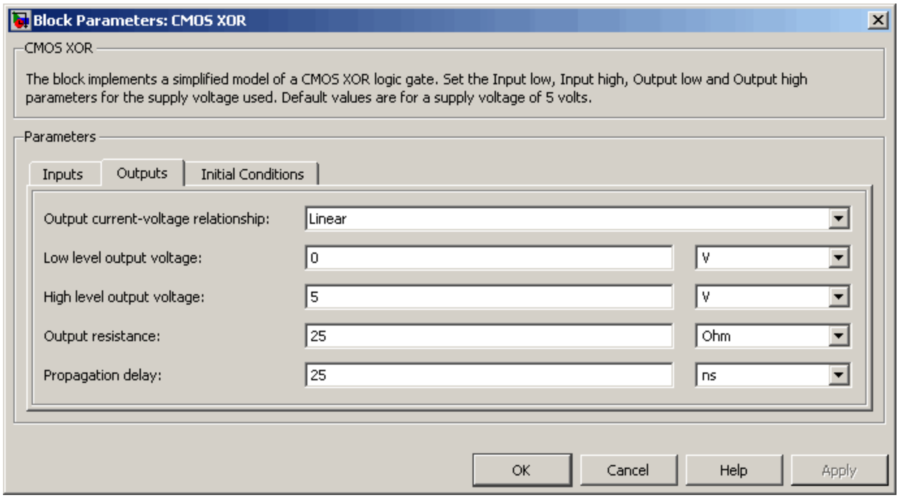
High level input voltage

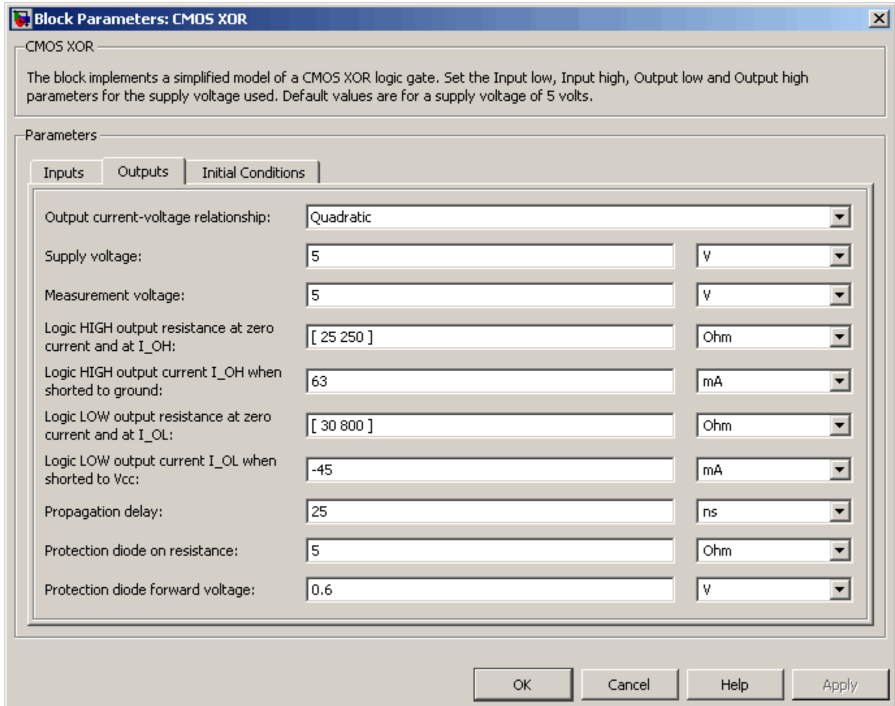
Voltage value above which the block interprets the input voltage as logic `HIGH`. The default value is `3` V.

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


Block Parameters: CMOS XOR
CMOS XOR
The block implements a simplified model of a CMOS XOR logic gate. Set the Input low, Input high, Output low and Output high parameters for the supply voltage used. Default values are for a supply voltage of 5 volts.
Parameters
Inputs
Outputs
Initial Conditions
Output current-voltage relationship:
Linear
Low level output voltage:
0
V
High level output voltage:
5
V
Output resistance:
25
Ohm
Propagation delay:
25
ns
OK
Cancel
Help
Apply



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` Ω. 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 ]` Ω. 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 ]` Ω. This parameter is available when you select the `Quadratic` option for the **Output current-voltage relationship** parameter.

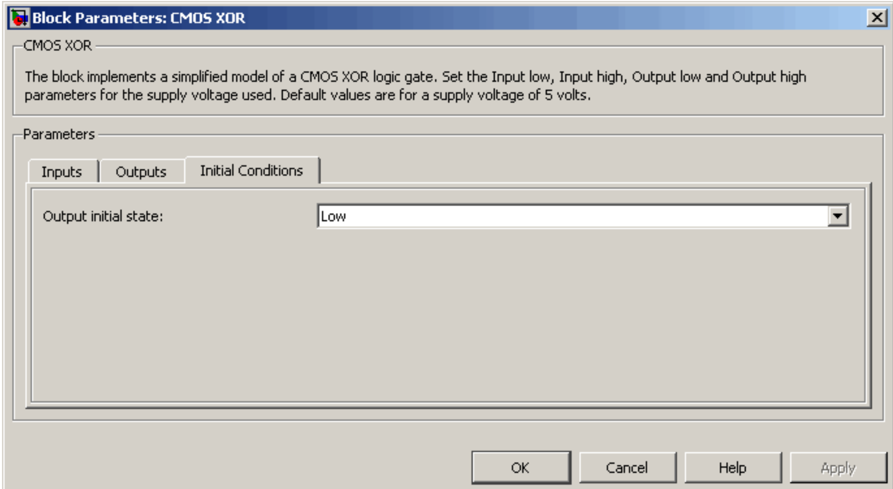
Logic LOW output current I_OL when shorted to Vcc
The resulting current when the gate is in the logic LOW state, but the load forces the output voltage to the supply voltage Vcc. The default value is `-45` mA. This parameter is available when you select the `Quadratic` option for the **Output current-voltage 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` Ω. 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.

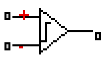
Purpose

Model a comparator behaviorally

Library

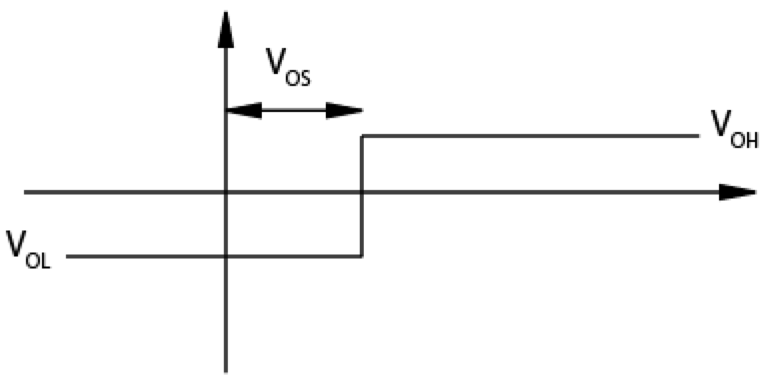
Integrated Circuits

Description

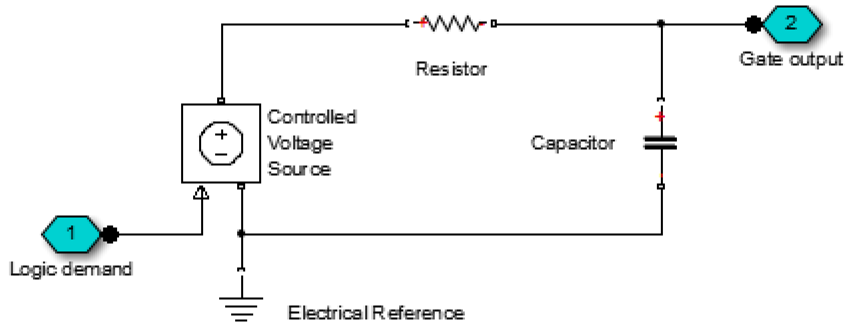


The Comparator block is an abstracted behavioral model of a comparator integrated circuit. It does not model an internal transistor-level implementation. Therefore, the block runs quickly during simulation but retains the correct I/O behavior. The block models differential inputs electrically as having 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.



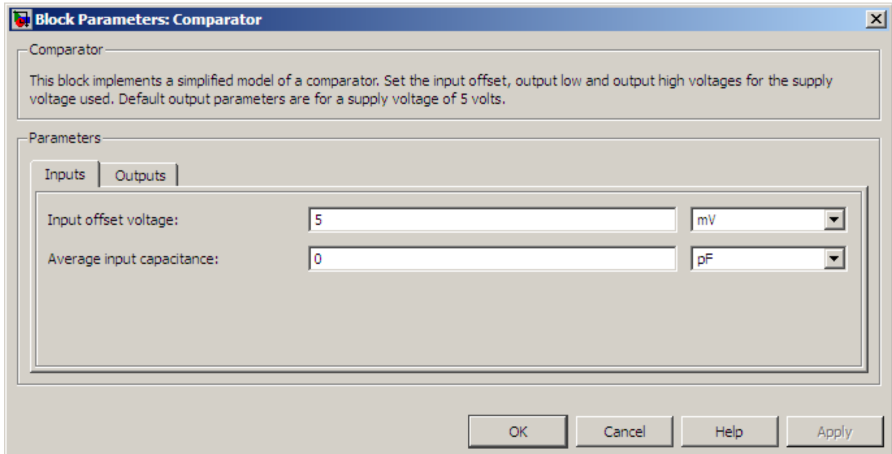
Basic Assumptions and Limitations

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.

Dialog Box and Parameters

Inputs Tab



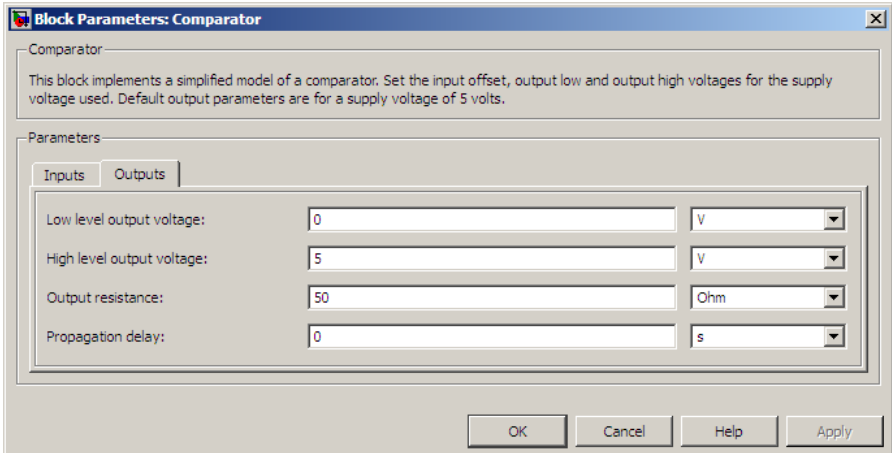
Input offset voltage

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 Ω.

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 .

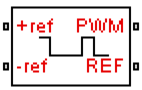
Purpose

Model pulse-width modulated voltage source

Library

Drivers

Description



The Controlled PWM Voltage block represents a pulse-width modulated (PWM) voltage source that depends on the reference voltage V_{ref} across its +ref and -ref ports. The demanded duty cycle is

$$100 * \frac{V_{ref} - V_{\min}}{V_{\max} - V_{\min}} \; percent$$

where:

- $V_{\min}$ 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".

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.

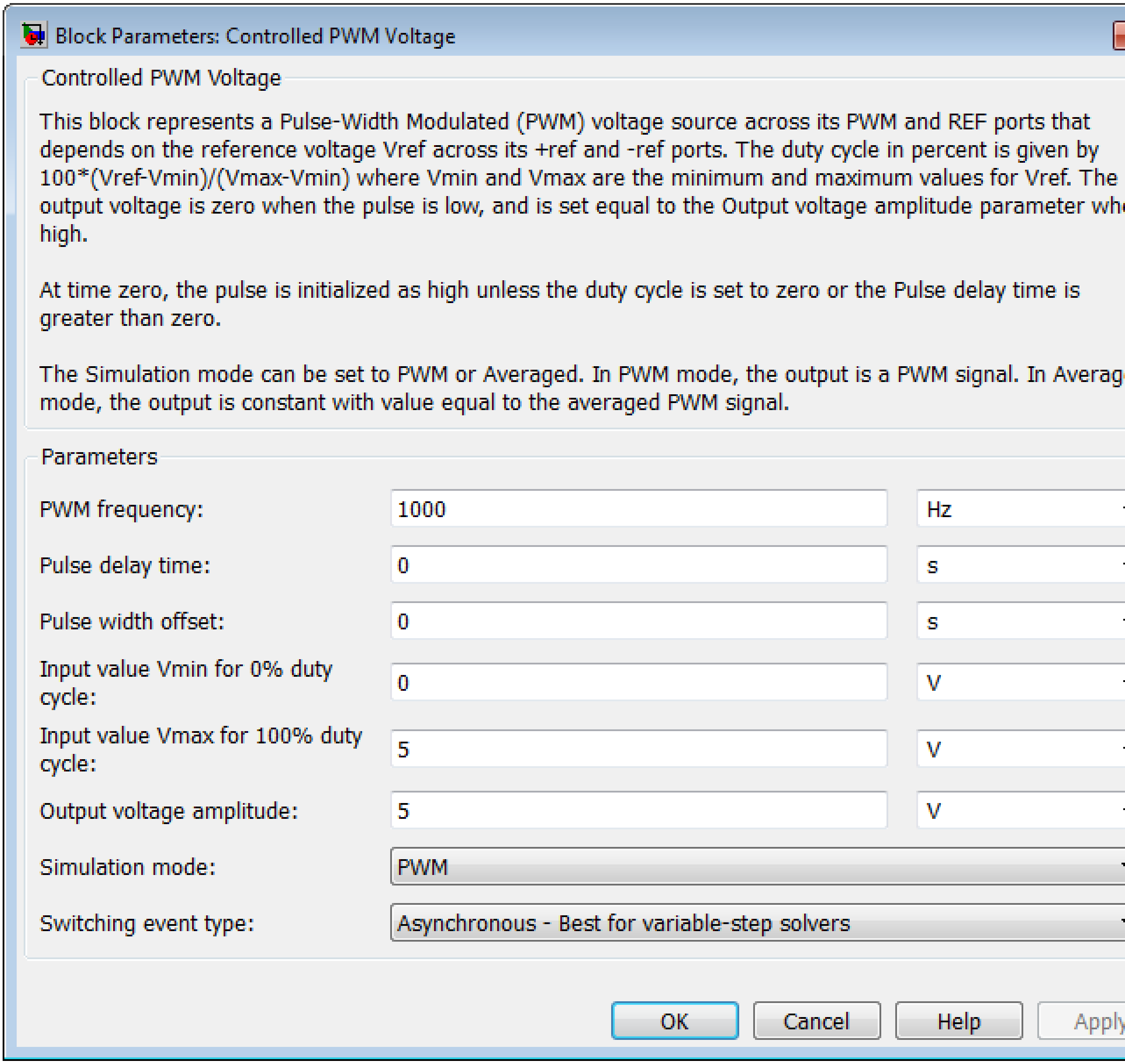
Basic Assumptions and Limitations

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 H-Bridge PWM and REF electrical ports, you must attach an Electrical Reference block to the REF connection line.
- 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.

Dialog Box and Parameters



PWM frequency
Frequency of the PWM output signal. The default value is `1000` Hz.

Pulse delay time
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. This parameter is only visible when you select `PWM` for the **Simulation mode** parameter. The default value is `0` s.

Pulse width offset
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. This parameter is only visible when you select `PWM` for the **Simulation mode** parameter. The default value is `0` s.

Input value Vmin for 0% duty cycle
Value of the input voltage at which the PWM signal has a 0% duty cycle. The default value is `0` V.

Input value Vmax for 100% duty cycle
Value of the input voltage at which the PWM signal has a 100% duty cycle. The default value is `5` V.

Output voltage amplitude
Amplitude of the PWM signal when the output is high. The default value is `5` V.

Simulation mode

The type of output voltage can be `PWM` or `Averaged`. The default mode, `PWM`, produces a pulse-width modulated signal. In `Averaged` mode, the output is a constant whose value is equal to the average value of the PWM signal.

Switching event type

This parameter is only visible when you select `PWM` for the **Simulation mode** parameter. 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. This is the default.
- `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".

Sample time

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.) This parameter is only visible when you select `Discrete time   Best for fixed-step solvers` for the **Switching event type** parameter. The default value is `1e-6` s.

Ports

The block has the following ports:

`+ref`

Positive electrical reference voltage.

`-ref`

Negative electrical reference voltage.

PWM
: Pulse-width modulated signal.

REF
: Floating zero volt reference.

Examples

See the Linear Electrical Actuator (System-Level Model) and Linear Electrical Actuator (Implementation Model) examples.

See Also

Stepper Motor Driver

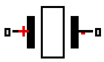
Purpose

Model stable resonator

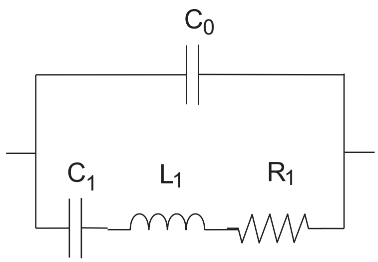
Library

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 *C0* corresponds to the capacitance you specify in the **Shunt capacitance, C0** parameter.
- The capacitor *C1* corresponds to the capacitance you specify in the **Motional capacitance, C1** parameter.
- The inductor *L1* 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:

  $$f_a = \frac{1}{2\pi\sqrt{L_1 C_1 \left(C_0 + C_L\right)/(C_1 + C_0 + C_L)}}$$

  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:

$$Q = \frac{2\pi f L_1}{R_1}$$

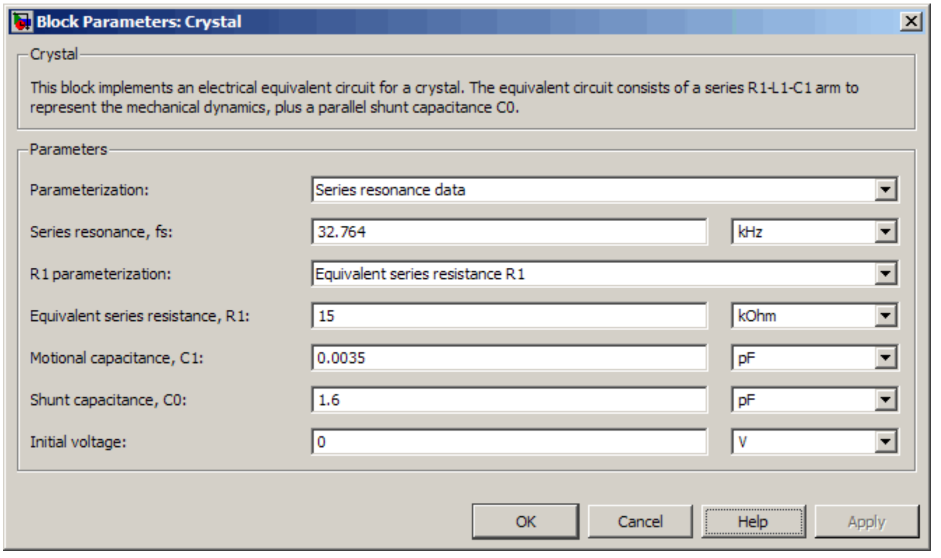
Where Q is the **Quality factor, Q** parameter value.

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.

Dialog Box and 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` kΩ.

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.

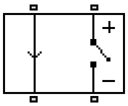
Purpose

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, IT** and **Hysteresis current, IH** parameter values, the switch is closed and has a resistance equal to the **On resistance, RON** parameter value.
- When the controlling current is less than the **Threshold current, IT** parameter value minus the **Hysteresis current, IH** parameter value, the switch is open and has a resistance equal to the **Off resistance, ROFF** parameter value.
- When the controlling current is greater than or less than the **Threshold current, IT** parameter value by an amount less than or equal to the **Hysteresis current, IH** parameter value, the current is in the *crossover region* and the state of the switch remains unchanged.

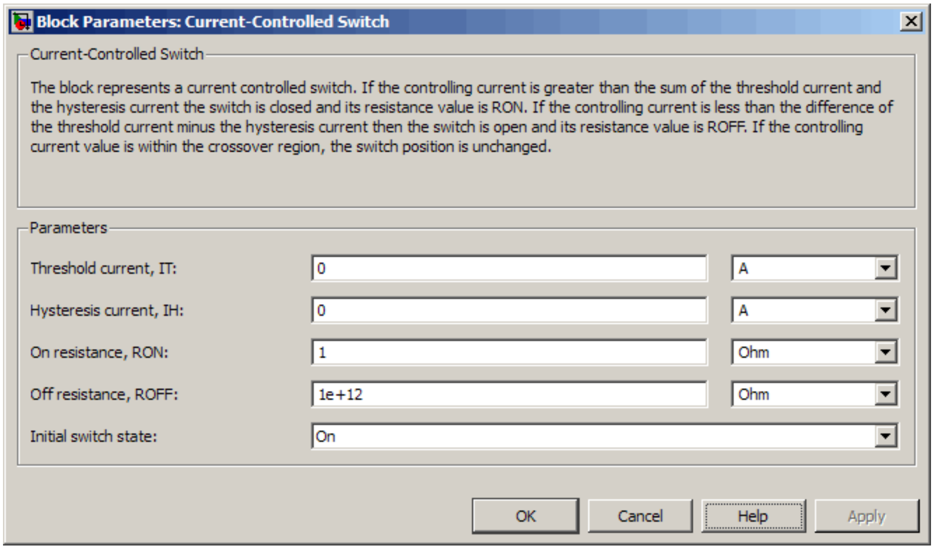
Basic Assumptions and Limitations

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, 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 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.

Dialog Box and 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` Ω.

Off resistance, ROFF
The resistance of the switch when it is open. The default value is `1e+12` Ω.

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 Source

Purpose

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:

$$i = i_{DC} + i_{AC}\sin\left(2\pi ft + \phi\right) + i_N$$

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.
- φ 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:

$$i_N = \sqrt{P_i/2}\,\frac{N\left(0,1\right)}{\sqrt{h}}$$

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);
  ```

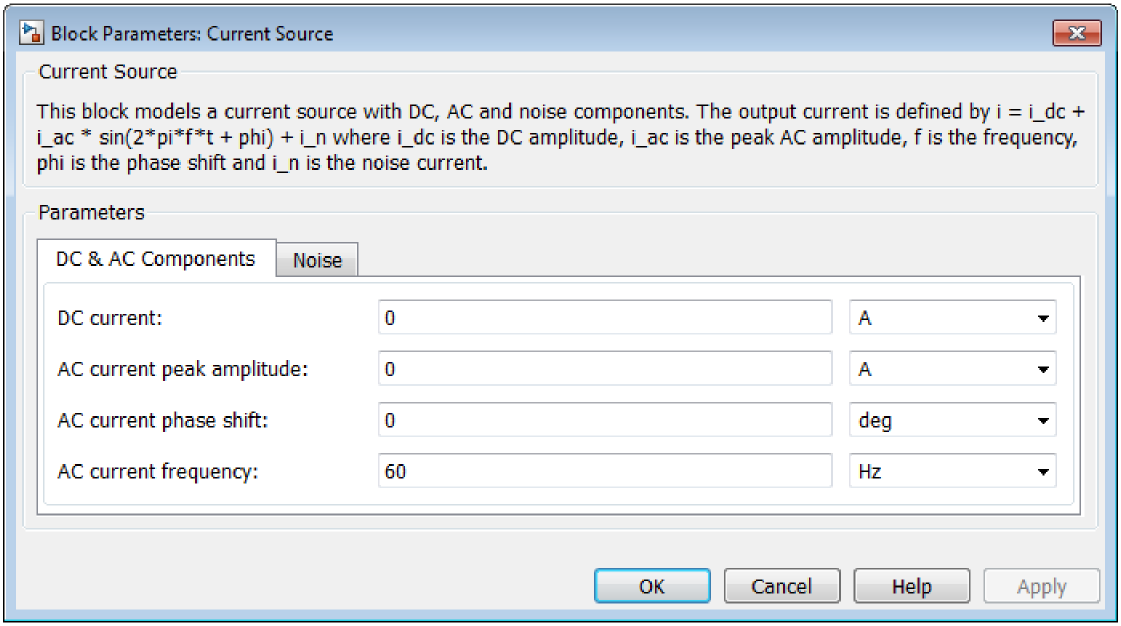
- `Repeatable` — The block uses a hidden parameter, called `auto_seed`, to always start the simulation with the same random number. The value of `auto_seed` is set whenever you copy the Resistor block from the block library to the model, or when you make a new copy of the Resistor block from an existing one in a model. 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.

Dialog Box and Parameters

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.

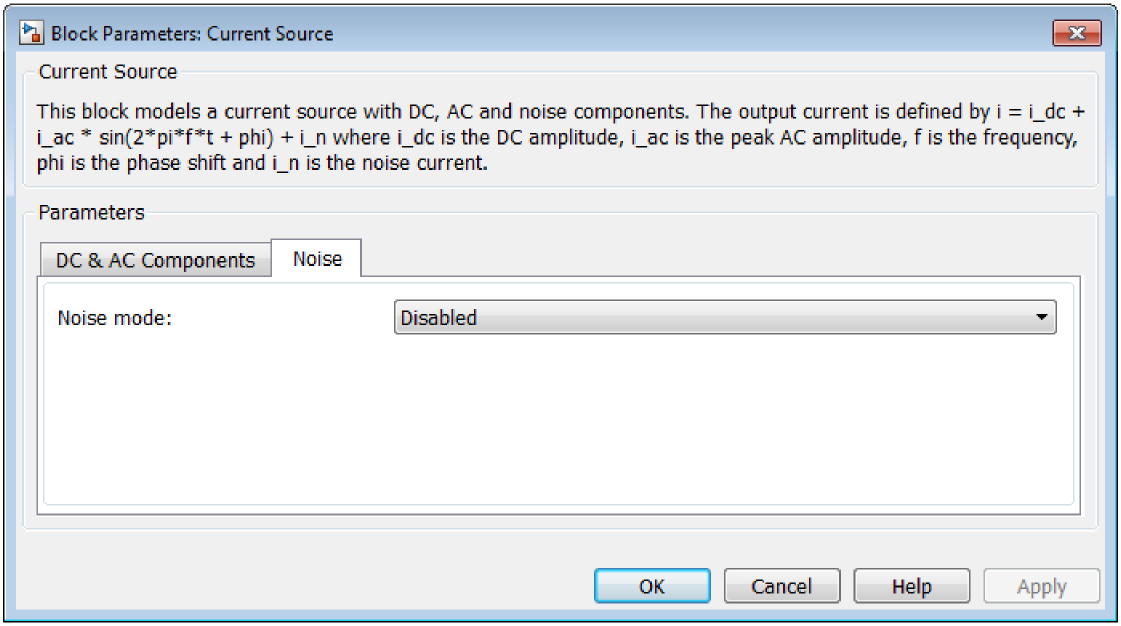
AC current phase shift

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/√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.

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.

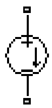
Purpose

Model constant current source

Library

SPICE-Compatible Components/Sources

Description

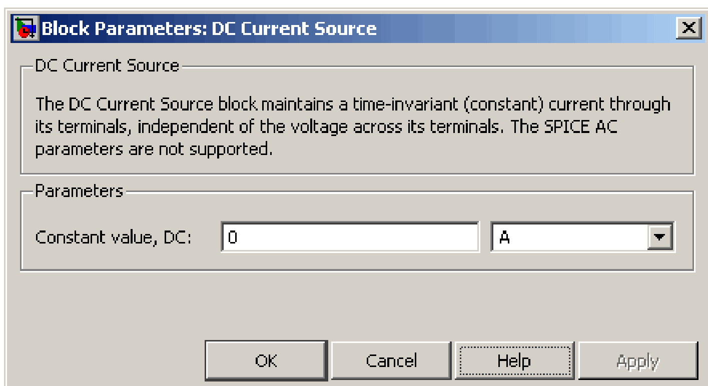


The DC Current Source block represents a constant current source whose output current value is independent of the voltage across its terminals.

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.

Dialog Box and 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

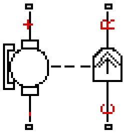
Purpose

DC motor model with electrical and torque characteristics

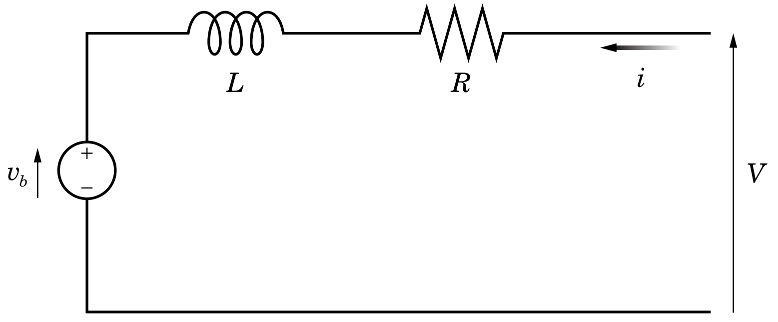
Library

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 ω 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:

$$T_E = \frac{k_t}{R}\left(V - k_v \omega\right)$$

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 λ for all values of the **Model parameterization** parameter. The resulting torque across the block is:

$$T = \frac{k_t}{R}\left(V - k_v\omega\right) - J\dot{\omega} - \lambda\omega$$

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:

$$k_t i_{noload} = \lambda\omega_{noload}$$

where i_{noload} 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 λ 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.

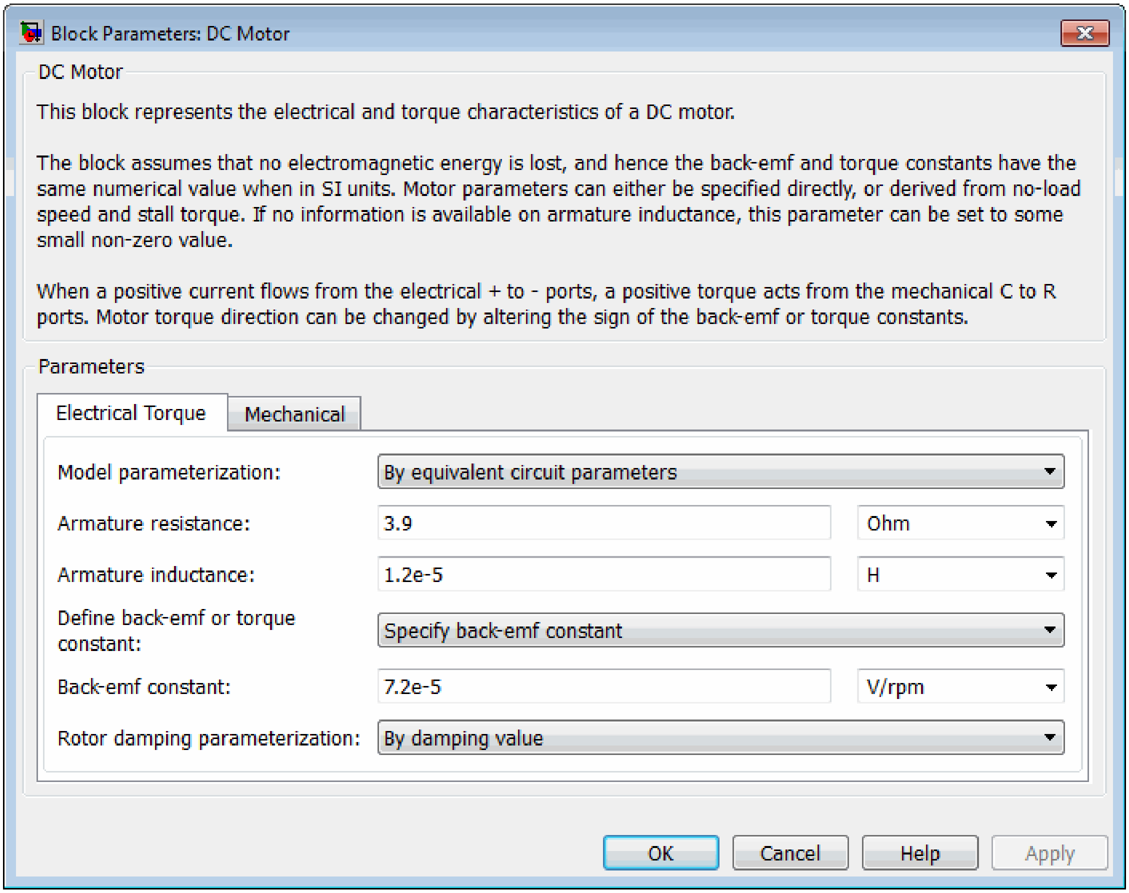
Thermal Port

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".

Dialog Box and Parameters

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` Ω.

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.

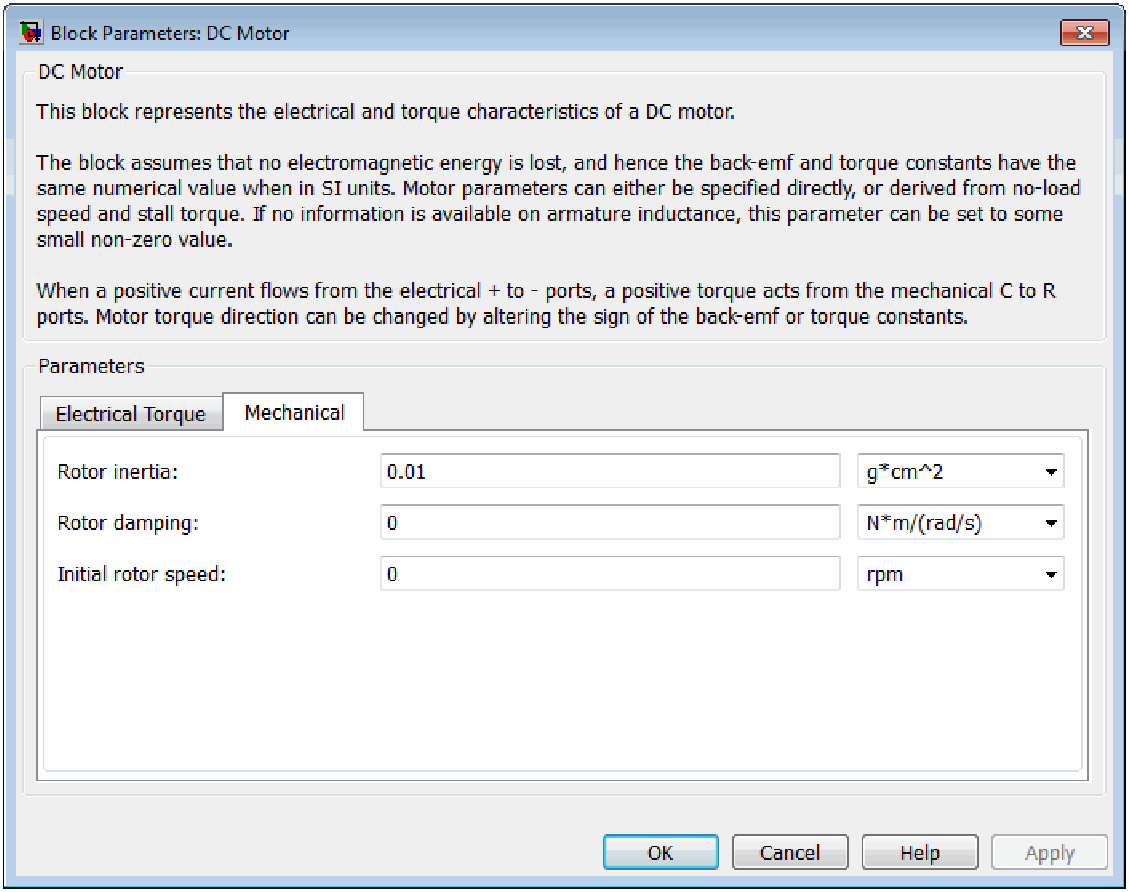
No-load current

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^2. 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:

- Linear Electrical Actuator (Motor Model)
- Linear Electrical Actuator (System-Level Model)
- Linear Electrical Actuator (Implementation Model)

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, and Universal Motor.

Purpose

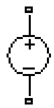
Model constant voltage source

Library

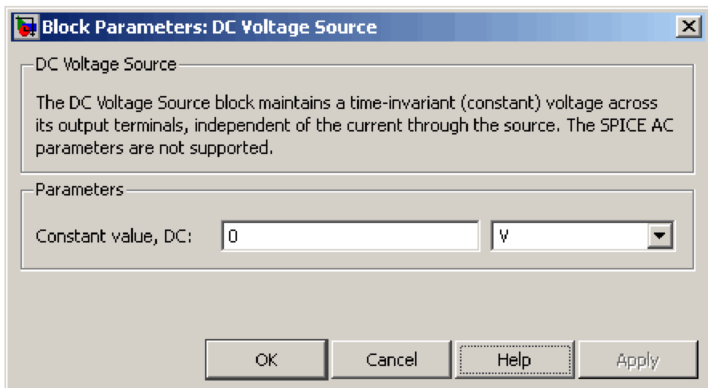
SPICE-Compatible Components/Sources

Description

The DC Voltage Source block represents a constant voltage source whose output voltage value is independent of the current through the source.



Dialog Box and 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

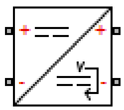
DC Current Source

DC-DC Converter

Purpose Behavioral model of power converter

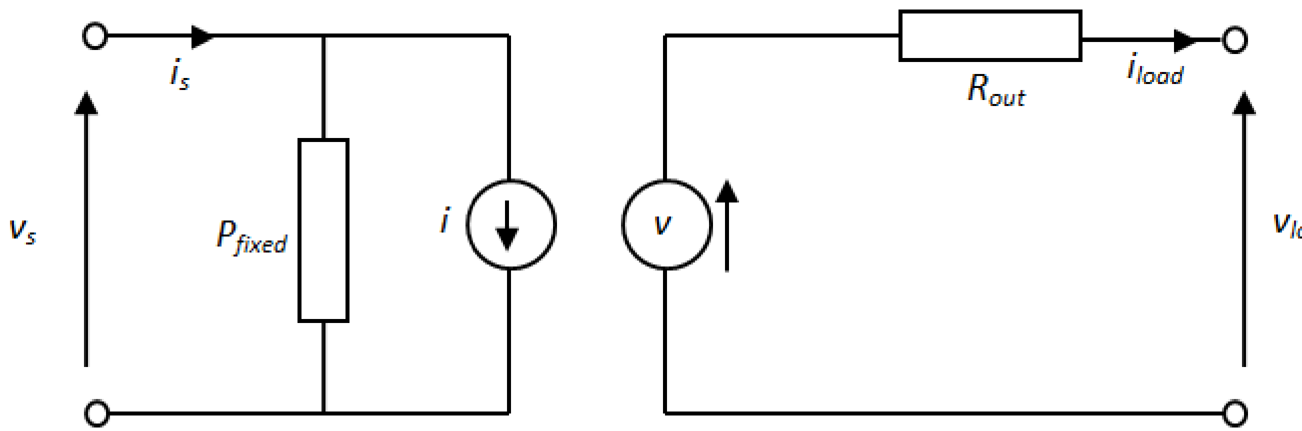
Library Sources

Description



The DC-DC Converter block represents a behavioral model of a power converter. This power converter regulates voltage on the load side, and the required amount of power is drawn from the supply side so as to balance input power, output power, and losses. Optionally the converter can support regenerative power flow from load to supply.

The following circuit illustrates the behavior of the converter.



The P_{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_{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:

$$v = v_{ref} - i_{load}D + i_{load}R_{out}$$

where:

- v_{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.

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.

If the voltage presented by the load is higher than the converter output voltage reference demand, then power will flow from the load to the converter. If you set the **Power direction** parameter to `Unidirectional power flow from supply to regulated side`, then the power is absorbed by the converter, and the current source current i is zero. If you set the **Power direction** parameter to `Bidirectional power flow`, then the 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 step change in load results in a transient change in output voltage, the time constant being defined by the **Voltage regulation time constant** parameter.

Basic Assumptions and Limitations

The model is based on the following 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:

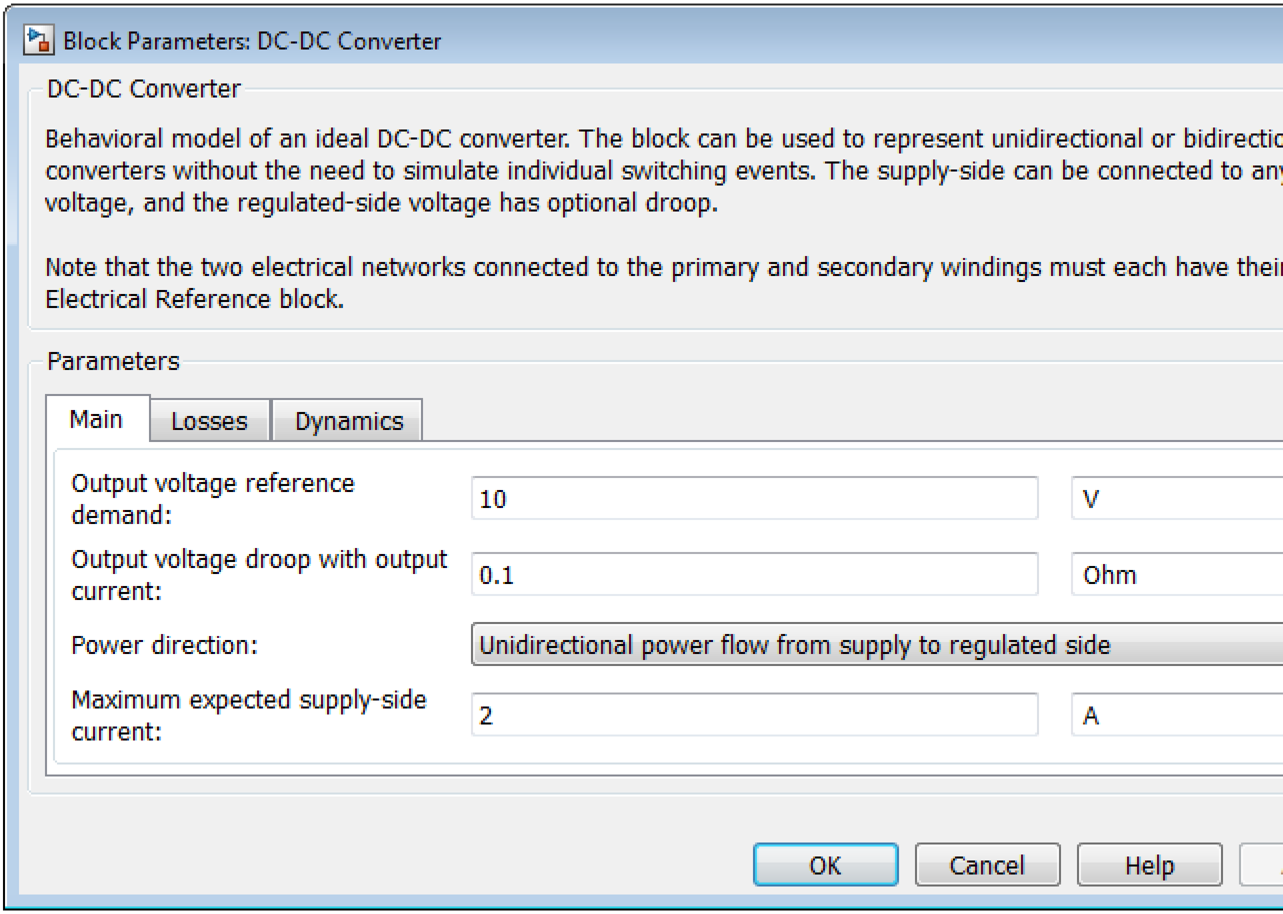
- $v_s > 0$
- $i_s < i_{max}$

The first assertion ensures that the sign of v_s is uniquely defined. The second 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.

Dialog Box and Parameters

- "Main Tab" on page 1-109
- "Losses Tab" on page 1-111
- "Dynamics Tab" on page 1-113

Main Tab



Output voltage reference demand

The set point for the voltage regulator, and the output voltage value when there is no output current. The default value is 10 V.

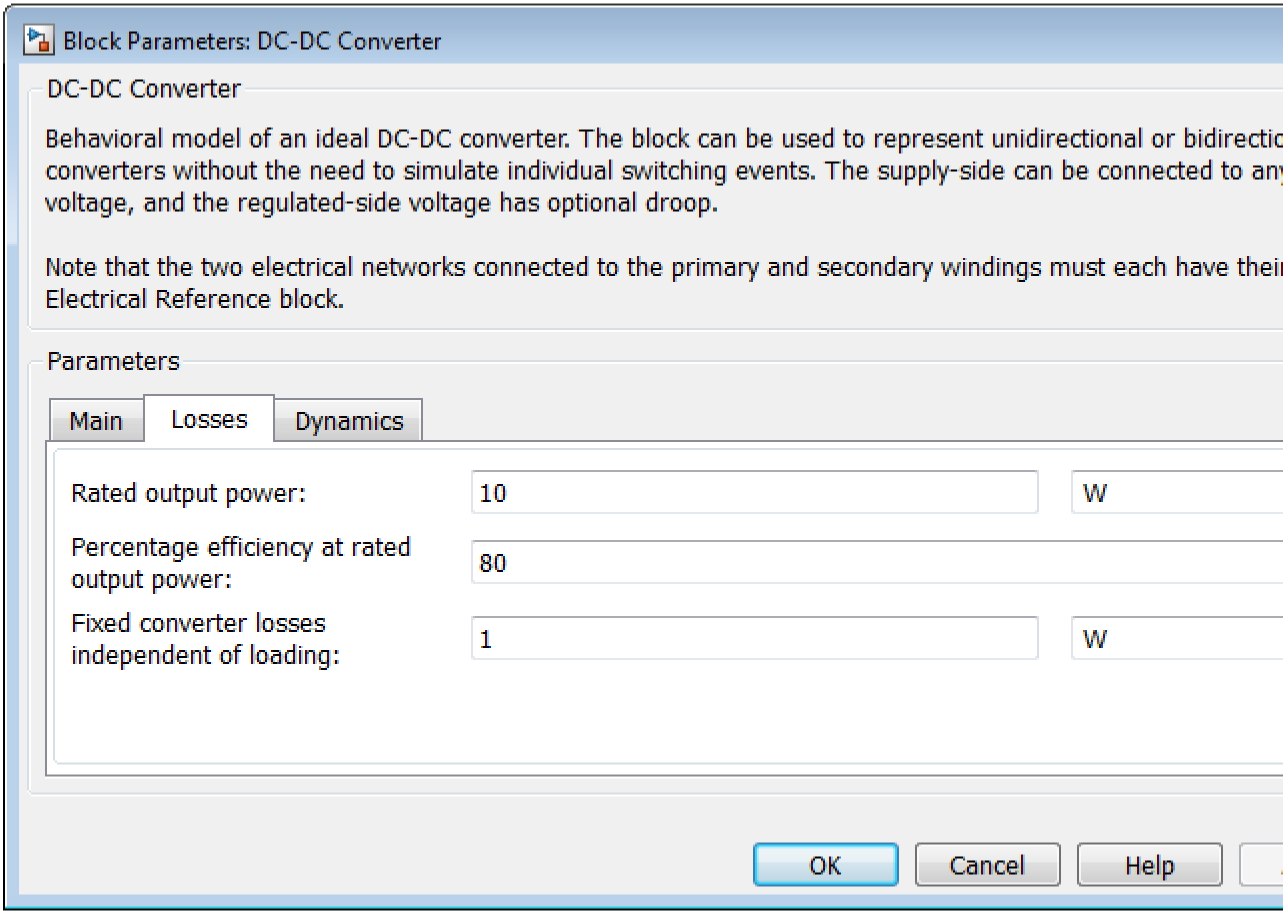
Output voltage droop with output current
The number of volts that the output voltage will drop from the set point for an output current of 1 A. The default value is `0.1` V/A.

Power direction
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

Maximum expected supply-side current
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 "Basic Assumptions and Limitations" on page 1-107. The default value is `2` A.

Losses Tab



Rated output power

The output power for which the percentage efficiency value is given. The default value is 10 W.

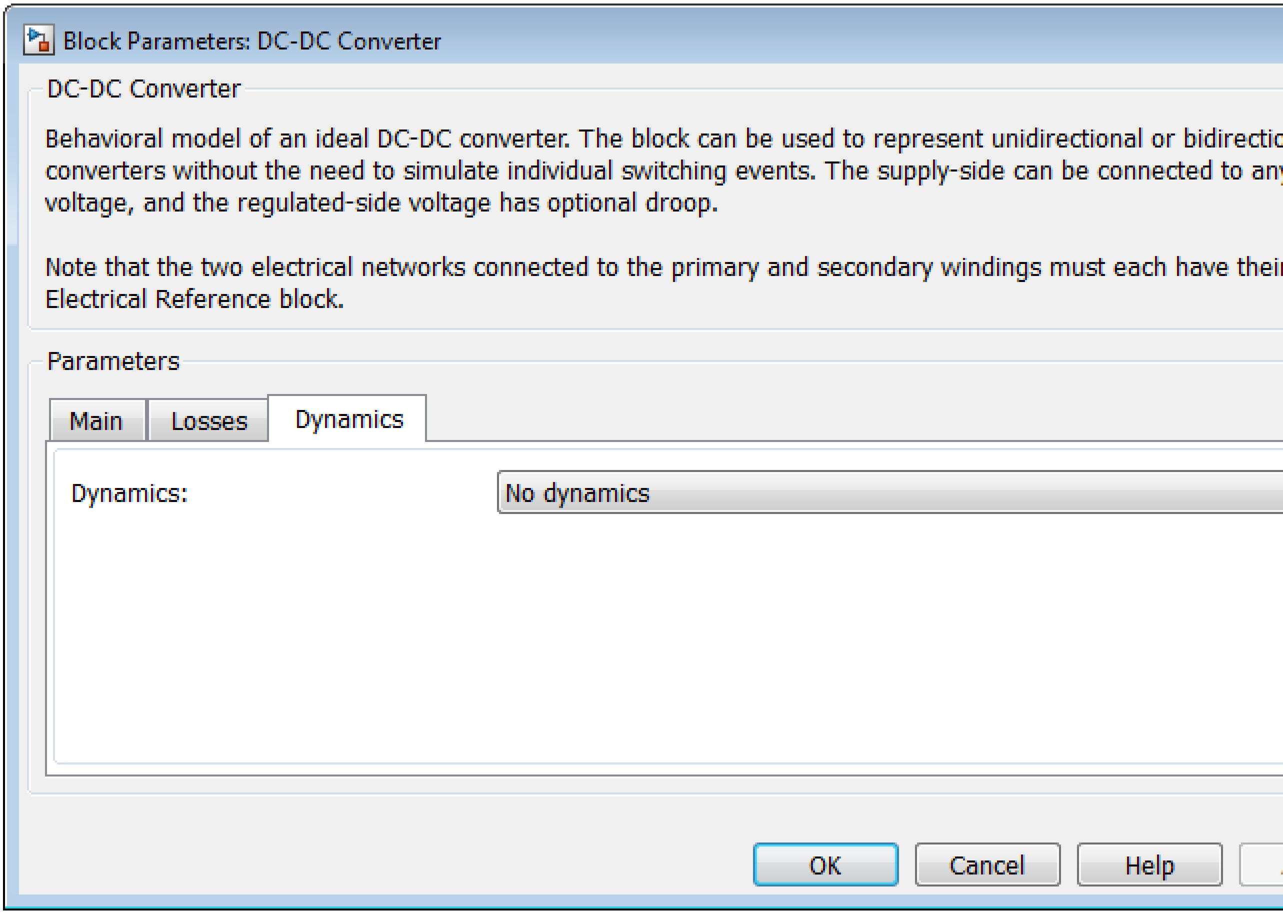
Percentage efficiency at rated output power

The efficiency as defined by 100 times the output load power divided by the input supply power. The default value is 80 percent.

Fixed converter losses independent of loading

The power drawn by the P_{fixed} component in the equivalent circuit diagram, which corresponds to converter losses that are independent of load current. The default value is 1 W.

Dynamics Tab



Dynamics

Specify whether to include voltage regulation dynamics:

- `No dynamics` — Do not consider the voltage regulation dynamics. This is the default option.

- `Specify voltage regulation time constant` — Add a first-order lag to the equation defining the voltage source value. With the dynamics enabled, a step change in load results in a transient change in output voltage.

Voltage regulation time constant
The time constant associated with voltage transients when the load current is stepped. This parameter is only visible when you select `Specify voltage regulation time constant` for the **Dynamics** parameter. The default value is `0.02` s.

Initial output voltage demand
This is the value of v_{ref} at time zero. Normally, v_{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_{ref} value by using this parameter, and increase it accordingly to take account of output resistance and droop. This parameter is only visible when you select `Specify voltage regulation time constant` for the **Dynamics** parameter. The default value is `10` V.

Ports

The block has four electrical conserving ports. Polarity is indicated by the `+` and `-` signs.

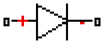
Purpose

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 1-115
- “Piecewise Linear Zener” on page 1-115
- “Exponential” on page 1-116

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 $-Vz$, where Vz is the **Reverse breakdown voltage Vz** parameter value. For voltages less than $-Vz$, the diode behaves as a linear resistor with the low Zener resistance specified in the **Zener resistance Rz** 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:

$$I = IS \cdot \left(e^{\frac{qV}{NkT_{m1}}} - 1 \right) \qquad V > -BV$$

$$I = -IS \cdot \left(e^{\frac{-q(V+Vz)}{kT_{m1}}} - e^{\frac{qV}{NkT_{m1}}} \right) \qquad V \le -BV$$

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:

- $\mathrm{N} = ((V_1 - V_2)/V_t)/(\log(I_1) - \log(I_2))$
- $\mathrm{IS} = \left(I_1/(\exp(V_1/(\mathrm{N}V_t)) - 1) + I_2/(\exp(V_2/(\mathrm{N}V_t)) - 1)\right)/2$

where:

- $V_\mathrm{t} = kT_\mathrm{m1} / q$.
- 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:

$$IS = I_1 / \left(\exp\left(V_1 / (NV_t) - 1 \right) \right)$$

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], 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.

Modeling Charge

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] to capture these effects. The three defining equations are:

$$I = \frac{q_E - q_M}{TT}$$

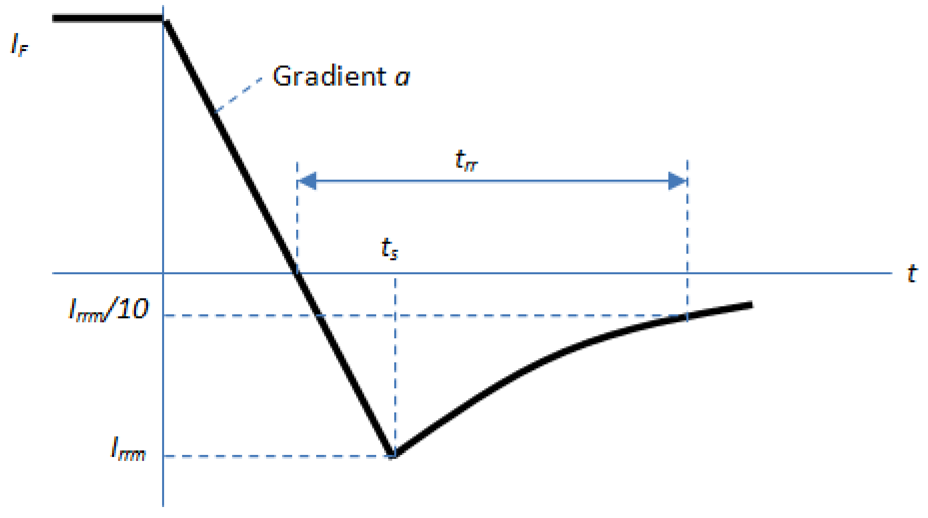
$$\frac{dq_M}{dt} + \frac{q_M}{\tau} - \frac{q_E - q_M}{TT} = 0$$

$$q_E = (\tau + TT) IS \left(\exp\left(\frac{V}{N \cdot V_t} \right) - 1 \right)$$

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.
- τ is the carrier lifetime.

Datasheets do not typically provide values for TT and τ. Therefore the Diode block provides an alternative parameterization in terms of **Peak reverse current, Irrm** and **Reverse recovery time, trr**. Equivalent values for TT and τ 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`

Modeling Temperature Dependence

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:

$$IS_{Ts} = IS_{Tm1} \cdot (T_s / T_{m1})^{XTI/N} \cdot \exp\left(-\frac{EG}{NkT_s}(1 - T_s / T_{m1}) \right)$$

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_{Tm1} is the saturation current at measurement temperature.
- IS_{Ts} 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):

$$BV_{\mathrm{Ts}} = BV_{\mathrm{Tm1}} - TCV \cdot (T_{\mathrm{s}} - T_{\mathrm{m1}})$$

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".

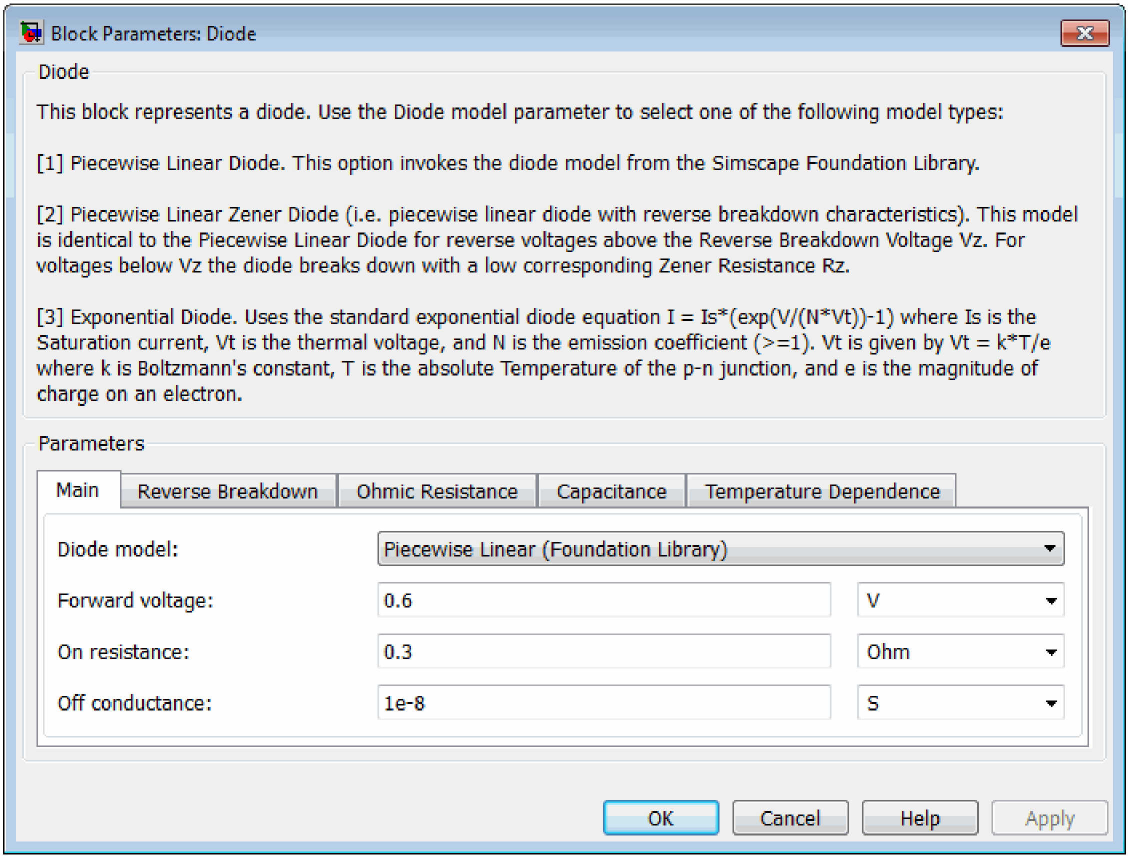
Basic Assumptions and Limitations

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 *IS* 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.

Dialog Box and Parameters

Main Tab



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 1-115. 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 1-115.
- `Exponential` — Use a standard exponential model for the diode, as described in "Exponential" on page 1-116.

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` Ω.

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/Ω.

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

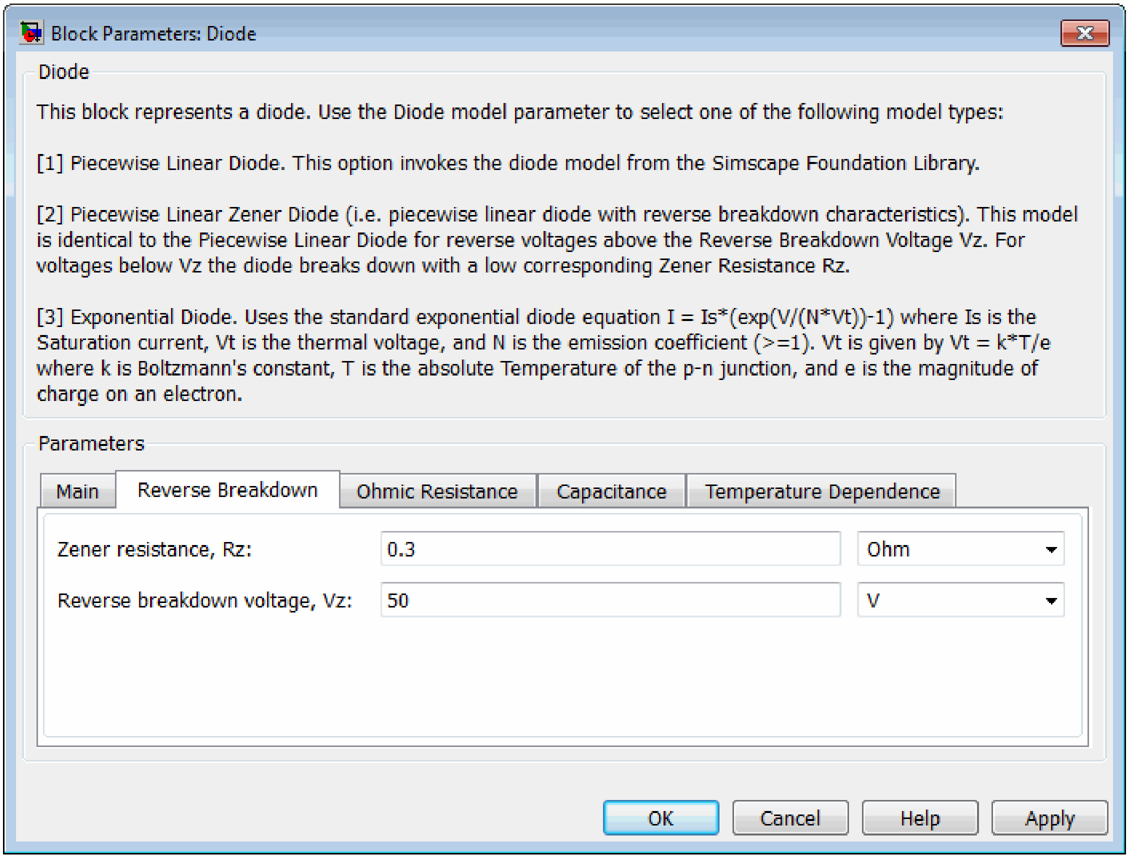
select `Exponential` for the **Diode model** parameter and either `Use an I-V data point and IS` or `Use an I-V data point and N` for the **Parameterization** parameter. Depending on the **Parameterization** value, the block uses this parameter to calculate either *N* or *IS*. The default value is `0.7` 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.

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.

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`.

Reverse Breakdown Tab



Zener resistance Rz

The resistance of the diode when the voltage is less than the **Reverse breakdown voltage Vz** value. This parameter is only visible when you select `Piecewise Linear Zener` for the **Diode model** parameter. The default value is `0.3` Ω.

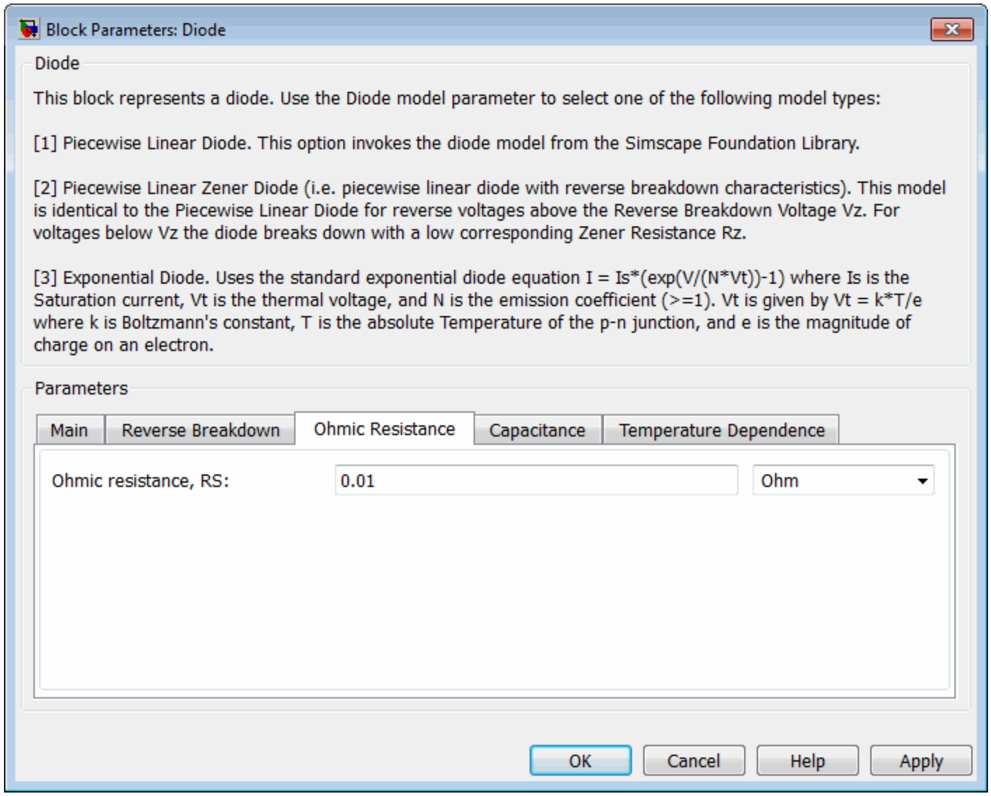
Reverse breakdown voltage Vz

The reverse voltage below which the diode resistance changes to the **Zener resistance Rz** 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.

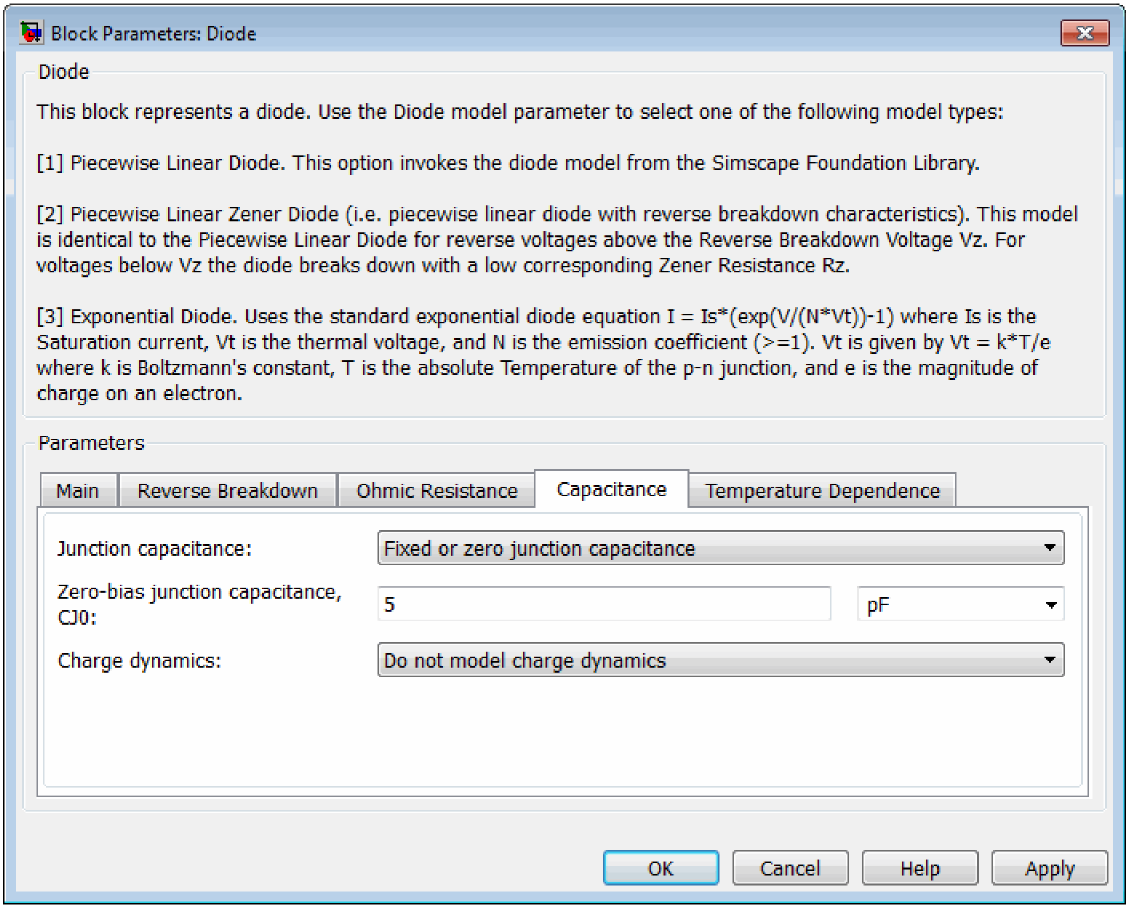
Ohmic Resistance Tab



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` Ω.

Capacitance Tab



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 CJO, 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 CJO, 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 CJO, 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 CJO, 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 CJO, 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, τ.
- `Use transit time and carrier lifetime` — Model charge dynamics by providing values for transit time, TT, and carrier lifetime, τ.

Peak reverse current, Irrm
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 Irrm
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 Irrm
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, trr
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.

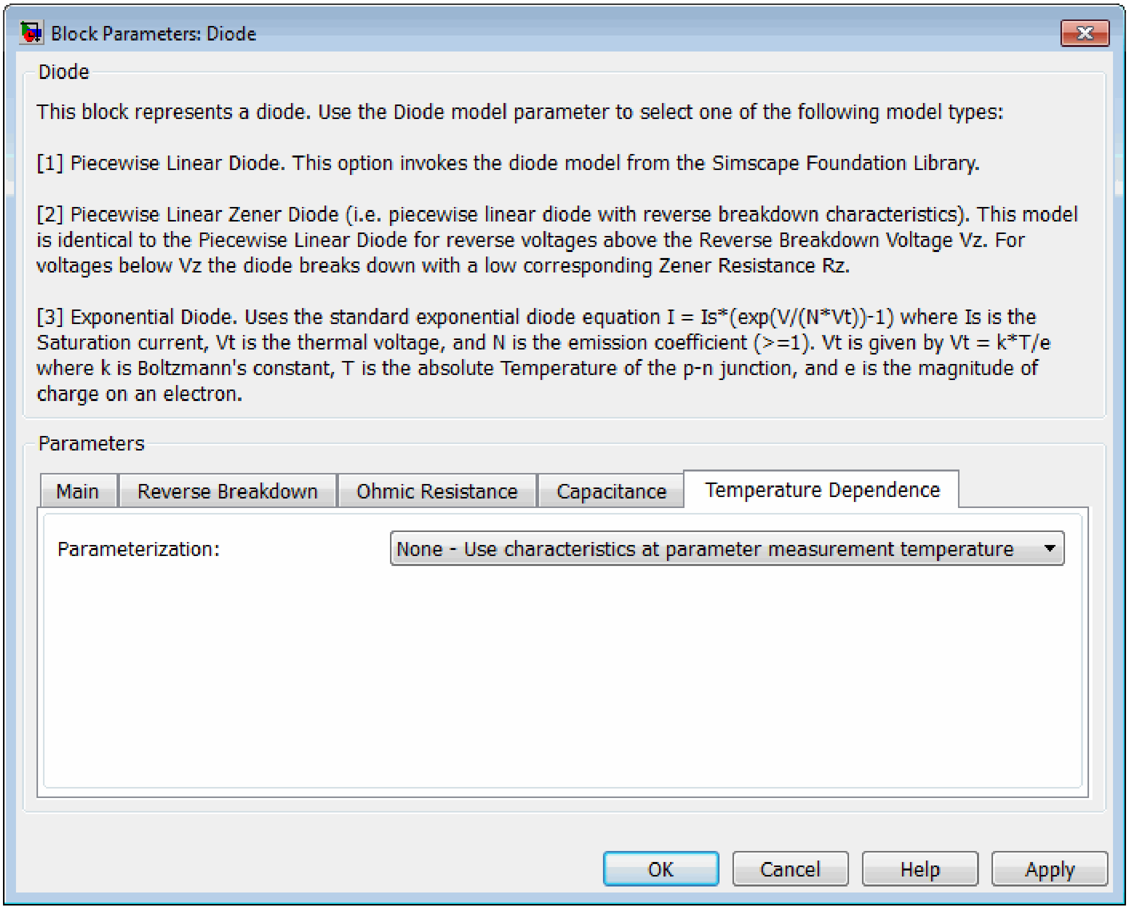
Transit time, TT
A measure of how long it takes carriers to cross the diode junction. This parameter is only visible when you select `Exponential` for the **Diode model** parameter and `Use transit time and carrier lifetime` for the **Charge model** parameter. The default value is `50` ns.

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 Tab

This tab 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 (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`.

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.

See Also

Simscape Diode, SPICE Diode

DPDT Switch

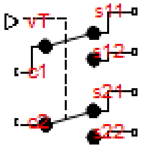
Purpose

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.

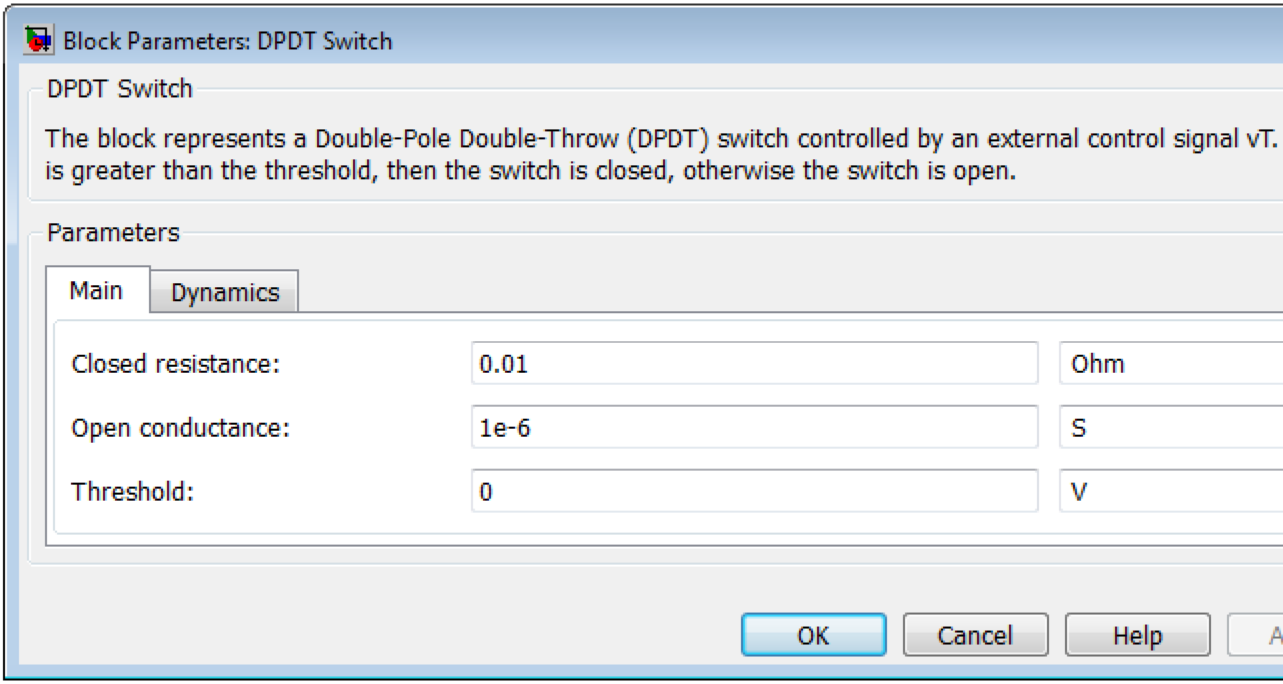
The switch is closed if the voltage presented at the `vT` control port exceeds the value of the **Threshold** parameter.

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`.

Dialog Box and Parameters

- "Main Tab" on page 1-143
- "Dynamics Tab" on page 1-144

Main Tab



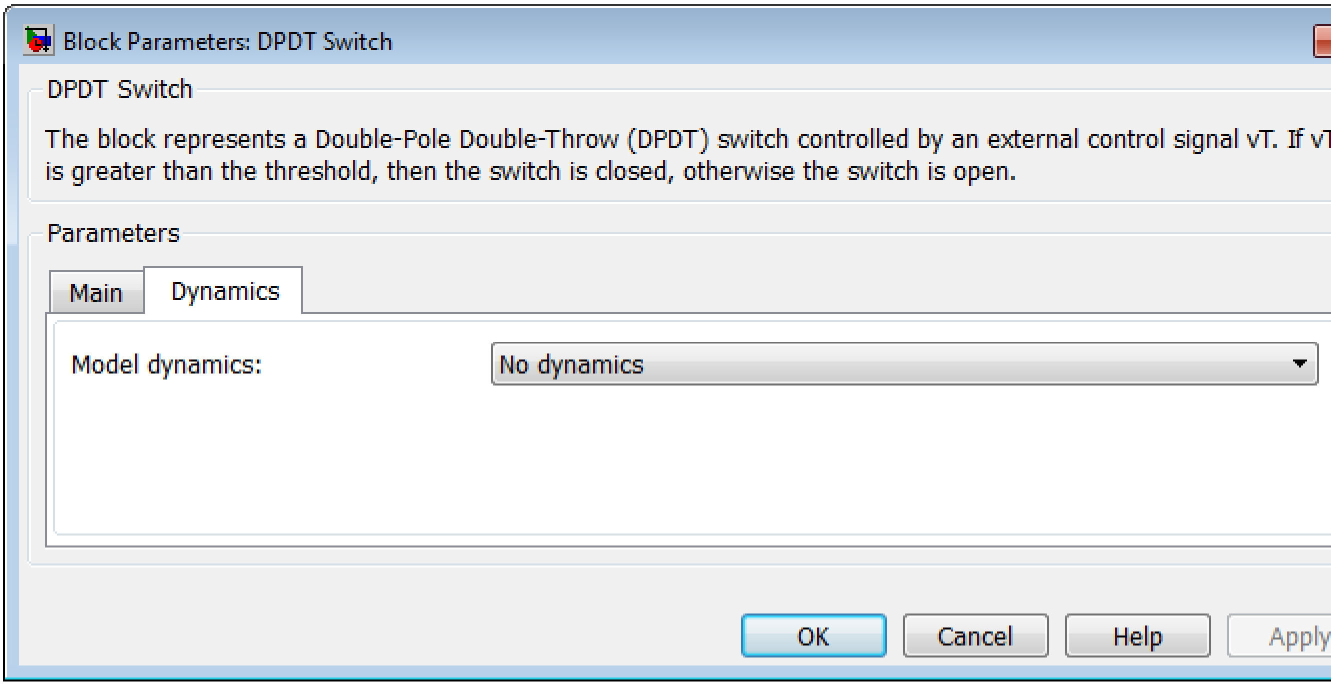
Closed resistance
Resistance between the `c` and `s` electrical ports when the switch is closed. The value must be greater than zero. The default value is `0.01` Ω.

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` 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 vT 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, vT
The value of the physical signal input `vT` 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:

`vT`
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

Simscape Switch

DPST Switch

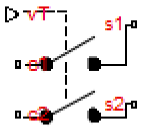
Purpose

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.

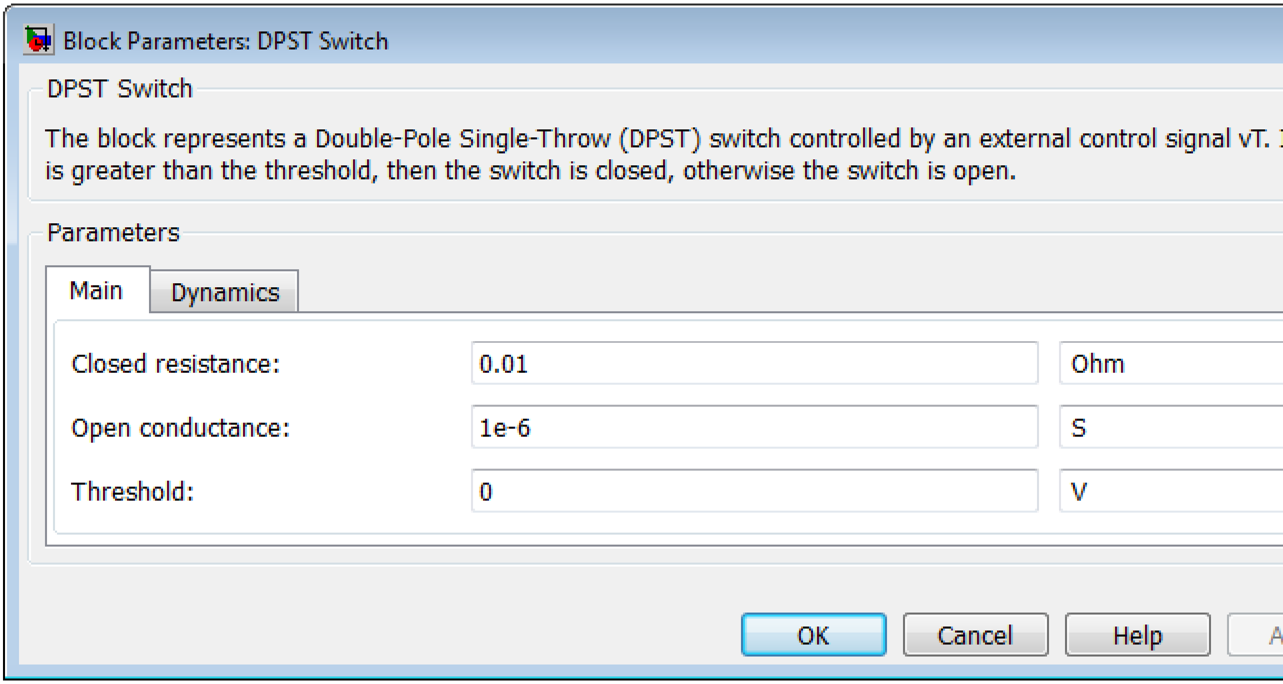
The switch is closed if the voltage presented at the `vT` control port exceeds the value of the **Threshold** parameter.

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`.

Dialog Box and Parameters

- "Main Tab" on page 1-147
- "Dynamics Tab" on page 1-148

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` Ω.

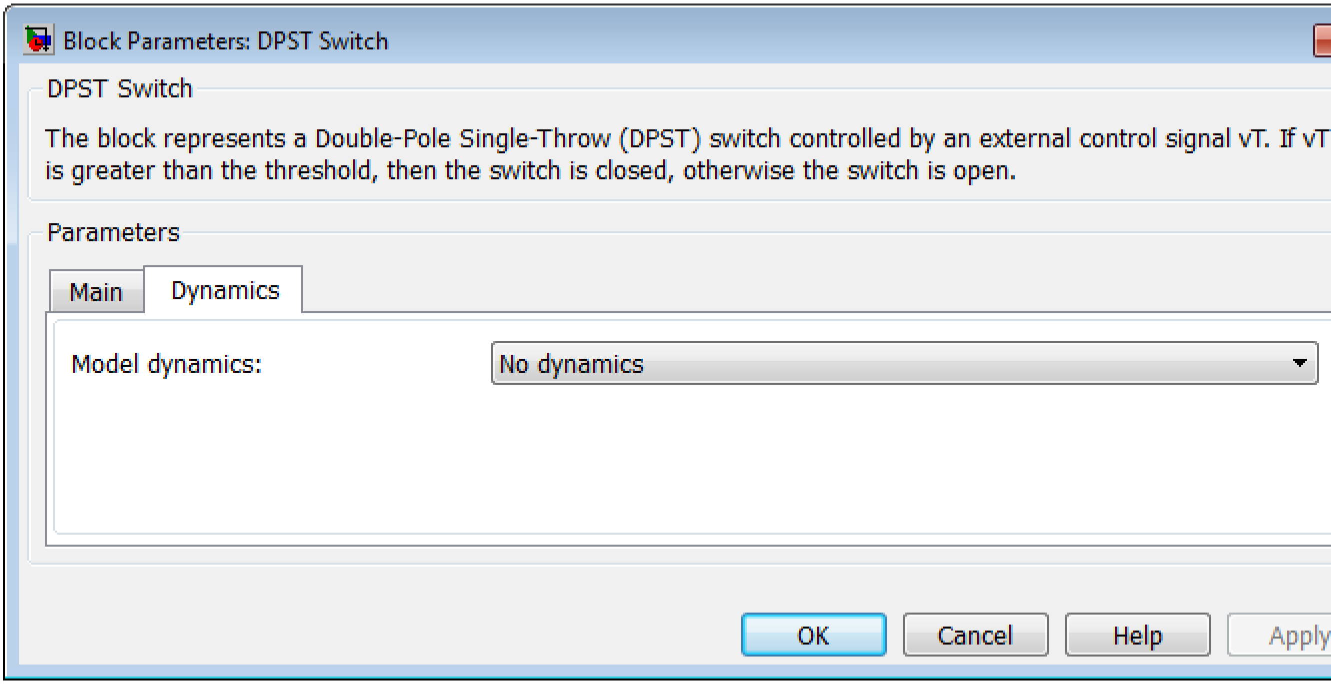
Open conductance

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 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 vT 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, vT
: The value of the physical signal input `vT` 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:

`vT`
: Physical signal that opens and closes the switch.

`c1`, `c2`, `s1`, `s2`
: Electrical conserving ports.

See Also

DPDT Switch

SPDT Switch

SPST Switch

Simscape Switch

Exponential Current Source

Purpose

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:

$$I_{out}\left(0 \le Time \le TDR\right)) = I1$$

$$I_{out}\left(TDR < Time \le TDF\right) = I1 + \left(I2 - I1\right) * \left(1 - e^{-(Time-TDR)/TR}\right)$$

$$I_{out}\left(TDF < Time\right) = I1 + \left(I2 - I1\right) * \left(e^{-(Time-TDF)/TF} - e^{-(Time-TDR)/TR}\right)$$

where:

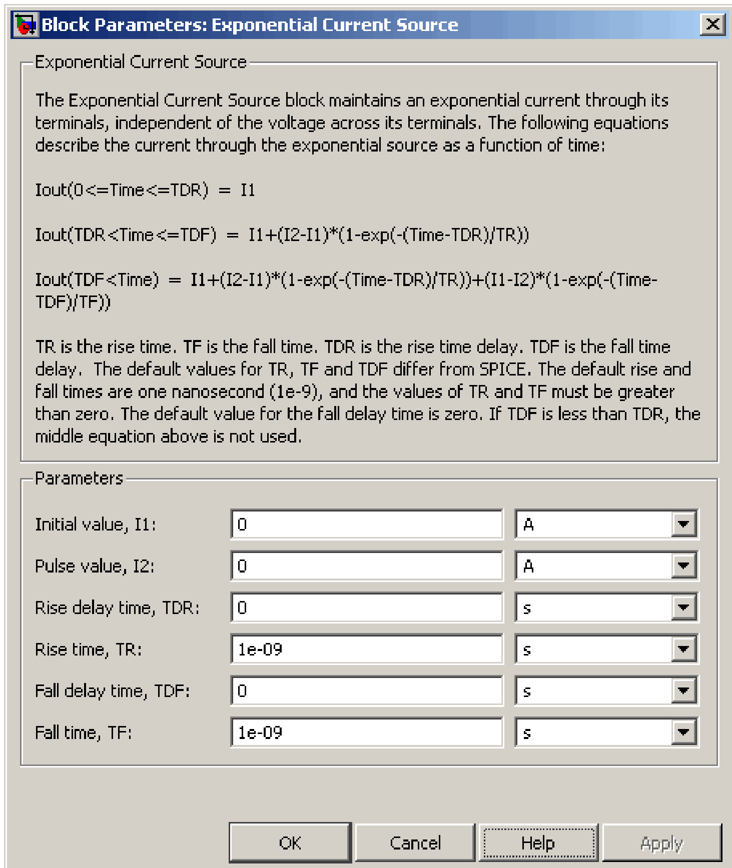
- *I1* is the **Initial value, I1** parameter value.
- *I2* is the **Pulse value, I2** parameter value.
- *TDR* is the **Rise delay time, TDR** parameter value.
- *TR* is the **Rise time, TR** parameter value.
- *TDF* is the **Fall delay time, TDF** parameter value.
- *TF* is the **Fall time, TF** parameter 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.

Dialog Box and 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

Purpose 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:

$$V_{out}\left(0 \le Time \le TDR\right)) = V1$$

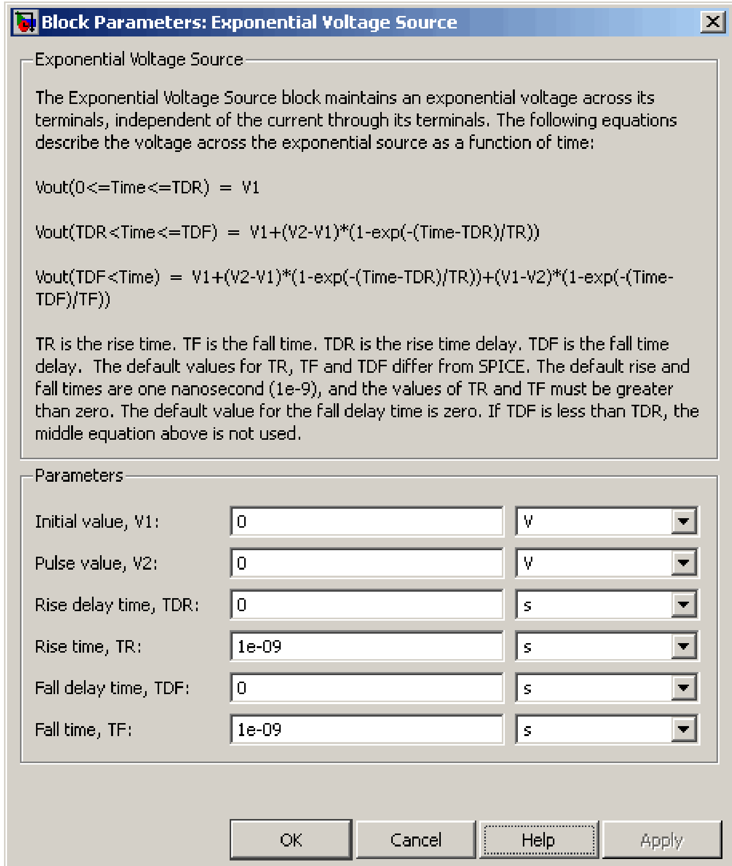
$$V_{out}\left(TDR < Time \le TDF\right) = V1 + \left(V2 - V1\right) * \left(1 - e^{-(Time-TDR)/TR}\right)$$

$$V_{out}\left(TDF < Time\right) = V1 + \left(V2 - V1\right) * \left(e^{-(Time-TDF)/TF} - e^{-(Time-TDR)/TR}\right)$$

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.

Dialog Box and 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

FEM-Parameterized Linear Actuator

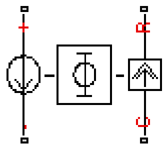
Purpose

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 dPhi(i,x)/dx and dPhi(i,x)/di`, defines the current in terms of partial derivatives of the magnetic flux (Φ) 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 specify the force as a function of current and position. If the finite element package does not provide force, then you can calculate it from the flux using the following equation:

$$F = \int_0^i \frac{\partial \Phi(x,i)}{\partial x} di$$

See the Finite Element Parameterized Solenoid example for more information on how to use this and other parameterization options.

You can define Φ 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”.

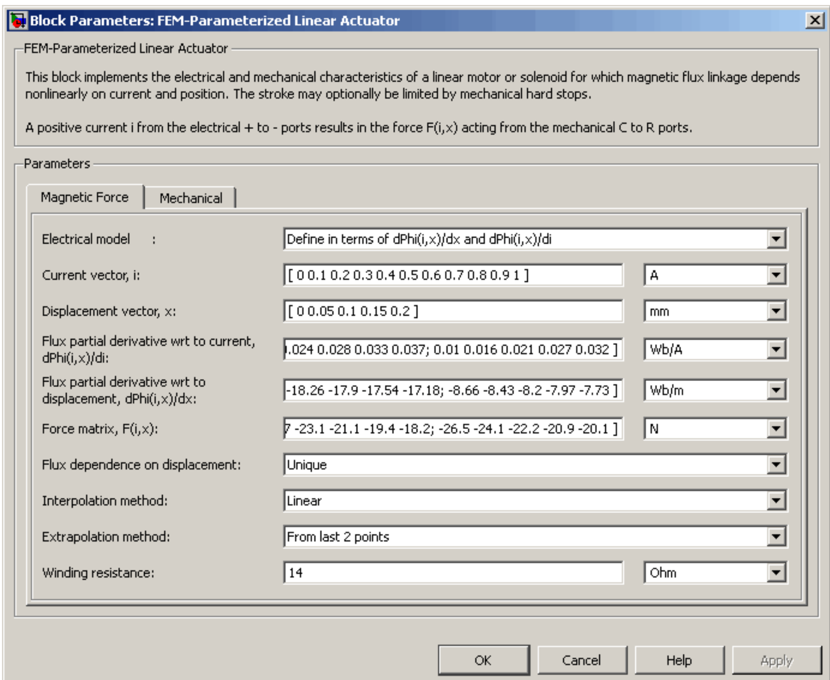
Basic Assumptions and Limitations

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.

Dialog Box and Parameters

Magnetic Force Tab



Electrical model

Select one of the following parameterization options, based on the underlying electrical model:

- `Define in terms of dPhi(i,x)/dx and dPhi(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, Phi(i,x)/di
: Specify a matrix of the flux partial derivatives with respect to current. This parameter is used if **Electrical model** is set to `Define in terms of dPhi(i,x)/dx and dPhi(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, Phi(i,x)/dx

Specify a matrix of the flux partial derivatives with respect to displacement. This parameter is used if **Electrical model** is set to `Define in terms of dPhi(i,x)/dx and dPhi(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 used 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 ]
```

Force matrix, F(i,x)

Specify a matrix of the electromagnetic force applied to the plunger or moving part. 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` — Uses a bilinear interpolation algorithm, which is an extension of linear interpolation for functions in two variables.
- `Cubic` — Uses the bicubic interpolation algorithm.
- `Spline` — Uses the bicubic spline interpolation algorithm.

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:

- `From last 2 points` — Extrapolates using the linear method (regardless of the interpolation method specified), based on the last two output values at the appropriate end of the range. That is, the block uses the first and second specified output values if the input value is below the specified range, and the two last specified output values if the input value is above the specified range.
- `From last point` — Uses the last specified output value at the appropriate end of the range. That is, the block uses the last specified output value for all input values greater than the last specified input argument, and the first specified output value for all input values less than the first specified input argument.

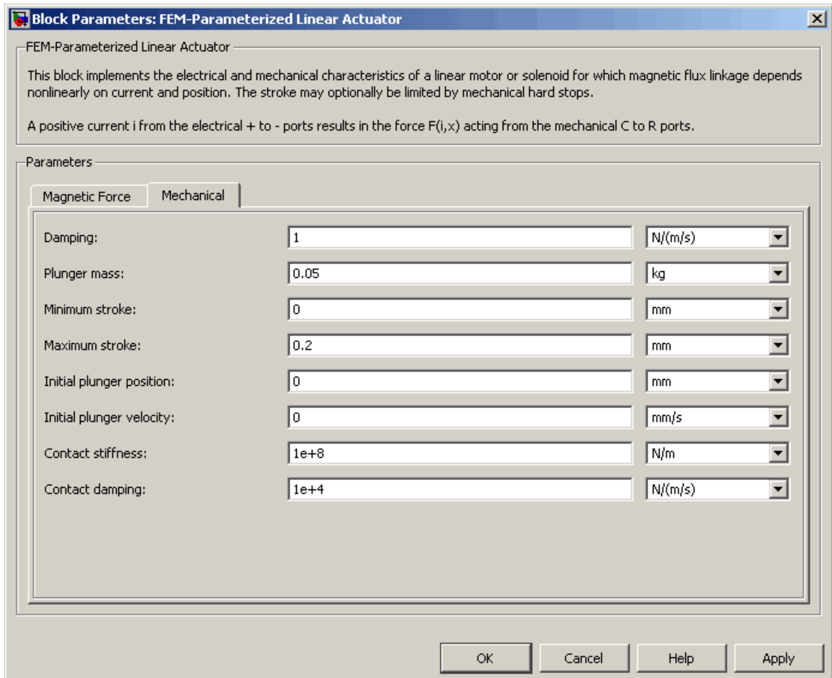
For more information on extrapolation algorithms, see the PS Lookup Table (2D) block reference page.

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 Finite Element Parameterized Solenoid example illustrates the use and parameterization options of this block.

See Also

FEM-Parameterized Rotary Actuator and Solenoid.

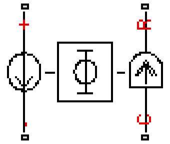
Purpose

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 dPhi(i,theta)/dtheta and dPhi(i,theta)/di`, defines the current in terms of partial derivatives of the magnetic flux (Φ) with respect to rotor angle (θ) 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 specify the torque as a function of current and rotor angle. If the finite element package does not provide torque, then you can calculate it from the flux using the 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.

You can define Φ 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”.

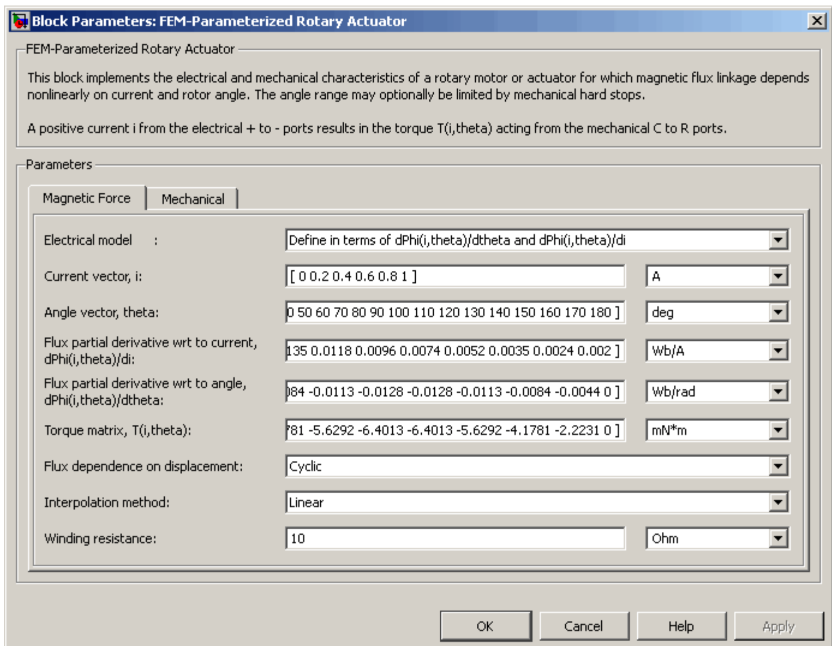
Basic Assumptions and Limitations

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.

Dialog Box and Parameters

Magnetic Force Tab



Electrical model

Select one of the following parameterization options, based on the underlying electrical model:

- `Define in terms of dPhi(i,theta)/dtheta and dPhi(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, Phi(i,theta)/di
: Specify a matrix of the flux partial derivatives with respect to current. This parameter is used if **Electrical model** is set to `Define in terms of dPhi(i,theta)/dtheta and dPhi(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;
  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, Phi(i,theta)/dtheta

Specify a matrix of the flux partial derivatives with respect to rotor angle. This parameter is used if **Electrical model** is set to `Define in terms of dPhi(i,theta)/dtheta and dPhi(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 used 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 ]
```

Torque matrix, T(i,theta)

Specify a matrix of the electromagnetic torque applied to the rotor. 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` — Uses a bilinear interpolation algorithm, which is an extension of linear interpolation for functions in two variables.
- `Cubic` — Uses the bicubic interpolation algorithm.
- `Spline` — Uses the bicubic spline interpolation algorithm.

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:

- `From last 2 points` — Extrapolates using the linear method (regardless of the interpolation method specified), based on the last two output values at the appropriate end of the range. That is, the block uses the first and second specified output values if the input value is below the specified range, and the two last specified output values if the input value is above the specified range.
- `From last point` — Uses the last specified output value at the appropriate end of the range. That is, the block uses the last specified output value for all input values greater than the last specified input argument, and the first specified output value for all input values less than the first specified input argument.

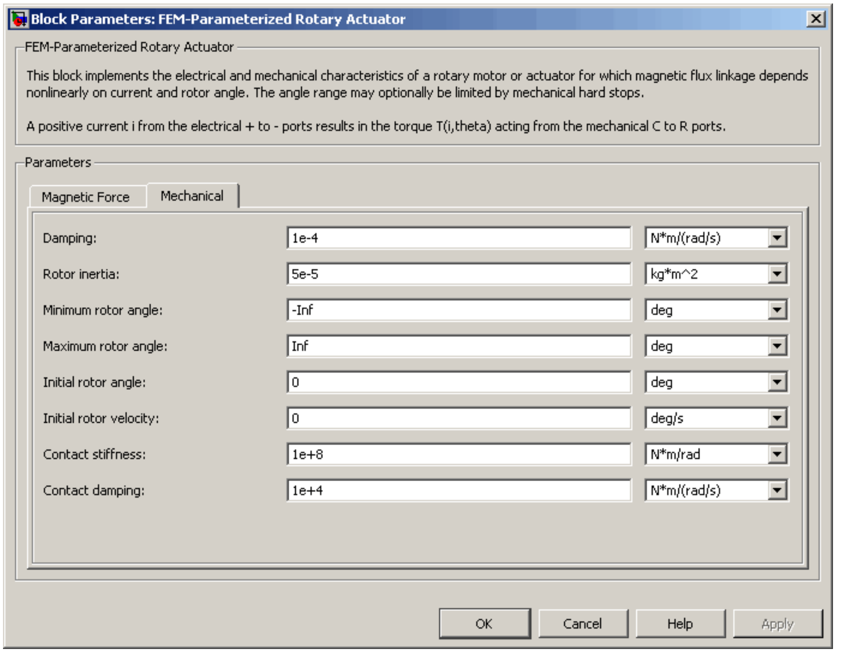
For more information on extrapolation algorithms, see the PS Lookup Table (2D) block reference page.

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` kg*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 `-Inf`.

Maximum rotor angle

The rotor angle at which the upper mechanical end stop is applied. The default value is `Inf`.

Initial rotor position
Position of the rotor at the start of the simulation. The default value is `0` deg.

Initial rotor velocity
Angular velocity of the rotor at the start of the simulation. The default value is `0` deg/s.

Contact stiffness
Contact stiffness between rotor and end stops. The default value is `1e8` N*m/rad.

Contact damping
Contact damping between rotor and end stops. The default value is `1e4` N*m/(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 and Solenoid.

Finite-Gain Op-Amp

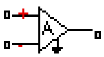
Purpose

Model gain-limited operational amplifier

Library

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 *Vp* and *Vm*, respectively, the output voltage is:

$$V_{out} = A\left(V_p - V_m\right) - I_{out} * R_{out}$$

where:

- A is the gain.
- R_{out} is the output resistance.
- I_{out} is the output current.

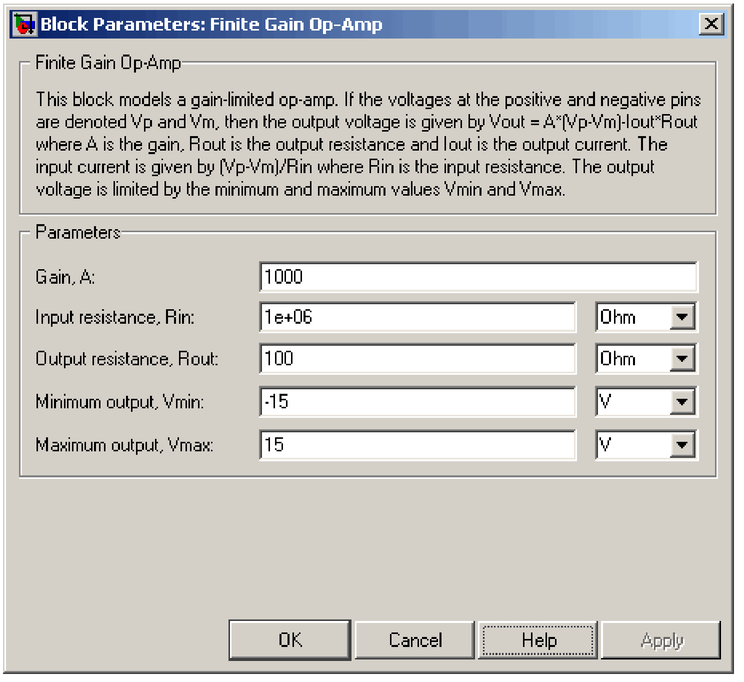
The input current is:

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

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.

Dialog Box and 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 Ω.

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 output current. The default value is 100 Ω.

Minimum output, Vmin
The lower limit on the operational amplifier output voltage. The default value is -15 V.

Maximum output, Vmax
The upper limit on the operational amplifier output voltage. The default value is 15 V.

Ports

The block has the following ports:

+
Positive electrical voltage.

-
Negative electrical voltage.

OUT
Output voltage.

See Also

Simscape Op-Amp, Band-Limited Op-Amp, Fully Differential Op-Amp

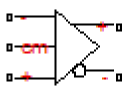
Purpose

Model operational amplifier with fully differential output, that is, not referenced to ground

Library

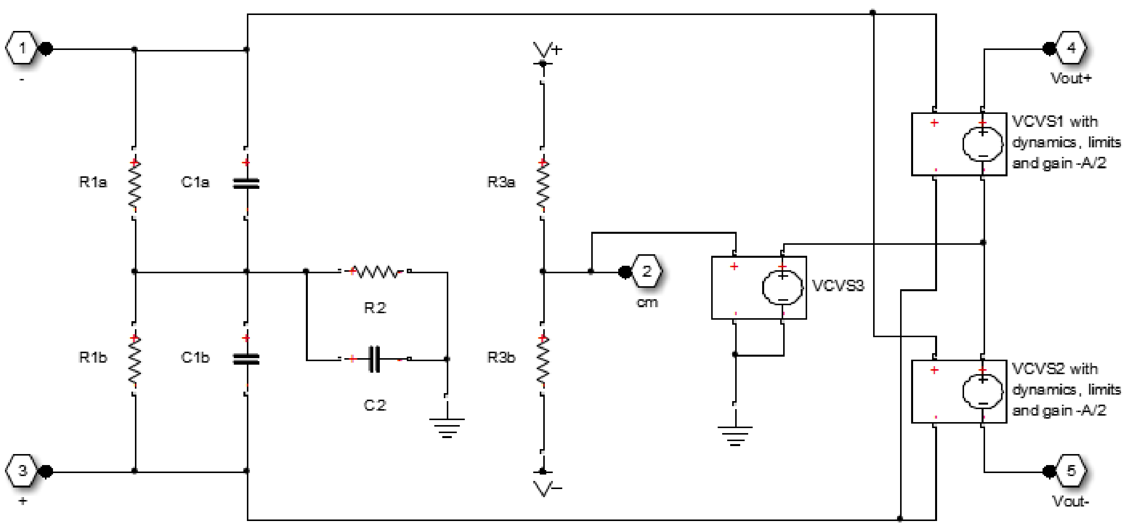
Integrated Circuits

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 `Vout+` and `Vout-` 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.

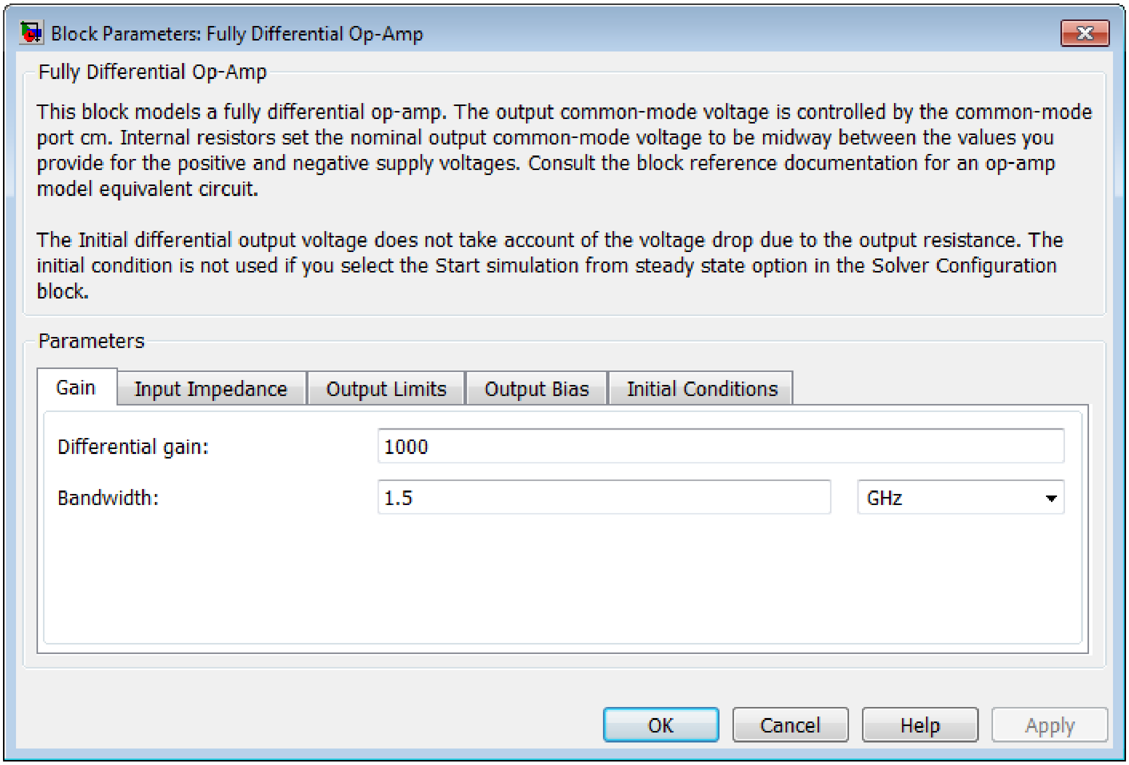
Note Physical Network block diagrams do not allow unconnected Conserving ports. If you want to leave pin `cm` open-circuit, connect it to an Open Circuit block from the Simscape Foundation library.

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.

Dialog Box and Parameters

Gain Tab



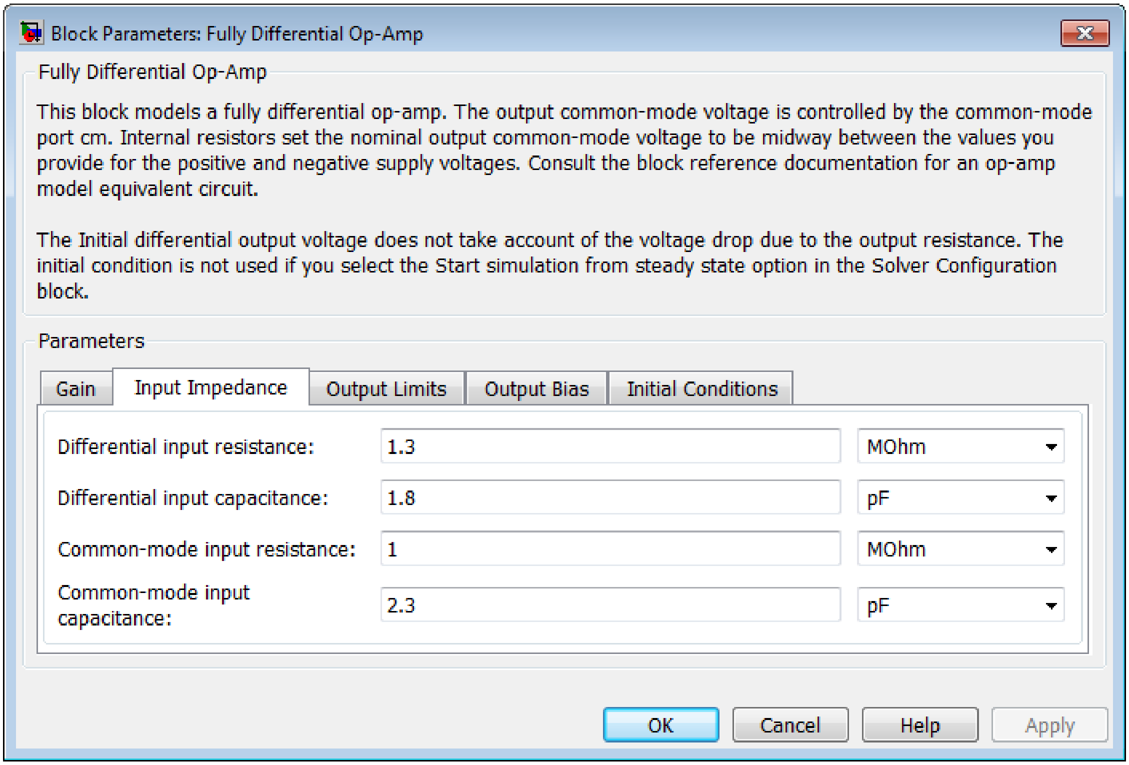
Differential gain

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.

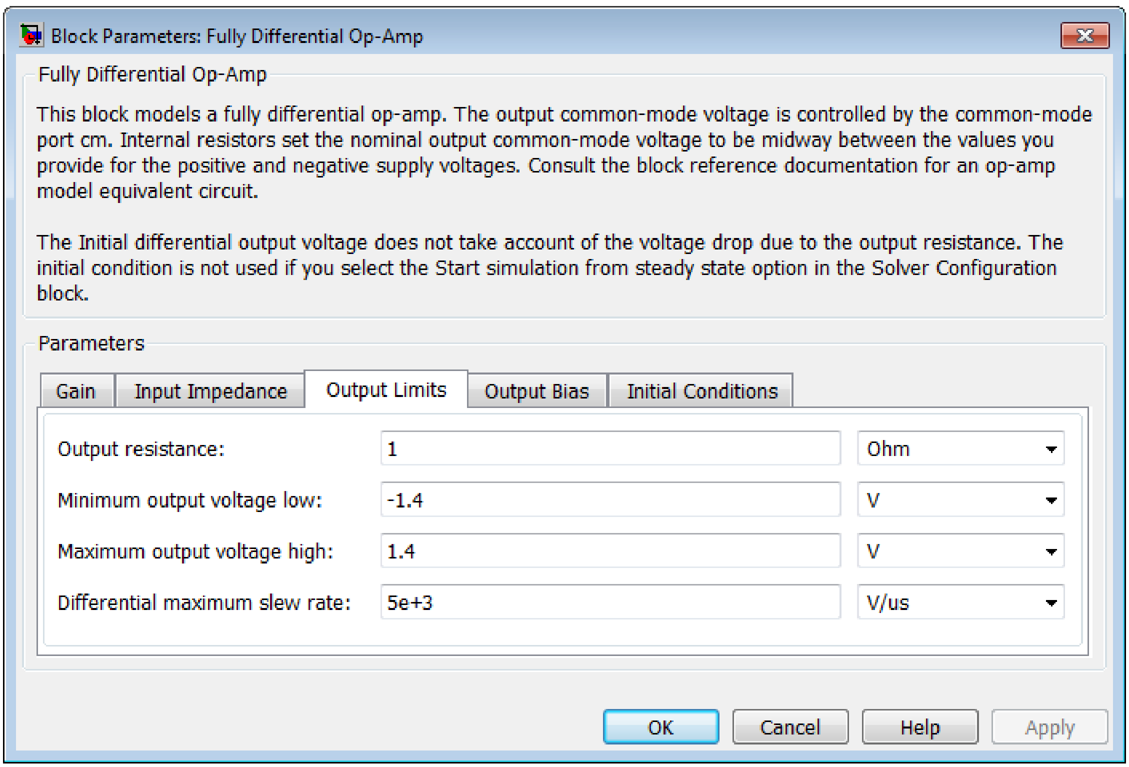
Common-mode input resistance

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_out. 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.

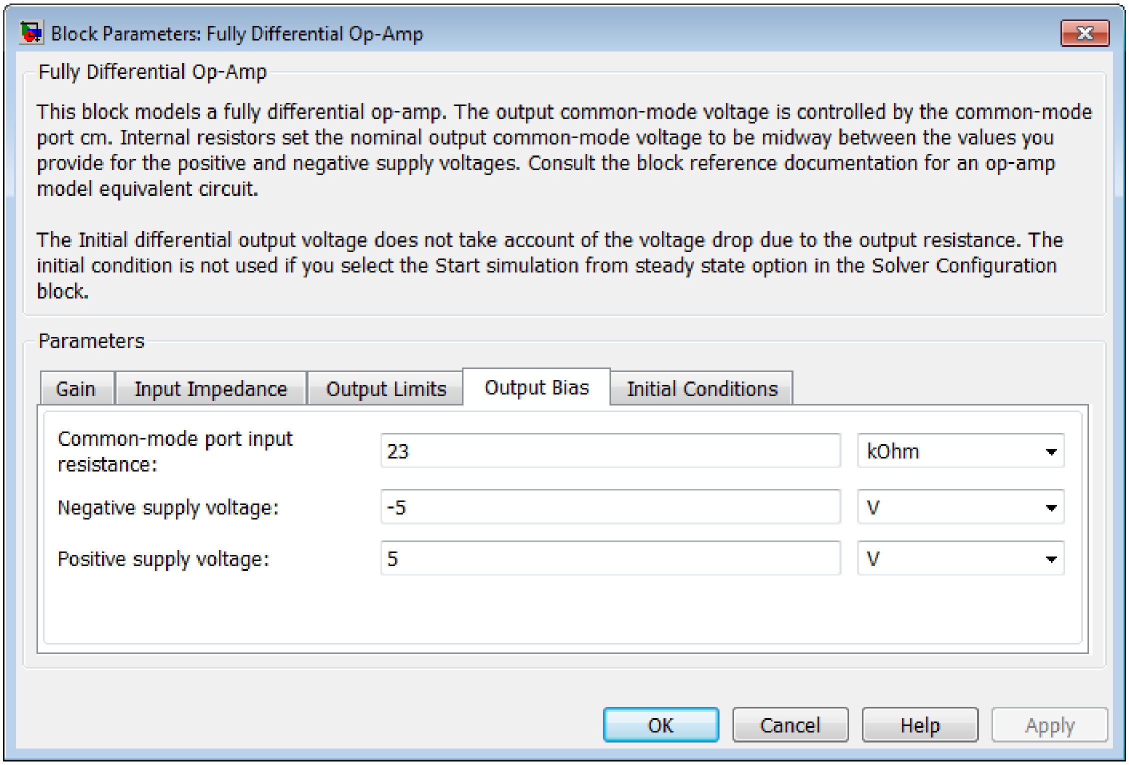
Maximum output voltage low

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/μs.

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.

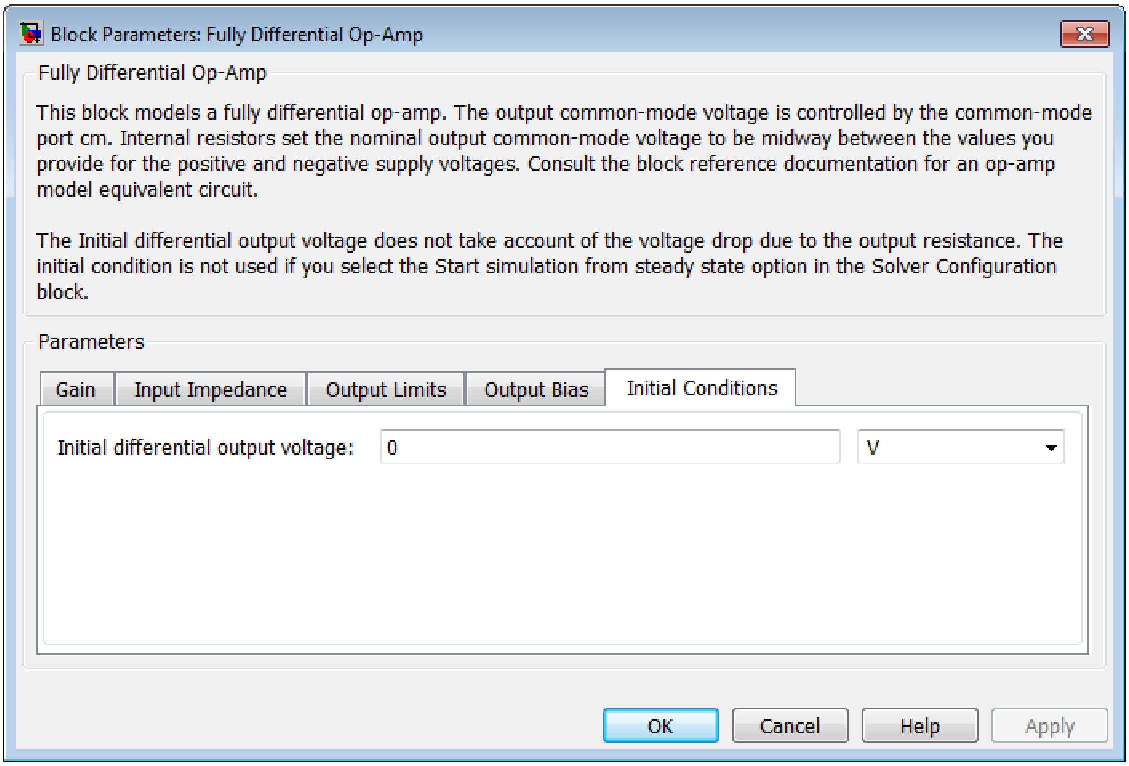
Negative supply voltage

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

Simscape Op-Amp, Band-Limited Op-Amp, Finite-Gain Op-Amp

Fuse

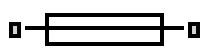
Purpose

Model fuse that protects against excessive current

Library

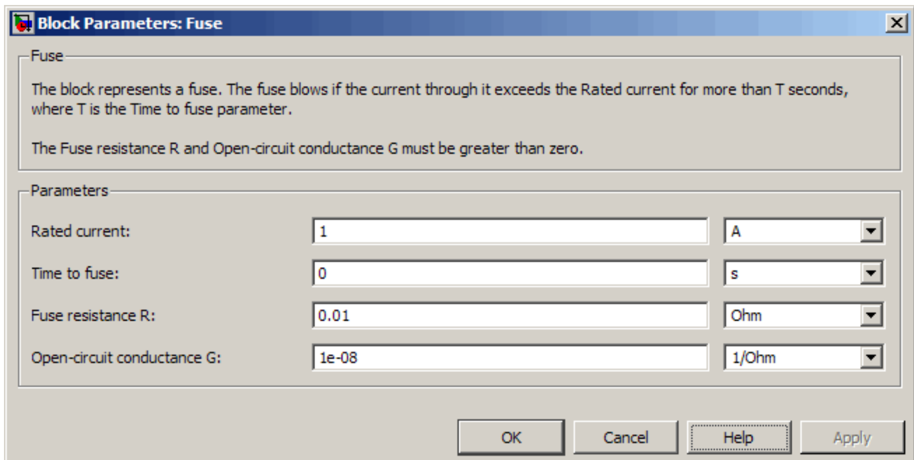
Passive Devices

Description



The Fuse block breaks the circuit in which it is connected. It does so when the current through the device exceeds the rated current at which the fuse is designed to blow and continues to exceed it for a specified amount of time.

Dialog Box and Parameters



Rated current

The current value at which the fuse blows when exceeded for a specified amount of time. The default value is `1` A.

Time to fuse

The time for which the current must exceed the rated current for the fuse to blow. The default value is `0` s.

Fuse resistance R

The fuse resistance. The parameter value must be greater than zero. The default value is `0.01` Ω.

Open-circuit conductance G
The open-circuit fuse conductance when the fuse has blown. The parameter value must be greater than zero. The default value is `1e-08` 1/Ω.

Ports

The block has the following ports:

+
Positive electrical port.

-
Negative electrical port.

Generic Battery

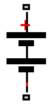
Purpose

Simple battery model

Library

Sources

Description



The Generic Battery block represents a simple 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 whose voltage as a function of charge has the following reciprocal relationship:

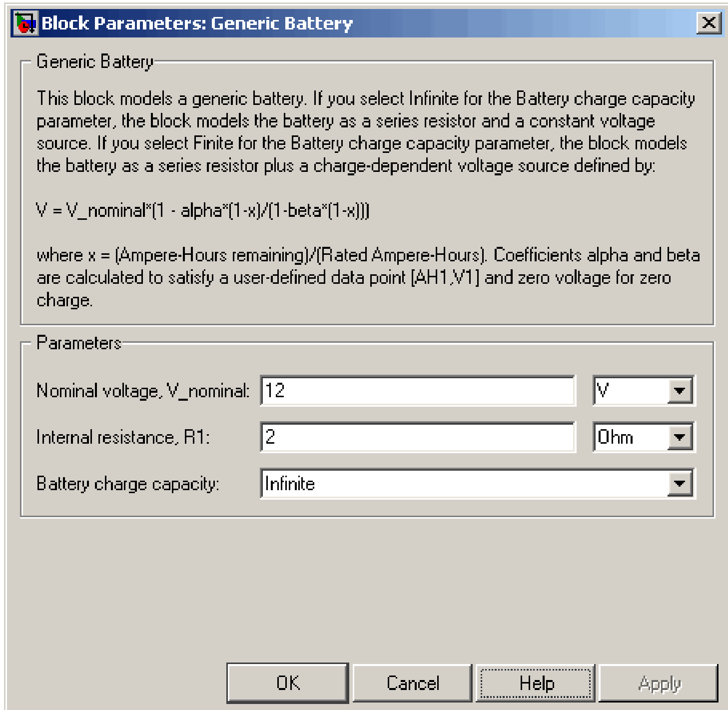
$$V = V_0 \left[1 - \left(\frac{\alpha(1-x)}{1-\beta(1-x)} \right) \right]$$

where:

- x is the ratio of the ampere-hours left to the number of ampere-hours, AH, for which the battery is rated.
- V_0 is the voltage when the battery is fully charged, as defined by the **Nominal voltage, V_nominal** parameter.
- The block calculates the constants α and β to satisfy the following battery conditions:
  - The battery voltage is zero when the charge is zero, that is, when $x = 0$.
  - The battery voltage is $V1$ (the **Voltage V1 < V_nominal when charge is AH1** parameter value) when the charge is the **Charge AH1 when no-load volts are V1** parameter value, that is, when $x = AH1/AH$.

The equation defines a reciprocal relationship between voltage and remaining charge. It is an approximation to what happens in a real battery, but it does replicate the increasing rate of voltage drop at low charge values. It also ensures that the battery voltage becomes zero when the charge level is zero. This simple model has the advantage of requiring very few parameters, and these are parameters that are readily available on most datasheets.

Dialog Box and Parameters



Nominal voltage, V_nominal

The voltage at the output port when the battery is fully charged. The default value is 12 V.

Internal resistance, R1

Internal connection resistance. The default value is 2 Ω.

Battery charge capacity

Select one of the following options for modeling the charge capacity of the battery:

- `Infinite` — The battery voltage is independent of charge drawn from the battery. This is the default option.
- `Finite` — The battery voltage decreases as charge decreases.

Ampere-Hour rating, AH
The maximum battery charge in ampere-hours. This parameter is only visible when you select `Finite` for the **Battery charge capacity** parameter. The default value is `50` hr*A.

Initial charge
The battery charge at the start of the simulation. This parameter is only visible when you select `Finite` for the **Battery charge capacity** parameter. The default value is `50` hr*A.

Voltage V1 < V_nominal when charge is AH1
The battery output voltage when the charge level is AH1 hr*A. This parameter is only visible when you select `Finite` for the **Battery charge capacity** parameter. The default value is `11.5` V.

Charge AH1 when no-load volts are V1
The battery charge level in hr*A when the no-load output voltage is V1. This parameter is only visible when you select `Finite` for the **Battery charge capacity** parameter. The default value is `25` hr*A.

Self-discharge resistance, R2
Select one of the following options for modeling the self-discharge resistance of the battery:

- `Omit` — Do not include resistance across the battery output terminals in the model.
- `Include` — Include resistance R2 across the battery output terminals in the model.

R2
The resistance across the battery output terminals that represents battery self-discharge. This parameter is only visible when you

select `Include` for the **Self-discharge resistance, R2** parameter. The default value is `2e+03` Ω.

Ports

The block has the following ports:

+
: Positive electrical voltage.

-
: Negative electrical voltage.

Examples

For an example of how you can create a detailed battery model, see the Simscape Lead-Acid Battery example.

See Also

Simscape DC Voltage Source, Simscape Controlled Voltage Source

Generic Linear Actuator

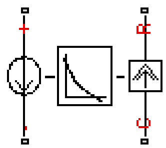
Purpose

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

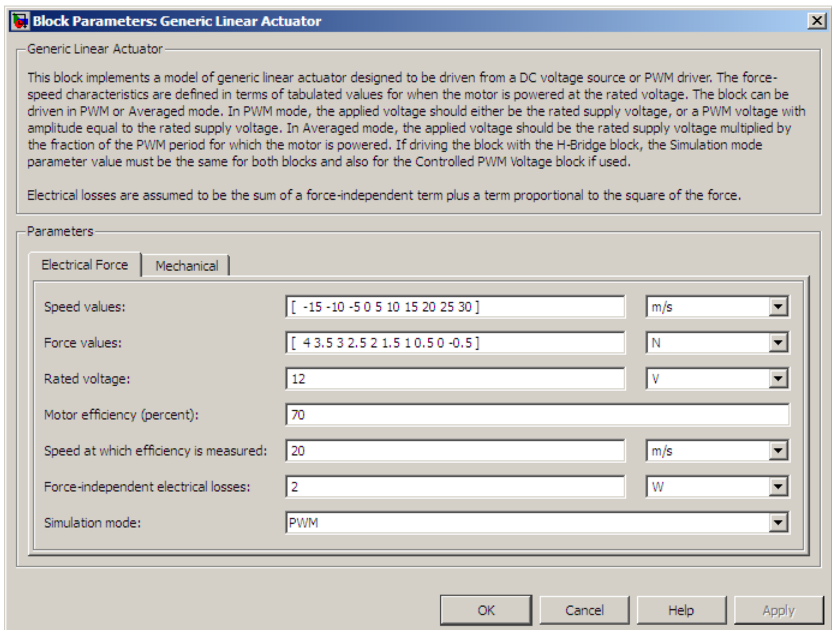
This block has the following limitations:

- 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 Rotary Actuator, and Controlled PWM block dialog boxes, ensure that the **Simulation mode** is the same for all three blocks.

Dialog Box and Parameters

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

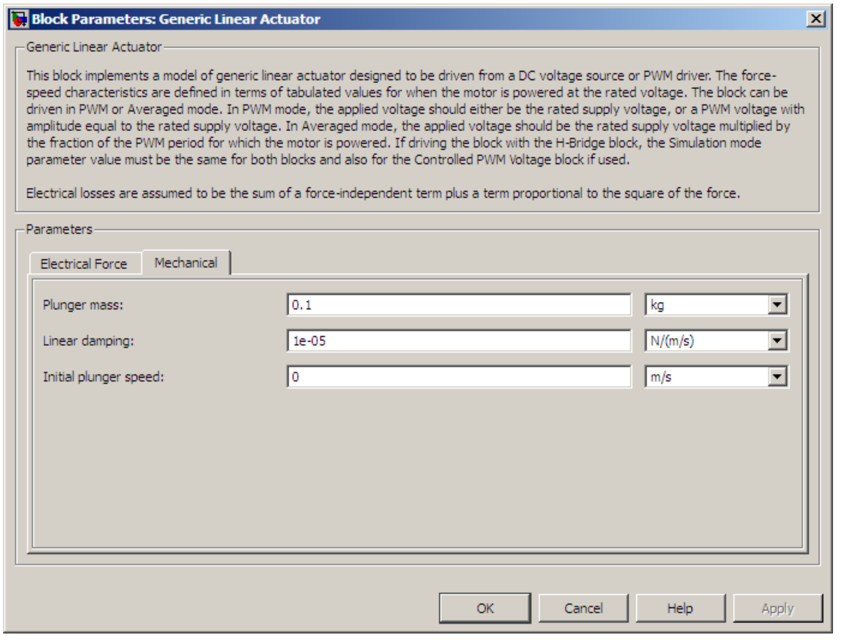
If you set the **Simulation mode** parameter to `PWM`, apply a PWM waveform switching between zero and rated volts to the block electrical terminals. The current drawn from the electrical supply is equal to the amount required to deliver the mechanical power and to compensate for electrical losses. If 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 and H-Bridge.

Generic Rotary Actuator

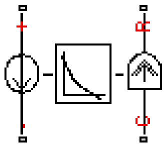
Purpose

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

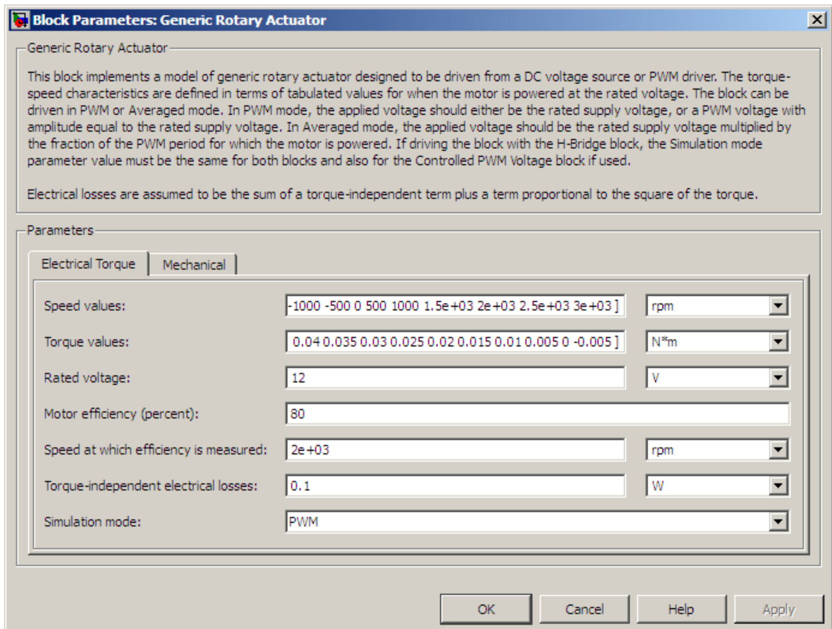
This block has the following limitations:

- 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 block dialog boxes, ensure that the **Simulation mode** is the same for all three blocks.

Dialog Box and Parameters

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

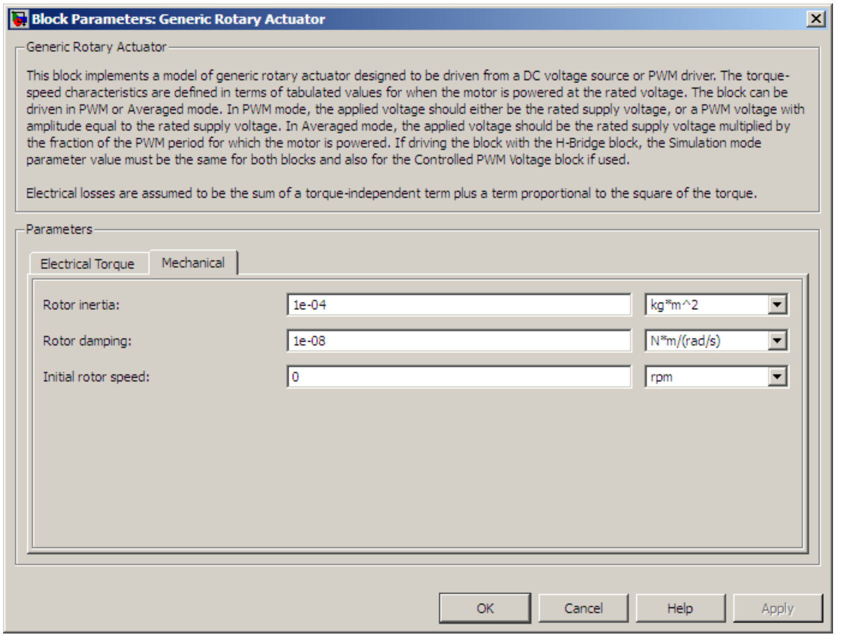
If you set the **Simulation mode** parameter to `PWM`, apply a PWM waveform switching between zero and rated volts to the block electrical terminals. The current drawn from the electrical supply is equal to the amount required to deliver the mechanical power and to compensate for electrical losses. If 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 ω. 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^2. 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 and H-Bridge.

Gyro

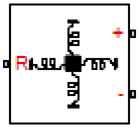
Purpose

Behavioral model of MEMS gyro

Library

Sensors

Description

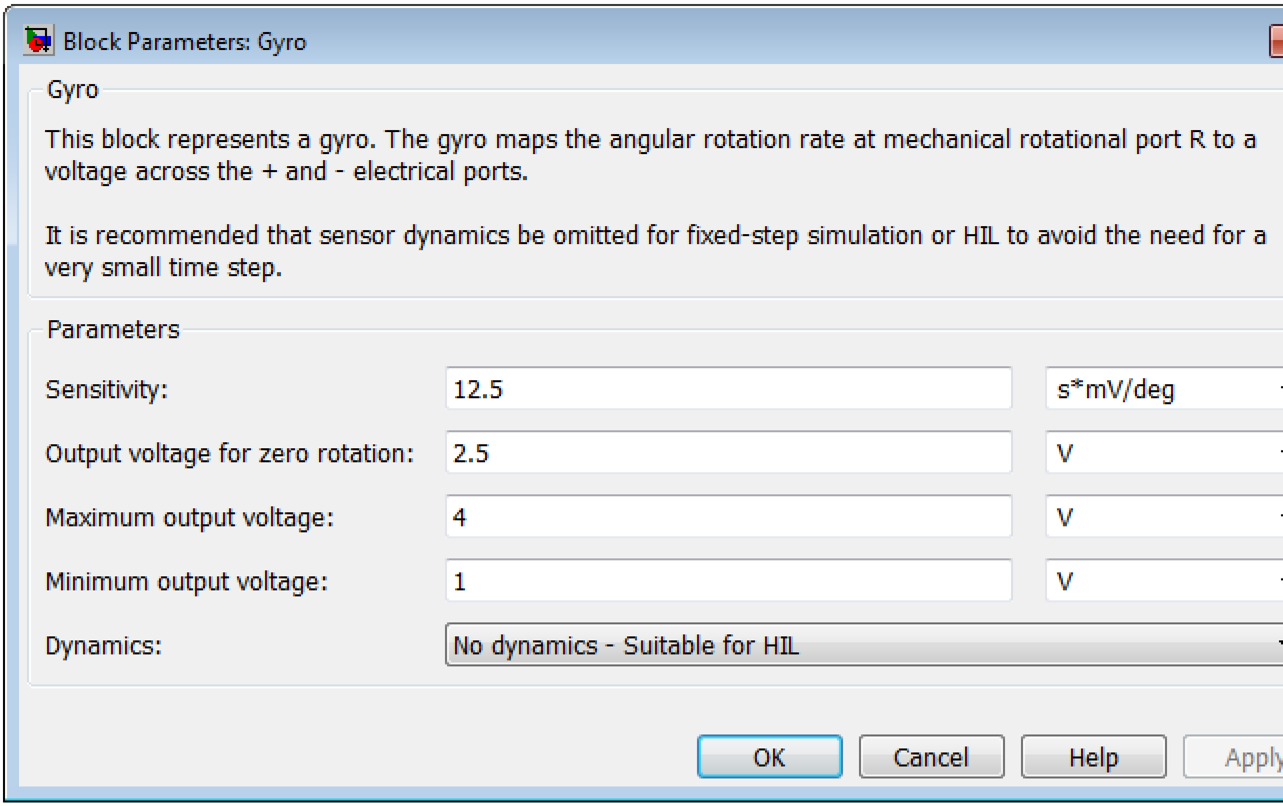


The Gyro block implements a behavioral model of a MicroElectroMechanical Systems (MEMS) gyro. The gyro provides an output voltage that is proportional to the angular rotation rate presented at the mechanical rotational physical port R. The output voltage is limited according to the values that you provide 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.

Dialog Box and 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** and the **Initial angular rate** parameter values.

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.

Initial angular rate
Determines the initial condition for the lag by specifying the initial output for the sensor, expressed in units of angular rotation rate. This parameter is only visible when you select `Model sensor bandwidth` for the **Dynamics** parameter. The default value is `0` deg/s.

Ports

The block has the following ports:

R
: Mechanical rotational port.

+
: Positive electrical port.

-
: Negative electrical port.

H-Bridge

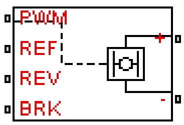
Purpose

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.

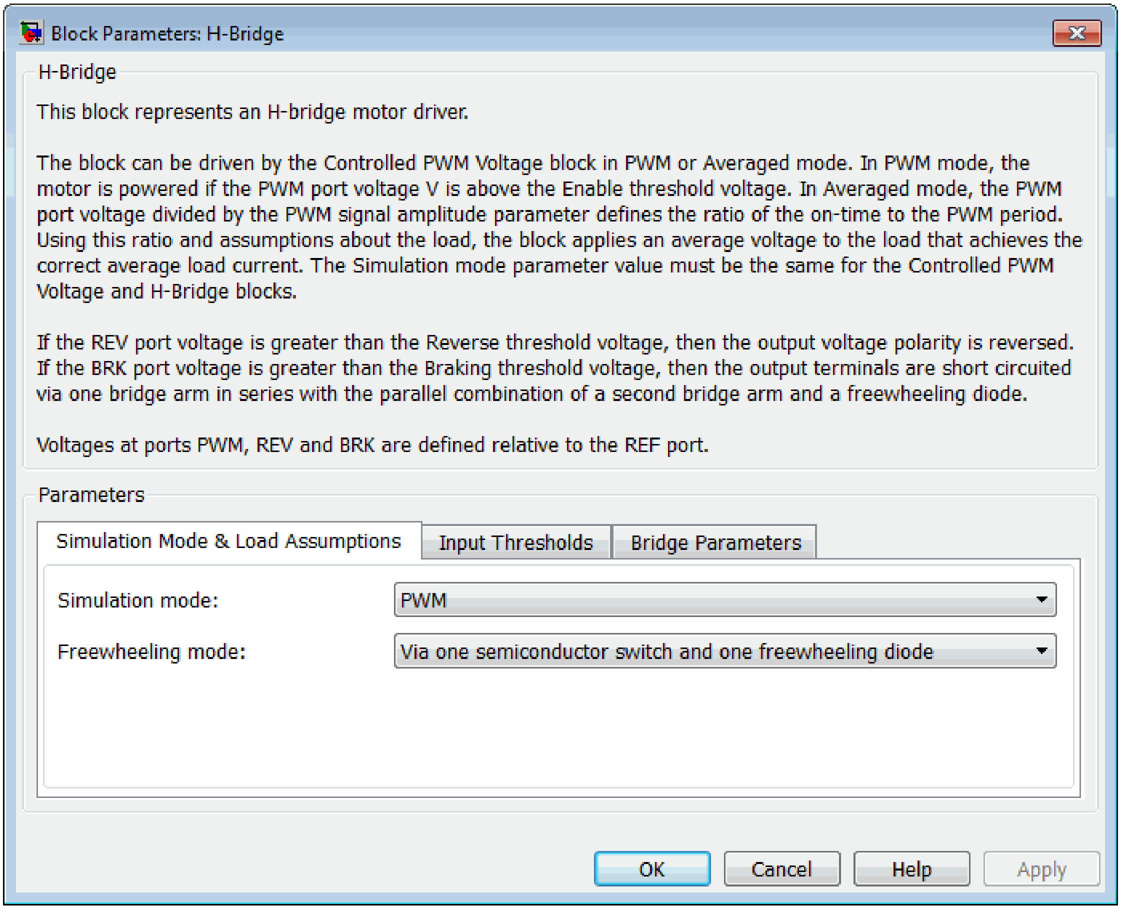
Basic Assumptions and Limitations

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.

Dialog Box and Parameters

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` Ω.

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.

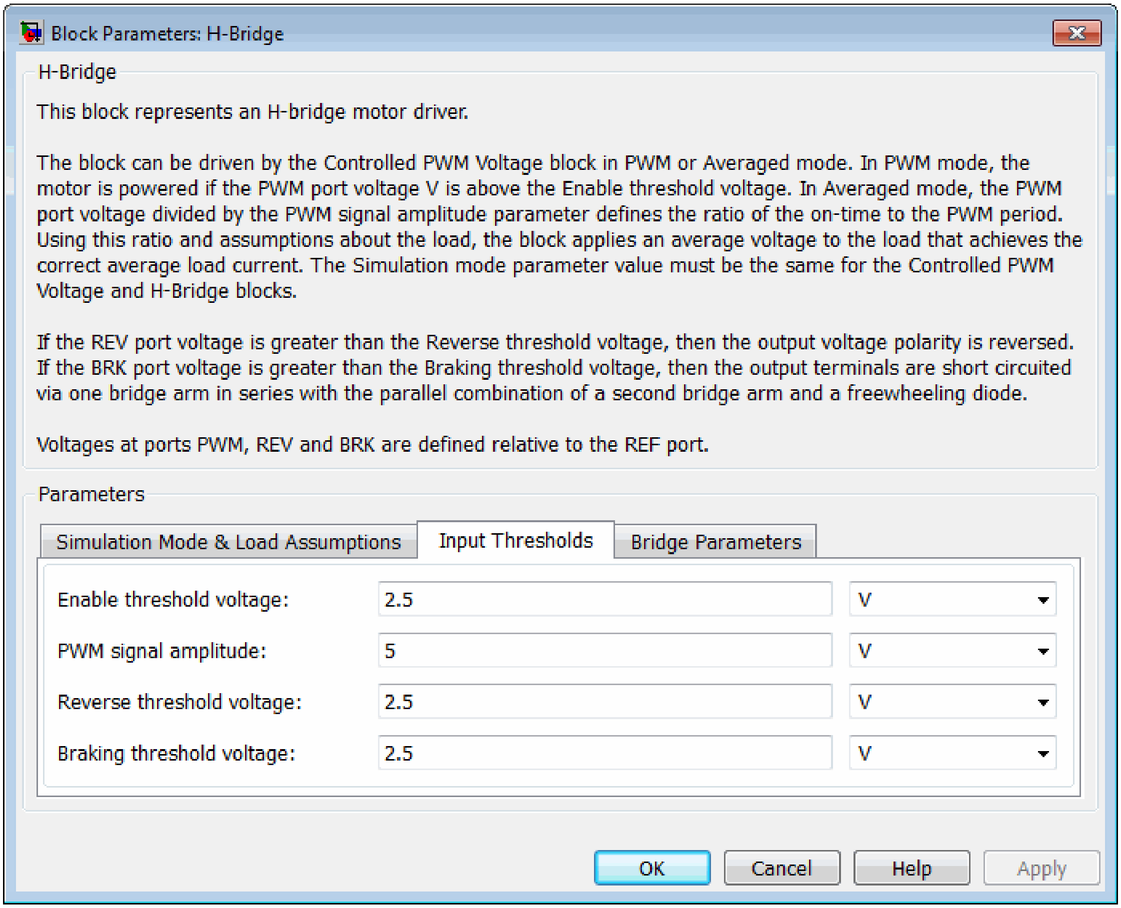
This parameter is only visible when you select `Averaged` for the **Simulation mode** parameter and `Unsmoothed or discontinuous` for the **Load current characteristics** parameter.

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.

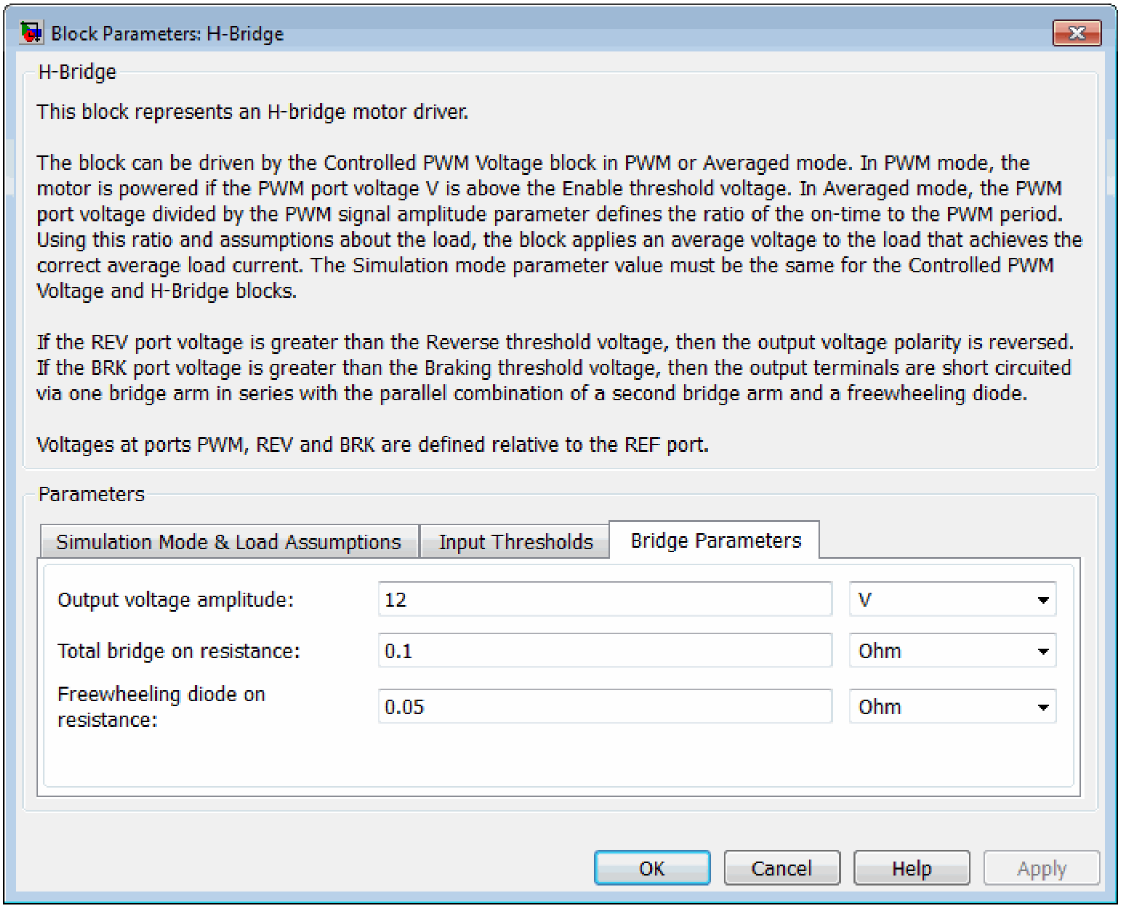
Reverse threshold voltage
When the voltage at the REV port is greater than this threshold, the output polarity becomes negative. The default value is `2.5` V.

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` Ω.

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. This parameter is only visible when you select `PWM` for the **Simulation mode** parameter on the **Simulation Mode & Load Assumptions** tab. The default value is `0.1` Ω.

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.

Examples

See the Controlled DC Motor, Linear Electrical Actuator (System-Level Model) and Linear Electrical Actuator (Implementation Model) examples.

Incandescent Lamp

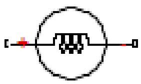
Purpose

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

$$R = R_0 \left(1 + \alpha T\right)$$

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.
- α 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:

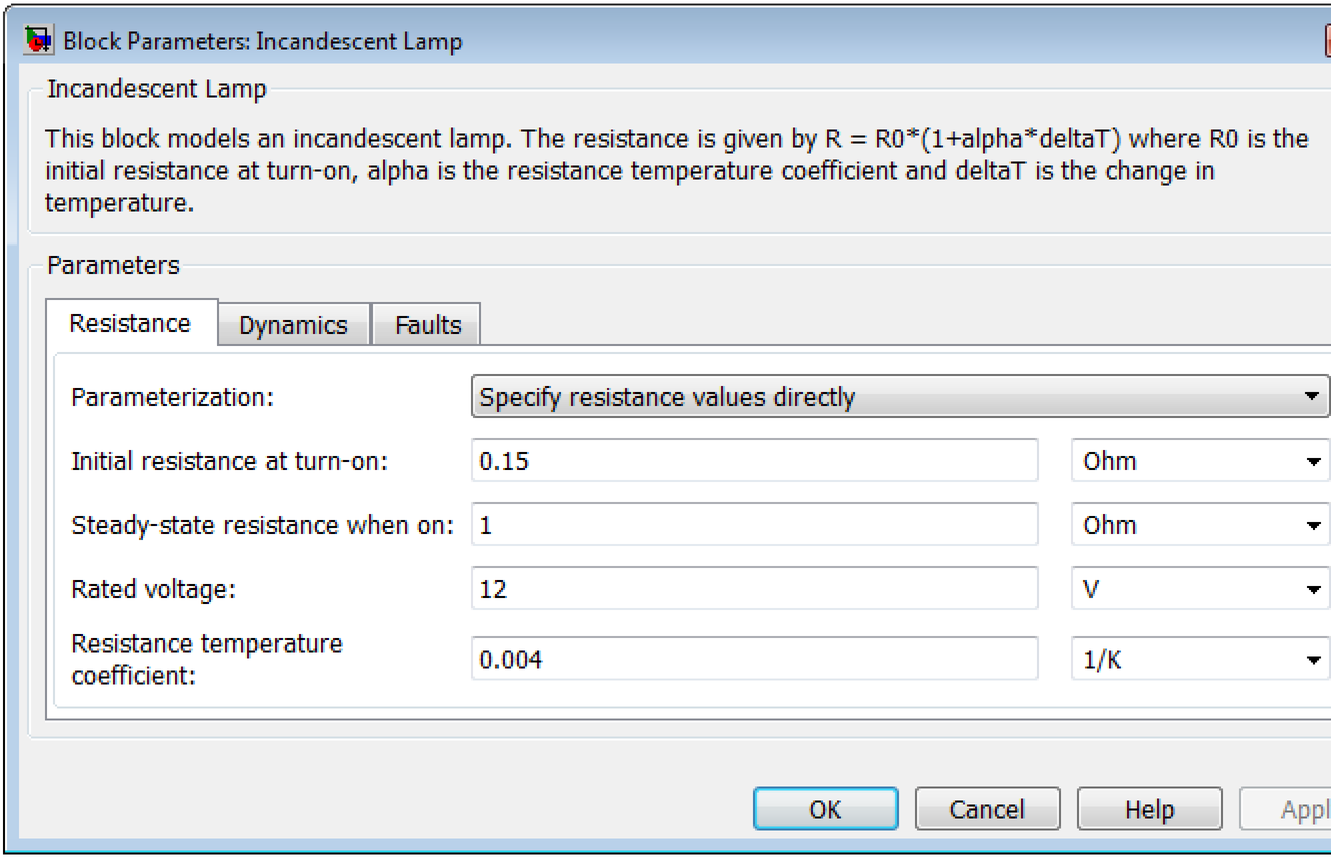
- If you select `Specify resistance values directly`, the block uses values that you provide for filament resistance when on and at turn-on to determine the value for the heat transfer coefficient.
- If you select `Specify currents`, the block uses values that you provide for filament current when on 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.

Dialog Box and Parameters

- "Resistance Tab" on page 1-222
- "Dynamics Tab" on page 1-224
- "Faults Tab" on page 1-225

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` Ω.

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` Ω.

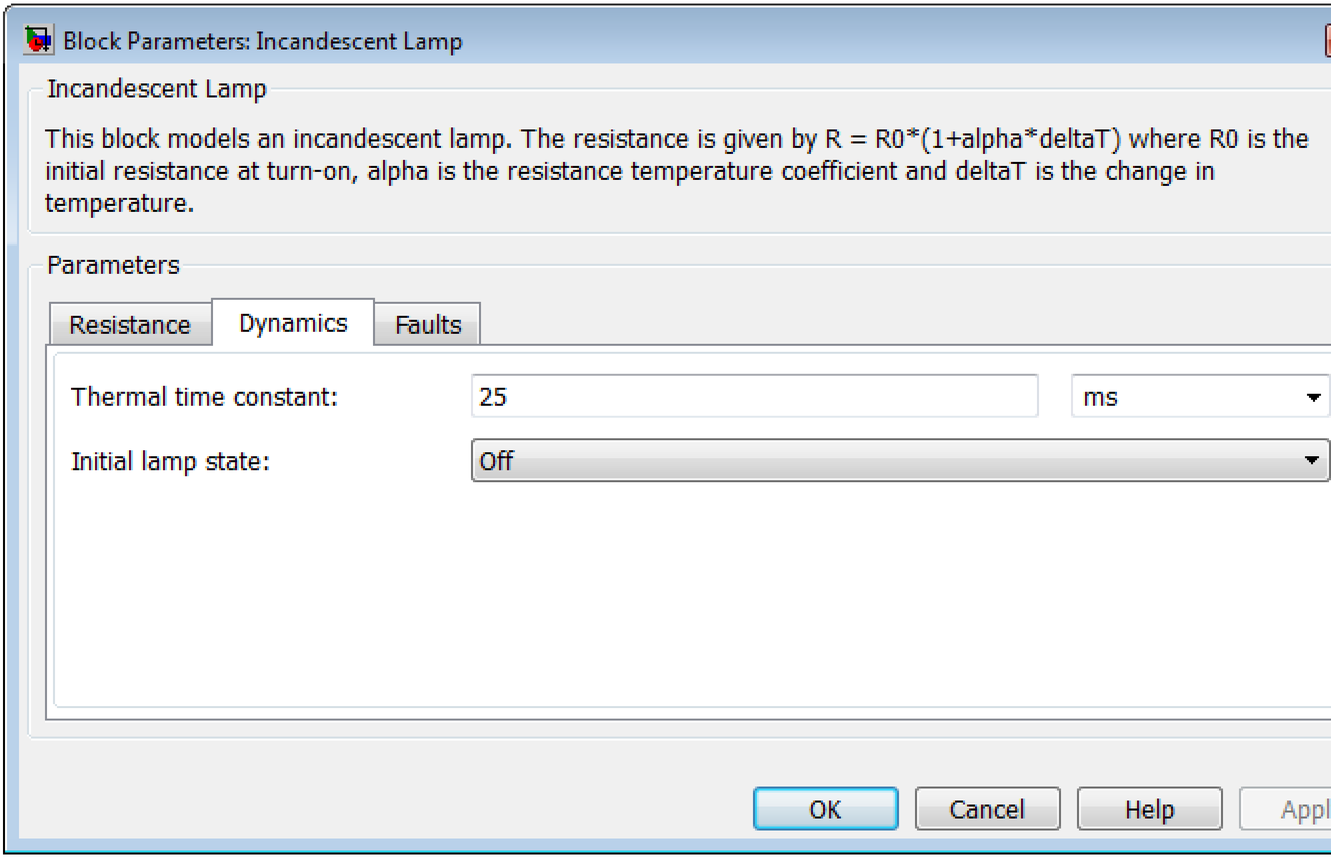
Inrush current at turn-on
The current through the lamp when it is initially turned on. This parameter is only visible when you select `Specify currents` for the **Parameterization** parameter. The default value is `70` A.

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` 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` V.

Resistance temperature coefficient
The fractional increase in resistance per unit increase in temperature. The default value is `0.004` 1/K.

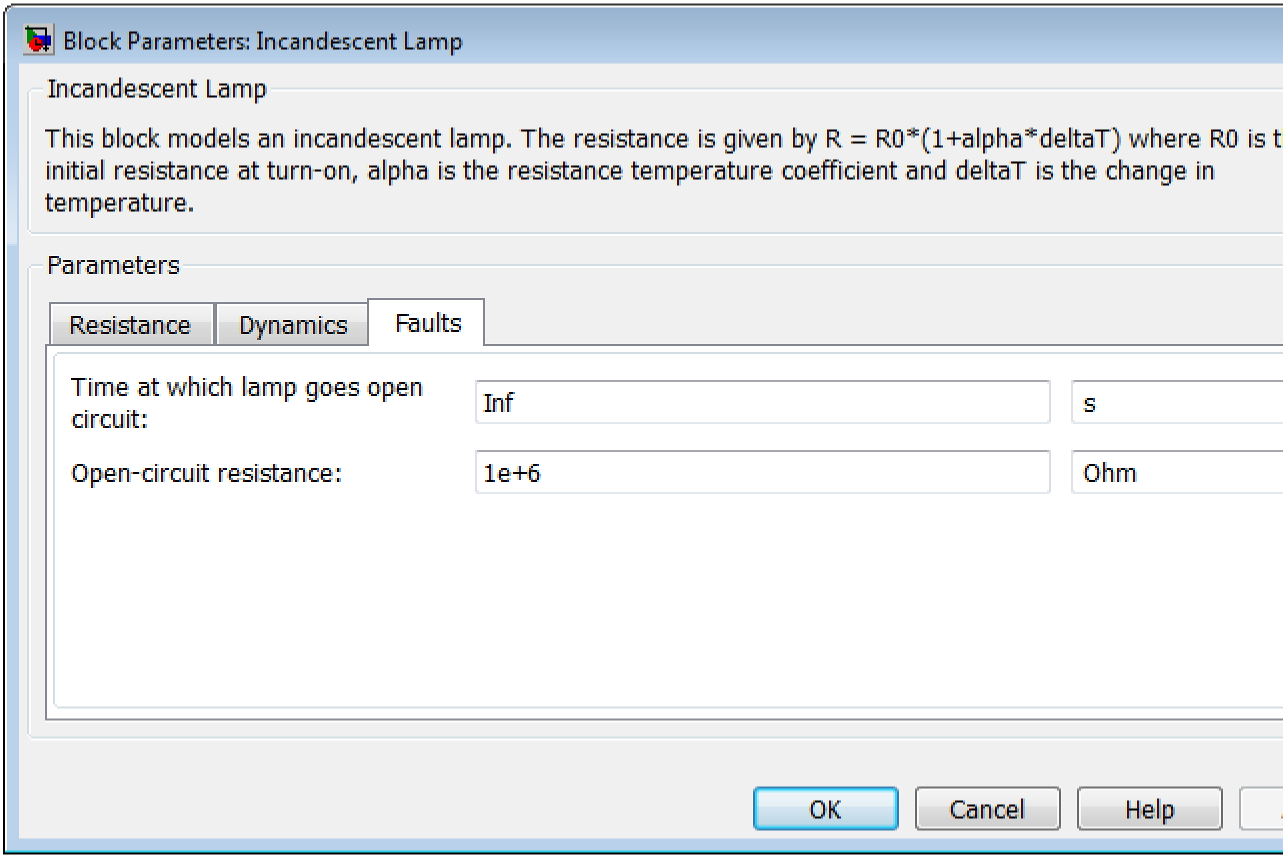
Dynamics Tab



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` 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` Ω.

Ports

The block has the following ports:

+

Positive electrical port.

-

Negative electrical port.

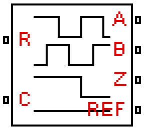
Purpose

Model device that converts information about angular shaft position into electrical pulses

Library

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. The voltages at output ports A, B, and Z are defined relative to the REF reference port voltage.

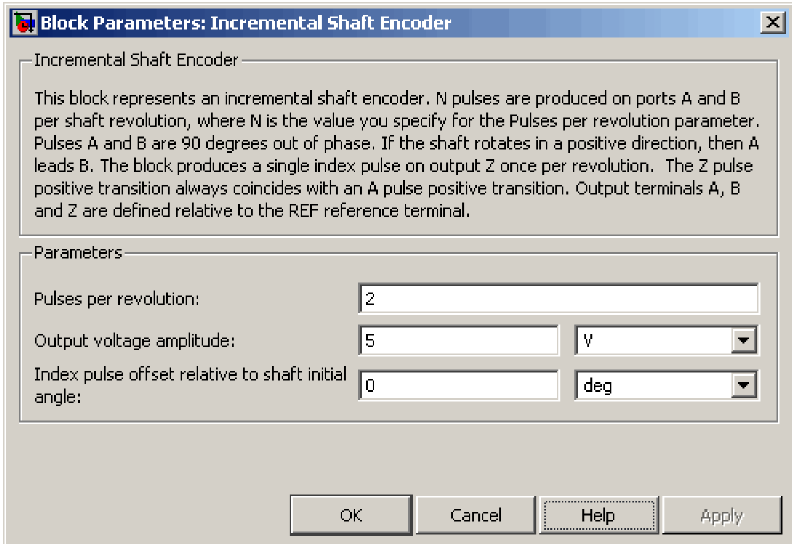
Use this block if you need to model the shaft encoder signals, either to support development of a decoding algorithm, or to include the quantization effects. Otherwise, use the Simscape Ideal Rotational Motion Sensor block.

Basic Assumptions and Limitations

The Incremental Shaft Encoder block has the following limitations:

- The Incremental Shaft Encoder block is not linearizable. Use the Simscape Ideal Rotational Motion Sensor block for control design studies where you need to linearize your model.

Dialog Box and Parameters



Pulses per revolution

The number of pulses produced on each of the A and B phases per revolution of the shaft. The default value is 2.

Output voltage amplitude

The amplitude of the shaft encoder output voltage when the output is high. The default value is 5 V.

Index pulse offset relative to shaft initial angle

The 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. The default value is 0°.

Ports

The block has the following ports:

R

Mechanical rotational conserving port associated with the sensor positive probe.

C
: Mechanical rotational conserving port associated with the sensor negative (reference) probe.

A
: Encoded electrical output.

B
: Encoded electrical output.

Z
: Index, or synchronization, electrical output.

REF
: Floating zero volt reference.

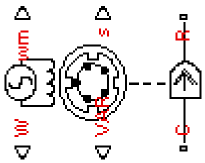
See Also

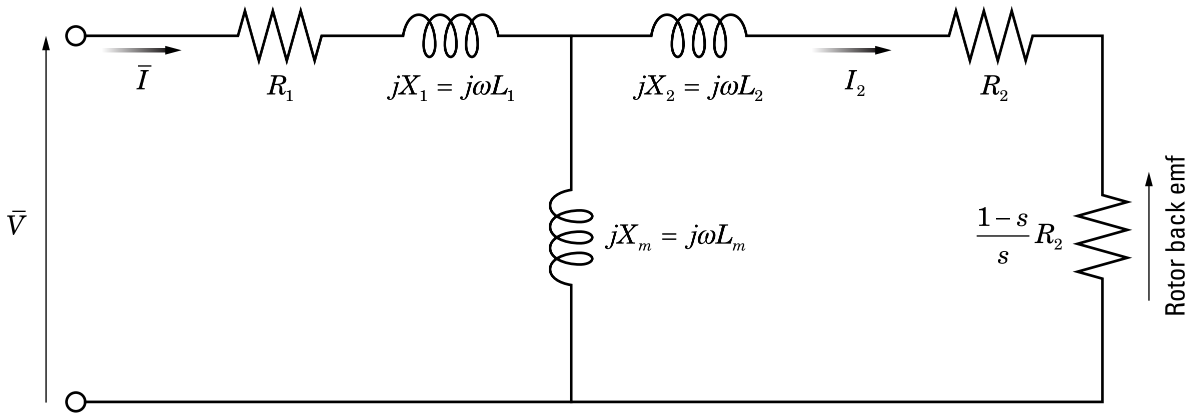
Simscape Ideal Rotational Motion Sensor

Purpose 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 ω_m, the number of pole pairs p, and the electrical supply frequency ω 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:

$$T = \frac{npR_2}{s\omega} \frac{V_{rms}^{\ 2}}{\left(R_1 + R_2 + \frac{1-s}{s}R_2\right)^2 + \left(X_1 + X_2\right)^2}$$

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

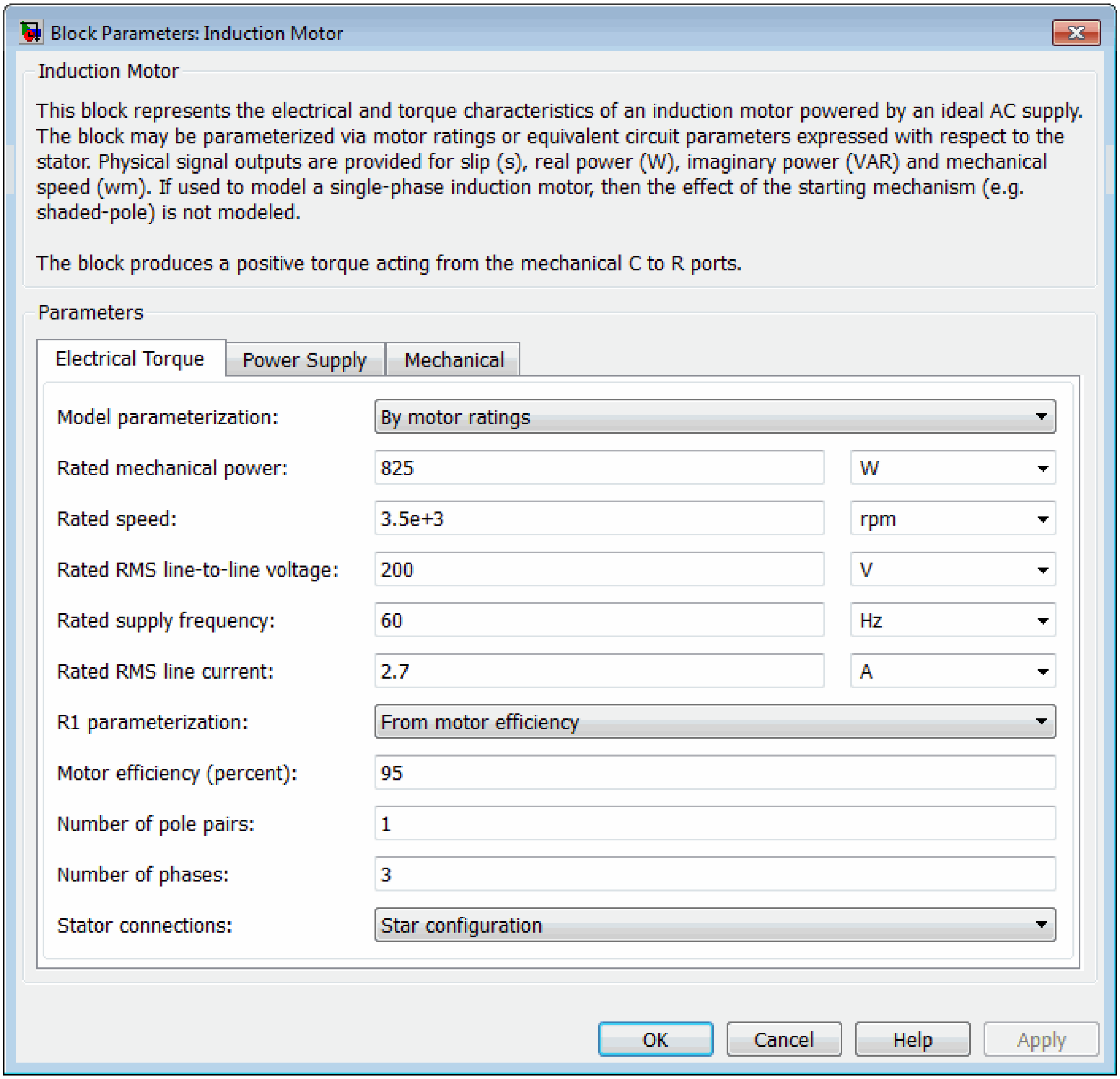
The model is based on the following assumptions:

- 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.

Dialog Box and Parameters

- "Electrical Torque Tab" on page 1-233
- "Power Supply Tab" on page 1-238
- "Mechanical Tab" on page 1-240
- "Temperature Dependence Tab" on page 1-241
- "Thermal Port Tab" on page 1-243

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` Ω. 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` Ω. 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` 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` 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` H.

Rated mechanical power
Mechanical power the motor delivers when running at the rated speed. The default value is `825` 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` 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` Ω. 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`.

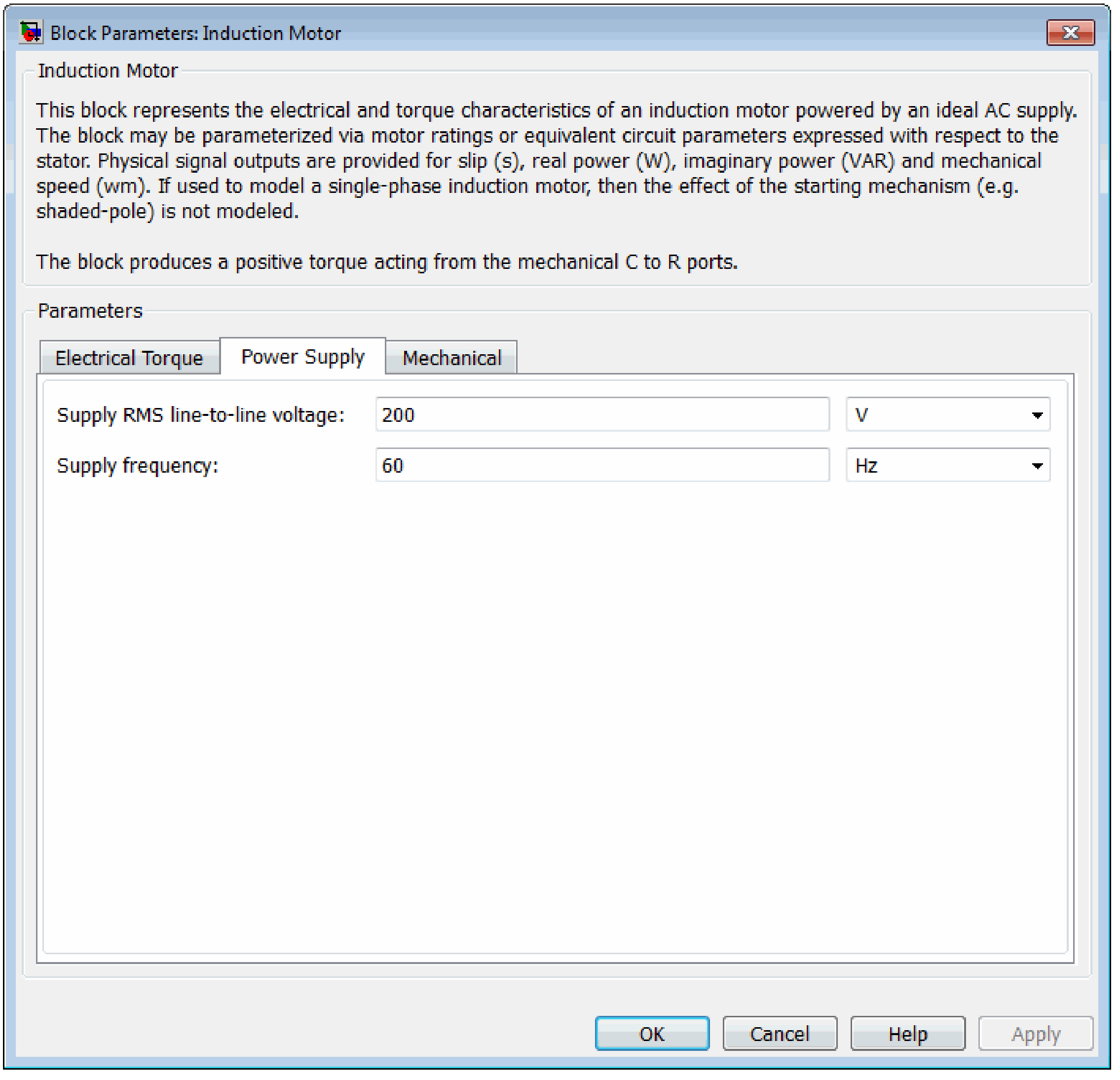
Number of phases
: Number of supply phases. The default value is `3`.

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



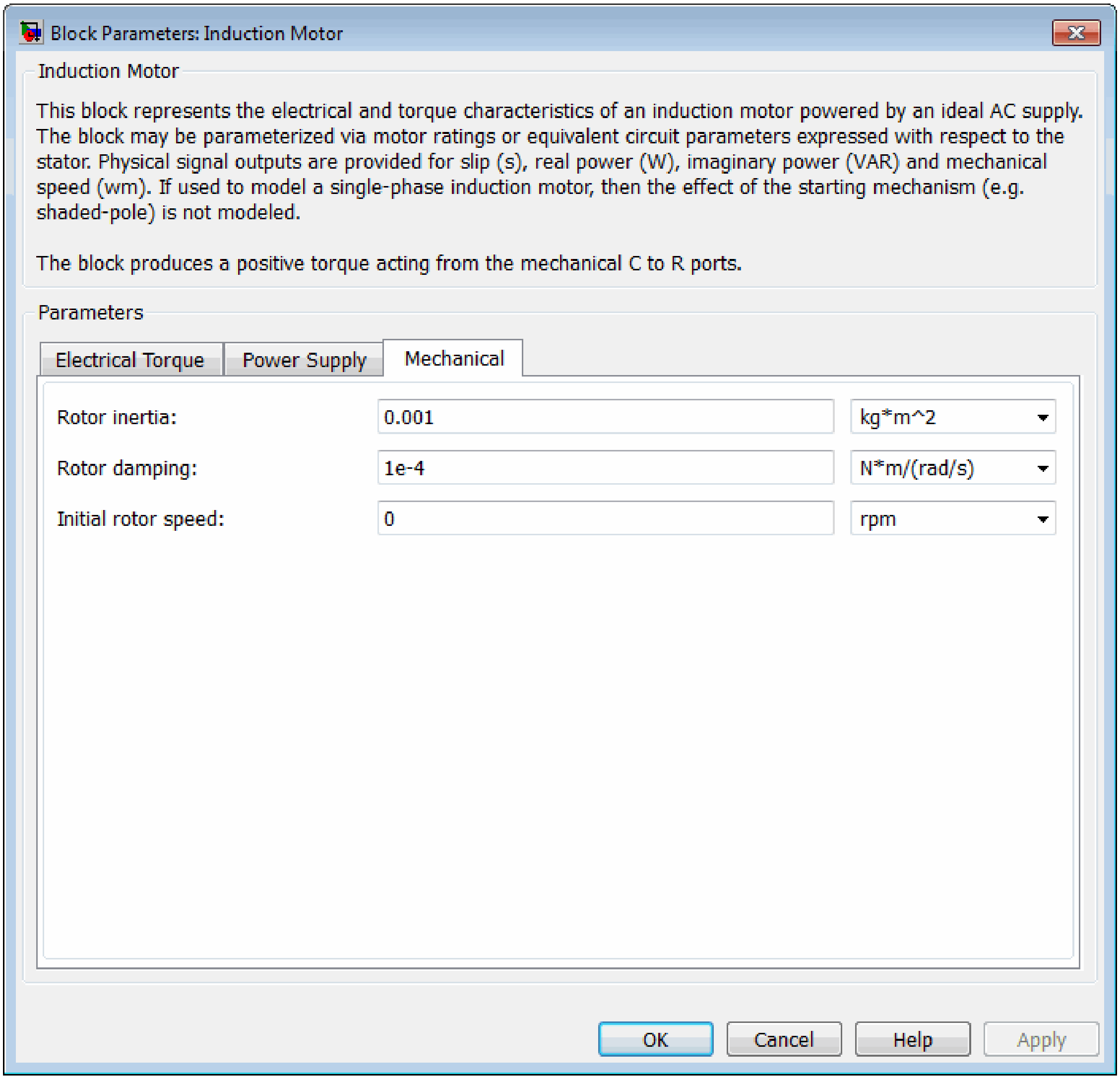
Supply RMS line-to-line voltage

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



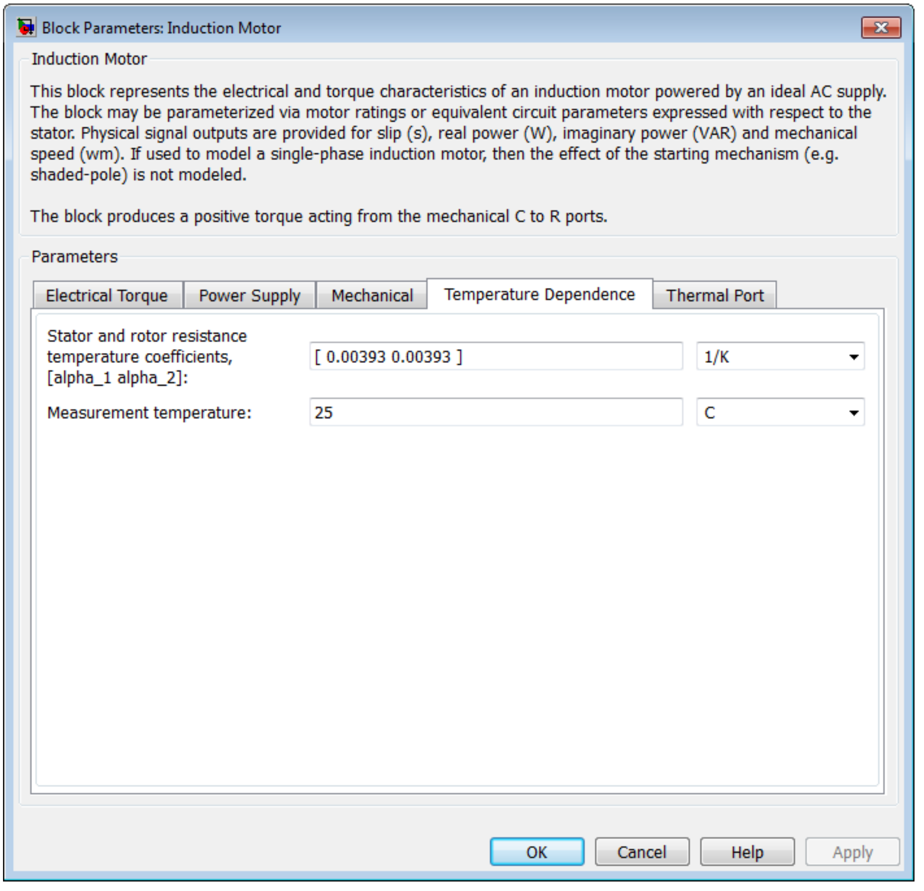
Rotor inertia
Rotor inertia. The default value is `0.1` kg*m^2. The value can be zero.

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 1-897.



Resistance temperature coefficients, [alpha_1 alpha_2]

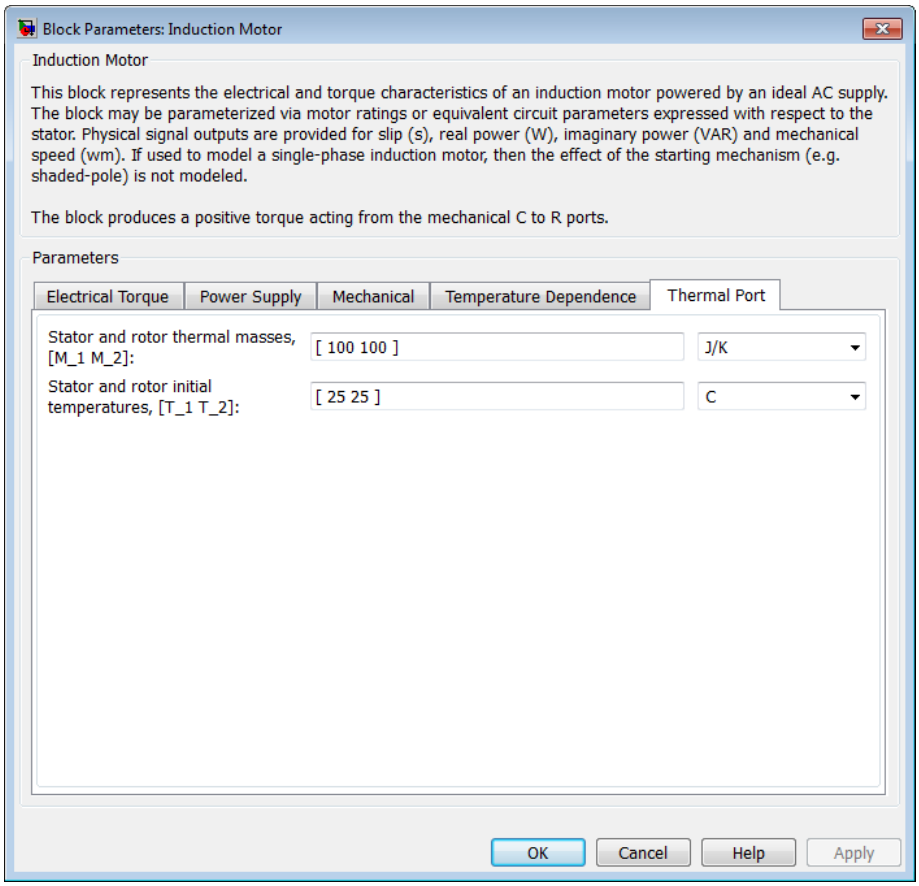
A 1 by 2 row vector defining the coefficient α in the equation relating resistance to temperature, as described in "Thermal Model for Actuator Blocks". The first element corresponds to the stator, and the second to rotor. The default value is for copper, and is `[ 0.00393 0.00393 ]` 1/K.

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 1-897.



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 1-897.

`H2`
: Rotor thermal port. For more information, see “Thermal Ports” on page 1-897.

References

[1] S.E. Lyshevski. *Electromechanical Systems, Electric Machines, and Applied Mechatronics*, CRC, 1999.

See Also

DC Motor, Servomotor, Shunt Motor, and Universal Motor.

Light-Emitting Diode

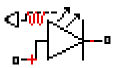
Purpose

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:

- $\mathrm{N} = ((V_1 - V_2)/V_t)/(\log(I_1) - \log(I_2))$
- $\mathrm{IS} = \left(I_1/(\exp(V_1/(\mathrm{N}V_t)) - 1) + I_2/(\exp(V_2/(\mathrm{N}V_t)) - 1)\right)/2$

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 *VJ* 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".

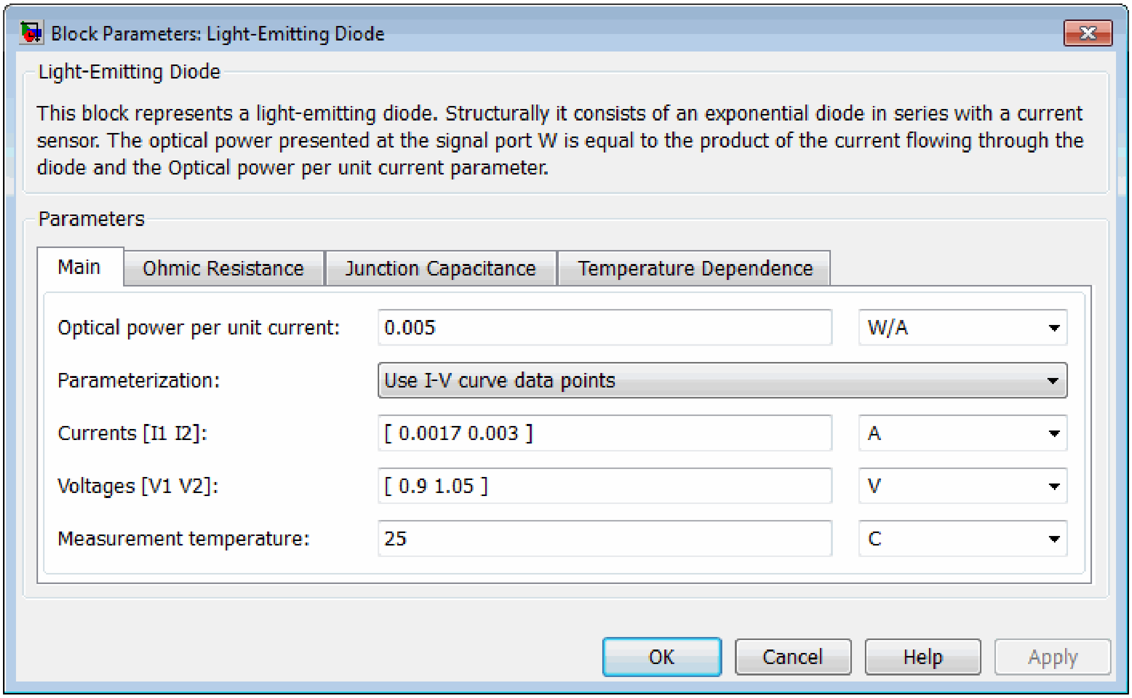
Basic Assumptions and Limitations

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 *IS* 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.

Dialog Box and Parameters

Main Tab



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 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 **Parameterization** parameter. The default value is `[ 0.0017 0.003 ]` A.

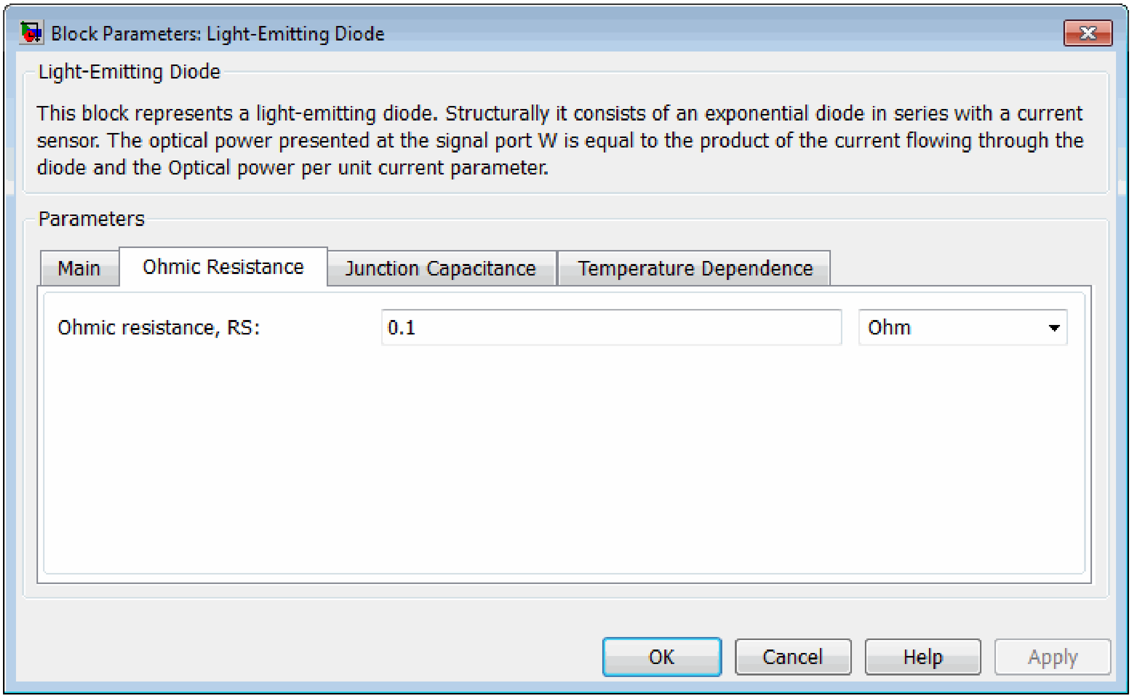
Voltages [V1 V2]
A vector of the voltage values at the two points on the diode I-V curve that the block uses to calculate *IS* and *N*. This parameter is only visible when you select `Use I-V curve data points` for the **Parameterization** parameter. The default value is `[ 0.9 1.05 ]` V.

Saturation current 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 `Use parameters IS and N` for the **Parameterization** parameter. The default value is `5e-5` A.

Measurement temperature
The temperature at which IS 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 IS and N` for the **Parameterization** parameter. The default value is `10`.

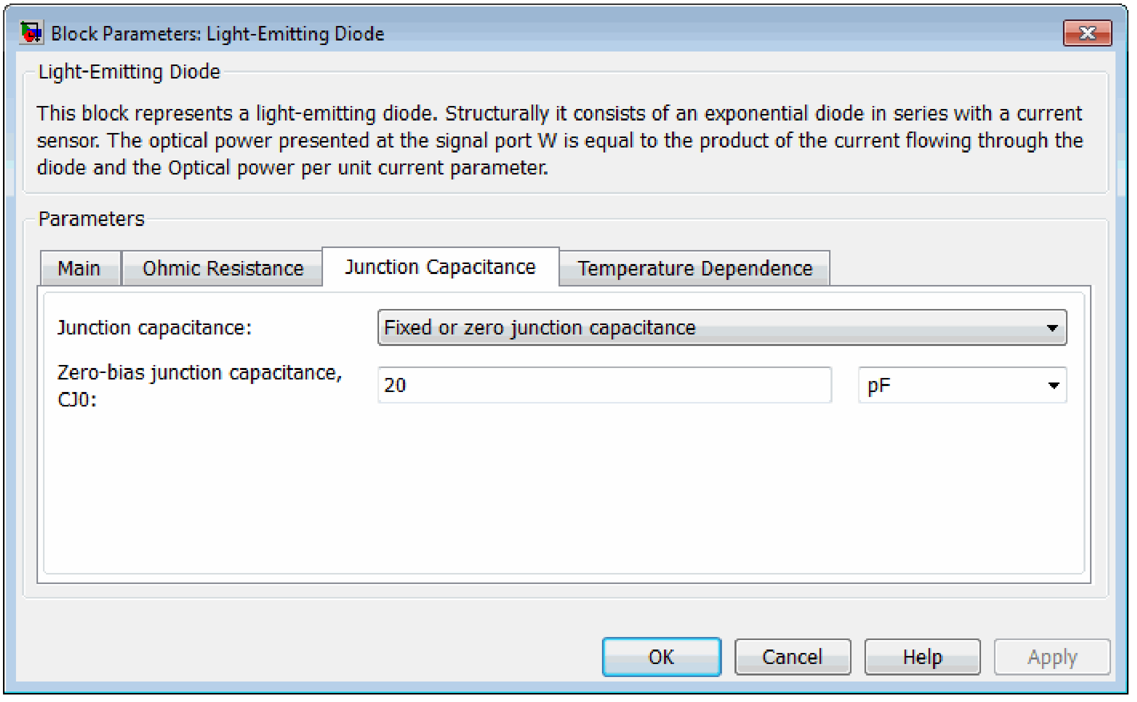
Ohmic Resistance Tab



Ohmic resistance RS

The series diode connection resistance. The default value is `0.1` Ω.

Junction Capacitance Tab



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` 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 ]` V.

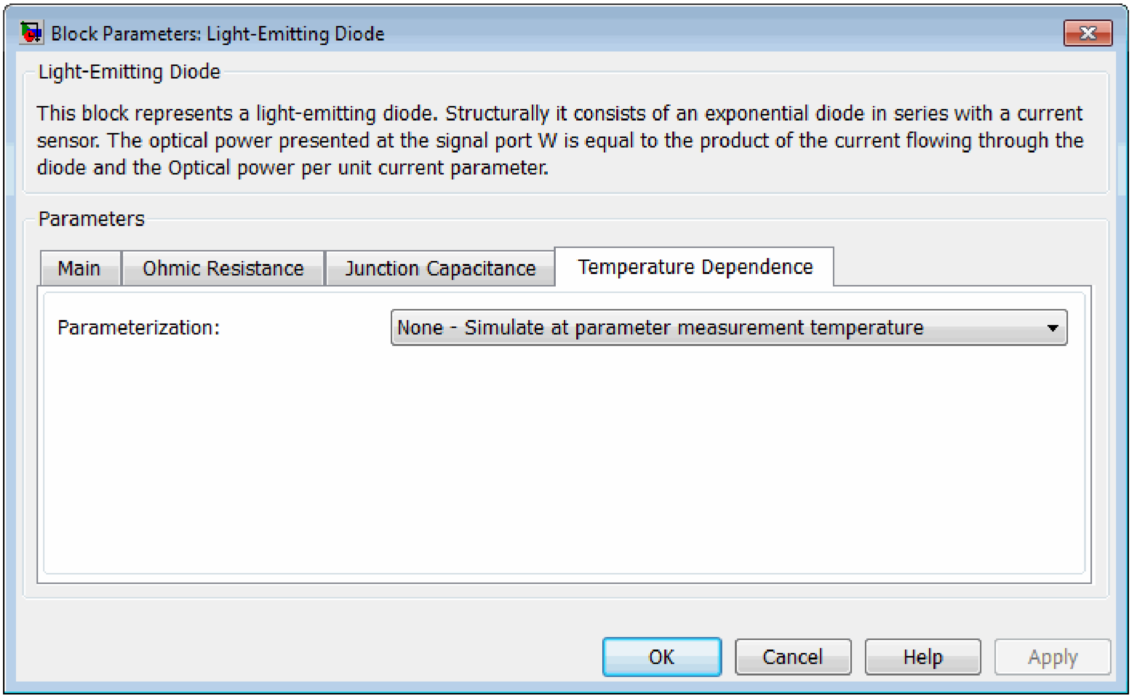
Corresponding capacitances [C1 C2 C3]
A vector of the capacitance values at the three points on the diode C-V curve that the block uses to calculate *CJ0*, *VJ*, and *M*. This parameter is only visible when you select `Use C-V curve data points` for the **Junction capacitance** parameter. The default 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 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.
- `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=1.74eV)
- 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:

`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

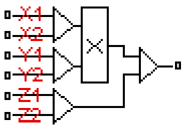
Purpose

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.

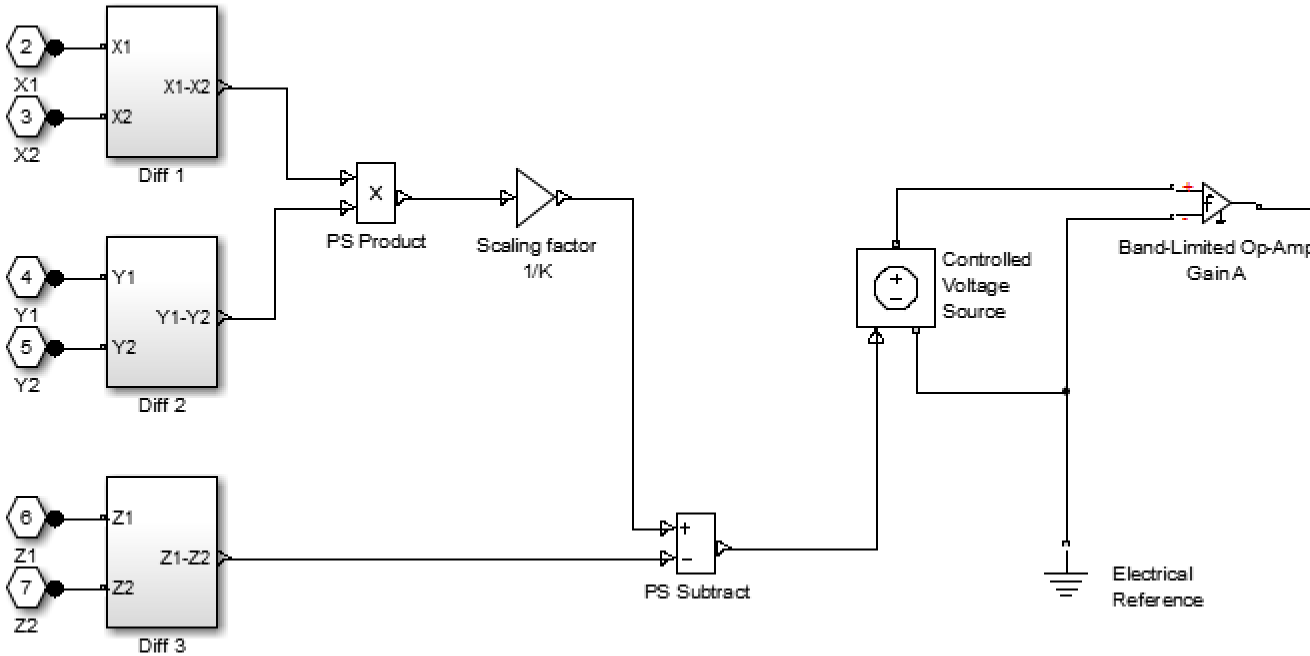
In a typical multiplication circuit, the output is fed back into input Z1, which results in the following gain (assuming that A is large):

$$V_{out} = \left(\frac{(X_1 - X_2)(Y_1 - Y_2)}{K} + Z_2\right)$$

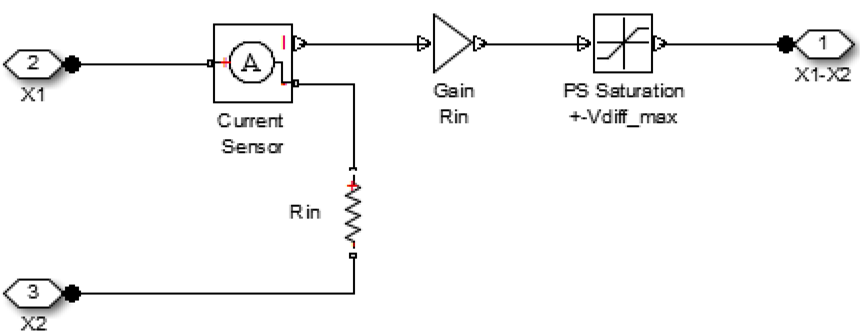
The value of the scale factor K is usually altered by an external resistor bias network. The Multiplier block implements K as an internal gain, and the external bias network is not necessary for system simulation. A typical value for K is 10, with a typical adjustment down to 3.

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.



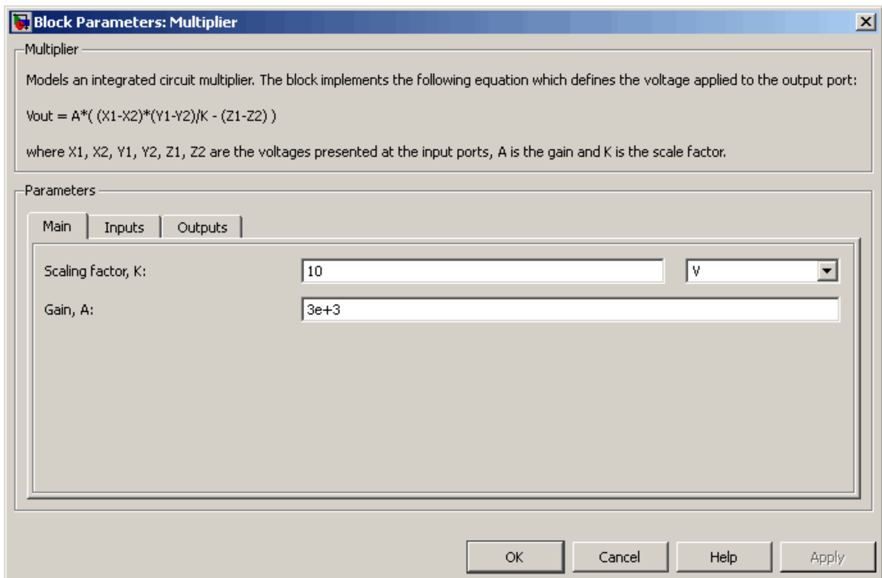
Basic Assumptions and Limitations

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.

Dialog Box and Parameters

Main Tab



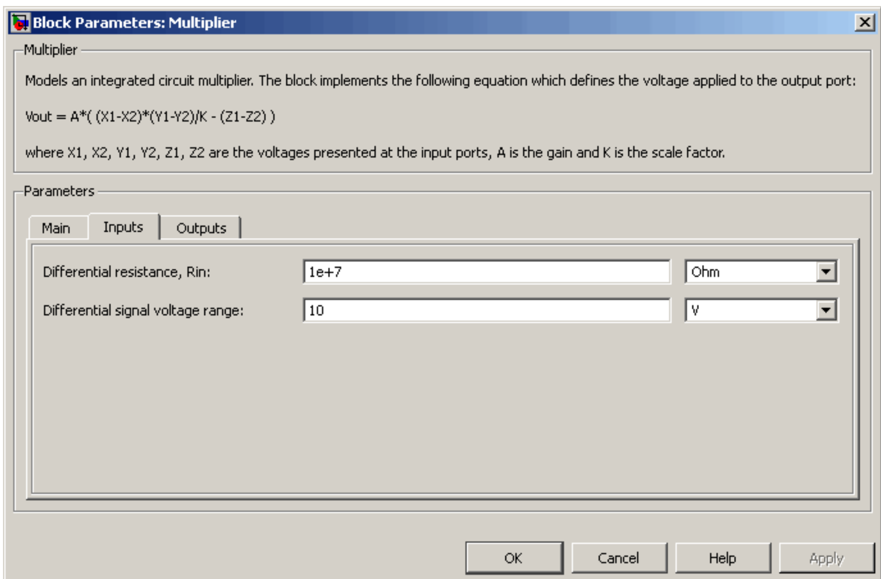
Scaling factor, K

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



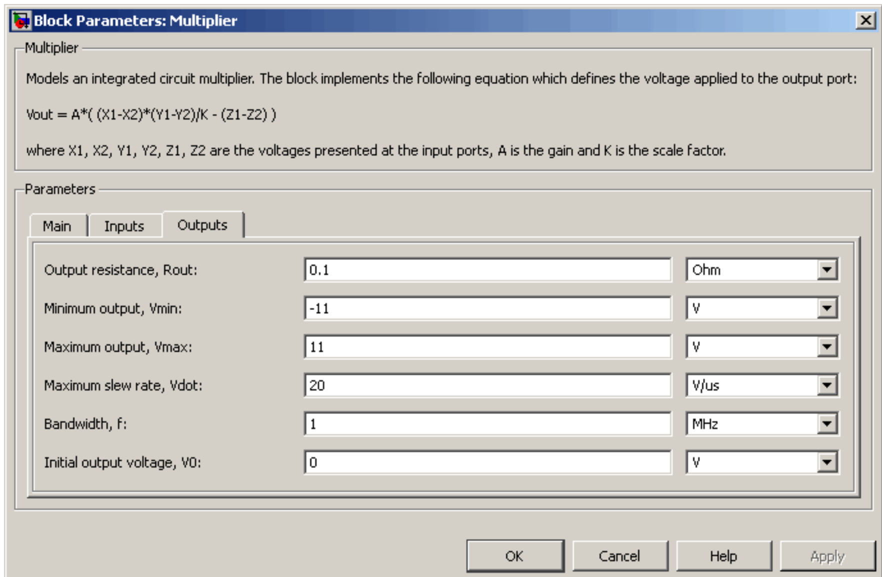
Differential resistance, Rin

Each of the differential inputs is approximated as a linear resistor with value *Rin*. Set this value to the datasheet value for differential resistance. The default value is `1e7` Ω.

Differential signal voltage range

This value, *Vdiff_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, Rout

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 Ω.

Minimum output, Vmin

The lower limit of the output voltage. The default value is -11 V.

Maximum output, Vmax

The upper limit of the output voltage. The default value is 11 V.

Maximum slew rate, Vdot

The maximum positive or negative rate of change of output voltage magnitude. The default value is 20 V/μs.

Bandwidth, f

The bandwidth of the Band-Limited Op-Amp block. The default value is 1 MHz.

Initial output voltage, V0

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.

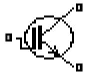
Purpose

Model N-Channel IGBT

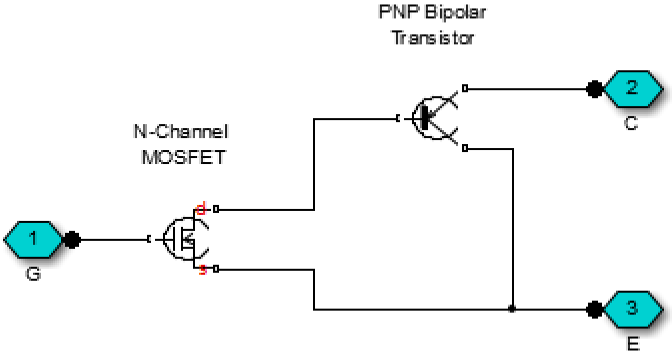
Library

Semiconductor Devices

Description



The N-Channel IGBT block models a PNP Bipolar transistor driven by an N-Channel MOSFET, 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:

$$I_b = 0 = -I_s \left[\frac{1}{\beta_F} (e^{-qV_{be}/(NkT)} - 1) - \frac{1}{\beta_R} \right]$$

$$I_c = -I_s \left[e^{-qV_{be}/(NkT)} \left(1 + \frac{V_{bc}}{V_{AF}} \right) + \frac{1}{\beta_R} \right]$$

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 β_R and β_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 β_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

$$K = \frac{I_{b(sat)}}{\left[(V_{GE(sat)}-V_{th})\left(V_{be(sat)}+V_{ce(sat)}\right)-\frac{\left(V_{be(sat)}+V_{ce(sat)}\right)^2}{2}\right]}$$

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.

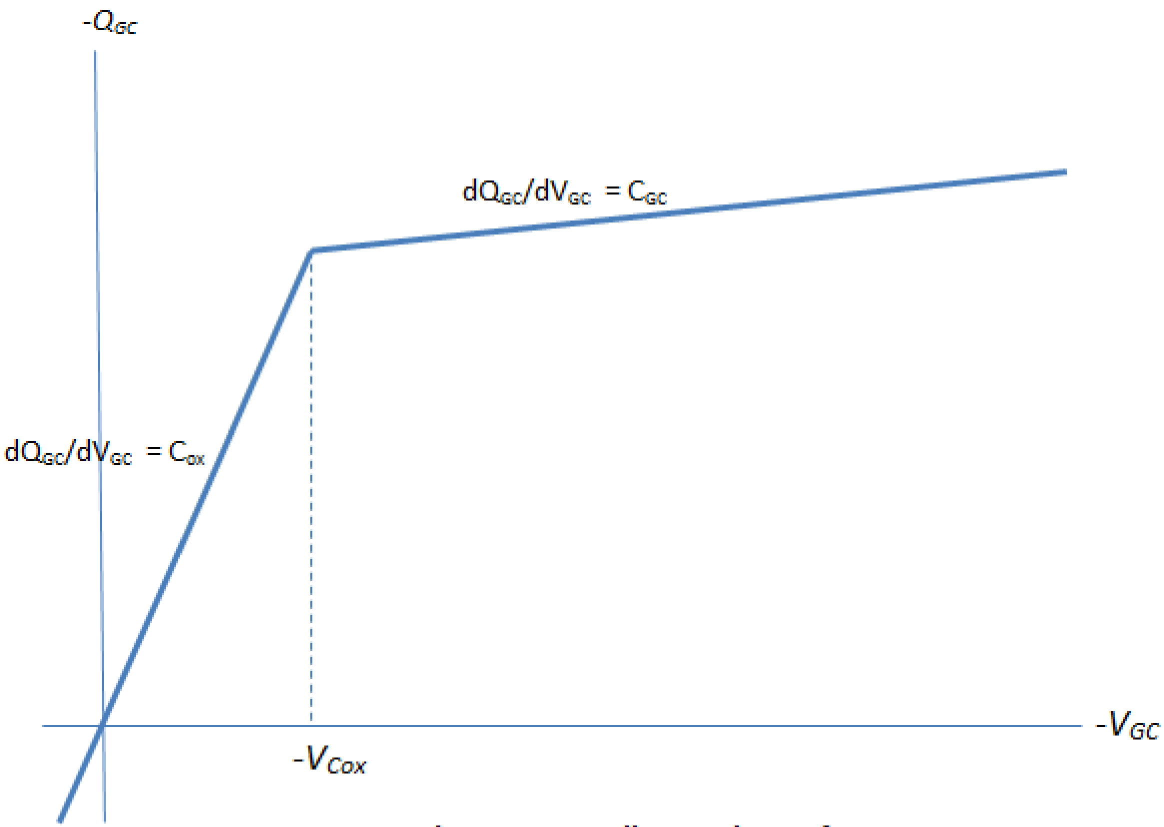
Charge Model

The block models 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 `Specify using equation parameters directly` for the **Parameterization** parameter in the **Junction Capacitance** tab, you specify the **Gate-emitter junction capacitance** and **Gate-collector junction capacitance** parameters directly. Otherwise, the block derives them from the **Input capacitance Cies** and **Reverse transfer capacitance Cres** parameter values. The two parameterizations are related as follows:

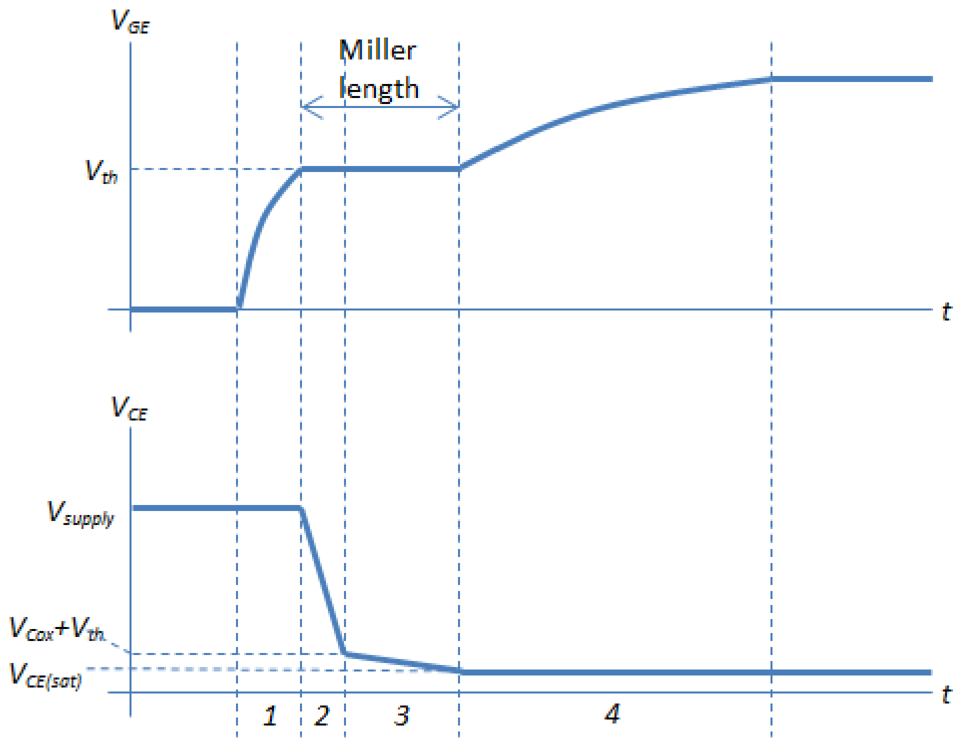
- $C_{GE} = Cres$
- $C_{GC} = Cies - Cres$

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.



Nonlinear gate-collector charge function

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 *Cies*, *Cres*, and C_{ox} as follows, assuming that the IGBT gate is driven through an external resistance R_G:

1. Set *Cies* to get correct time-constant for V_{GE} in Region 1. The time constant is defined by the product of *Cies* and R_G. Alternatively, you can use a datasheet value for *Cies*.

2. Set *Cres* so as to achieve the correct V_{CE} gradient in Region 2. The gradient is given by $(V_{GE} - V_{th})/(Cres \cdot R_G)$.

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.

Fine-Tuning the Current-Voltage Characteristics

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

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.

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, *Ices*, 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.

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".

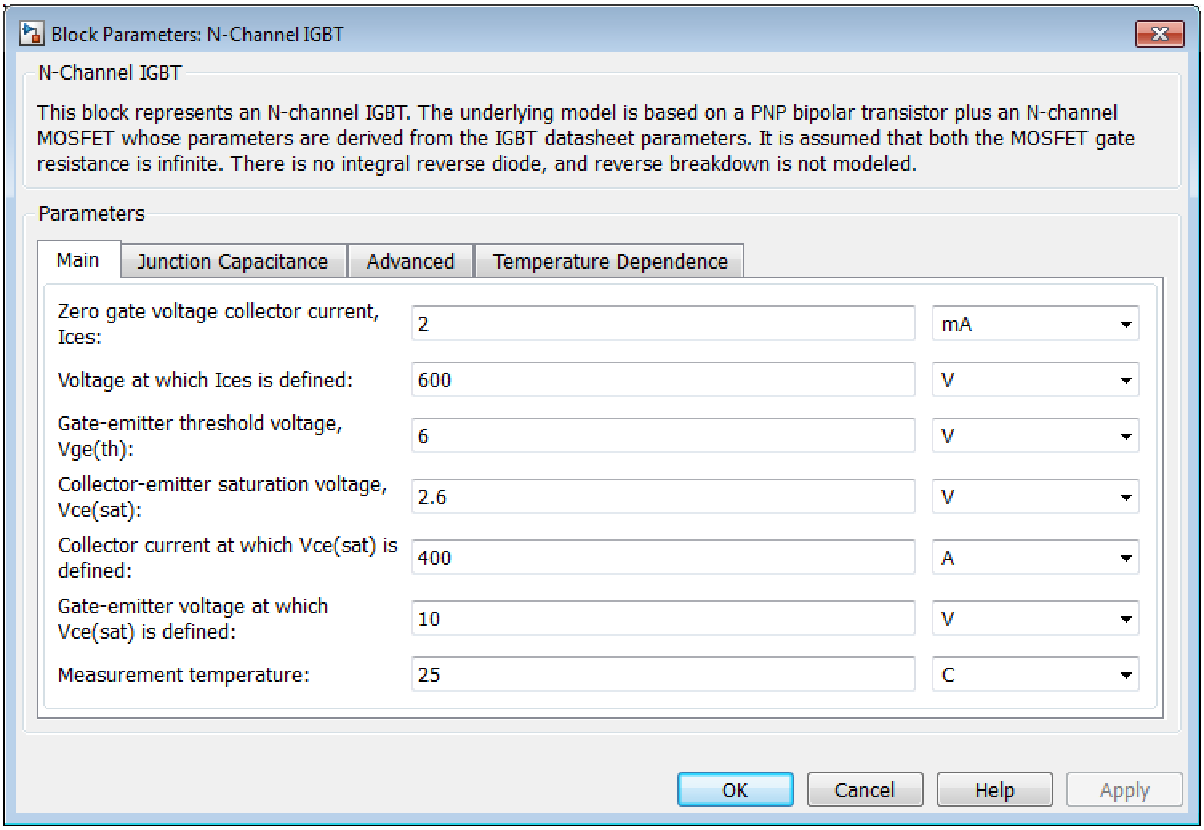
Basic Assumptions and Limitations

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 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.

Dialog Box and Parameters

Main Tab



Zero gate voltage collector current, Ices

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 Ices is defined**. The default value is 2 mA.

Voltage at which Ices is defined
The voltage used when measuring the **Zero gate voltage collector current, Ices**. The default value is 600 V.

Gate-emitter threshold voltage, Vge(th)
The threshold voltage used in the MOSFET equations. The default value is 6 V.

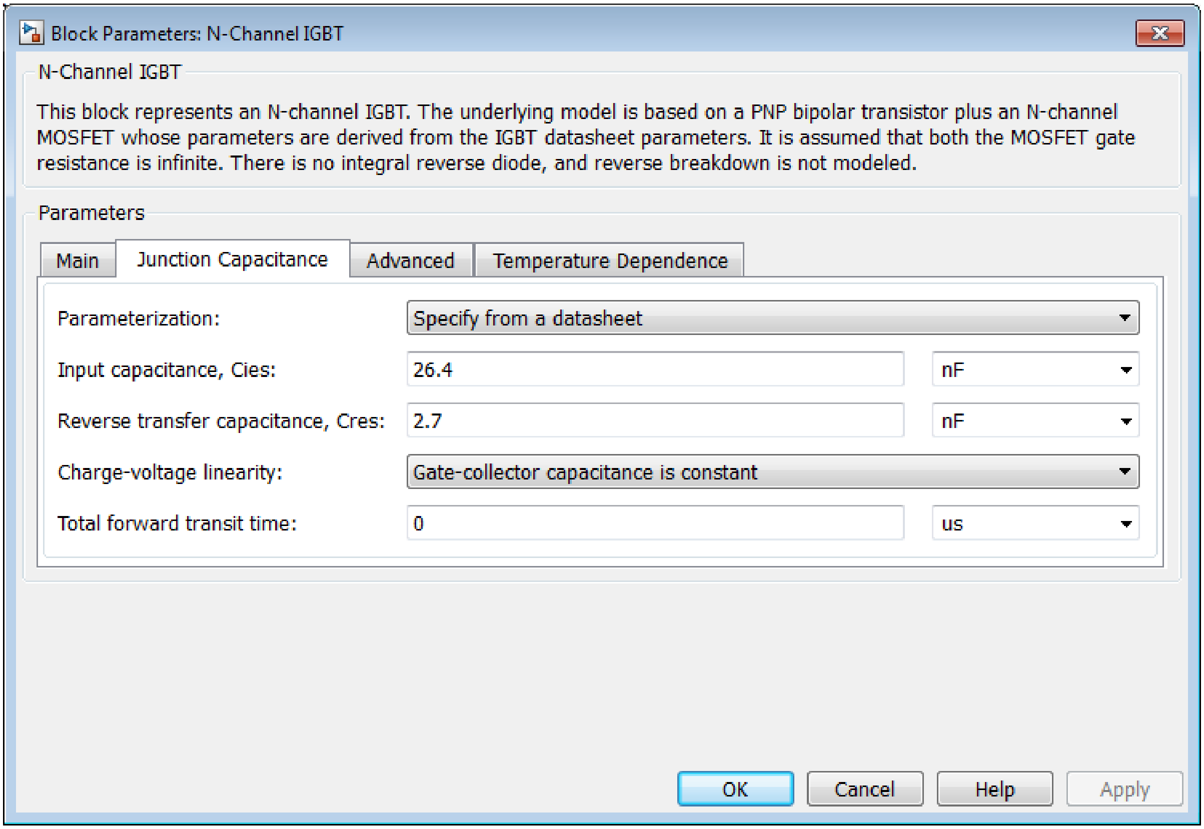
Collector-emitter saturation voltage, Vce(sat)
The collector-emitter voltage for a typical on-state as specified by the manufacturer. The default value is 2.8 V.

Collector current at which Vce(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.

Gate-emitter voltage at which Vce(sat) is defined
The gate voltage used when measuring $V_{ce(sat)}$ and $I_{ce(sat)}$. The default value is 10 V.

Measurement temperature
The temperature for which the parameters are quoted. The default value is 25 C.

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, Cies
The gate-emitter capacitance with the collector 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 `26.4` nF.

Reverse transfer capacitance, Cres
The collector-gate capacitance with the emitter 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 `2.7` nF.

Gate-emitter junction capacitance
The value of the capacitance placed between the gate and the emitter. This parameter is only visible when you select `Specify using equation parameters directly` for the **Model junction capacitance** parameter. The default value is `23.7` nF.

Gate-collector junction capacitance
The value of the capacitance placed between the gate and the collector. This parameter is only visible when you select `Specify using equation parameters directly` for the **Model junction capacitance** parameter. The default value is `2.7` nF.

Charge-voltage linearity
Select whether gate-drain capacitance is fixed or nonlinear:

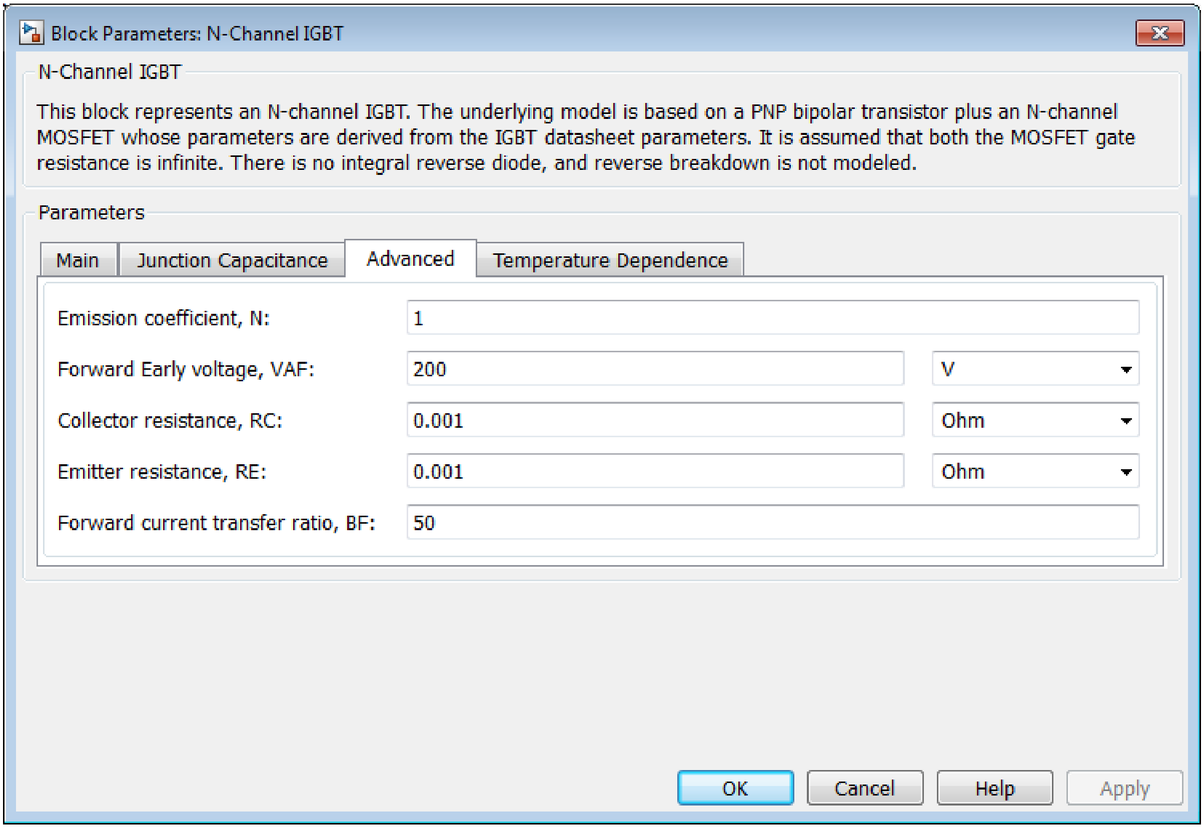
- `Gate-collector capacitance is constant` — The capacitance value is constant and defined according to the selected parameterization option, either directly or derived from a datasheet. This is the default method.
- `Gate-collector charge function is nonlinear` — The gate-collector charge relationship is defined according to the piecewise-nonlinear function described in “Charge Model” on page 1-270. Two additional parameters appear to let you define the gate-collector charge function.

Gate-collector oxide capacitance
The gate-collector capacitance when the device is on and the collector-gate voltage is small. This parameter is only visible 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 only visible 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` μs.

Advanced Tab



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.

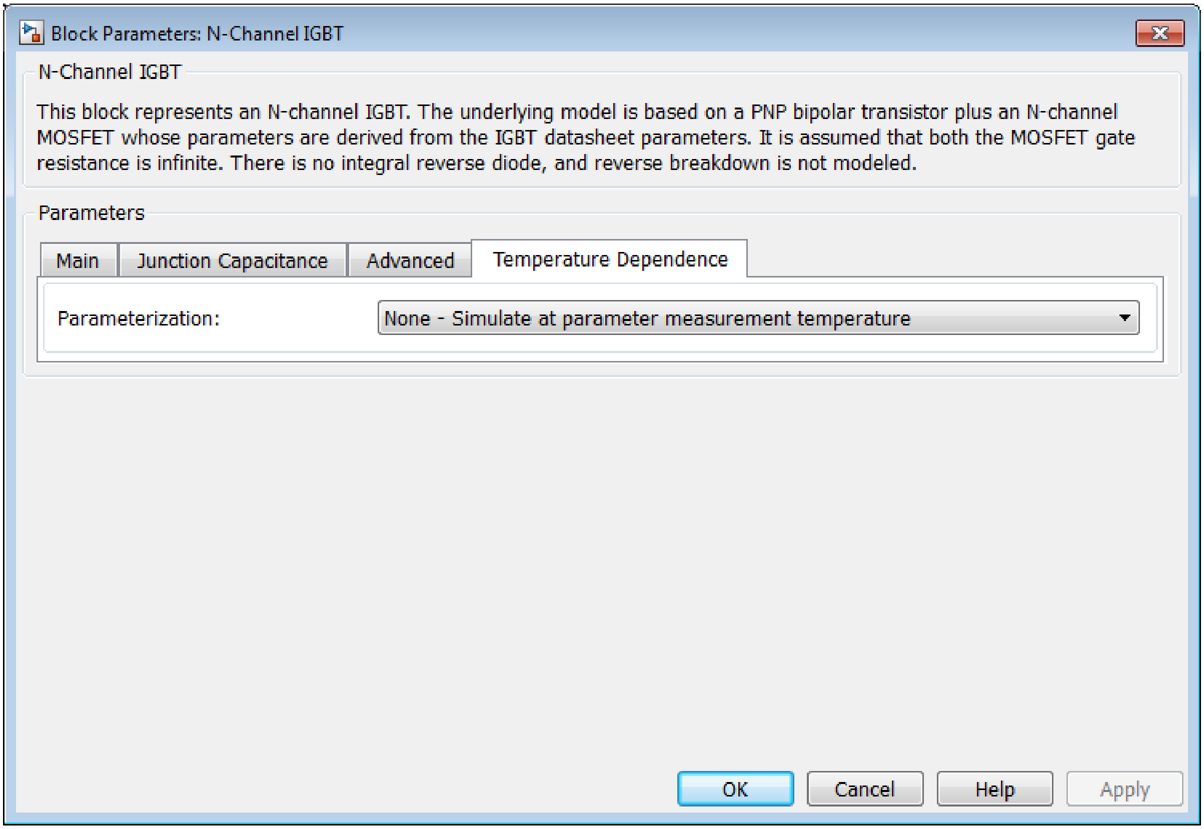
Emitter resistance, RE

Resistance at the emitter. The default value is `0.001` Ohm.

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



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 Ices and Vce(sat) at second measurement temperature` — Model temperature-dependent effects by providing values for the zero gate voltage collector current, *Ices*, and collector-emitter voltage, $V_{ce(sat)}$, at the second measurement temperature.
- `Specify Vce(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, *Ices*, at a higher measurement temperature.

Energy gap, EG

Energy gap value. This parameter is only visible when you select `Specify Vce(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, Ices, at second measurement temperature

The zero gate collector current value at the second measurement temperature. This parameter is only visible when you select `Specify Ices and Vce(sat) at second measurement temperature` for the **Parameterization** parameter. The default value is `100` mA.

Collector-emitter saturation voltage, Vce(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, Ices, 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 *Ices* 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`.

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:

`C`

Electrical conserving port associated with the PNP emitter terminal.

`G`

Electrical conserving port associated with the MOSFET gate terminal.

`E`

Electrical conserving port associated with the PNP collector terminal.

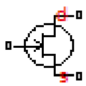
Purpose

Model N-Channel JFET

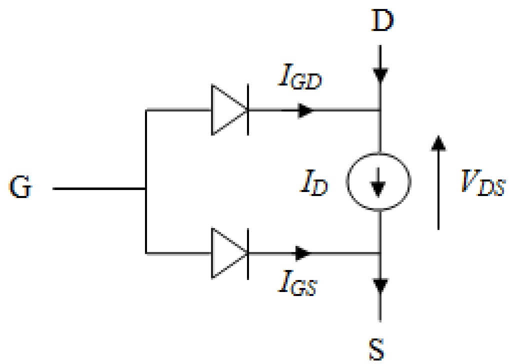
Library

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	$V_{GS} - V_{t0} \leq 0$	$I_D = 0$
Linear	$0 < -V_{DS} < V_{GS} - V_{t0}$	$I_D = \beta V_{DS}(2(V_{GD} - V_{t0}) + V_{DS})(1 - \lambda V_{DS})$
Saturated	$0 < V_{GS} - V_{t0} \leq -V_{DS}$	$I_D = \beta\,(V_{GD} - V_{t0})^2\,(1 - \lambda V_{DS})$

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.
- β is the transconductance parameter. If you select `Specify using equation parameters directly` for the **Parameterization** parameter, β is the **Transconductance parameter** parameter value. Otherwise, the block calculates β from the datasheet parameters you specify.
- λ is the channel-length modulation parameter. If you select `Specify using equation parameters directly` for the **Parameterization** parameter, λ is the **Channel-length modulation** parameter value. Otherwise, the block calculates λ 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, β, and the threshold voltage, V_{t0}, become a function of temperature according to the following equations:

$$\beta_{Ts} = \beta_{Tm1}\left(\frac{T_s}{T_{m1}}\right)^{BEX}$$

$$V_{t0s} = V_{t01} + \alpha\,(T_s - T_{m1})$$

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.
- β_{Tm1} is JFET transconductance at the measurement temperature.
- β_{Ts} 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.
- a 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 a, 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 a:

- 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 a based on this data.
- If you parameterize by specifying equation parameters, you have to provide the value for a 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 a and *BEX*.

In addition, the saturation current term, *IS*, in the gate-drain and gate-source current equations depends on temperature

$$IS_{Ts} = IS_{Tm1} \cdot (T_s / T_{m1})^{XTI} \cdot \exp\left(-\frac{EG}{kT_s}(1 - T_s / T_{m1}) \right)$$

where:

- IS_{Tm1} is the saturation current at the measurement temperature.
- IS_{Ts} 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 *a*, 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

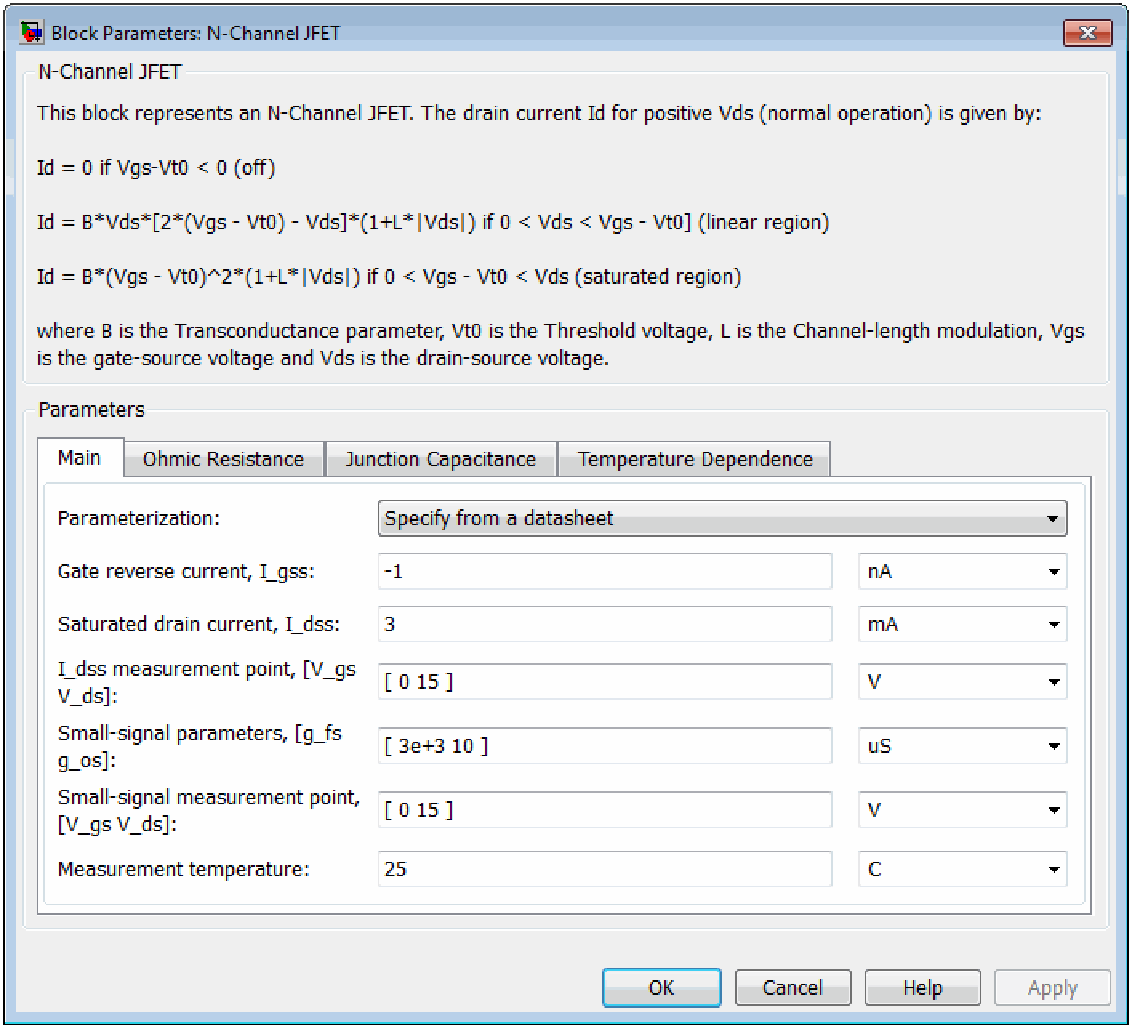
The model is based on the following assumptions:

- 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 a 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.

Dialog Box and Parameters

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 β, IS, V_{t0}, and λ.

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_gs V_ds]

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_fs g_os]

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_gs V_ds]
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^2.

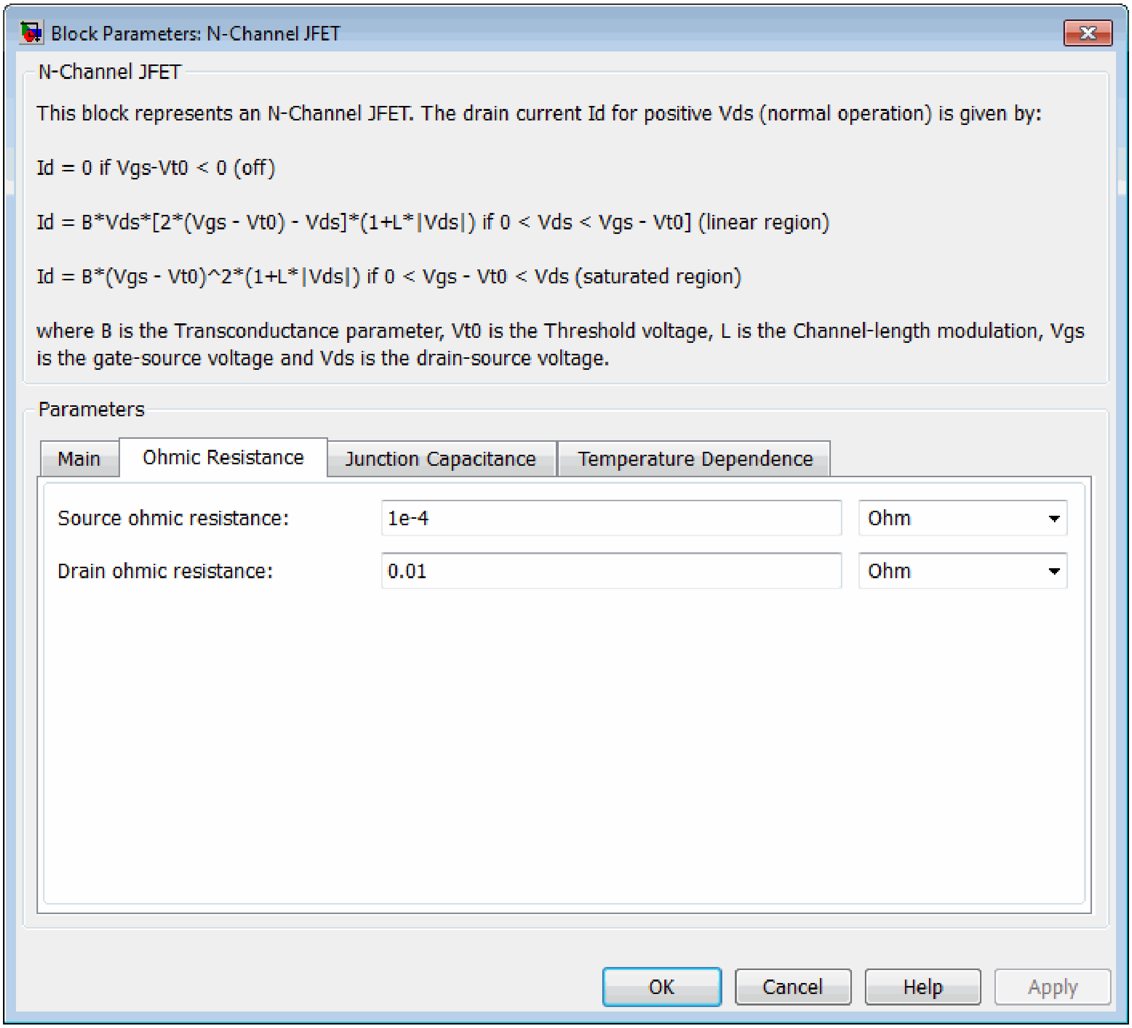
Saturation current
The magnitude of the current that the ideal diode equation approaches asymptotically for very large reverse bias levels. This parameter is only visible when you select `Specify using equation parameters directly` for the **Parameterization** parameter. The default value is `1e-14` A.

Threshold voltage
The gate-source voltage above which the transistor produces a nonzero drain current. For an enhancement device, Vt0 should be positive. For a depletion mode device, Vt0 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` 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.

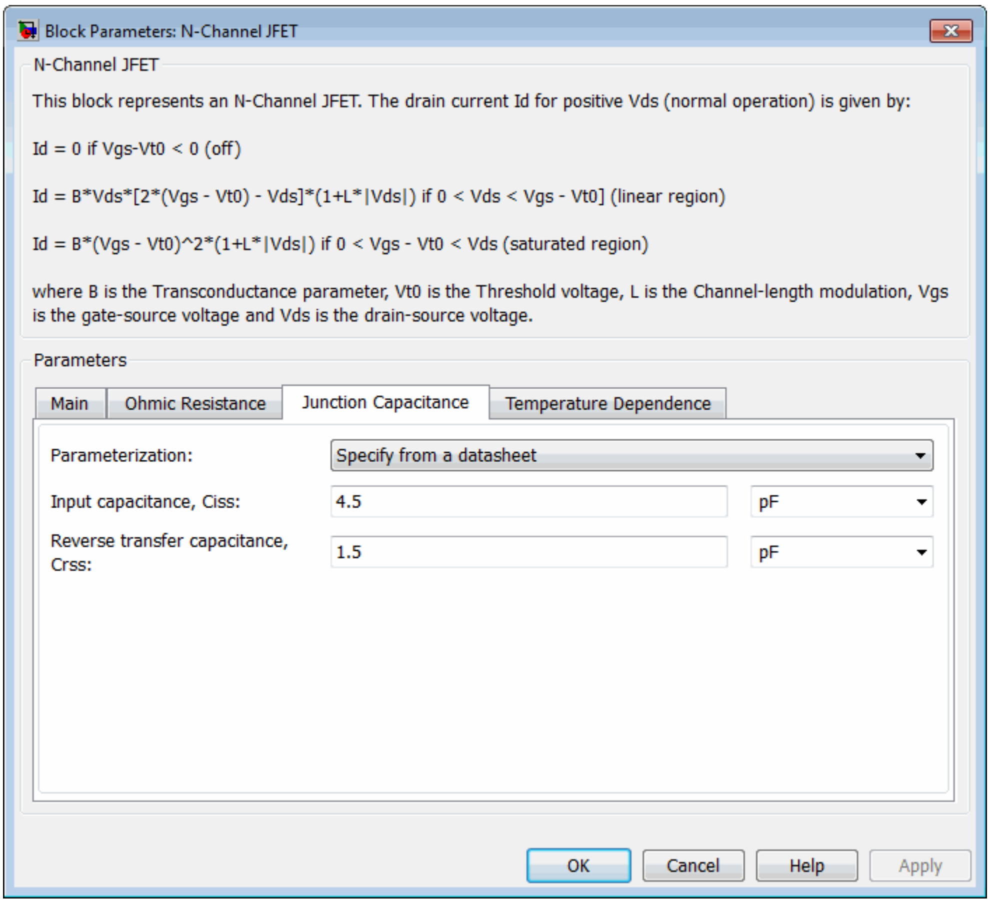
Ohmic Resistance Tab



Source ohmic resistance
The transistor source resistance. The default value is 1e-4 Ω. The value must be greater than or equal to 0.

Drain ohmic resistance
The transistor drain resistance. The default value is 0.01 Ω. 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.

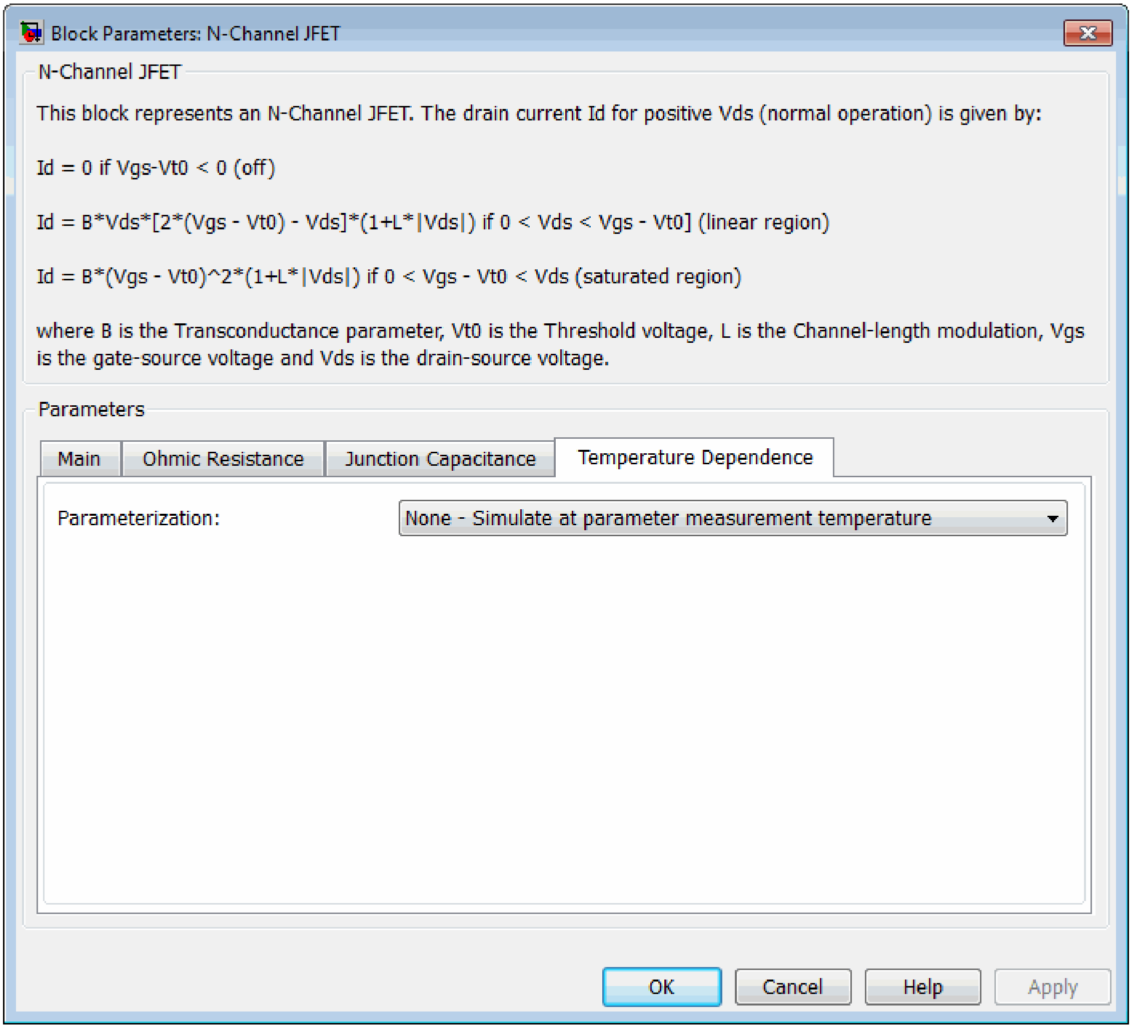
Gate-source junction capacitance

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, 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 `-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 1-290 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

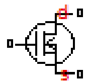
P-Channel JFET

N-Channel MOSFET

Purpose Model N-Channel MOSFET using Shichman-Hodges equation

Library Semiconductor Devices

Description The N-Channel MOSFET 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:

$$I_{DS} = K\left((V_{GS} - V_{th})V_{DS} - V_{DS}^{\ 2}/2\right)\left(1 + \lambda\,|V_{DS}|\right)$$

- In the saturated region ($0 < V_{GS} - V_{th} < V_{DS}$) the drain-source current is:

$$I_{DS} = (K/2)(V_{GS} - V_{th})^2\left(1 + \lambda\,|V_{DS}|\right)$$

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.
- λ is the channel modulation.

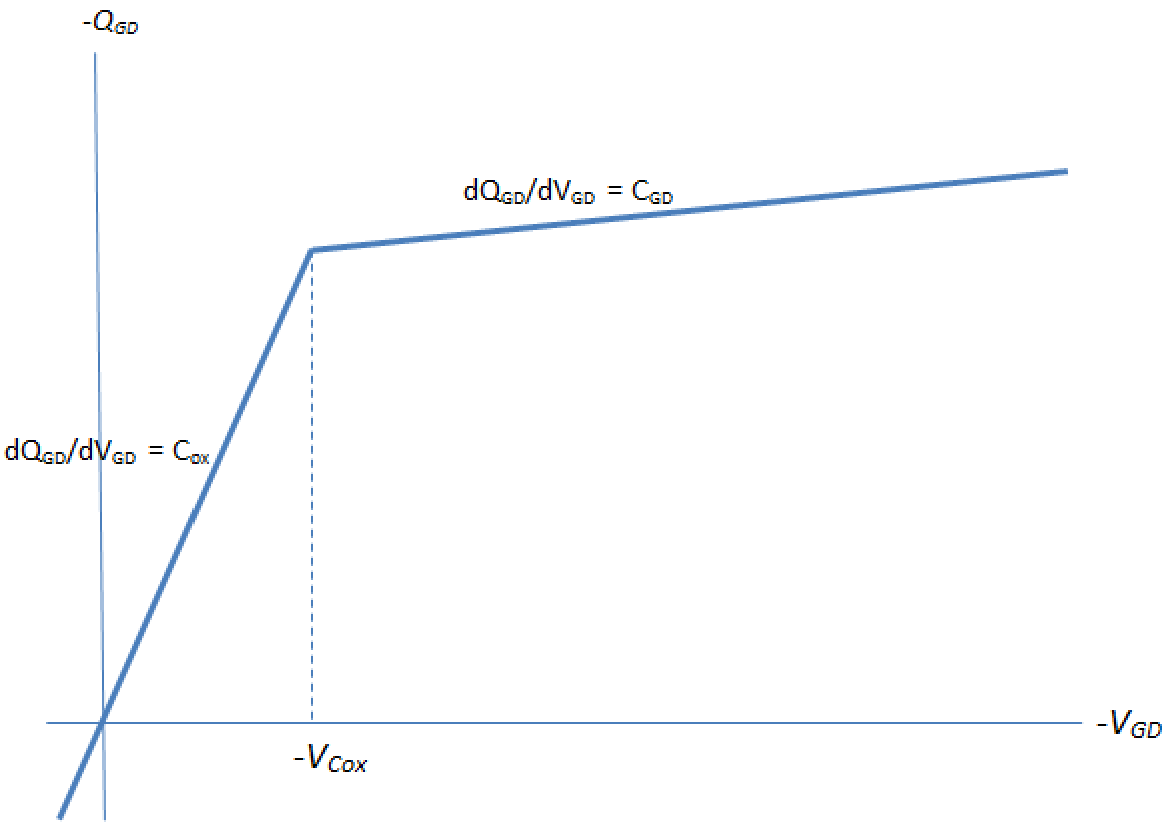
Charge Model

The block models 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 `Specify using equation parameters directly` for the **Parameterization** parameter in the **Junction Capacitance** tab, you specify the **Gate-drain junction capacitance** and **Gate-source junction capacitance** parameters directly. Otherwise, the block derives them from the **Input capacitance Ciss** and **Reverse transfer capacitance Crss** parameter values. The two parameterizations are related as follows:

- $C_{GD} = Crss$
- $C_{GS} = Ciss - Crss$

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.

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 gain, K, and the threshold voltage, V_{th}, become a function of temperature according to the following equations:

$$K_{Ts} = K_{Tm1}\left(\frac{T_s}{T_{m1}}\right)^{BEX}$$

$$V_{ths} = V_{th1} + a\,(T_s - T_{m1})$$

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_{Tm1} is the transistor gain at the measurement temperature.
- K_{Ts} 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_{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.
- a 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 a, 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 a:

- 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 a based on this data.
- If you parameterize by specifying equation parameters, you have to provide the value for a 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 a and *BEX*.

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

When modeling temperature dependence, 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.

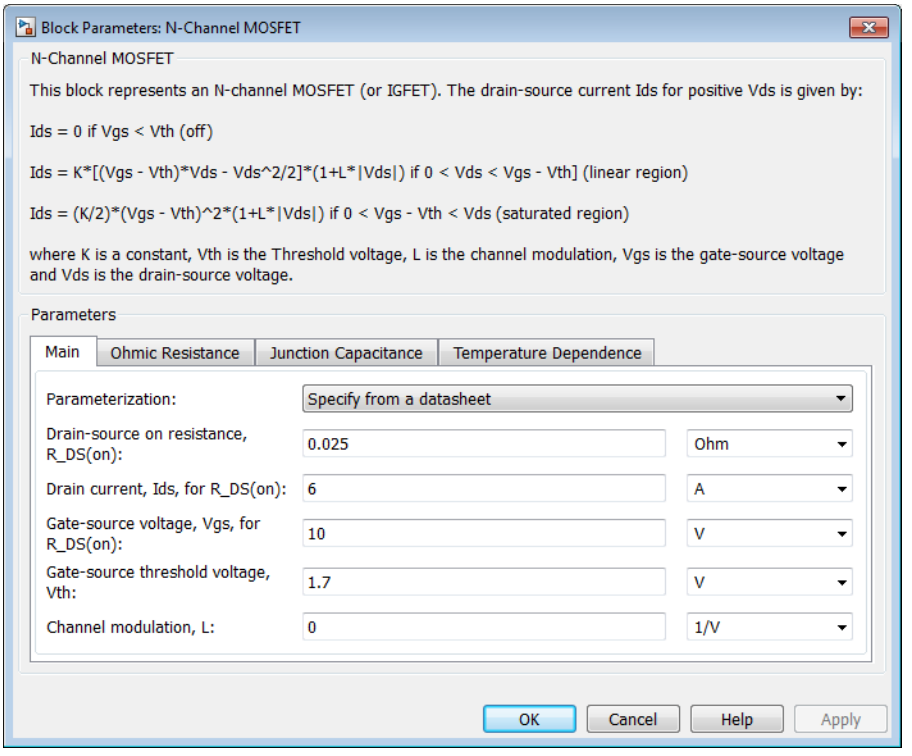
Inconsistent values for R_{DS}*(on)* at the higher temperature will result in unphysical values for a and unrepresentative simulation results. Typically R_{DS}*(on)* increases by a factor of about 1.5 for a hundred degree increase in temperature.

- You may need to tune the values of *BEX* and threshold voltage, V_{th}, to replicate the V_{DS}-V_{GS} relationship (if available) for a given device. Increasing V_{th} moves the V_{DS}-V_{GS} plots to the right. The value of *BEX* affects whether the V_{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*.

N-Channel MOSFET

Dialog Box and Parameters

Main Tab



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` Ω.

Drain current, Ids, 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, Vgs, 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 Vth

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.

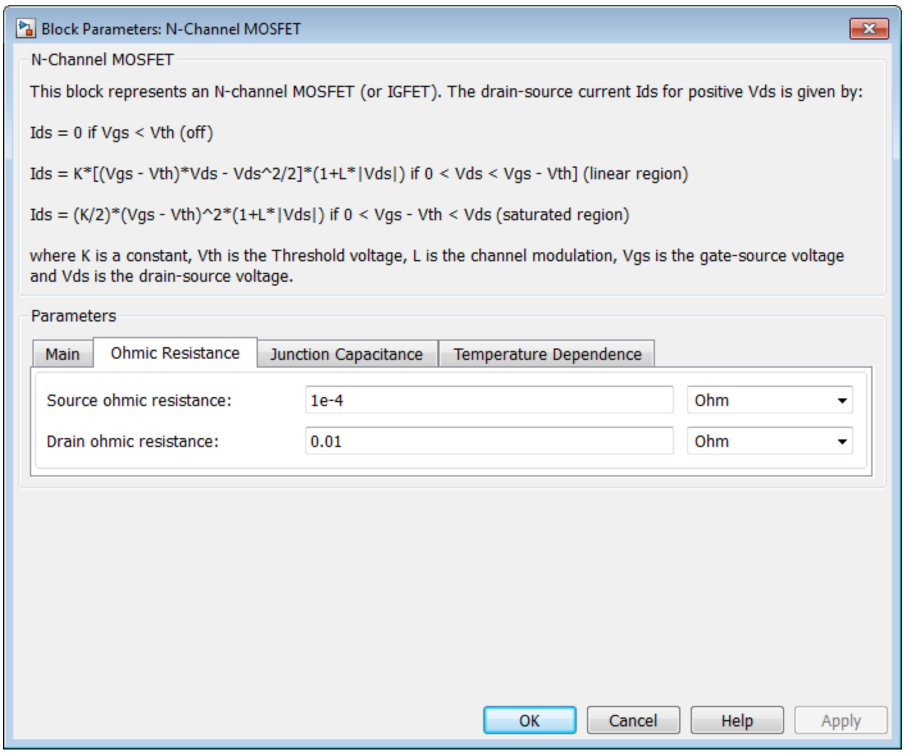
Channel modulation, L

The channel-length modulation, usually denoted by the mathematical symbol λ. 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. This parameter is only visible when you select `Model temperature dependence` for the **Parameterization** parameter on the **Temperature Dependence** tab. The default value is `25` C.

Ohmic Resistance Tab



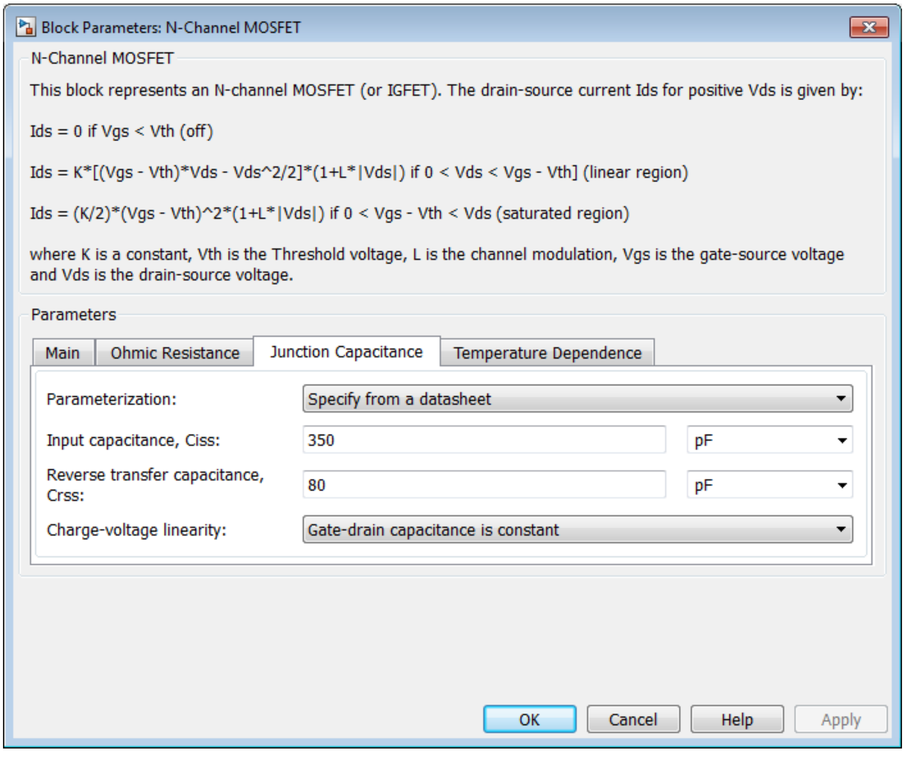
Source ohmic resistance

The transistor source resistance. The default value is 1e-4 Ω. The value must be greater than or equal to 0.

Drain ohmic resistance

The transistor drain resistance. The default value is 0.001 Ω. The value must be greater than or equal to 0.

Junction Capacitance Tab



Parameterization

Select one of the following methods for capacitance 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 **Parameterization** parameter. The default value is `350` 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 **Parameterization** parameter. The default value is `80` 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 **Parameterization** parameter. The default value is `270` 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 **Parameterization** parameter. The default value is `80` pF.

Charge-voltage linearity
Select whether 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” on page 1-303. Two additional parameters appear to let you define the gate-drain charge function.

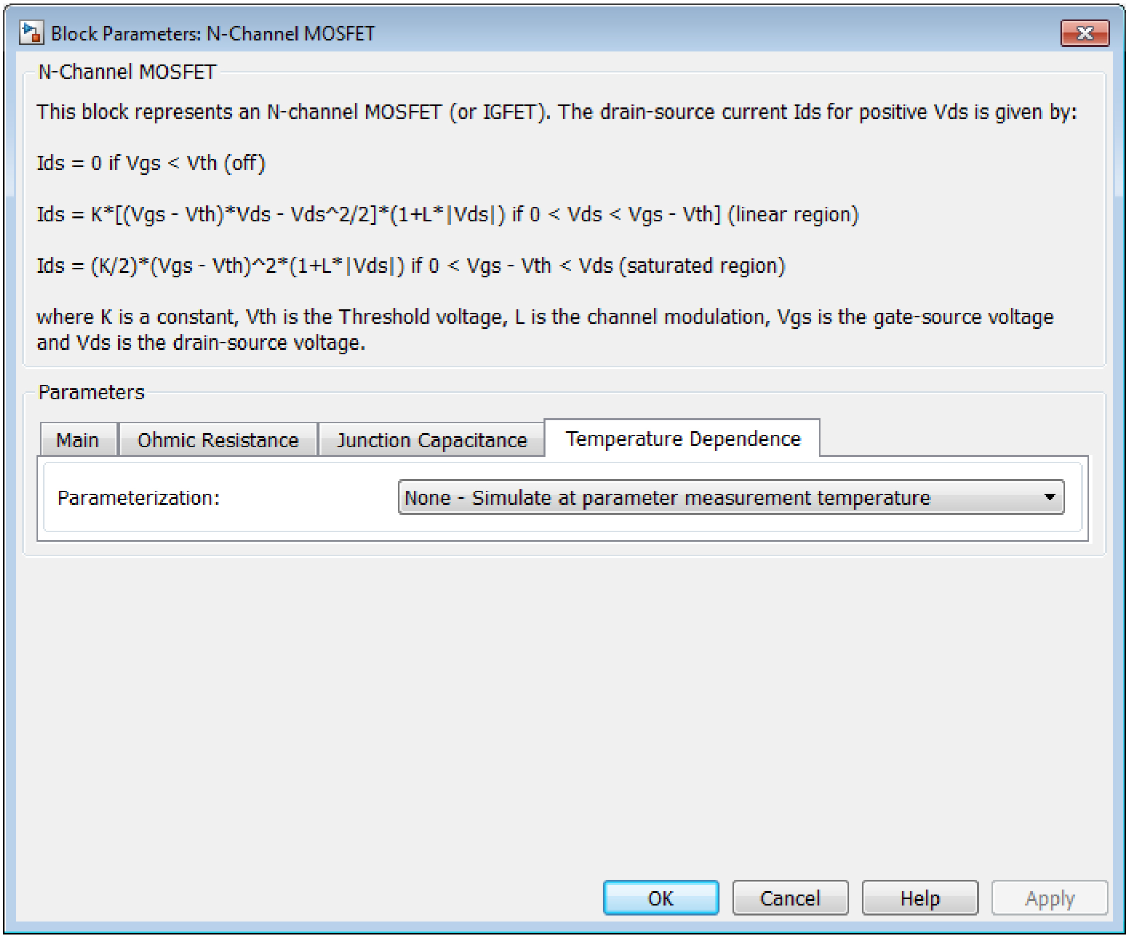
Gate-drain oxide capacitance

The gate-drain capacitance when the device is on and the drain-gate voltage is small. 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.

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 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 a 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 a based on that. If you parameterize by specifying equation parameters, you have to provide the value for a 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` Ω.

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` 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 `-6` 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 1-306 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.

See Also

P-Channel MOSFET

Purpose

Model ideal negative supply rail

Library

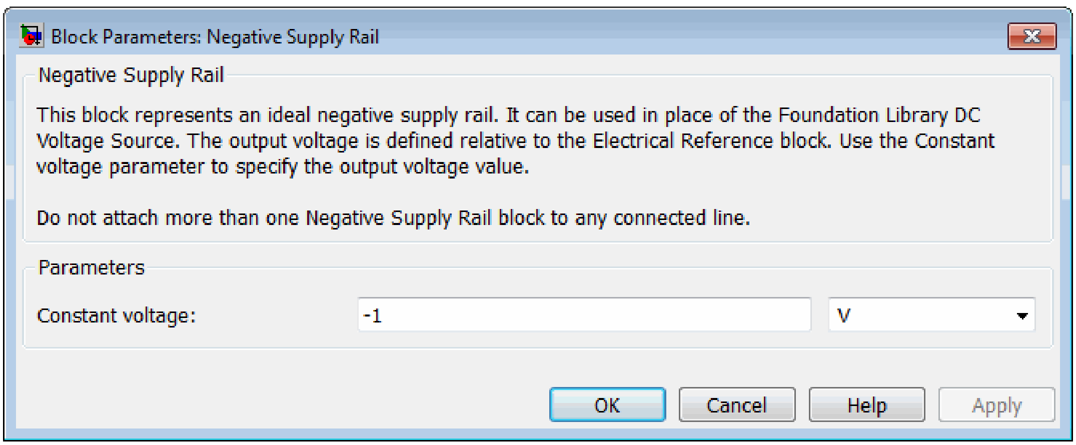
Sources

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.

Dialog Box and 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

Simscape DC Voltage Source, Positive Supply Rail

Nonlinear Inductor

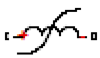
Purpose

Model inductor with nonideal core

Library

Passive Devices

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)
- Single saturation point
- Magnetic flux versus current characteristic
- Magnetic flux density versus magnetic field strength characteristic

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.

- Φ 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.
- Φ is the magnetic flux.
- Φ_{offset} is the magnetic flux saturation offset.
- L is the unsaturated inductance.
- L_{sat} is the saturated inductance.

Magnetic Flux Versus Current Characteristic

The relationships between voltage, current and flux are defined by the following equations:

$$i = i_L + vG_p$$

$$v = N_w \frac{d\Phi}{dt}$$

$$\Phi = f\left(i_L\right)$$

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.
- Φ 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.
- Φ is the magnetic flux.
- H is the magnetic field strength.
- B is the magnetic flux density.
- l_e is the effective core length.
- A_e is the effective core cross-sectional area.

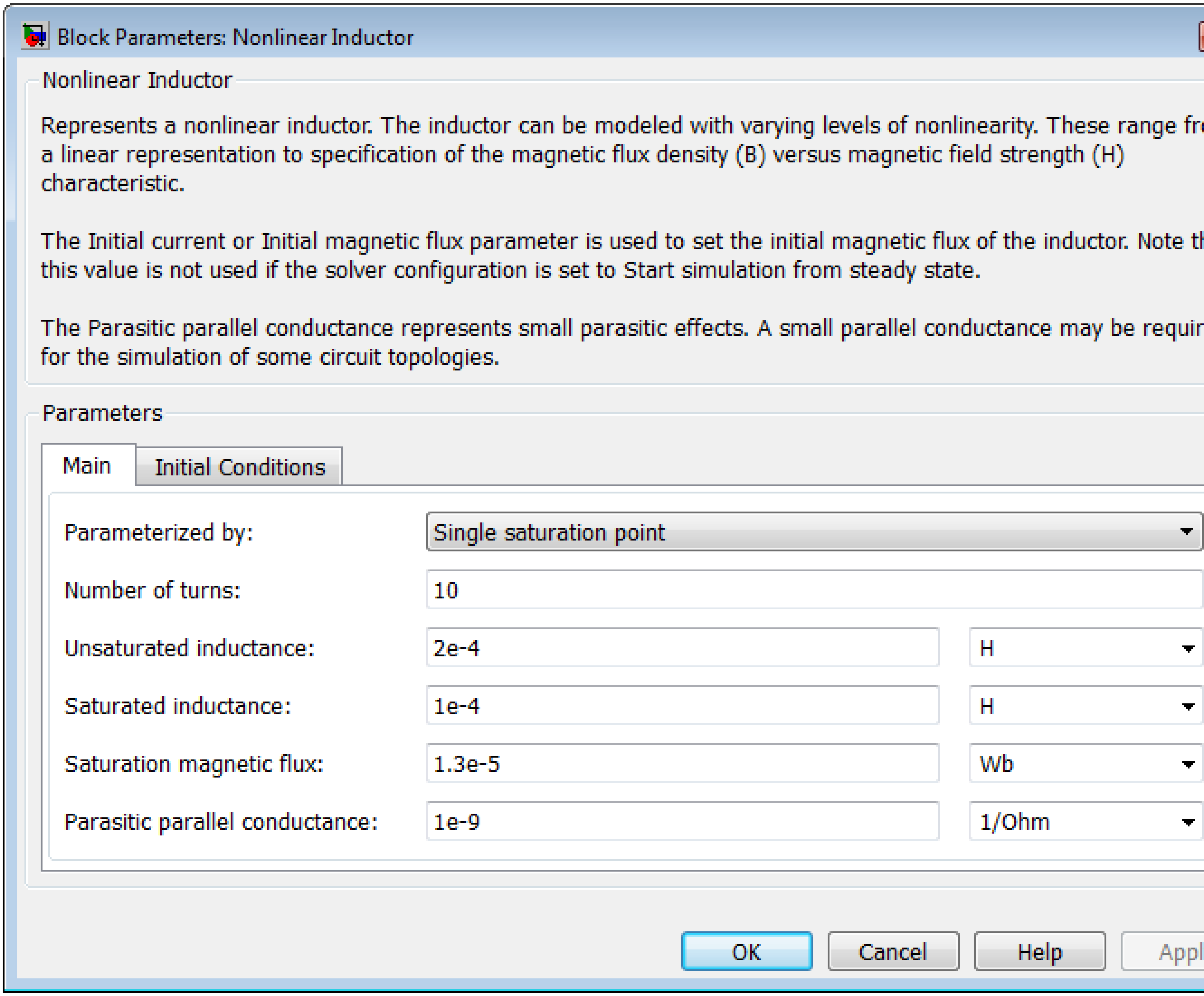
Magnetic flux density is determined by one-dimensional table lookup, based on the vector of magnetic field strength values and the vector of corresponding magnetic flux density values that you provide. You can construct these vectors using either negative and positive data, or positive data only. If using positive data only, the vector must start at

0, and the negative data will be automatically calculated by rotation about (0,0).

Dialog Box and Parameters

- "Main Tab" on page 1-324
- "Initial Conditions Tab" on page 1-328

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.

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 only visible 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 only visible 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 only visible 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 only visible 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 only visible 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 only visible 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 only visible 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 only visible when you select `Magnetic flux density versus magnetic field strength characteristic` 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 only visible when you select `Magnetic flux density versus magnetic field strength characteristic` for the **Parameterized by** parameter. The default value is `1.6e-5` m^2.

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/Ω.

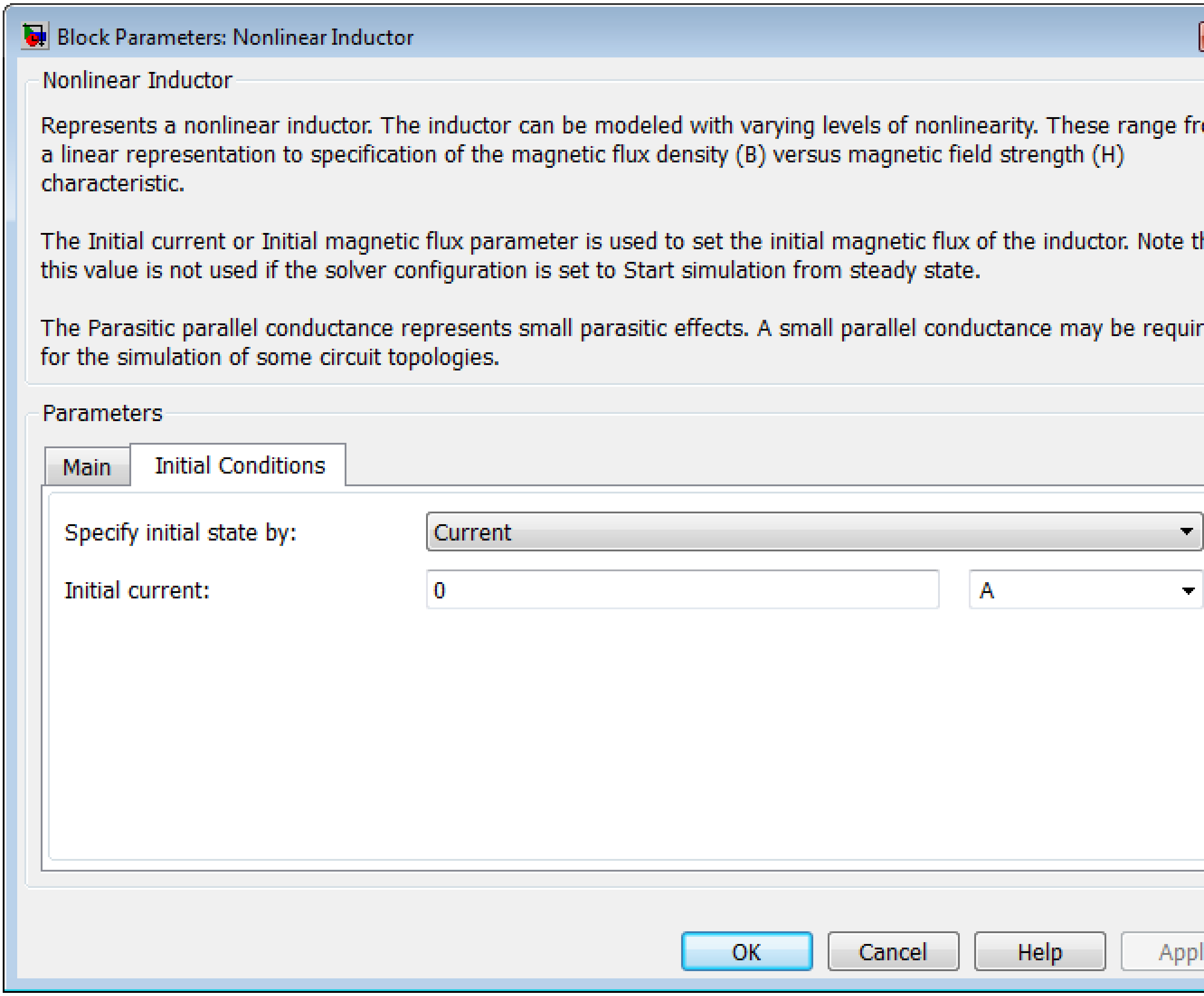
Interpolation option

The lookup table interpolation option. This parameter is only visible 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` — Uses a linear interpolation function.
- `Cubic` — Uses the Piecewise Cubic Hermite Interpolation Polinomial (PCHIP).

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.

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 only visible 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 only visible when you select `Magnetic flux` for the **Specify initial state by** parameter. The default is `0` Wb.

Examples

For comparison of nonlinear inductor behavior with different parameterization options, see the Nonlinear Inductor Characteristics example.

Ports

The block has the following ports:

+

Positive electrical port.

-

Negative electrical port.

See Also

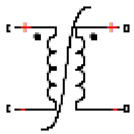
Simscape Inductor

Nonlinear Transformer

Purpose Model transformer with nonideal core

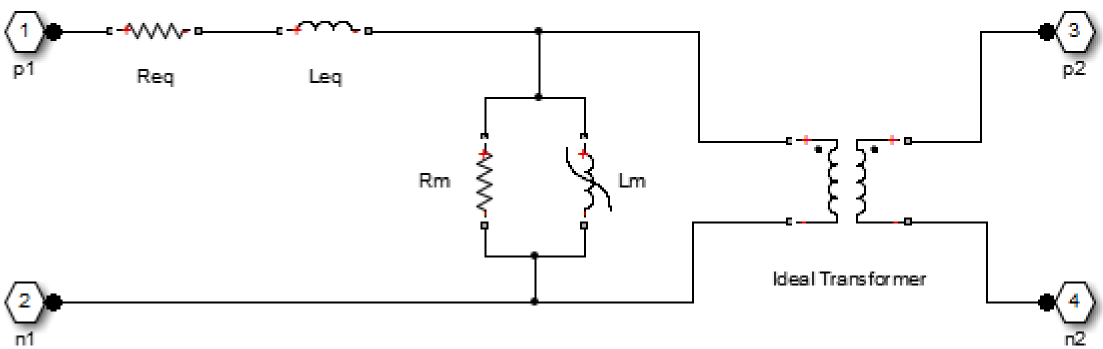
Library Passive Devices

Description

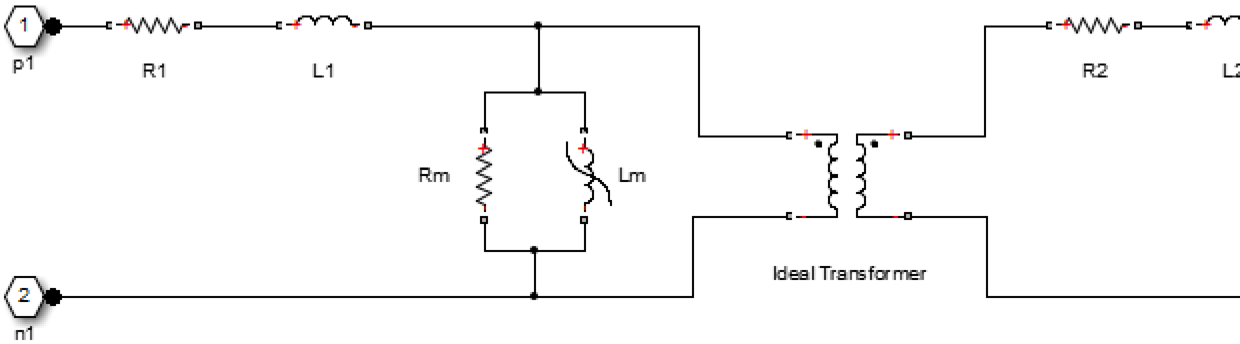


The Nonlinear Transformer block represents a transformer with a nonideal core. A core may be nonideal due to its magnetic properties and dimensions. The equivalent circuit topology depends upon which of the two winding leakage parameterization options you select:

- `Combined primary and secondary values`



- `Separate primary and secondary values`



where:

- *Req* is the combined leakage resistance.
- *Leq* 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.
- *Rm* is the magnetization resistance.
- *Lm* 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`

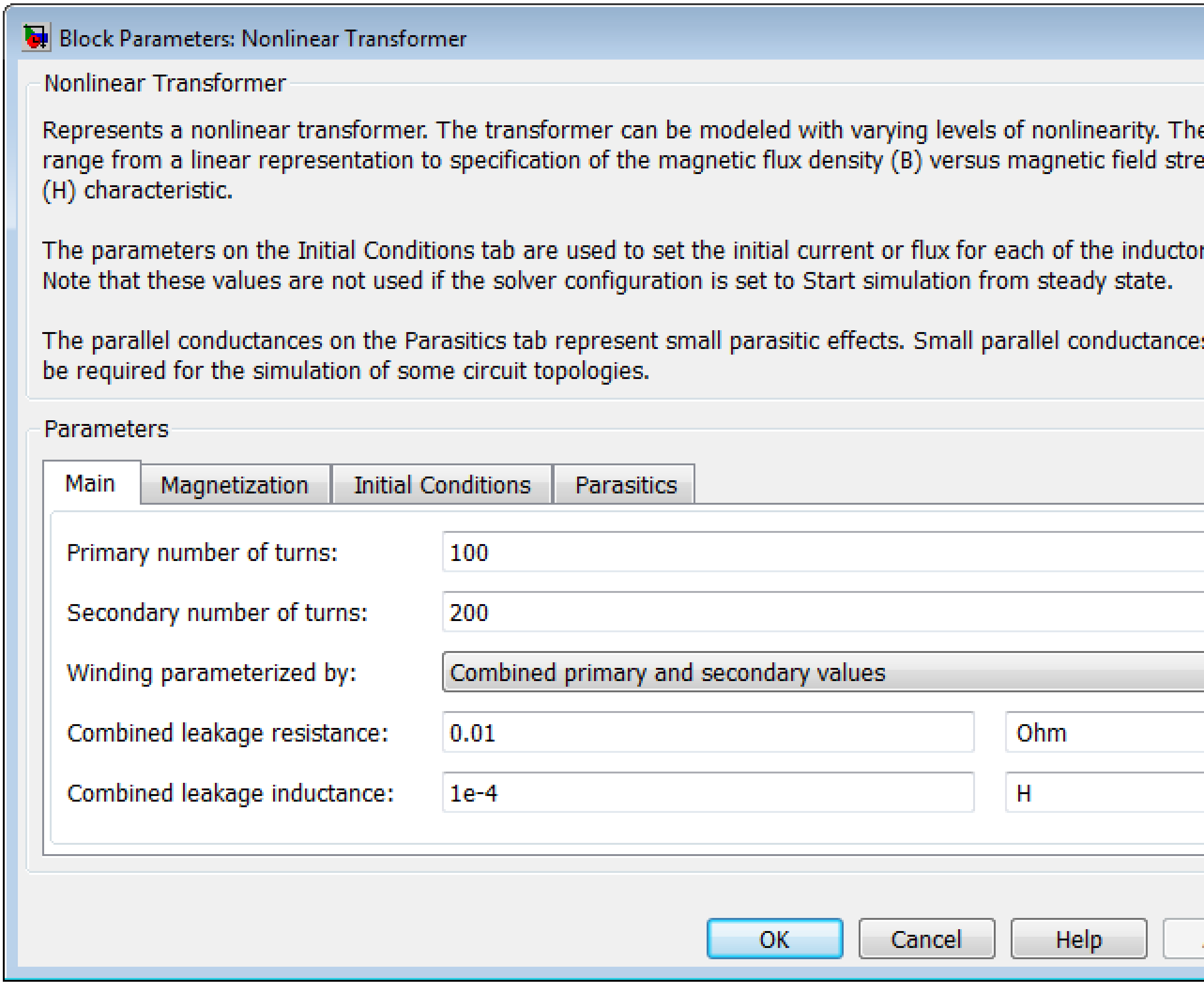
- `Magnetic flux density versus magnetic field strength characteristic`

For more information, see the Nonlinear Inductor block reference page.

Dialog Box and Parameters

- “Main Tab” on page 1-333
- “Magnetization Tab” on page 1-336
- “Initial Conditions Tab” on page 1-340
- “Parasitics Tab” on page 1-343

Main Tab


Block Parameters: Nonlinear Transformer
Nonlinear Transformer
Represents a nonlinear transformer. The transformer can be modeled with varying levels of nonlinearity.
range from a linear representation to specification of the magnetic flux density (B) versus magnetic field
(H) characteristic.
The parameters on the Initial Conditions tab are used to set the initial current or flux for each of the
Note that these values are not used if the solver configuration is set to Start simulation from steady state.
The parallel conductances on the Parasitics tab represent small parasitic effects. Small parallel
be required for the simulation of some circuit topologies.
Parameters
Main
Magnetization
Initial Conditions
Parasitics
Primary number of turns:
100
Secondary number of turns:
200
Winding parameterized by:
Combined primary and secondary values
Combined leakage resistance:
0.01
Ohm
Combined leakage inductance:
1e-4
H
OK
Cancel
Help

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 only visible when you select `Combined primary and secondary values` for the **Winding parameterized by** parameter. The default value is `0.01` Ω.

Combined leakage inductance

The lumped equivalent inductance *Leq*, which represents the combined magnetic flux loss of the primary and secondary windings. This parameter is only visible 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 only visible when you select `Separate primary and secondary values` for the **Winding parameterized by** parameter. The default value is `0.01` Ω.

Primary leakage inductance

The inductance *L1*, which represents the magnetic flux loss of the primary winding. This parameter is only visible when you select `Separate primary and secondary values` for the **Winding parameterized by** parameter. The default value is `1e-4` H.

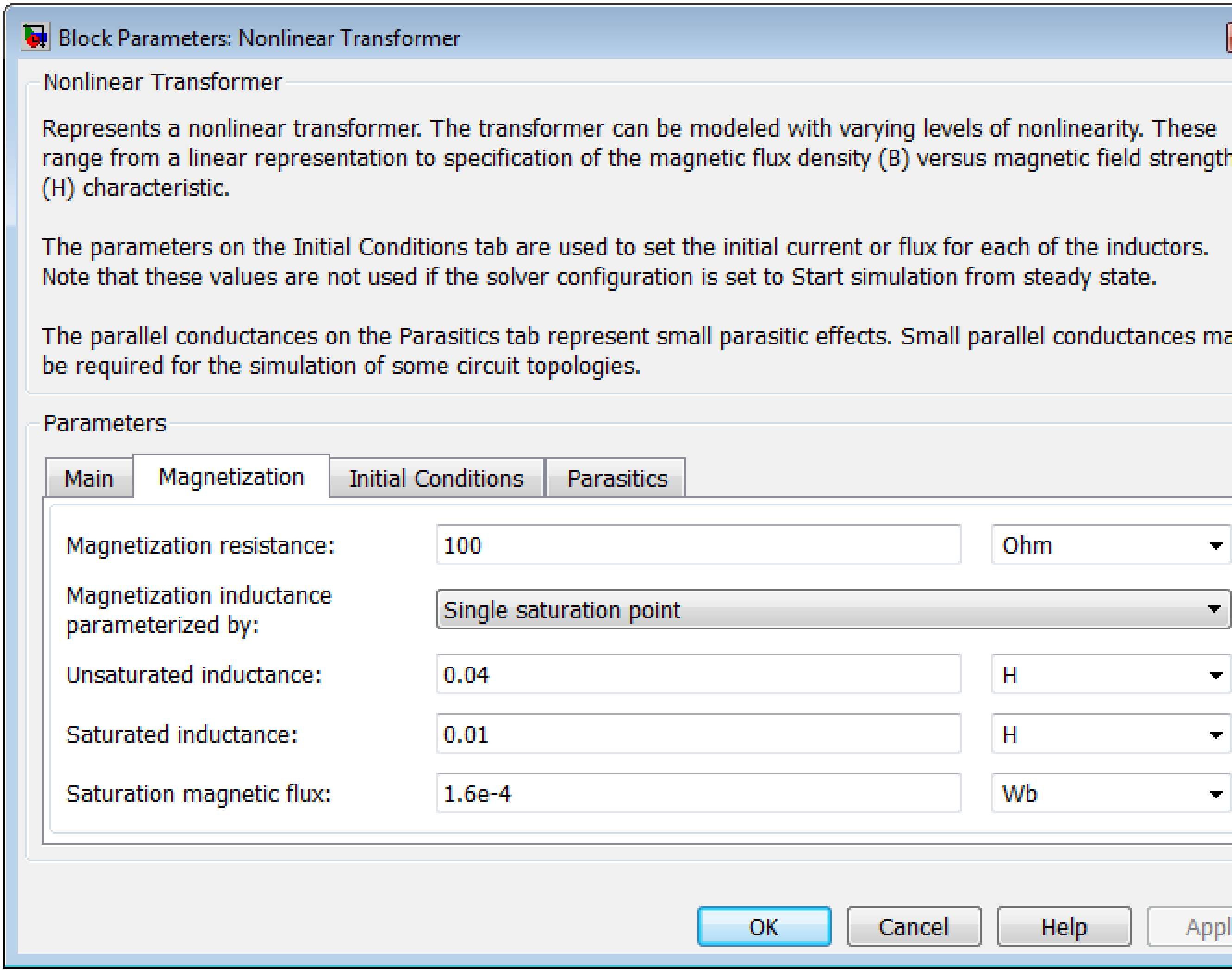
Secondary leakage resistance

The resistance *R2*, which represents the power loss of the secondary winding. This parameter is only visible when you select `Separate primary and secondary values` for the **Winding parameterized by** parameter. The default value is `0.01` Ω.

Secondary leakage inductance

The inductance *L2*, which represents the magnetic flux loss of the secondary winding. This parameter is only visible 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` Ω.

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.

Unsaturated inductance

The value of inductance used when the magnetization inductance *Lm* is operating in its linear region. This parameter is only visible 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 *Lm* is operating beyond its saturation point. This parameter is only visible 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 *Lm* saturates. This parameter is only visible 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 only visible 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 only visible 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 only visible 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 only visible 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 only visible 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 only visible 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^2.

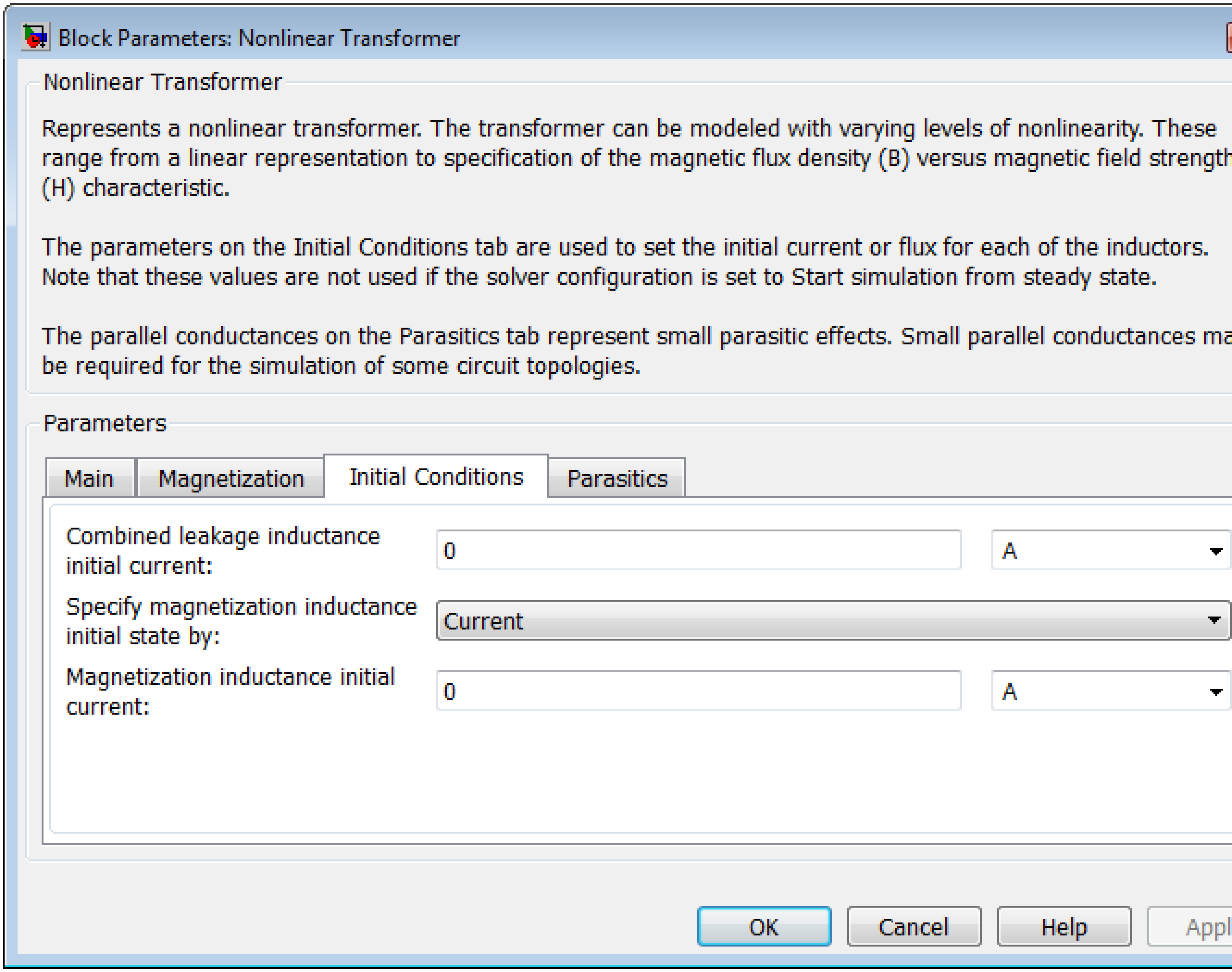
Interpolation option

The lookup table interpolation option. This parameter is only visible 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` — Uses a linear interpolation function.
- `Cubic` — Uses the Piecewise Cubic Hermite Interpolation Polinomial (PCHIP).

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 only visible 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 only visible 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 only visible 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:

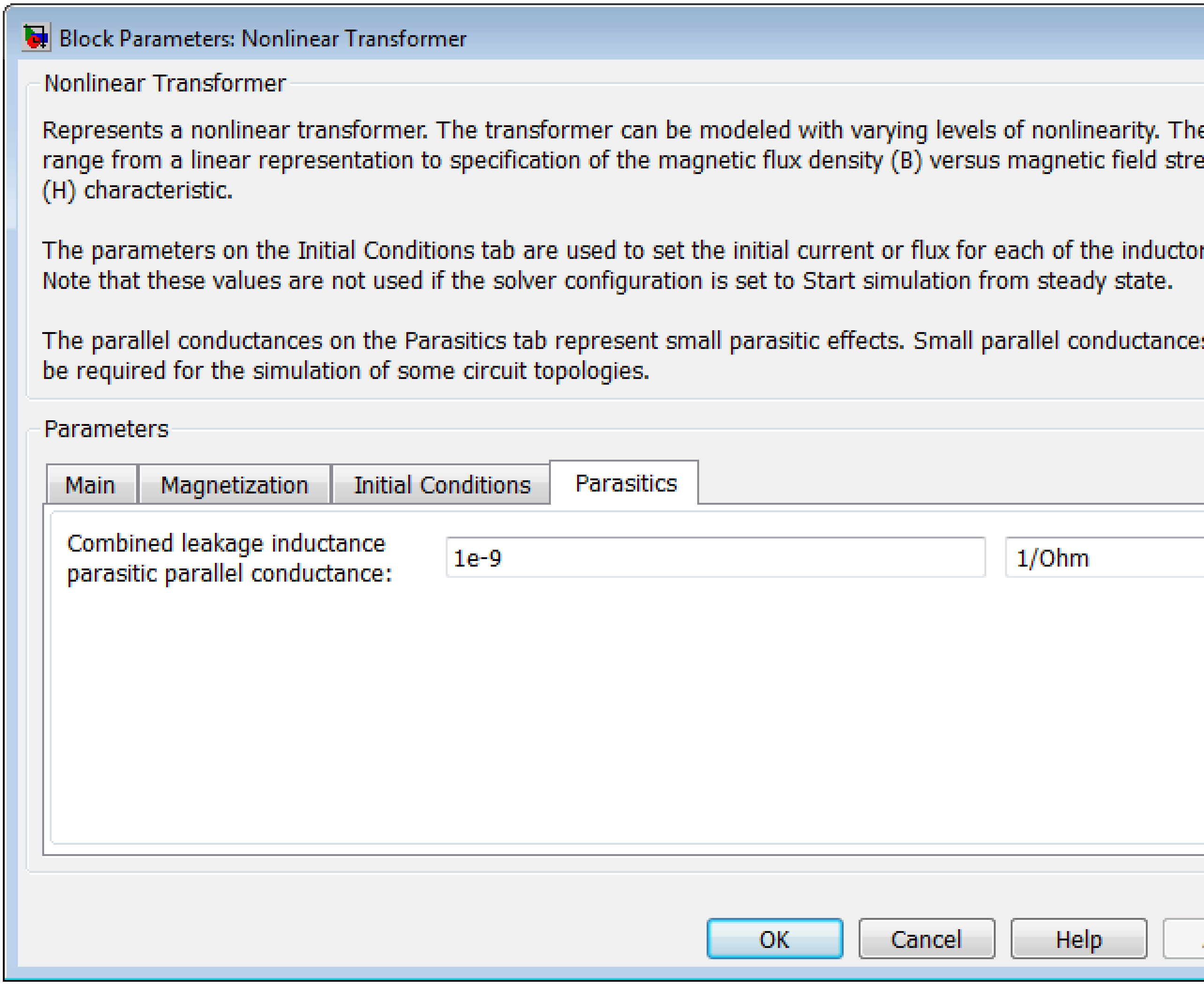
- `Current` — Specify the initial state of the magnetization inductance *Lm* by the initial current. This is the default option.
- `Magnetic flux` — Specify the initial state of the magnetization inductance *Lm* by the magnetic flux.

Magnetization inductance initial current

The initial current value used to calculate the value of magnetic flux within the magnetization inductance *Lm* at time zero. This is the current passing through the magnetization inductance *Lm*. Total magnetization current consists of current passing through the magnetization resistance *Rm* and current passing through the magnetization inductance *Lm*. This parameter is only visible 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 *Lm* at time zero. This parameter is only visible when you select `Magnetic flux` for the **Specify magnetization inductance initial state by** parameter. The default is `0` Wb.

Parasitics Tab



Combined leakage inductance parasitic parallel conductance
Use this parameter to represent small parasitic effects in parallel to the combined leakage inductance *Leq*. A small parallel conductance may be required for the simulation of some circuit topologies. This parameter is only visible 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 only visible 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/Ω.

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 only visible 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/Ω.

Examples

For comparison of nonlinear transformer behavior with different parameterization options, see the Nonlinear Transformer Characteristics example.

Ports

The block has four electrical conserving ports. Polarity is indicated by the `+` and `-` signs.

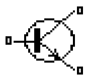
See Also

Simscape Ideal Transformer, Nonlinear Inductor

Purpose 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.
- β_{F} is the ideal maximum forward current gain BF
- β_{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 (β_F and β_R) become a function of temperature according to the following equations:

$$IS_{Ts} = IS_{Tm1} \cdot (T_s / T_{m1})^{XTI} \cdot \exp\left(-\frac{EG}{kT_s}(1 - T_s / T_{m1}) \right)$$

$$\beta_{Fs} = \beta_{Fm1}\left(\frac{T_s}{T_{m1}}\right)^{XTB}$$

$$\beta_{Rs} = \beta_{Rm1}\left(\frac{T_s}{T_{m1}}\right)^{XTB}$$

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_{Tm1} is the saturation current at the measurement temperature.
- IS_{Ts} 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.
- β_{Fm1} and β_{Rm1} are the forward and reverse gains at the measurement temperature.
- β_{Fs} and β_{Rs} 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".

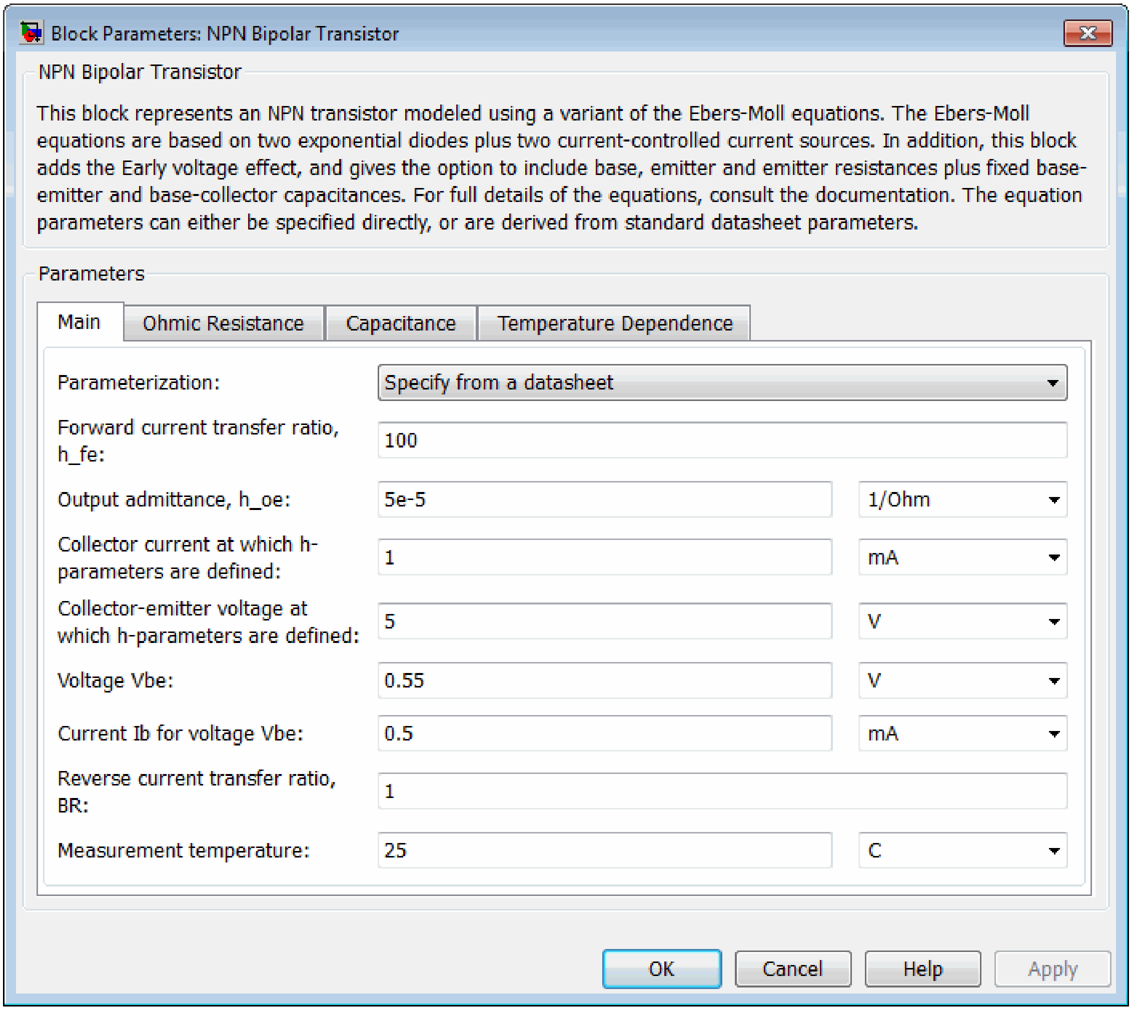
Basic Assumptions and Limitations

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.

Dialog Box and Parameters

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 *Ic/h_oe*, where *Ic* 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 Vbe** value and the corresponding **Current Ib for voltage Vbe** value when *Ic* 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/Ω.

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 Vbe
Base-emitter voltage when the base current is *Ib*. The [*Vbe Ib*] 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 Ib for voltage Vbe
Base current when the base-emitter voltage is *Vbe*. The [*Vbe Ib*] 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 *Ic* versus *Vce* curve is zero in the normal active region. The additional forward Early voltage term increases this gradient. The intercept on the *Vce*-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.

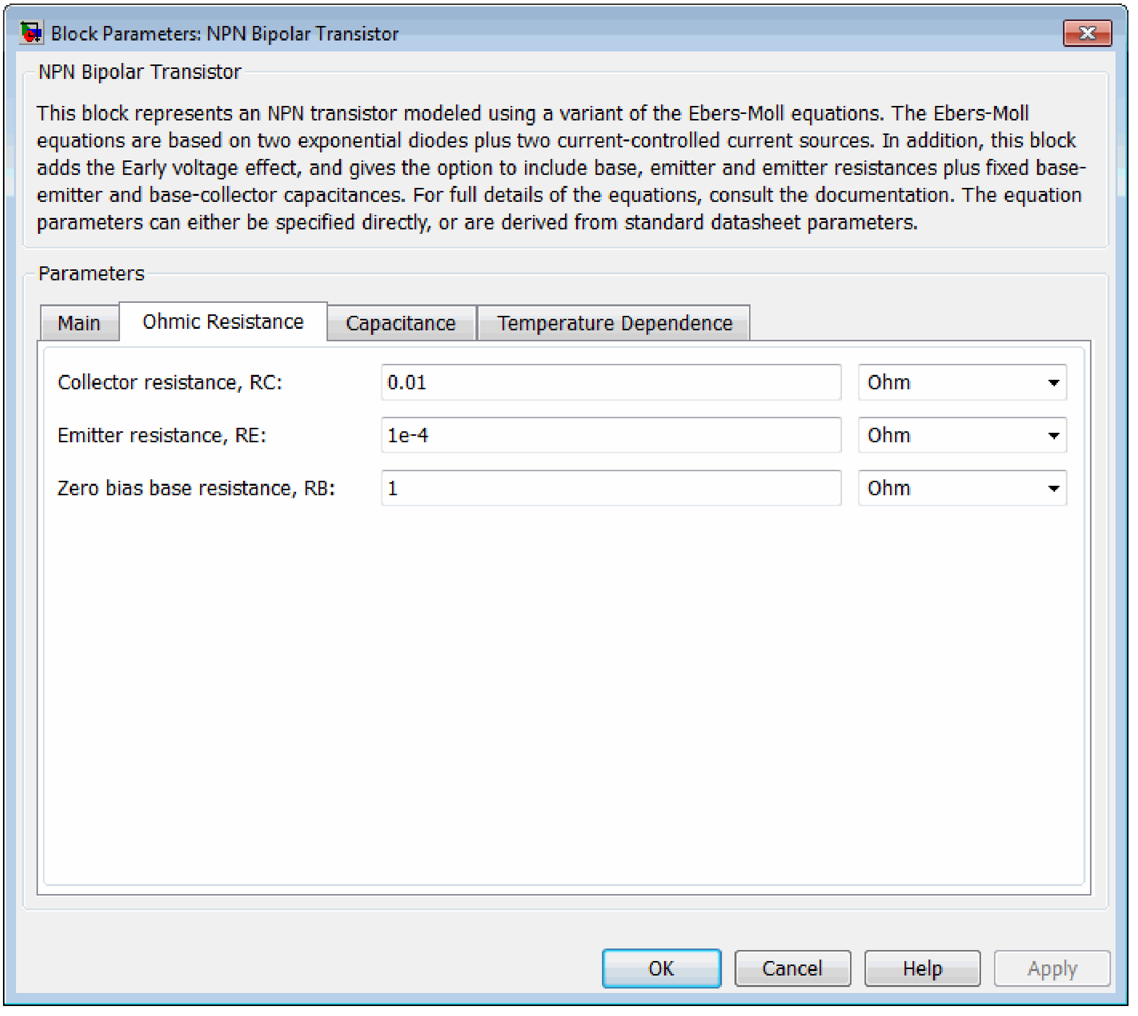
Reverse current transfer ratio BR
Ideal maximum reverse current gain. This value is often not quoted in manufacturer datasheets, because it is not significant when the transistor is biased to operate in the normal active

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 *Vbe* and *Ib*, or *IS*, are measured. The default value is 25 C.

Ohmic Resistance Tab



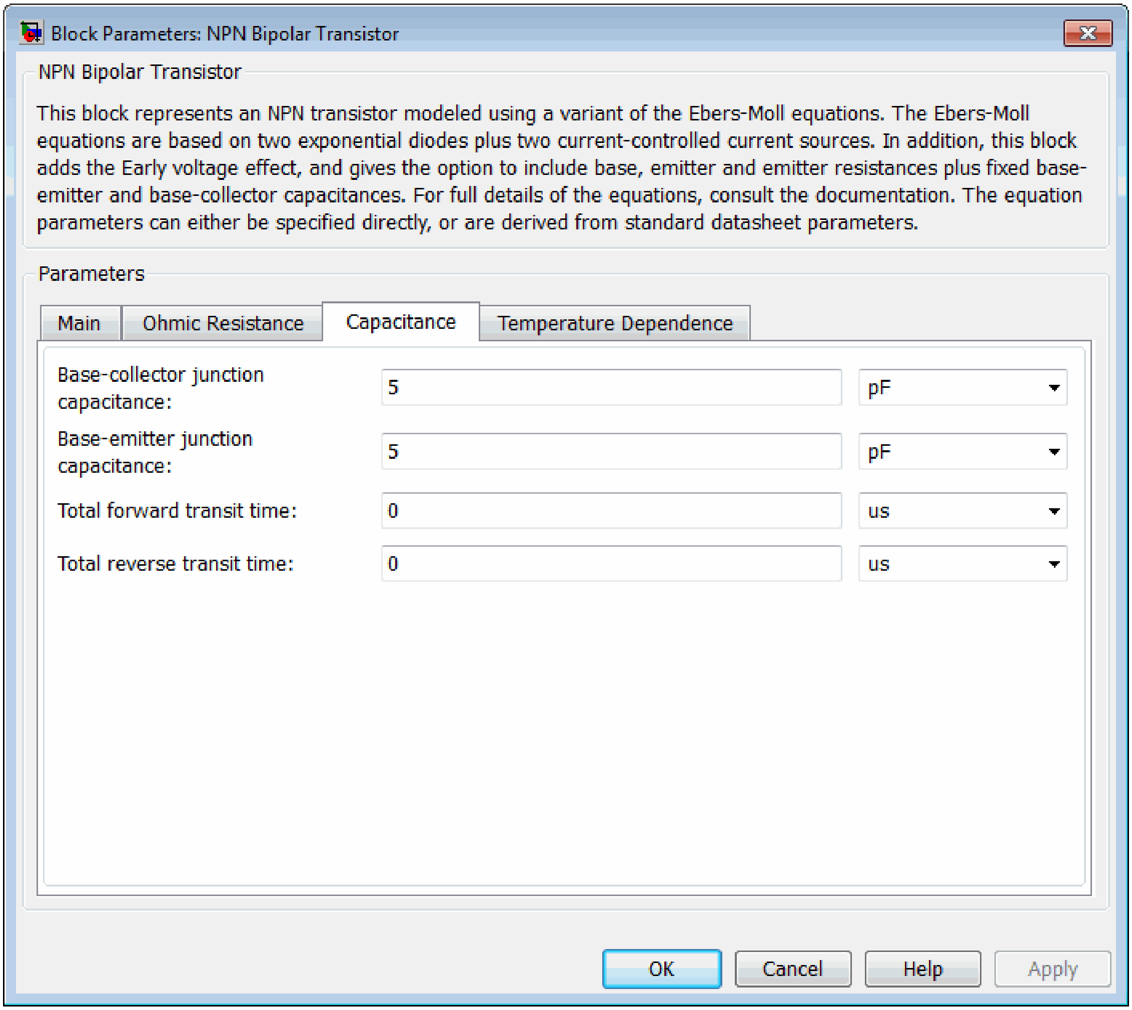
Collector resistance RC

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



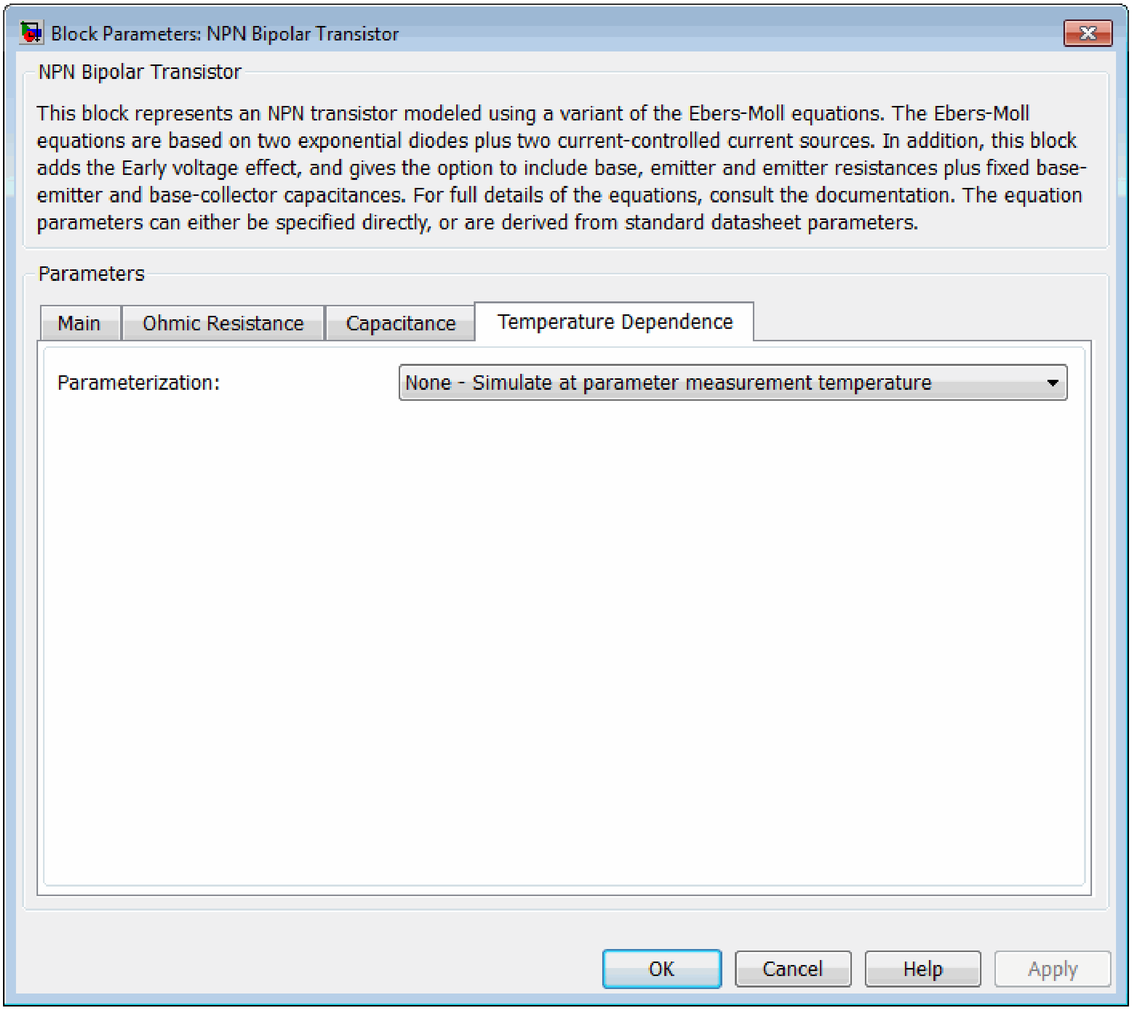
Base-collector junction capacitance
Parasitic capacitance across the base-collector junction. The default value is 5 pF.

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 μs.

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 0μs.

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 Vbe at second measurement temperature

Base-emitter voltage when the base current is *Ib* and the temperature is set to the second measurement temperature. The [*Vbe Ib*] 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 Ib for voltage Vbe at second measurement temperature
Base current when the base-emitter voltage is *Vbe* and the temperature is set to the second measurement temperature. The [*Vbe Ib*] 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*,*Vbe*, and *Ib* 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 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

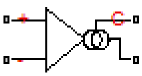
Purpose

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 500µA, a resistor connected between the +15V rail and the control current pin must have a value of (15 - (-14.5)) / 500e-6 = 59kOhm.

Transconductance

The relationship between input voltage, v, and transconductance current, i_{gm}, is:

$$v = v_+ - v_-$$
$$i_{gm} = g_m \cdot v$$
$$g_m = \frac{g_{m0} \cdot i_c}{i_{c0}}$$

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:

$$i_{gm} + i_o = \frac{v_o}{R_{out}}$$
$$R_{out} = \frac{R_{out0} \cdot i_{c0}}{i_c}$$

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:

$$V_{\min} \leq v_{\mathrm{o}} \leq V_{\max}$$

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.

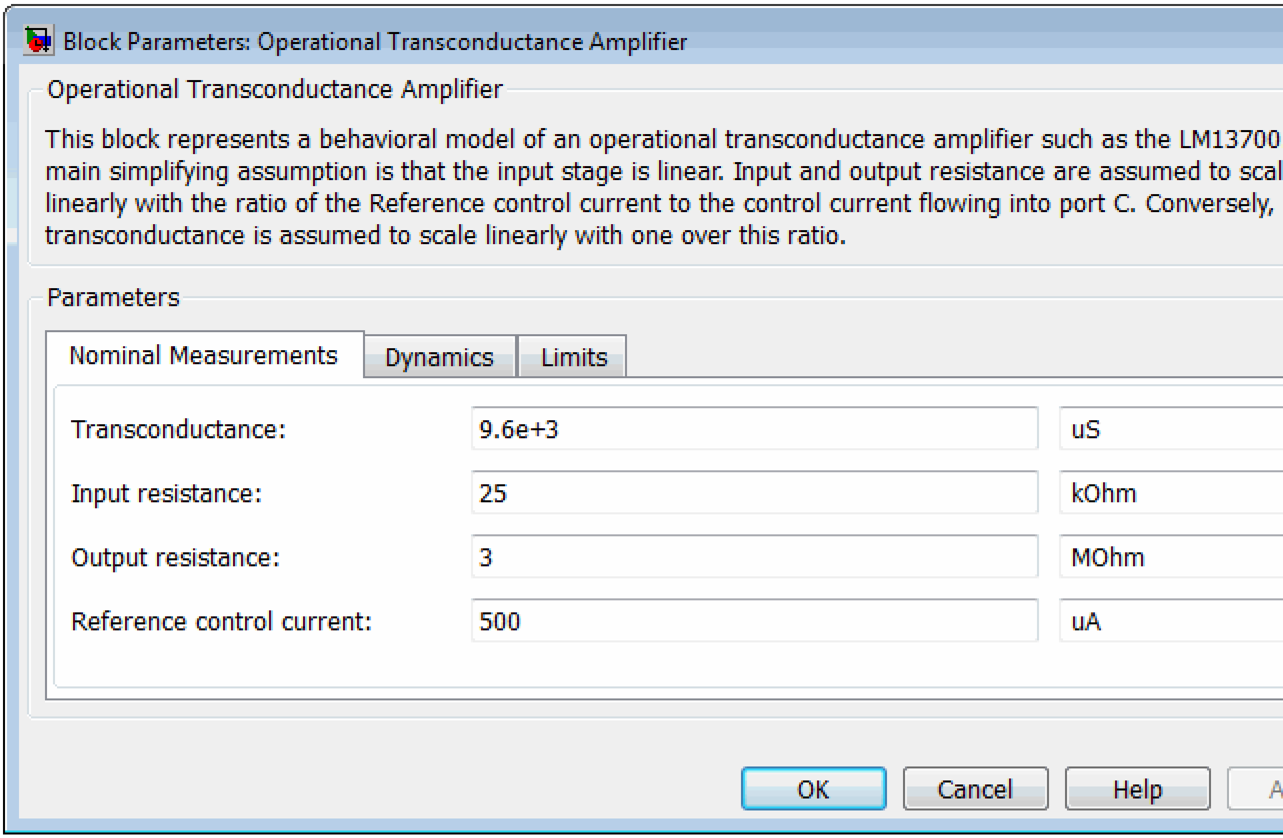
The transconductance current is also slew-rate limited, a value for slew rate limiting typically being given on datasheets:

$$-\mu \leq \frac{di_{gm}}{dt} \leq \mu$$

where μ is the **Maximum current slew rate**.

Dialog Box and Parameters

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 µS.

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.

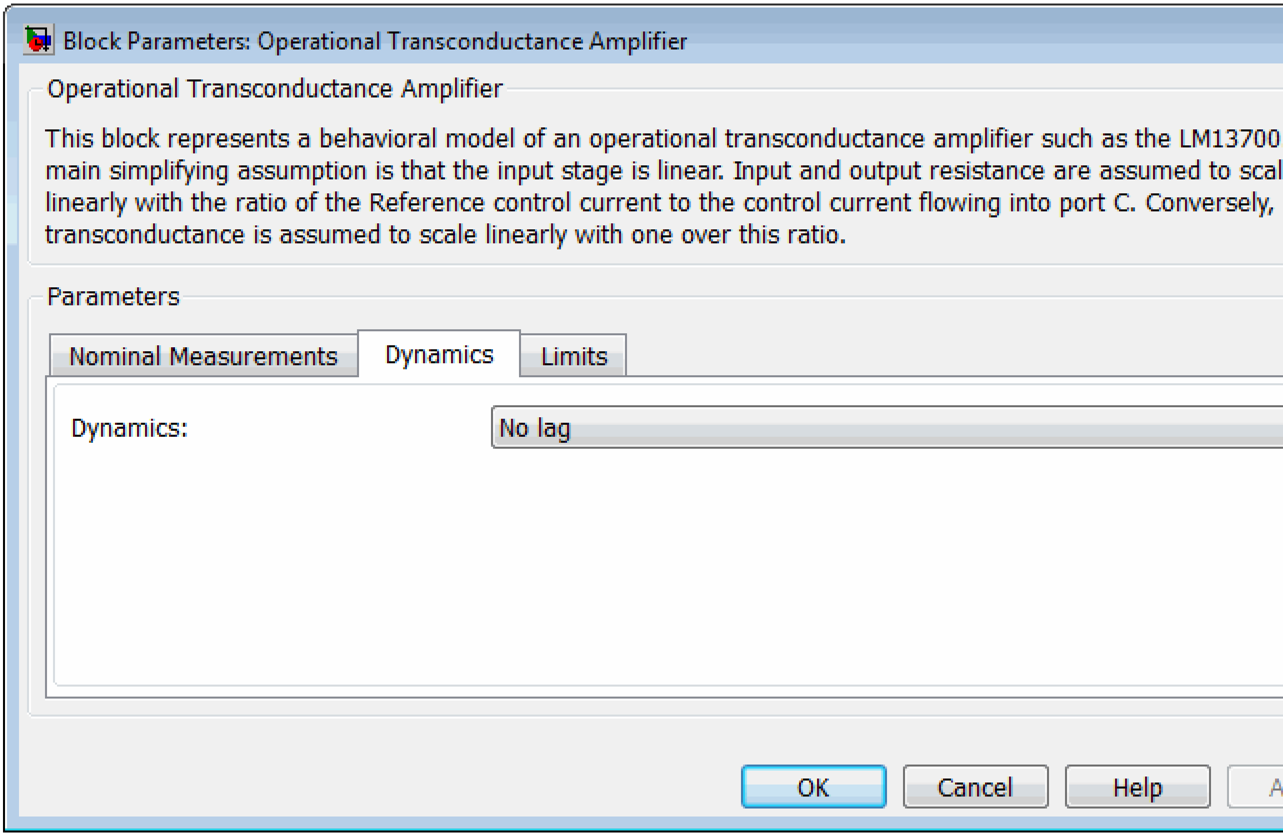
Output resistance

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 µA.

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

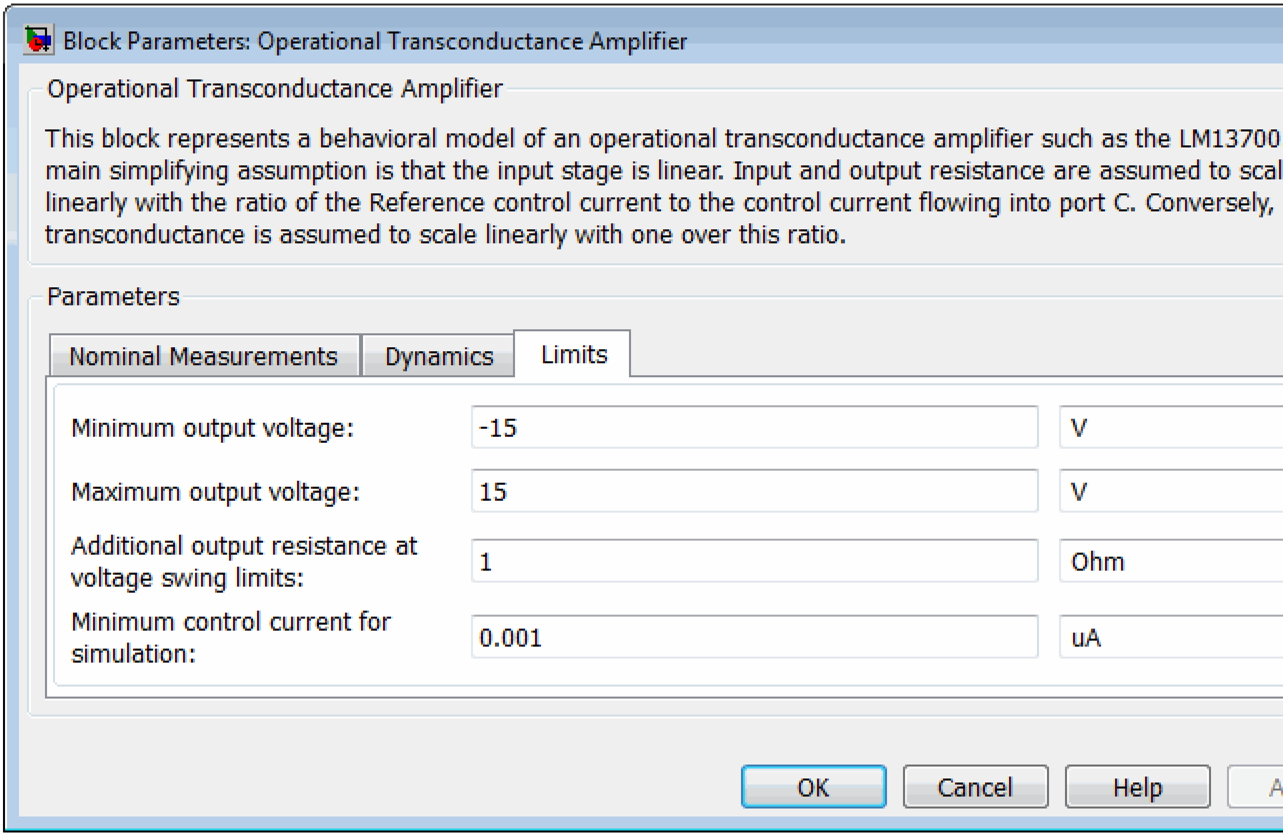
Bandwidth, **Maximum current slew rate**, and **Initial current** parameters appear on the **Dynamics** tab.

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 SimElectronics® 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/μs.

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_{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` μA.

Ports

The block has the following ports:

`+`
: Positive electrical voltage.

`-`
: Negative electrical voltage.

`C`
: Control current.

`OUT`
: Output current.

See Also

Simscape Op-Amp, Band-Limited Op-Amp, Finite-Gain Op-Amp

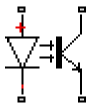
Purpose

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".

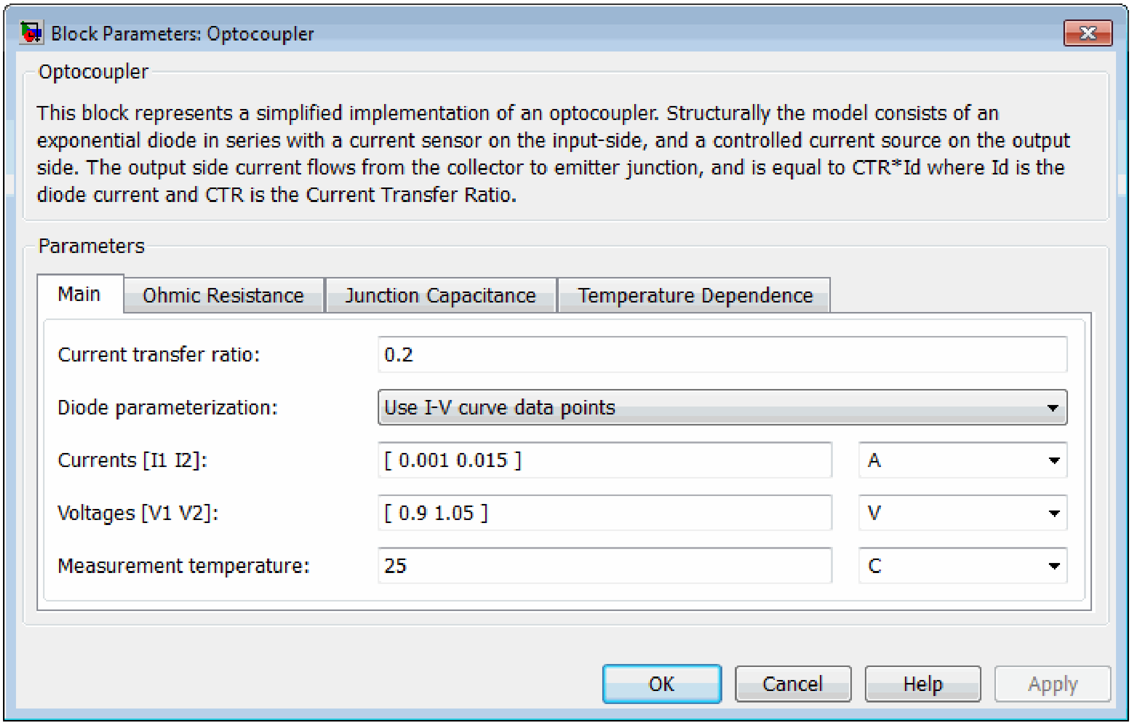
Basic Assumptions and Limitations

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.

Dialog Box and Parameters

Main Tab



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.

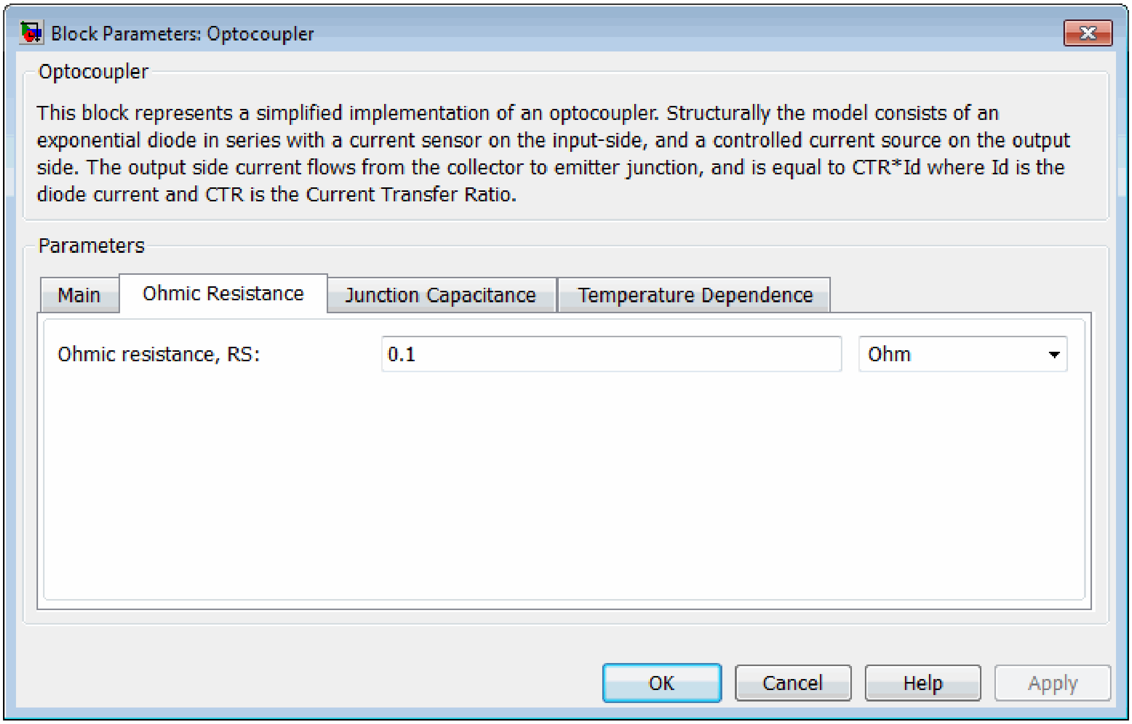
Voltages [V1 V2]
: A vector of the voltage values at the two points on the diode I-V curve that the block uses to calculate *IS* and *N*. This parameter is only visible when you select `Use I-V curve data points` for the **Diode parameterization** parameter. The default value is `[ 0.9 1.05 ]` V.

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 `Use parameters IS and N` for the **Diode parameterization** parameter. The default value is `1e-10` A.

Measurement temperature
: The temperature at which IS 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 IS and N` for the **Diode parameterization** parameter. The default value is `2`.

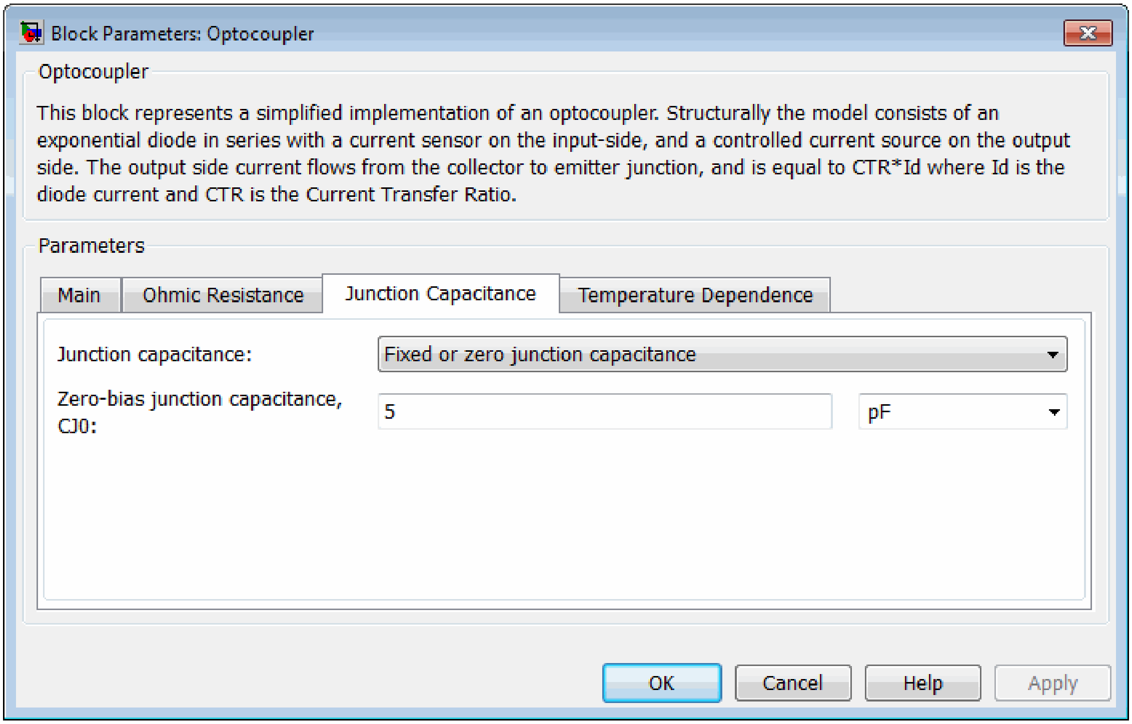
Ohmic Resistance Tab



Ohmic resistance RS

The series diode connection resistance. The default value is `0.1` Ω.

Junction Capacitance Tab



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 CJO, 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 `5` 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.

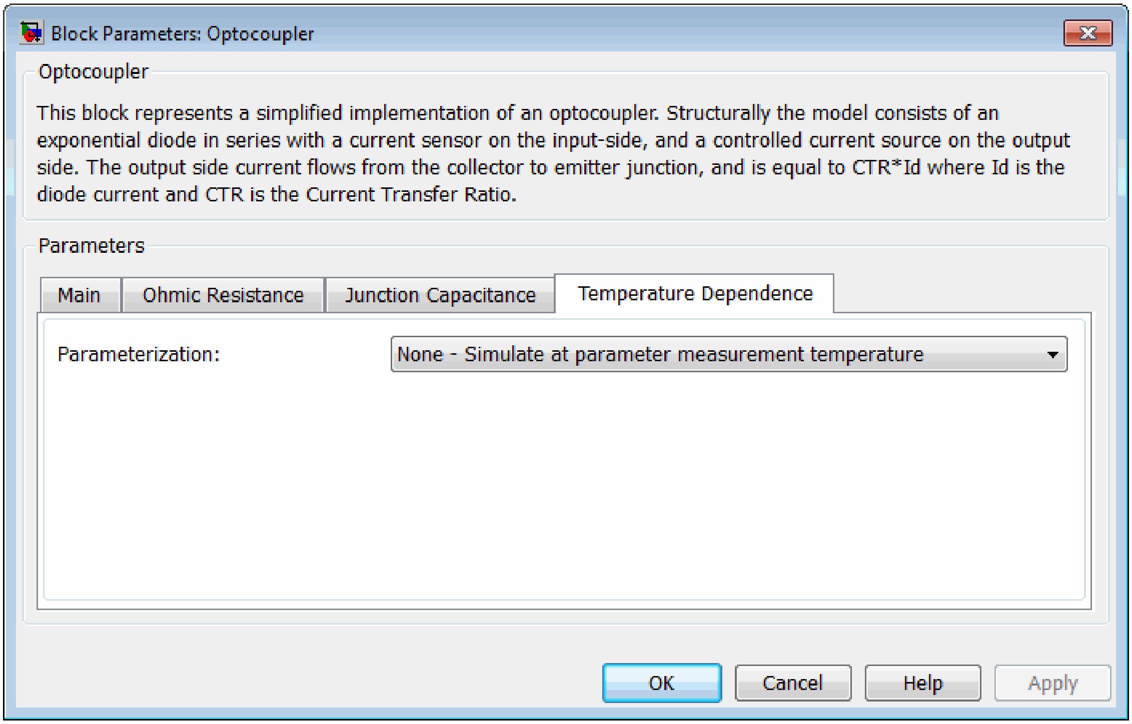
Grading coefficient M
The coefficient that quantifies the grading of the junction. This parameter is only visible when you select `Use parameters CJ0, VJ, M & FC` for the **Junction capacitance** parameter. The default value is `0.5`.

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 `[ 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 CJ0, VJ, M & FC` for the **Junction capacitance** parameter. The default value is `0.5`.

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.
- `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

Diode, NPN Bipolar Transistor, Simscape Controlled Current Source

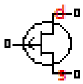
Purpose

Model P-Channel JFET

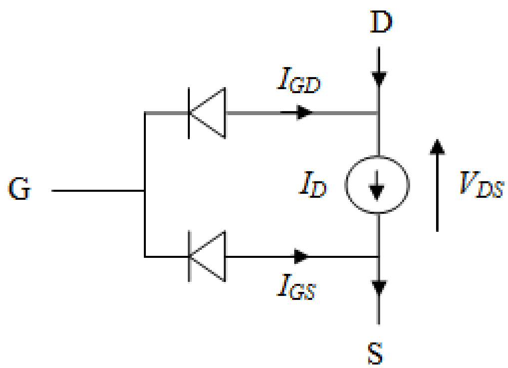
Library

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	$-V_{GS} < -V_{t0}$	$I_D = 0$
Linear	$0 < -V_{DS} < -V_{GS} + V_{t0}$	$I_{ds} = \beta V_{ds}\left(2(-V_{gs} + V_{t0}) + V_{ds}\right)(1 - \lambda V_{ds})$ $I_D = \beta V_{DS}(2(-V_{GS} + V_{t0}) + V_{DS})(1 - \lambda V_{DS})$
Saturated	$0 < -V_{GS} + V_{t0} < -V_{DS}$	$I_{ds} = -\beta(-V_{gs} + V_{t0})^2(1 - \lambda V_{ds})$ $I_D = -\beta (-V_{GS} + V_{t0})^2 (1 - \lambda V_{DS})$

- 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	$-V_{GD} < -V_{t0}$	$I_D = 0$
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	$0 < -V_{GD} + V_{t0} < V_{DS}$	$I_D = \beta (-V_{GD} + V_{t0})^2 (1 + \lambda V_{DS})$

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.

- β is the transconductance parameter. If you select `Specify using equation parameters directly` for the **Parameterization** parameter, β is the **Transconductance parameter** parameter value. Otherwise, the block calculates β from the datasheet parameters you specify.
- λ is the channel-length modulation parameter. If you select `Specify using equation parameters directly` for the **Parameterization** parameter, λ is the **Channel-length modulation** parameter value. Otherwise, the block calculates λ 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, β, and the threshold voltage, V_{t0}, become a function of temperature according to the following equations:

$$\beta_{Ts} = \beta_{Tm1}\left(\frac{T_s}{T_{m1}}\right)^{BEX}$$

$$V_{\mathrm{t0s}} = V_{\mathrm{t01}} + \alpha\,(T_{\mathrm{s}} - T_{\mathrm{m1}})$$

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.
- β_{Tm1} is JFET transconductance at the measurement temperature.

- β_{Ts} 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.
- a 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 a, 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 a:

- 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 a based on this data.
- If you parameterize by specifying equation parameters, you have to provide the value for a 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 a and *BEX*.

In addition, the saturation current term, *IS*, in the gate-drain and gate-source current equations depends on temperature

$$IS_{Ts} = IS_{Tm1} \cdot (T_s / T_{m1})^{XTI} \cdot \exp\left(-\frac{EG}{kT_s}(1 - T_s / T_{m1}) \right)$$

where:

- IS_{Tm1} is the saturation current at the measurement temperature.
- IS_{Ts} 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 *a*, 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”.

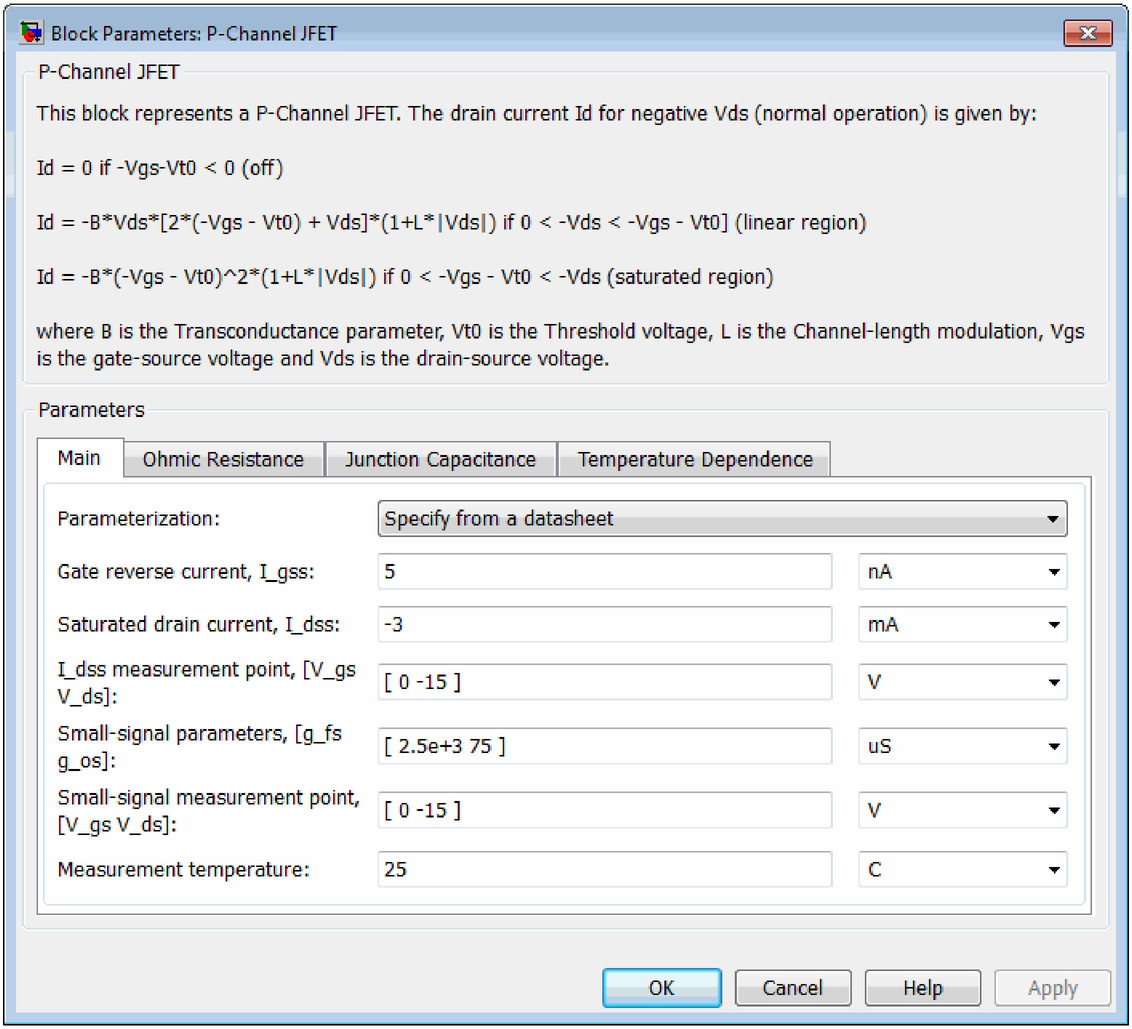
Basic Assumptions and Limitations

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 a 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.

Dialog Box and Parameters

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 β, IS, V_{t0}, and λ.

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_gs V_ds]

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_fs g_os]

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_gs V_ds]
: 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^2.

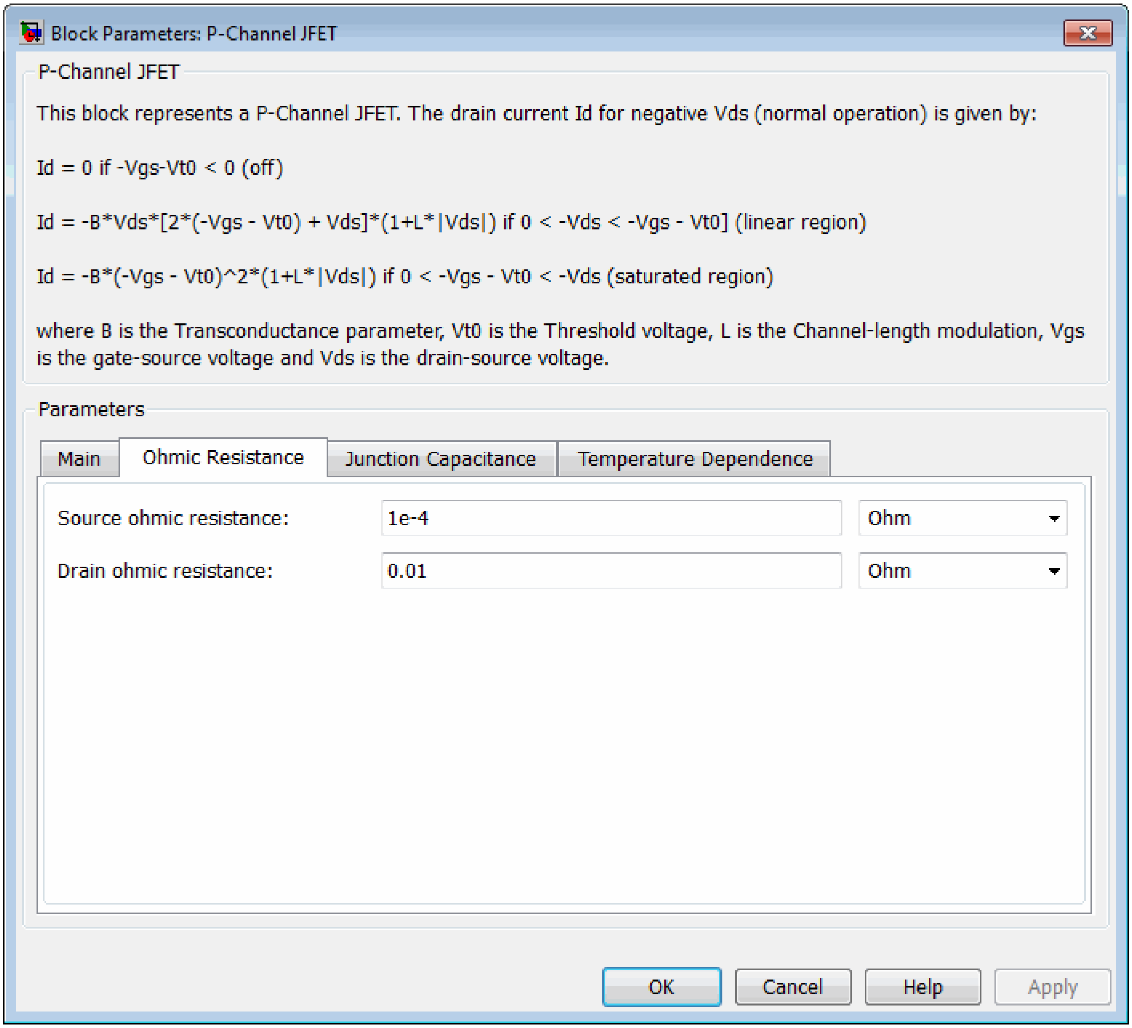
Saturation current
: The magnitude of the current that the ideal diode equation approaches asymptotically for very large reverse bias levels. This parameter is only visible when you select `Specify using equation parameters directly` for the **Parameterization** parameter. The default value is `1e-14` A.

Threshold voltage
: The gate-source voltage above which the transistor produces a nonzero drain current. For an enhancement device, Vt0 should be negative. For a depletion mode device, Vt0 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



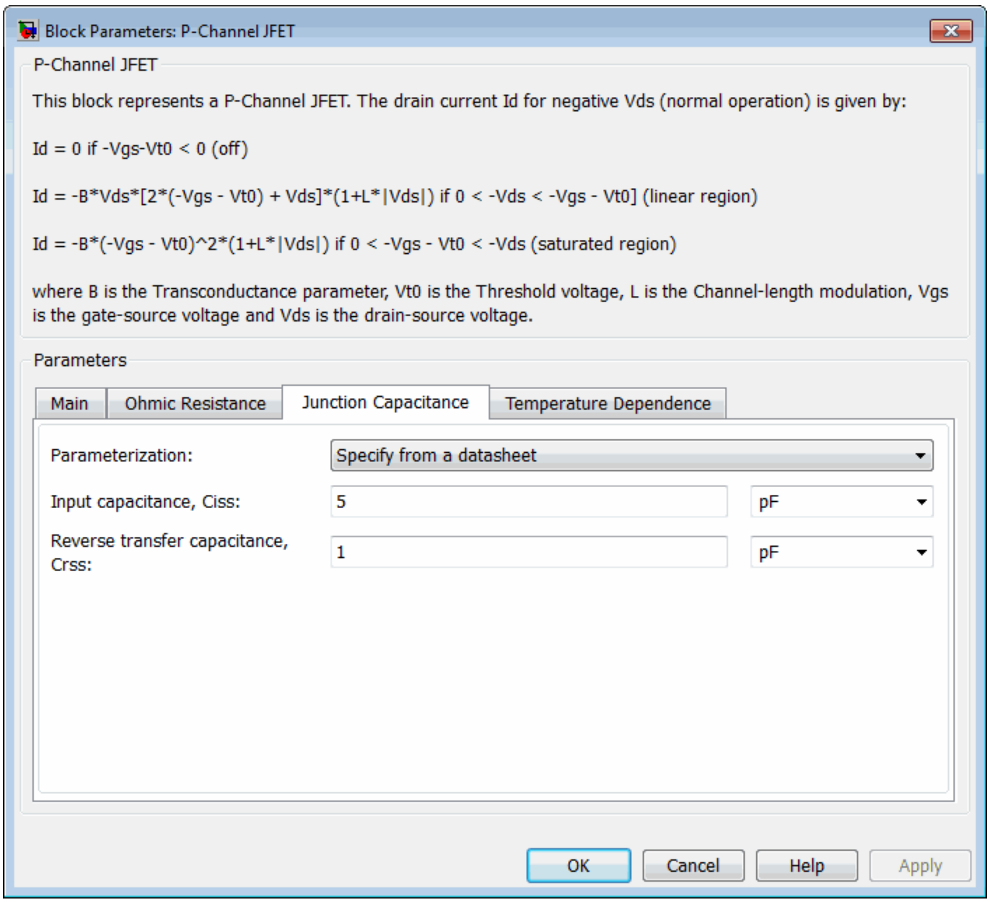
Source ohmic resistance

The transistor source resistance. The default value is `1e-4` Ω. The value must be greater than or equal to `0`.

Drain ohmic resistance

The transistor drain resistance. The default value is `0.01` Ω. 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.

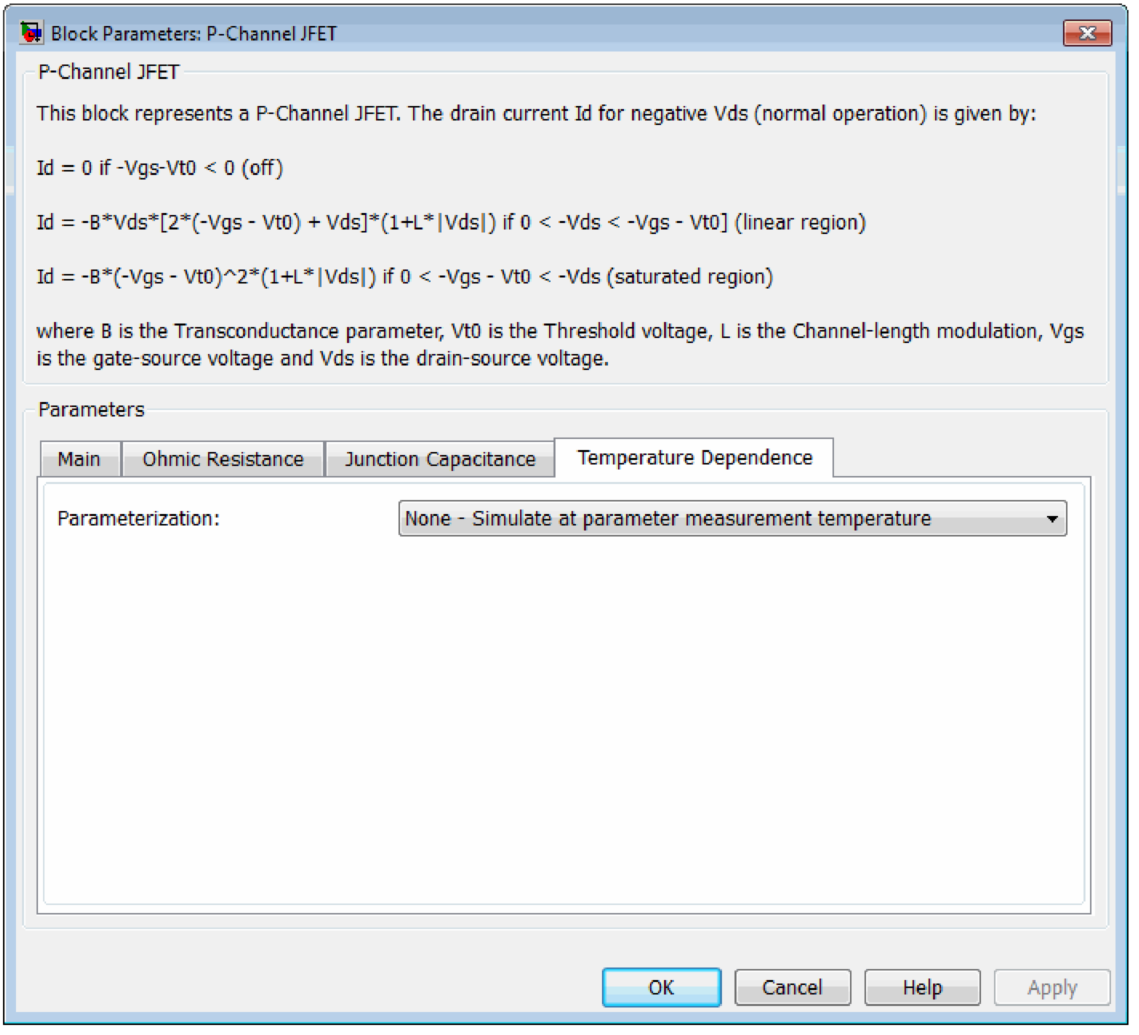
Input capacitance, Ciss
The gate-source capacitance with the drain shorted to the source. This parameter is only visible when you select `Specify from a datasheet` for the **Model junction capacitance** parameter. The default value is `4.5` pF.

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_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 `-2.3` 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, 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 1-389 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 MOSFET

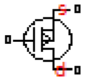
Purpose

Model P-Channel MOSFET using Shichman-Hodges equation

Library

Semiconductor Devices

Description



The P-Channel MOSFET 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:

$$I_{DS} = -K\left((V_{GS} - V_{th})V_{DS} - {V_{DS}}^2/2\right)\left(1 + \lambda\,|V_{DS}|\right)$$

- In the saturated region ($0 < -V_{GS} + V_{th} < -V_{DS}$) the drain-source current is:

$$I_{DS} = -(K/2)(V_{GS} - V_{th})^2\left(1 + \lambda\,|V_{DS}|\right)$$

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.
- λ is the channel modulation.

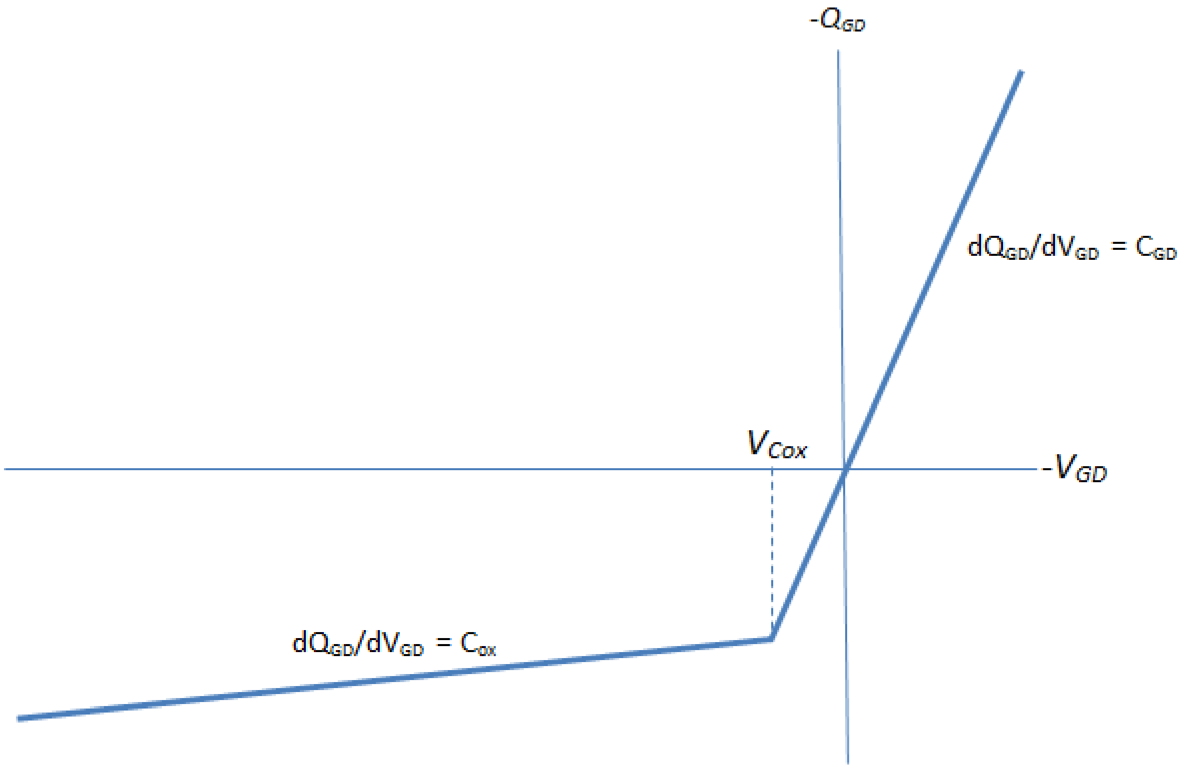
Charge Model

The block models 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 `Specify using equation parameters directly` for the **Parameterization** parameter in the **Junction Capacitance** tab, you specify the **Gate-drain junction capacitance** and **Gate-source junction capacitance** parameters directly. Otherwise, the block derives them from the **Input capacitance Ciss** and **Reverse transfer capacitance Crss** parameter values. The two parameterizations are related as follows:

- $C_{GD} = Crss$
- $C_{GS} = Ciss - Crss$

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.

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 gain, K, and the threshold voltage, V_{th}, become a function of temperature according to the following equations:

$$K_{Ts} = K_{Tm1}\left(\frac{T_s}{T_{m1}}\right)^{BEX}$$

$$V_{ths} = V_{th1} + a\,(T_s - T_{m1})$$

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_{Tm1} is the transistor gain at the measurement temperature.
- K_{Ts} 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_{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.
- a 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 a, 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 a:

- 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 a based on this data.
- If you parameterize by specifying equation parameters, you have to provide the value for a 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 a and *BEX*.

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

When modeling temperature dependence, consider the following:

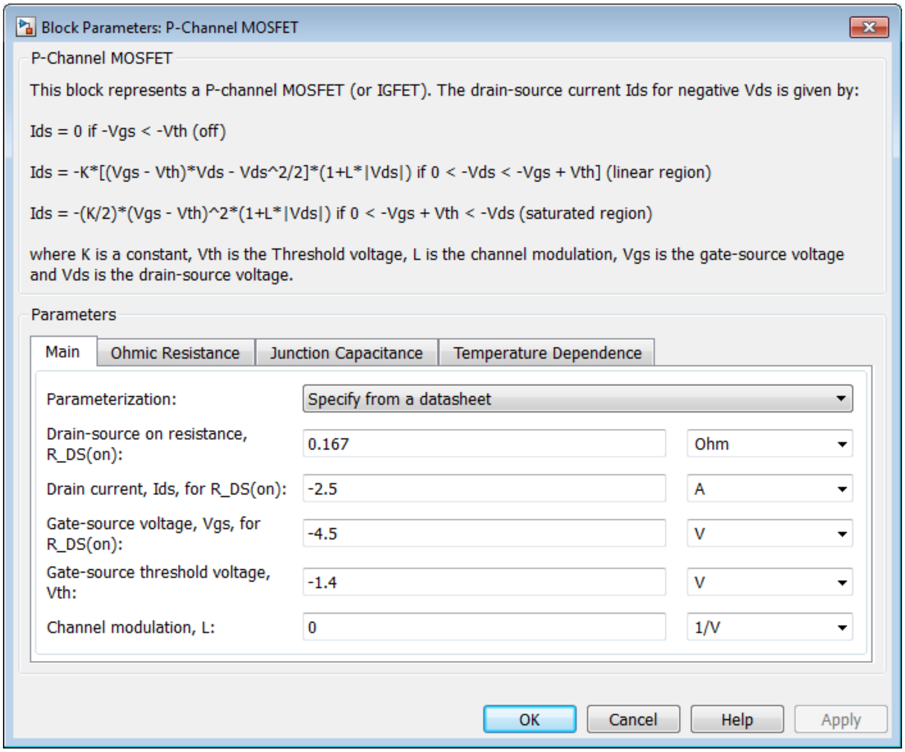
- The block does not account for temperature-dependent effects on the junction capacitances.
- When you specify $R_{DS}(on)$ at a second measurement temperature, it must be quoted for the same working point (that is, the same drain current and gate-source voltage) as for the other $R_{DS}(on)$ value.

Inconsistent values for $R_{DS}(on)$ at the higher temperature will result in unphysical values for a 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 V_{DS}-V_{GS} relationship (if available) for a given device. Increasing V_{th} moves the V_{DS}-V_{GS} plots to the right. The value of *BEX* affects whether the V_{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*.

Dialog Box and Parameters

Main Tab



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` Ω.

Drain current, Ids, 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` A.

Gate-source voltage, Vgs, 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` 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` A/V^2.

Gate-source threshold voltage Vth
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` V.

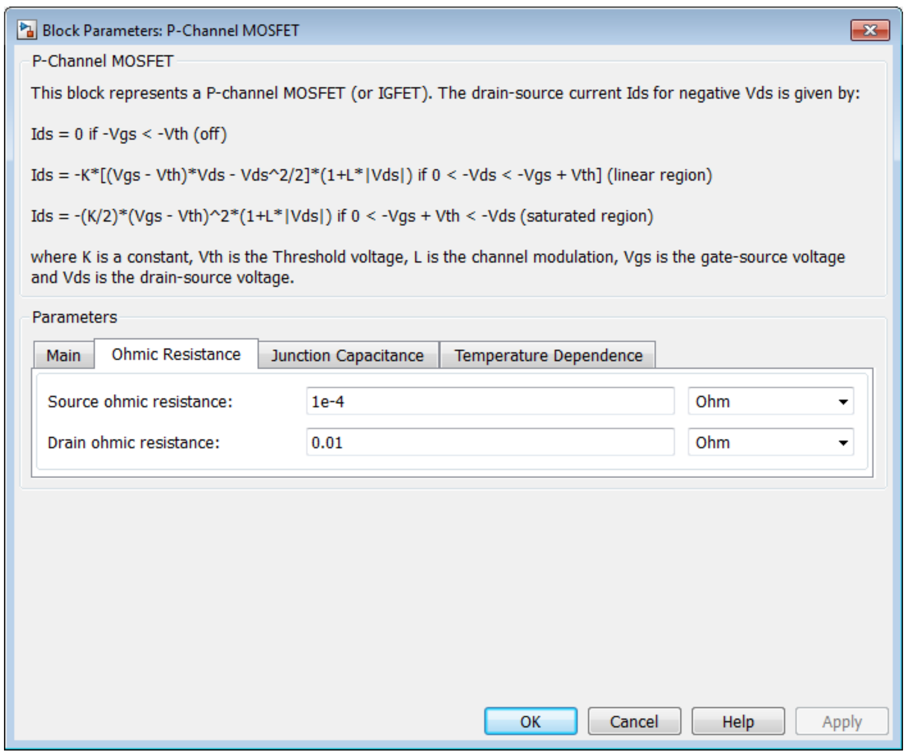
Channel modulation, L
The channel-length modulation, usually denoted by the mathematical symbol λ. When in the saturated region, it is minus the rate of change of drain current with drain-source voltage. The effect on drain current is typically small, and the effect is neglected if calculating transistor 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` 1/V.

Measurement temperature

Temperature T_{m1} at which **Drain-source on resistance, R_DS(on)** is measured. This parameter is only visible when you select `Model temperature dependence` for the **Parameterization** parameter on the **Temperature Dependence** tab. The default value is `25` C.

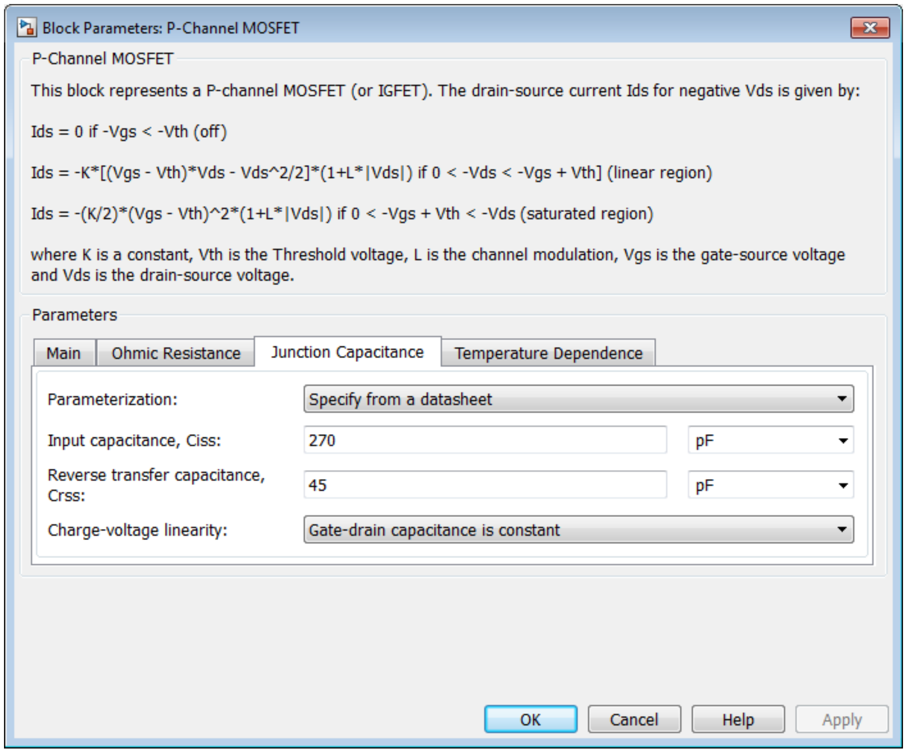
Ohmic Resistance Tab



Source ohmic resistance
The transistor source resistance. The default value is 1e-4 Ω. The value must be greater than or equal to 0.

Drain ohmic resistance
The transistor drain resistance. The default value is 0.001 Ω. The value must be greater than or equal to 0.

Junction Capacitance Tab



Parameterization
Select one of the following methods for capacitance 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 **Parameterization** parameter. The default value is `270` 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 **Parameterization** parameter. The default value is `45` 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 **Parameterization** parameter. The default value is `225` 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 **Parameterization** parameter. The default value is `45` pF.

Charge-voltage linearity
: Select whether gate-drain capacitance is fixed or nonlinear:

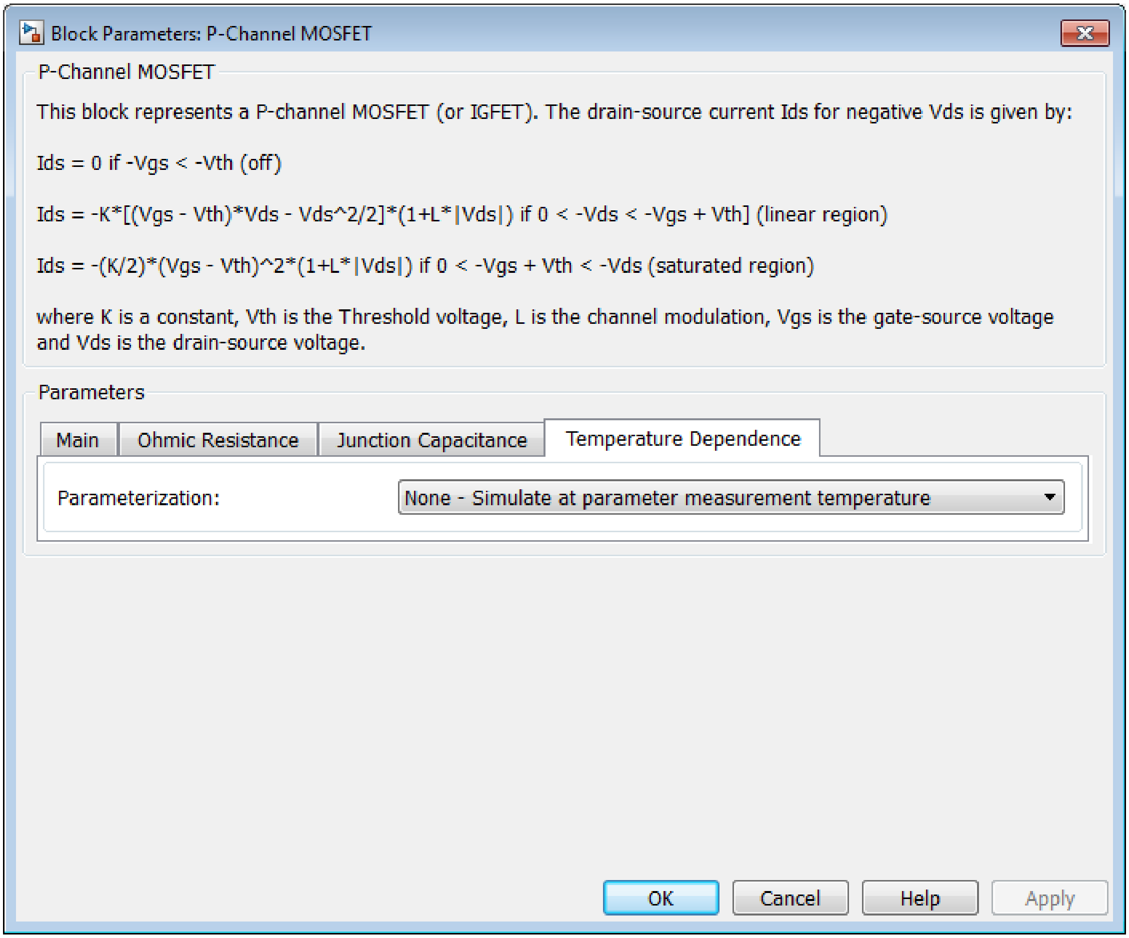
- `Gate-drain capacitance is constant` — The capacitance value is constant and defined according to the selected parameterization option, either directly or derived from a datasheet. This is the default method.
- `Gate-drain charge function is nonlinear` — The gate-drain charge relationship is defined according to the

piecewise-nonlinear function described in "Charge Model" on page 1-401. Two additional parameters appear to let you define the gate-drain charge function.

Gate-drain oxide capacitance
The gate-drain capacitance when the device is on and the drain-gate voltage is small. 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.

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 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 a 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 a based on that. If you parameterize by specifying equation parameters, you have to provide the value for a 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` Ω.

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` 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` 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 1-404 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.

See Also

N-Channel MOSFET

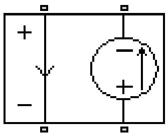
Purpose

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:

$$I_{out} = p(0) + p(1) * I_{in} + \ldots + p(n-1) * I_{in}^{n-1} + p(n) * I_{in}^{n}$$

- If you specify a scalar coefficient for the **Polynomial coefficients** parameter:

$$I_{out} = p * I_{in}$$

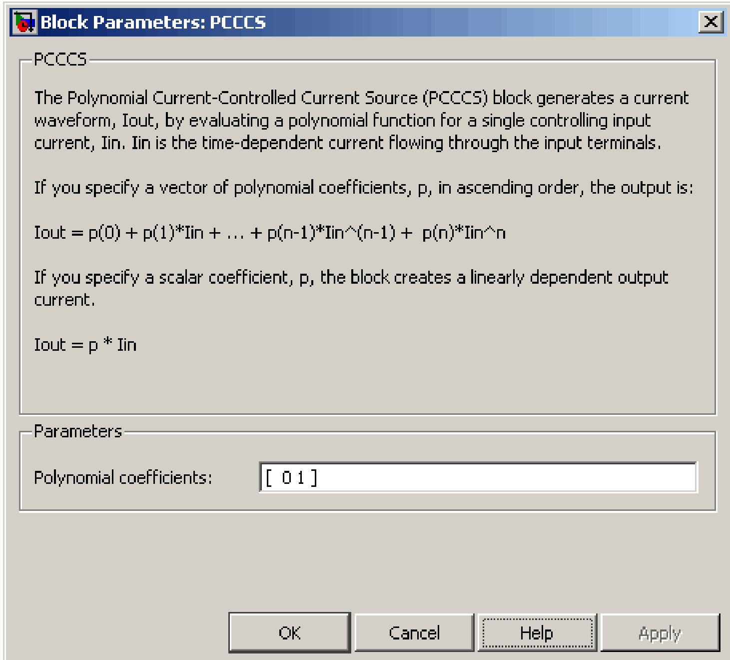
where:

- 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.

Dialog Box and 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

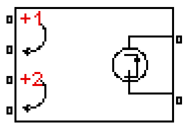
Purpose

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:

$$I_{out} = p_1 + p_2 * I_{in1} + p_3 * I_{in2} + p_4 * I_{in1}^2 + p_5 I_{in1} * I_{in2} + p_6 * I_{in2}^2 + \ldots$$

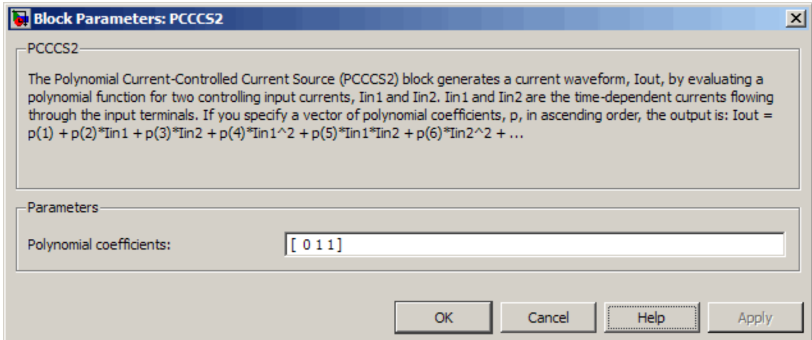
where:

- 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.

Dialog Box and 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.

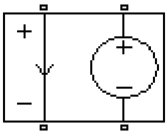
See Also

PCCCS, PCCVS2, PVCCS2, and PVCVS2

Purpose 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:

$$V_{out} = p(0) + p(1) * I_{in} + \ldots + p(n-1) * I_{in}^{n-1} + p(n) * I_{in}^{n}$$

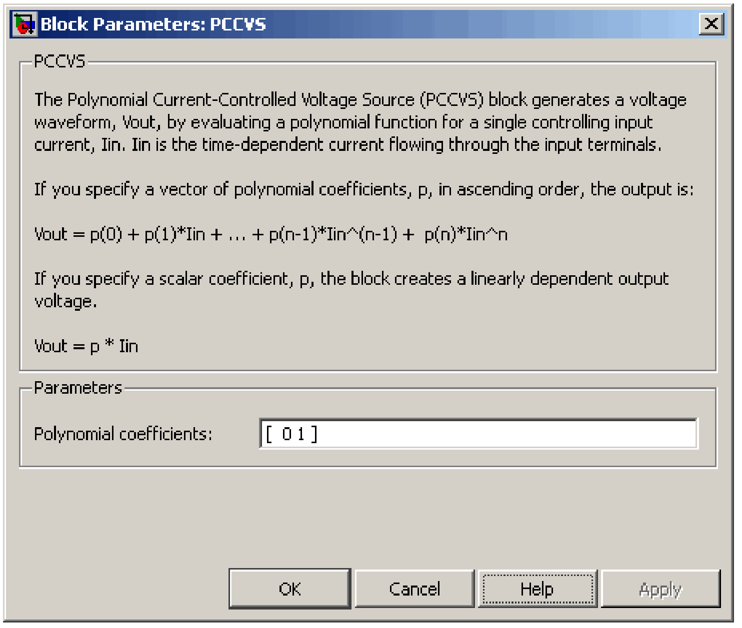
- If you specify a scalar coefficient for the **Polynomial coefficients** parameter:

$$V_{out} = p * I_{in}$$

where:

- I_{in} is the current through the input ports.
- p is the **Polynomial coefficients** parameter value.

Dialog Box and 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

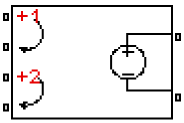
Purpose

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:

$$V_{out} = p_1 + p_2 * I_{in1} + p_3 * I_{in2} + p_4 * I_{in1}^2 + p_5 I_{in1} * I_{in2} + p_6 * I_{in2}^2 + \ldots$$

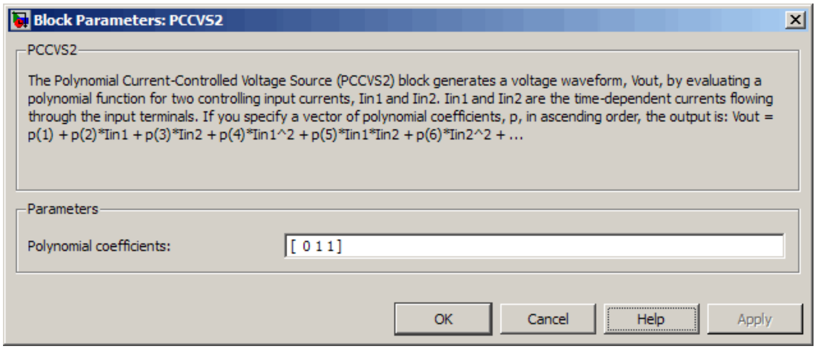
where:

- 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.

Dialog Box and 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

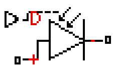
Purpose

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:

$$I_{\mathrm{p}} = DeviceSensitivity \cdot RadiantFluxDensity$$

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:

  - $CJ0 = C_1((V_{R2} - V_{R1})/(V_{R2} - V_{R1}(C_2/C_1)^{-1/M}))^M$
  - $VJ = -(-V_{R2}(C_1/C_2)^{-1/M} + V_{R1})/(1-(C_1/C_2)^{-1/M})$
  - $M = \log(C_3/C_2)/\log(V_{R2}/V_{R3})$

  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 *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".

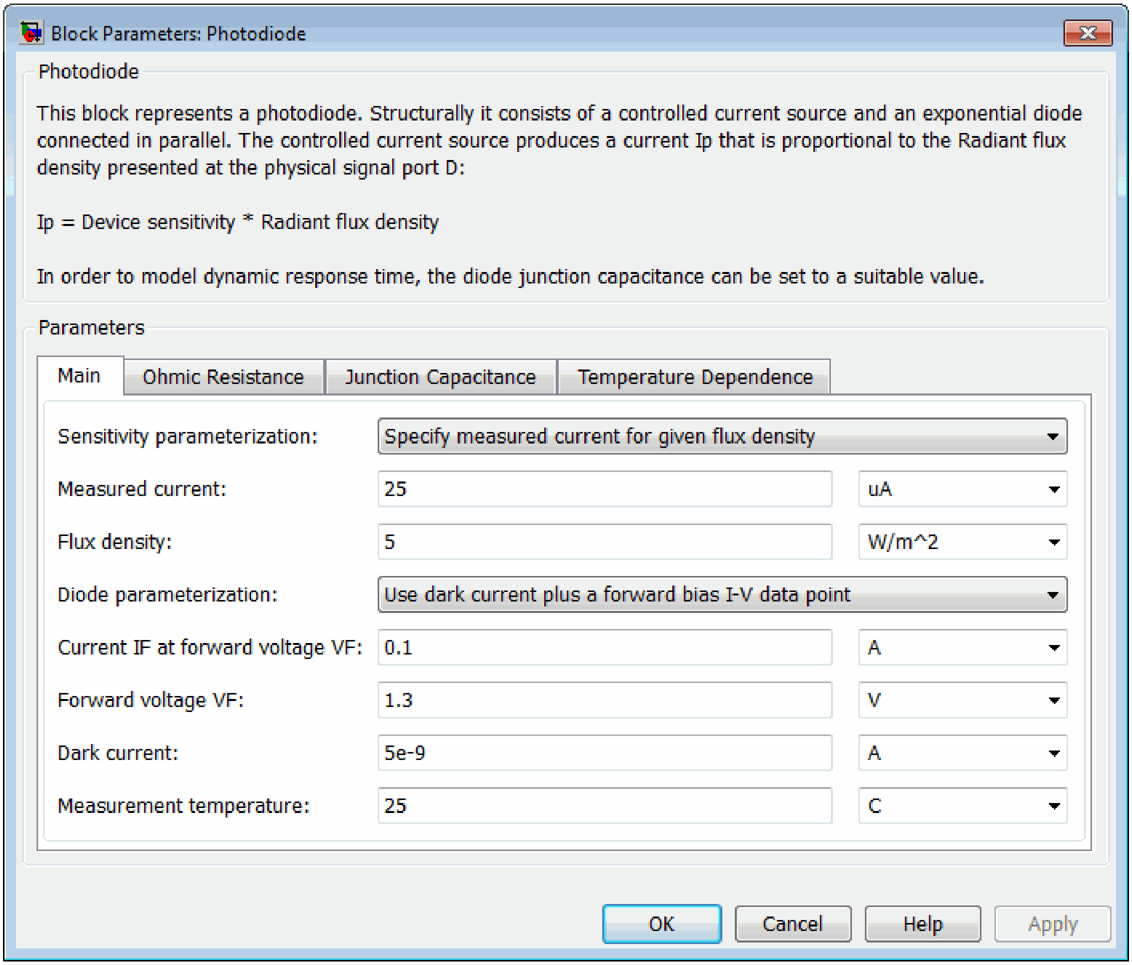
Basic Assumptions and Limitations

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.

Dialog Box and Parameters

Main Tab



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` μA.

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` W/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 `5e-06` m^2*A/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 IF at forward voltage VF
The current at the forward-biased point on the diode I-V curve that the block uses to calculate *IS* 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.

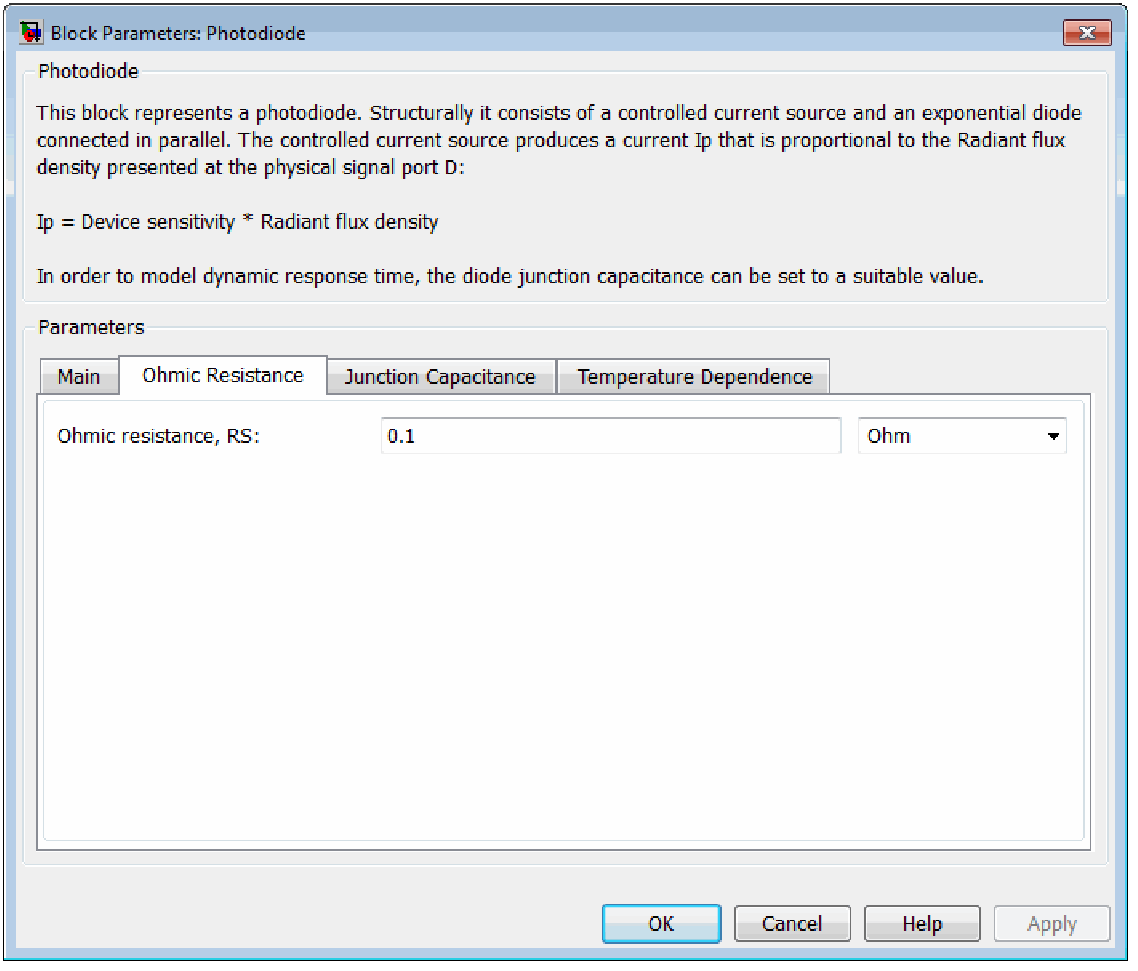
Forward voltage VF
The corresponding voltage at the forward-biased point on the diode I-V curve that the block uses to calculate *IS* and *N*. This parameter is only visible when you select and `Use dark current plus a forward bias I-V data point` for the **Diode parameterization** parameter. The default value is `1.3` V.

Dark current
The current through the diode when it is not exposed to light. The default value is `5e-9` A.

Measurement temperature
The temperature at which the I-V curve or dark current 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 dark current and N` for the **Diode parameterization** parameter. The default value is `3`.

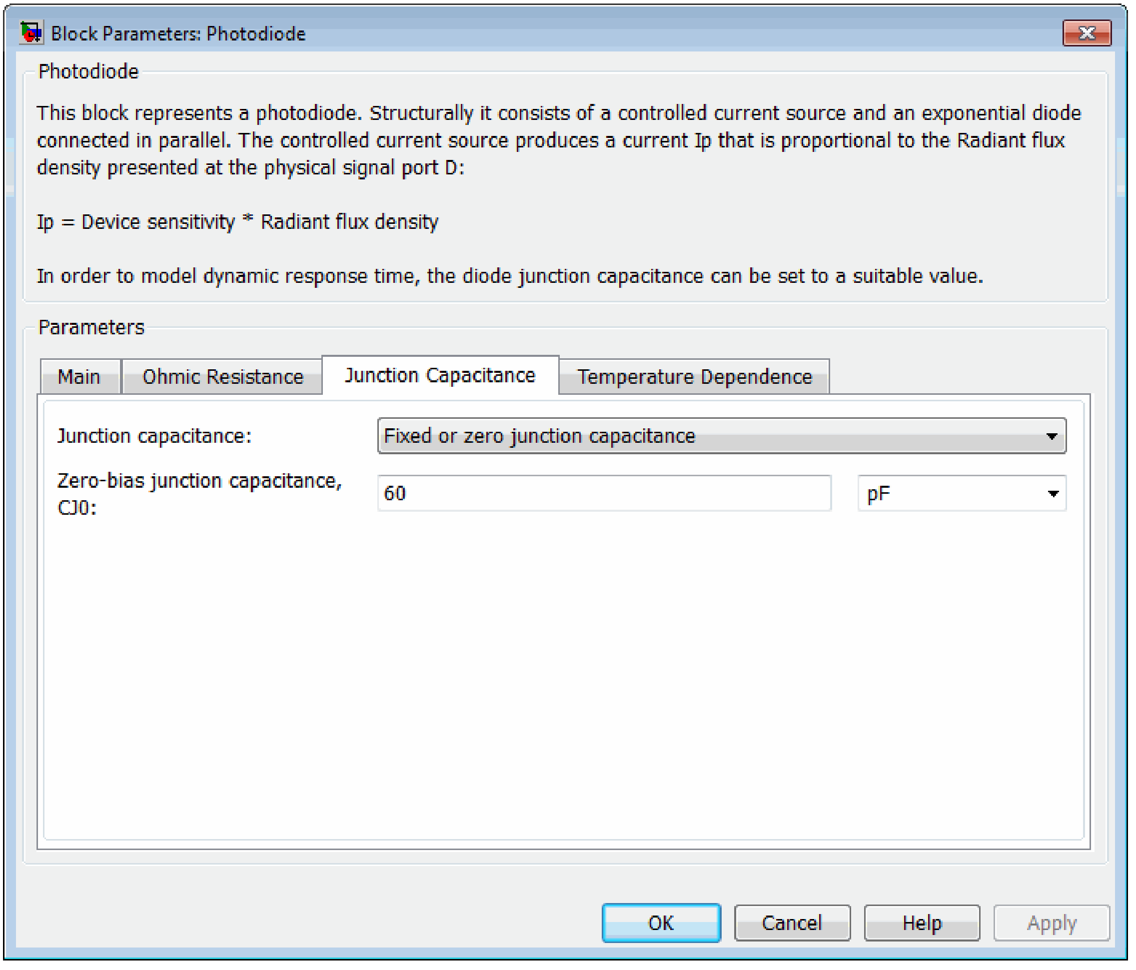
Ohmic Resistance Tab



Ohmic resistance RS

The series diode connection resistance. The default value is `0.1` Ω.

Junction Capacitance Tab



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 CJO, 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 CJO, 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 CJO, VJ, M & FC` for the **Junction capacitance** parameter. The default value is `1` V.

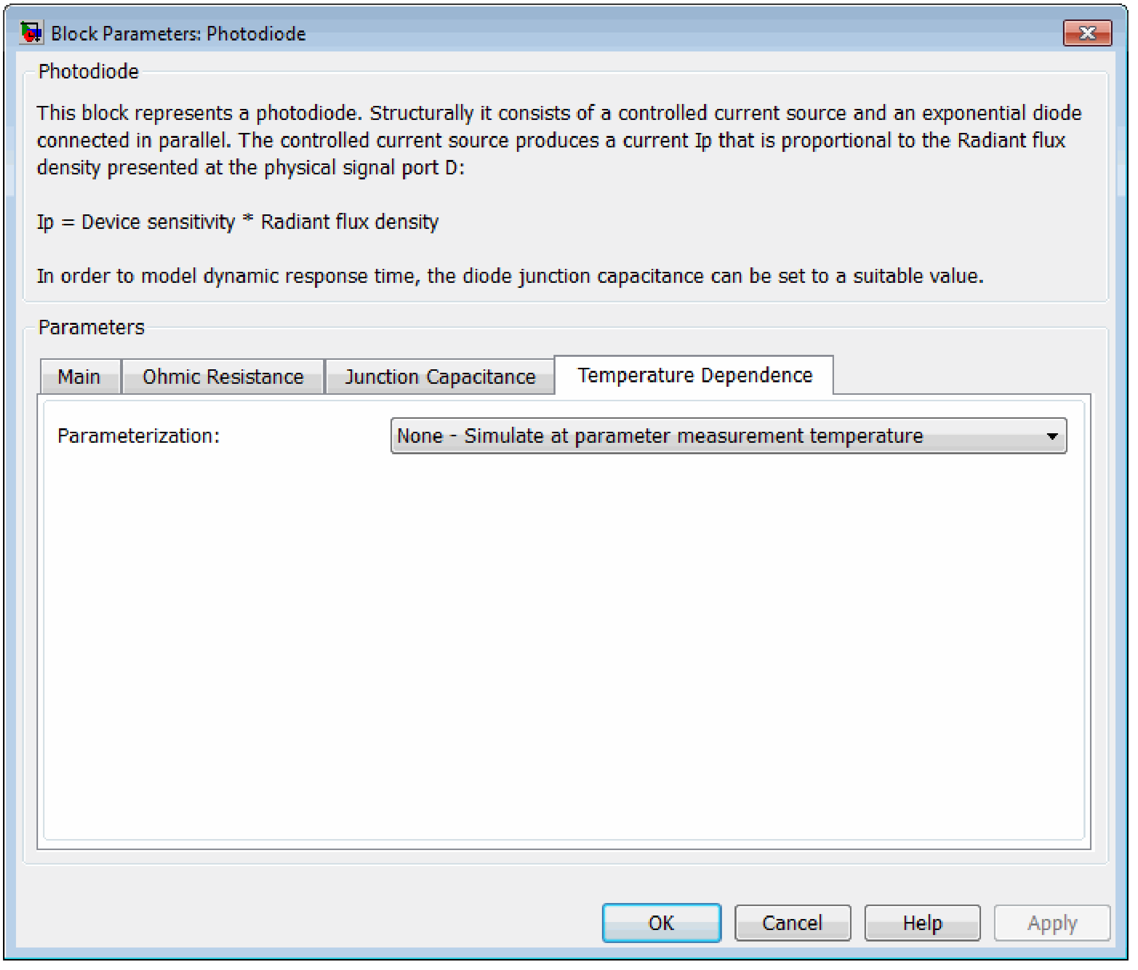
Grading coefficient M

The grading coefficient. This parameter is only visible when you select `Use parameters CJO, 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 CJO, VJ, M & FC` for the **Junction capacitance** parameter. The default value is `0.5`.

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.
- `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

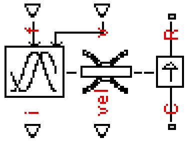
Purpose

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 1-442
- "Resonant Circuit Model for Powered Motor" on page 1-442

Mass and Friction Model for Unpowered Motor

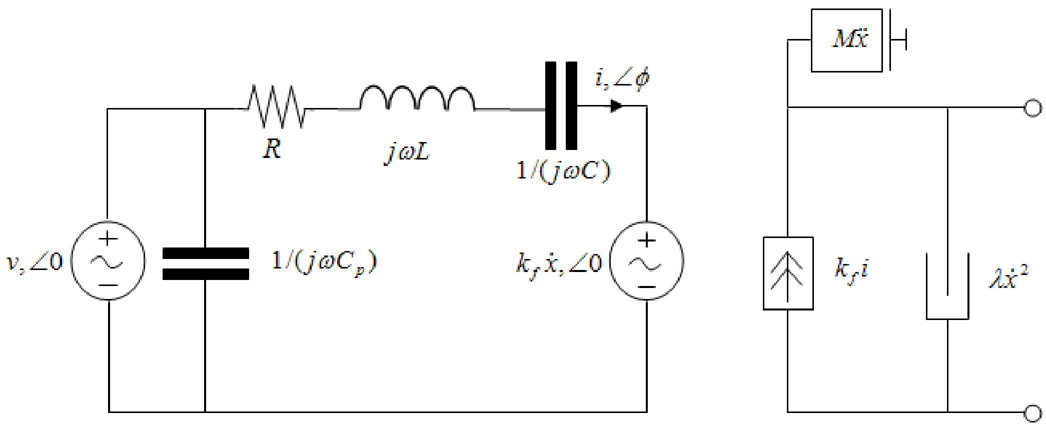
The motor is unpowered when the physical signal input v is zero. This corresponds to applying zero RMS volts to the motor. In this scenario, the block models the motor using the following elements:

- An 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 λ 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

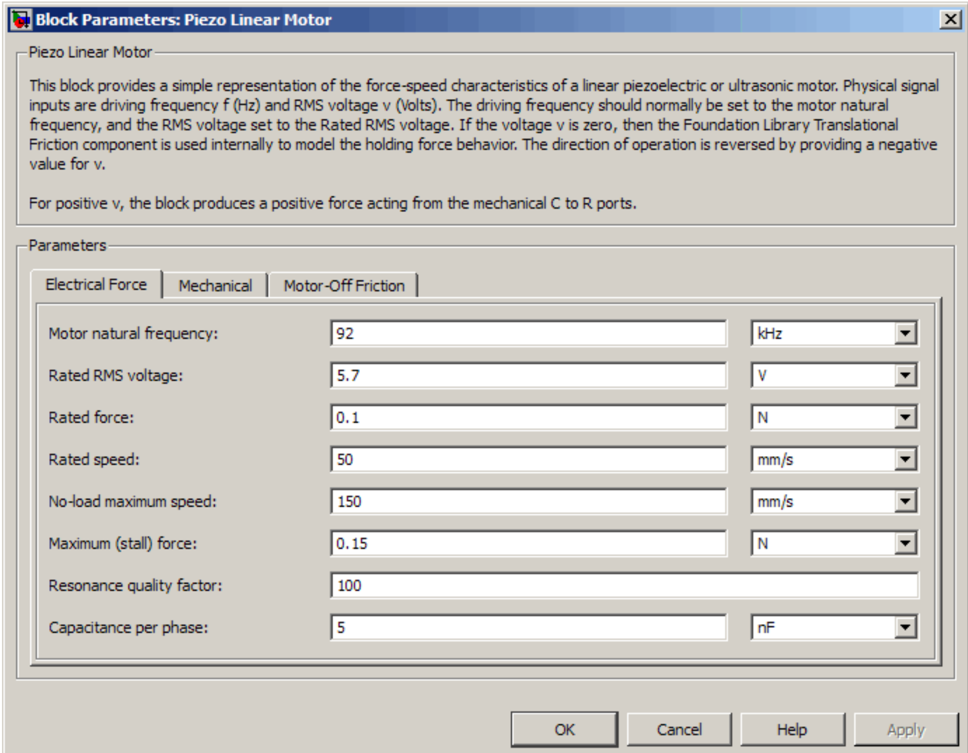
The block has the following limitations:

- 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.

Dialog Box and Parameters

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.

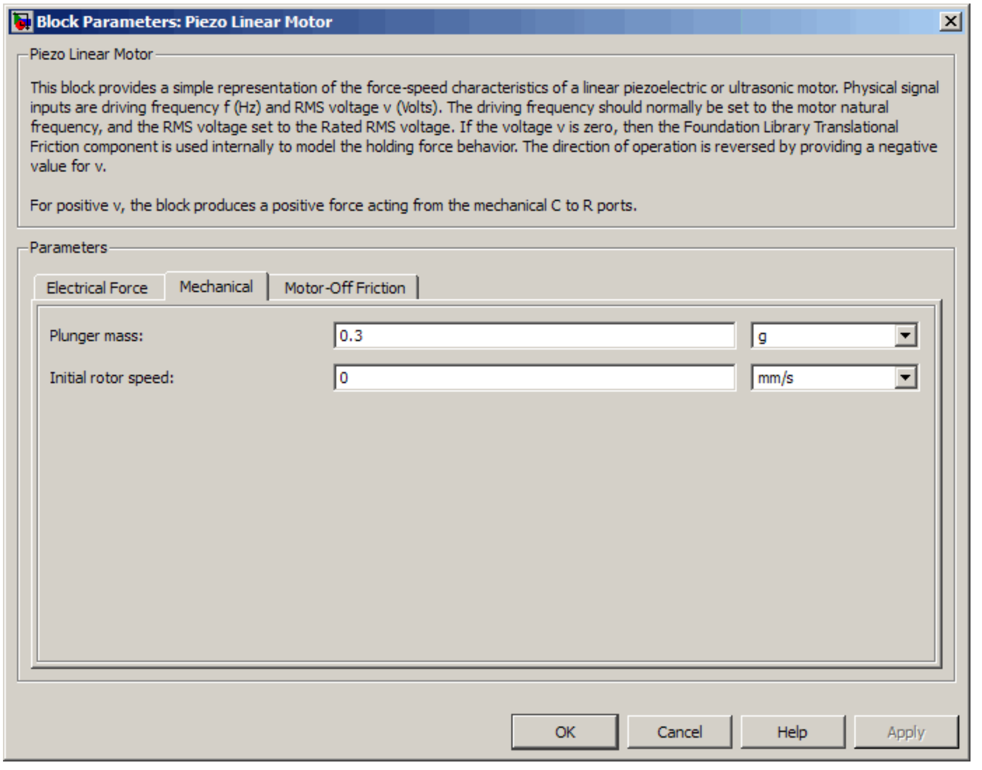
Maximum (stall) force
Maximum force the motor delivers when actively driving a load and powered at the rated voltage and frequency. The default value is `0.15` N.

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.

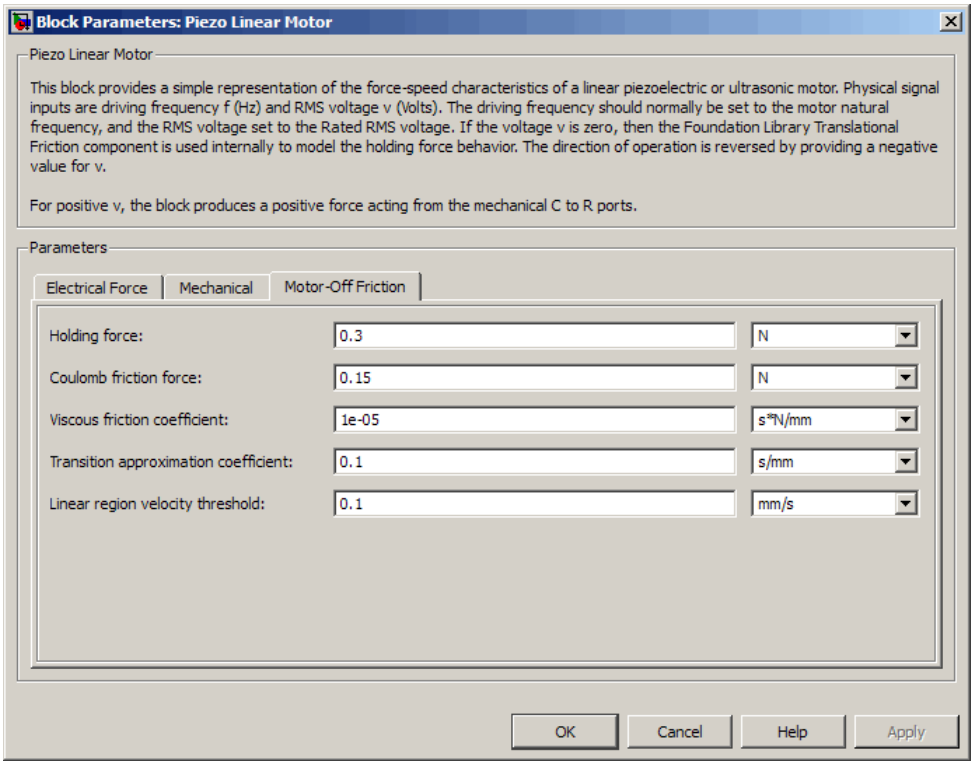
Mechanical Tab



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.

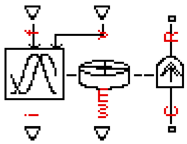
Purpose

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 1-451
- "Resonant Circuit Model for Powered Motor" on page 1-451

Inertia and Friction Model for Unpowered Motor

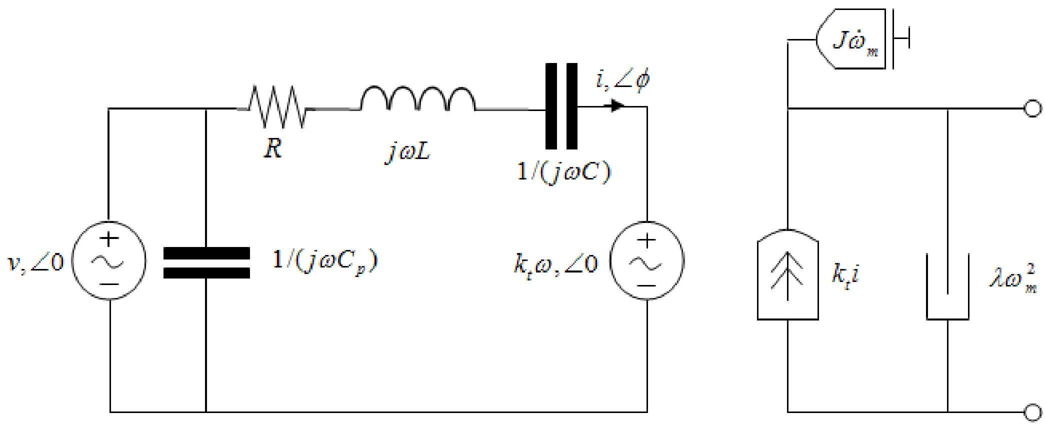
The motor is unpowered when the physical signal input v is zero. This corresponds to applying zero RMS volts to the motor. In this scenario, the block models the motor using the following elements:

- 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. ω_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 λ 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

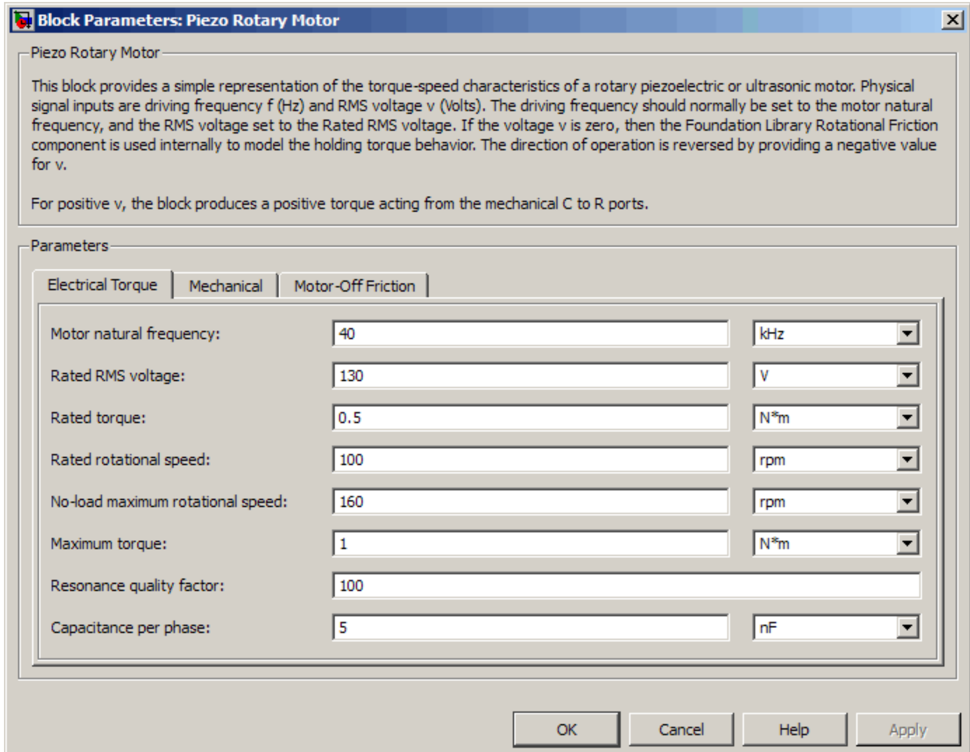
The block has the following limitations:

- 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.

Dialog Box and Parameters

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.

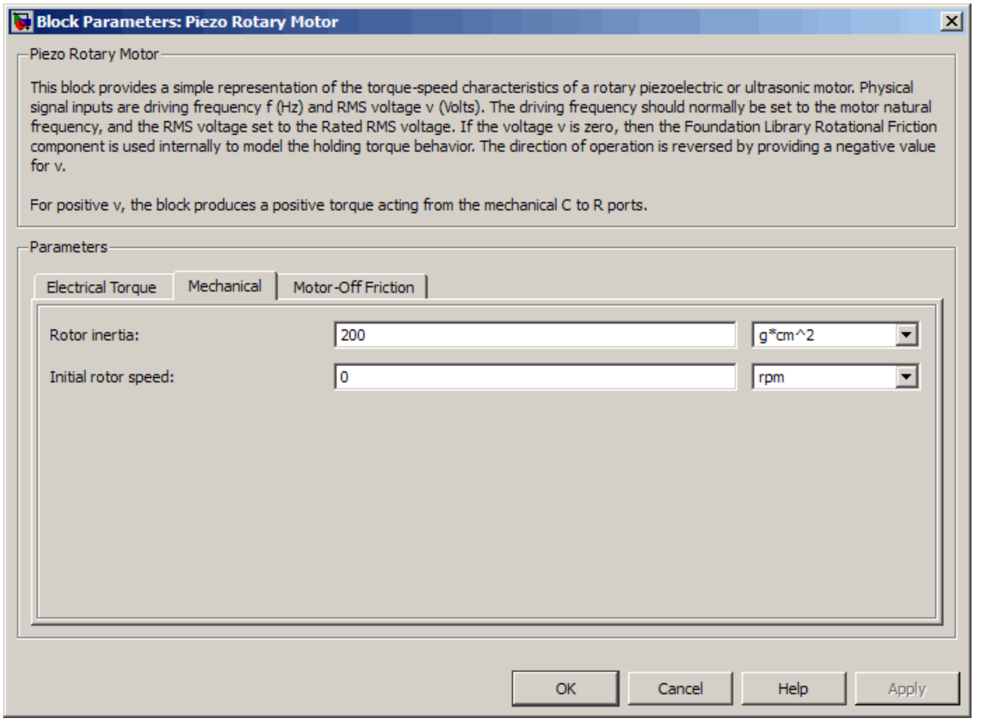
Maximum torque
Maximum torque that the motor delivers when actively driving a load and powered at the rated voltage and frequency. The default value is 1 N*m.

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.

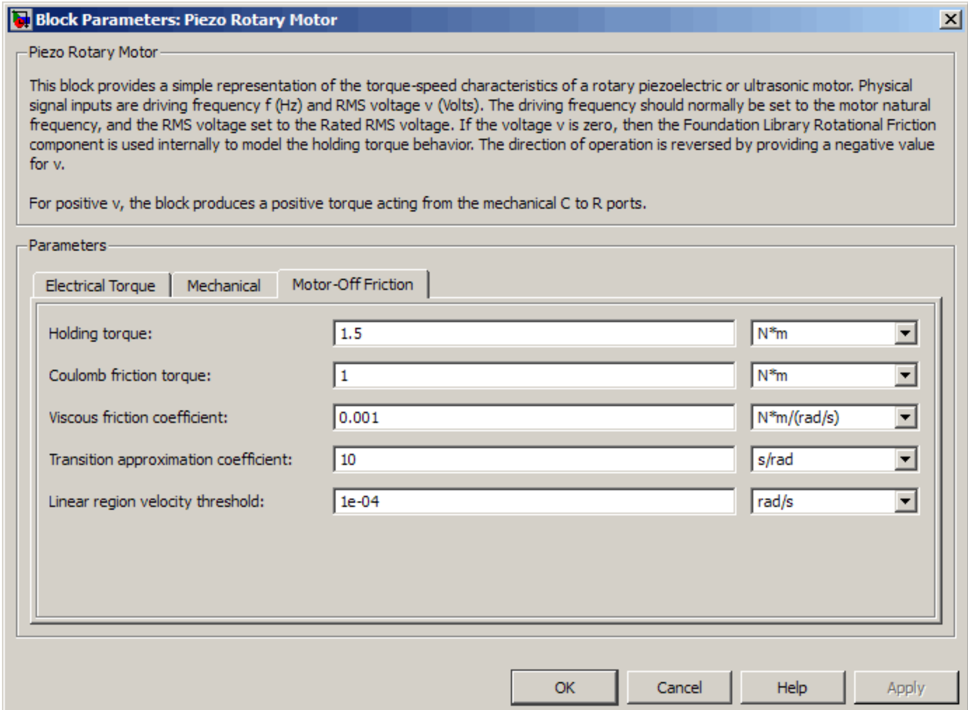
Mechanical Tab



Rotor inertia
: Rotor resistance to change in motor motion. The default value is 200 g*cm^2.

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` N*m/(rad*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. The default value is `10` 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 `1e-5` and `1e-3` rad/s. The default value is `1e-04` 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.

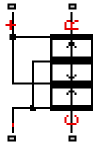
Purpose

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:

$$S = s^E T + d' E$$

$$D = dT + \varepsilon^T E$$

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.
- ε^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.

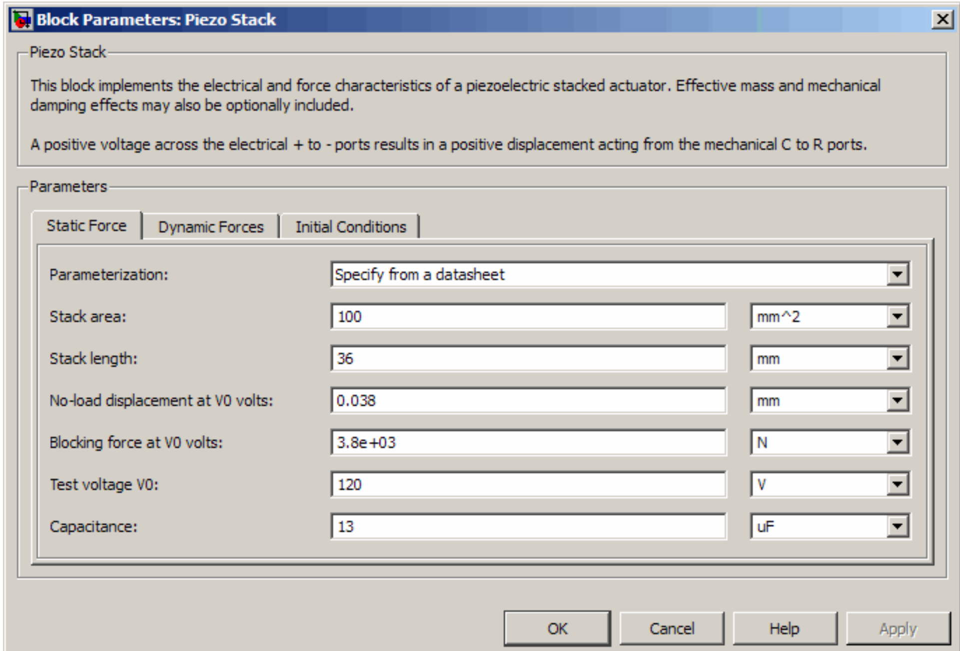
A positive voltage across the electrical + to – ports creates a positive displacement acting from the mechanical C to R ports.

Basic Assumptions and Limitations

The model does not include hysteresis effects.

Dialog Box and Parameters

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^2.

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

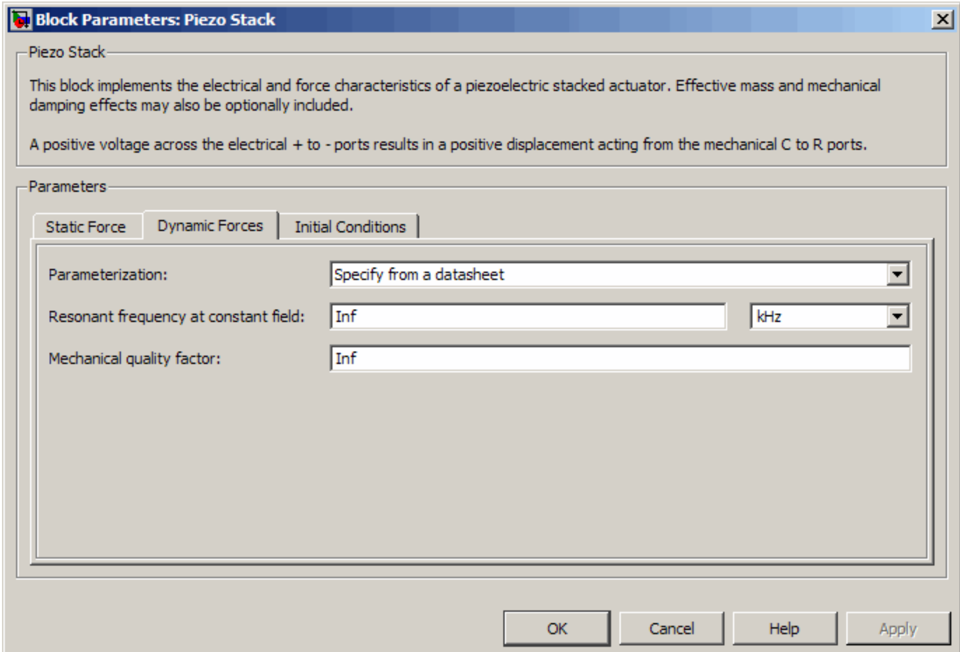
Number of layers in the piezo stack. This parameter is only visible when you select `Specify material properties` for the **Parameterization** parameter. The default value is `50`.

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^2/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

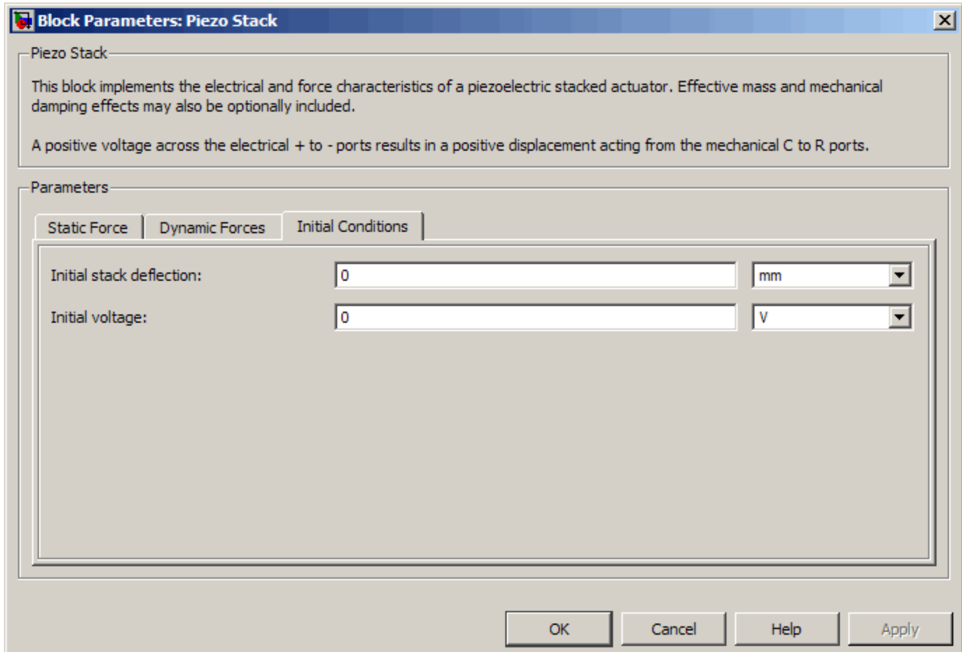
Frequency at which the actuator naturally resonates if mechanically perturbed with the electrical ports shorted. This parameter is only visible when you select `Specify from a datasheet` for the **Parameterization** parameter. The default value is `Inf` kHz.

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.

PNP Bipolar Transistor

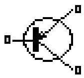
Purpose

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.
- β_F is the ideal maximum current gain BF
- β_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 (β_F and β_R) become a function of temperature according to the following equations:

$$IS_{Ts} = IS_{Tm1} \cdot (T_s / T_{m1})^{XTI} \cdot \exp\left(-\frac{EG}{kT_s}(1 - T_s / T_{m1}) \right)$$

$$\beta_{Fs} = \beta_{Fm1}\left(\frac{T_s}{T_{m1}}\right)^{XTB}$$

$$\beta_{Rs} = \beta_{Rm1}\left(\frac{T_s}{T_{m1}}\right)^{XTB}$$

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_{Tm1} is the saturation current at the measurement temperature.
- IS_{Ts} 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.
- β_{Fm1} and β_{Rm1} are the forward and reverse gains at the measurement temperature.
- β_{Fs} and β_{Rs} 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".

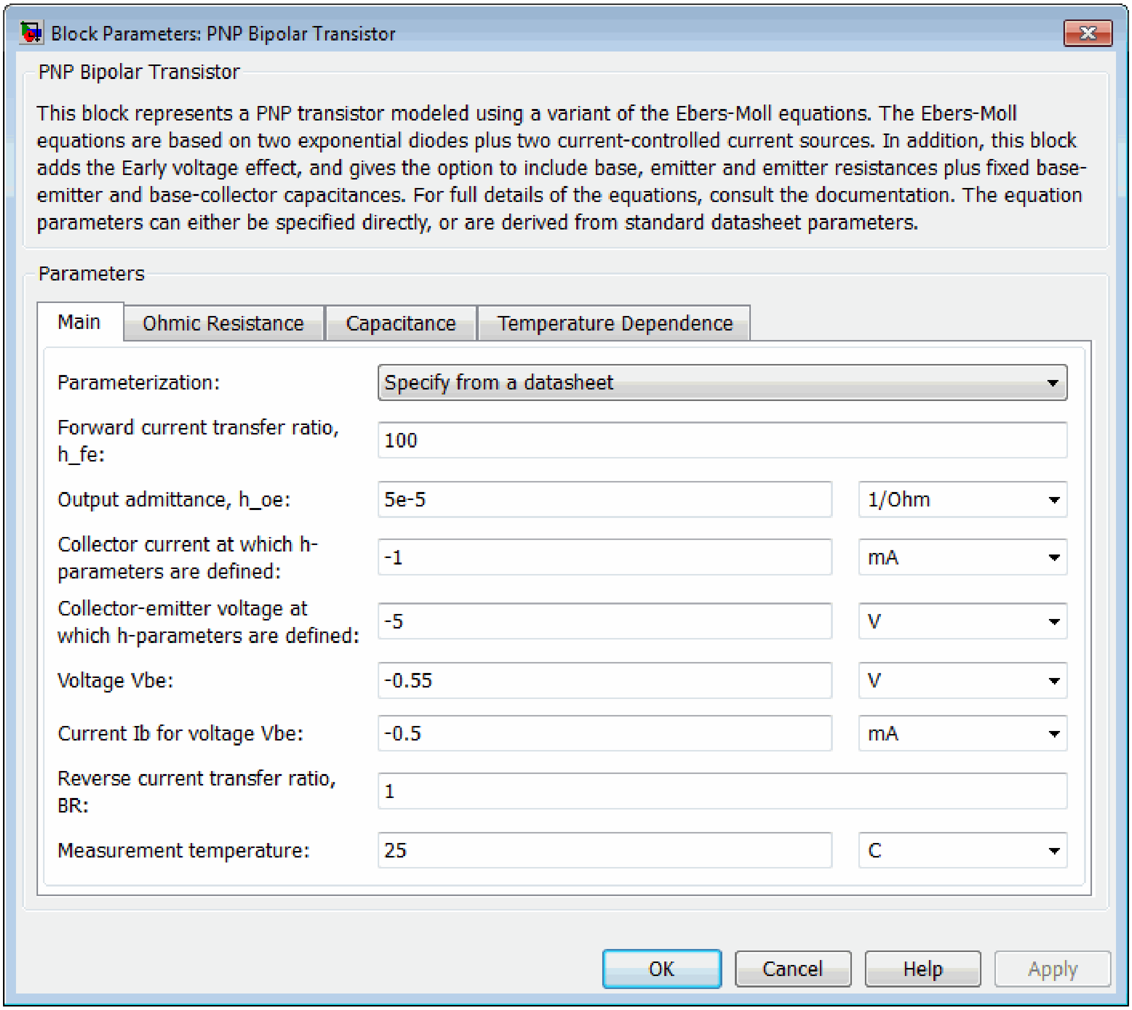
Basic Assumptions and Limitations

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.

Dialog Box and Parameters

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 *Ic*/*h_oe*, where *Ic* 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 Vbe** value and the corresponding **Current Ib for voltage Vbe** value when *Ic* 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/Ω.

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 Vbe

Base-emitter voltage when the base current is *Ib*. The [*Vbe Ib*] 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 Ib for voltage Vbe

Base current when the base-emitter voltage is *Vbe*. The [*Vbe Ib*] 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 *Ic* versus *Vce* curve is zero in the normal active region. The additional forward Early voltage term increases this gradient. The intercept on the *Vce*-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.

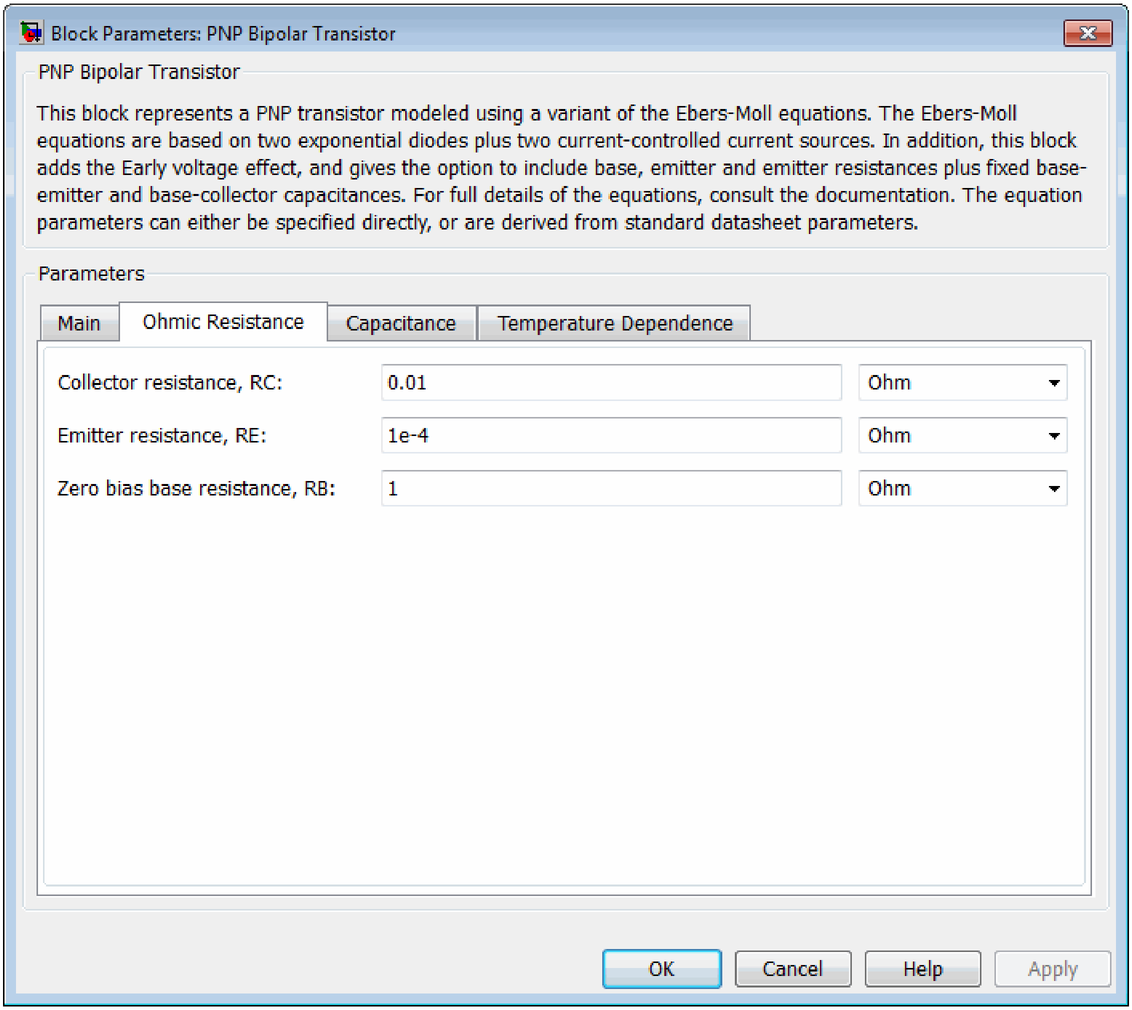
Reverse current transfer ratio BR

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 *Vbe* and *Ib*, 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.

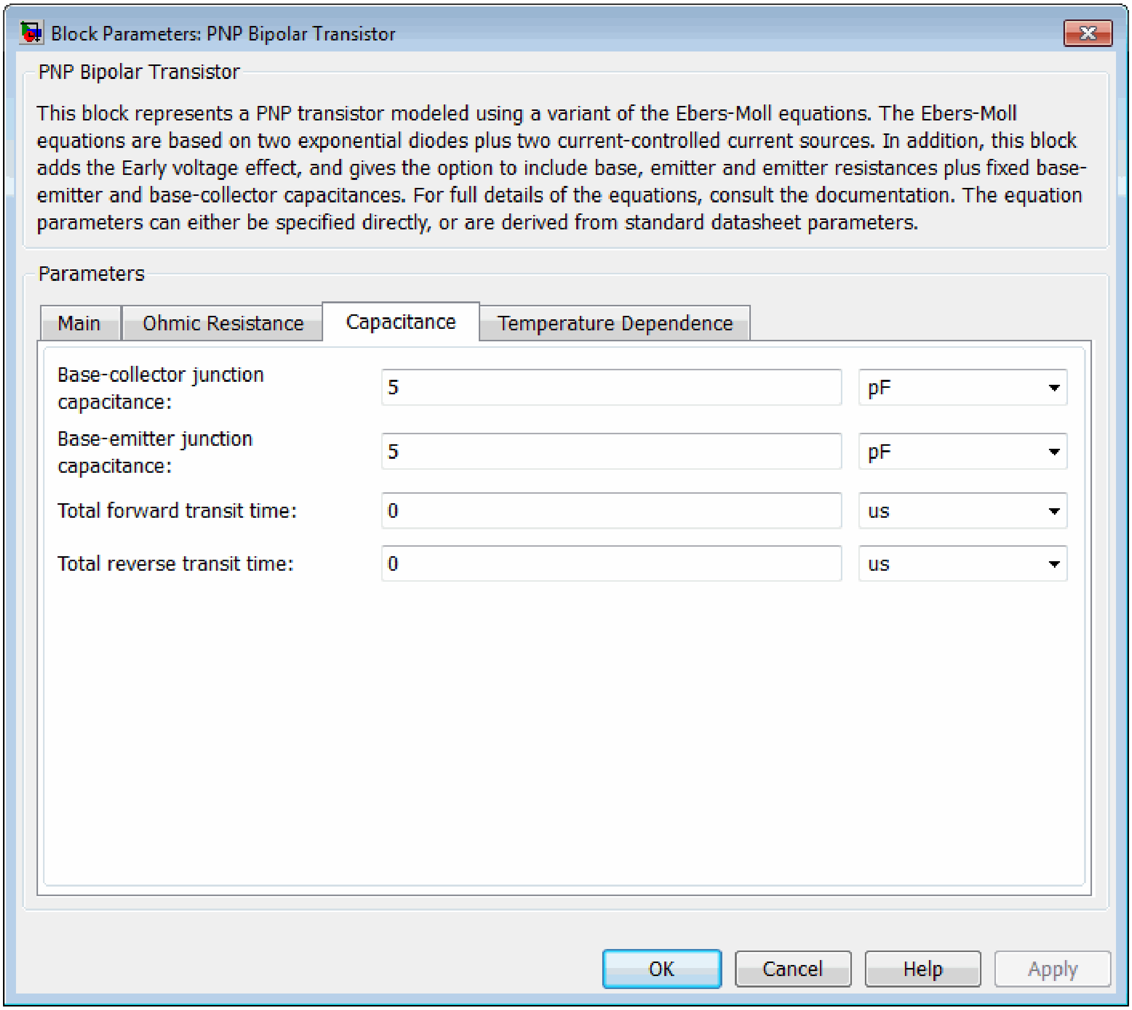
Emitter resistance RE

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



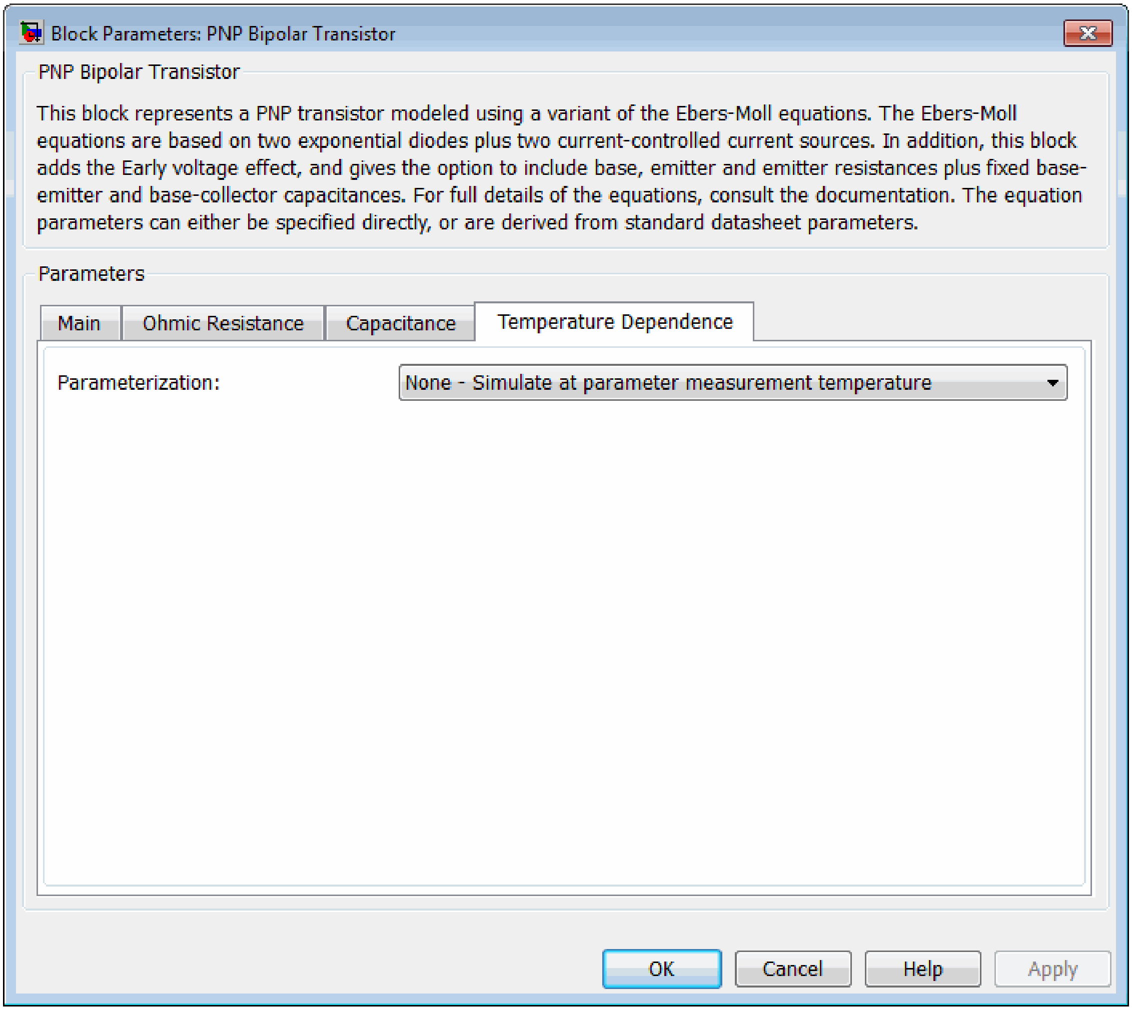
Base-collector junction capacitance
Parasitic capacitance across the base-collector junction. The default value is 5 pF.

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 μs.

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 0μs.

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 Vbe at second measurement temperature

Base-emitter voltage when the base current is *Ib* and the temperature is set to the second measurement temperature. The [*Vbe Ib*] 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 Ib for voltage Vbe at second measurement temperature
Base current when the base-emitter voltage is *Vbe* and the temperature is set to the second measurement temperature. The [*Vbe Ib*] 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*,*Vbe*, and *Ib* 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

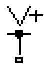
Purpose

Model ideal positive supply rail

Library

Sources

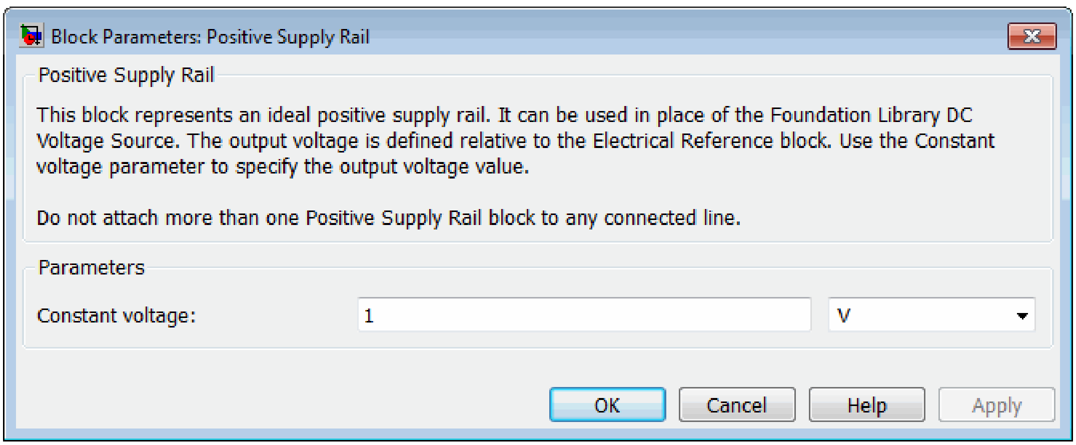
Description



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.

Dialog Box and 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

Simscape DC Voltage Source, Negative Supply Rail

Potentiometer

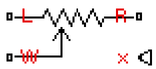
Purpose

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 λ 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.

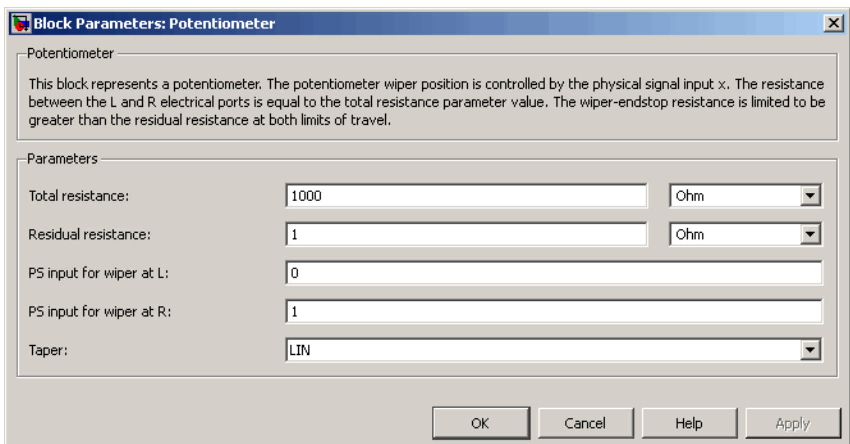
For both linear and logarithmic tapers, the resistance between the wiper position and port R is:

$$R_{WR} = R_0 - R_{WL}$$

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.

Dialog Box and 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 `1` Ω.

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 `200` Ω.

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

Simscape Variable Resistor

Power Sensor

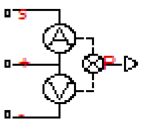
Purpose

Measure instantaneous or cycle-average power

Library

Sensors

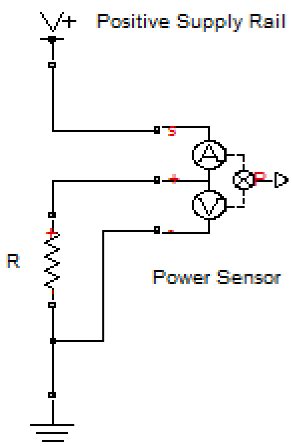
Description



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.

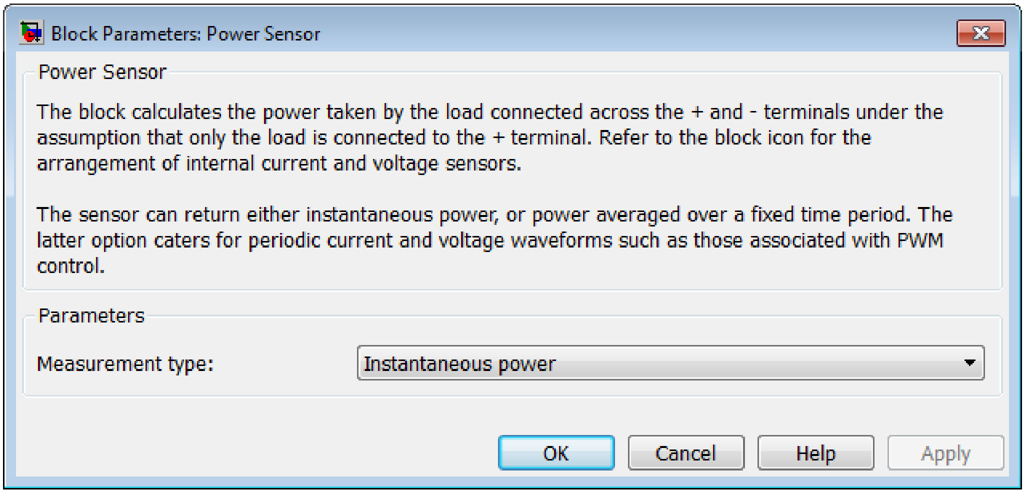
The sensor can return either instantaneous power, or power averaged over a fixed time period. Use the latter option for periodic current and voltage waveforms such as those associated with PWM control.

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 Synchronous Buck Converter example.

Dialog Box and 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.

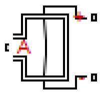
Purpose

Model generic pressure transducer that turns pressure measurement into voltage

Library

Sensors

Description



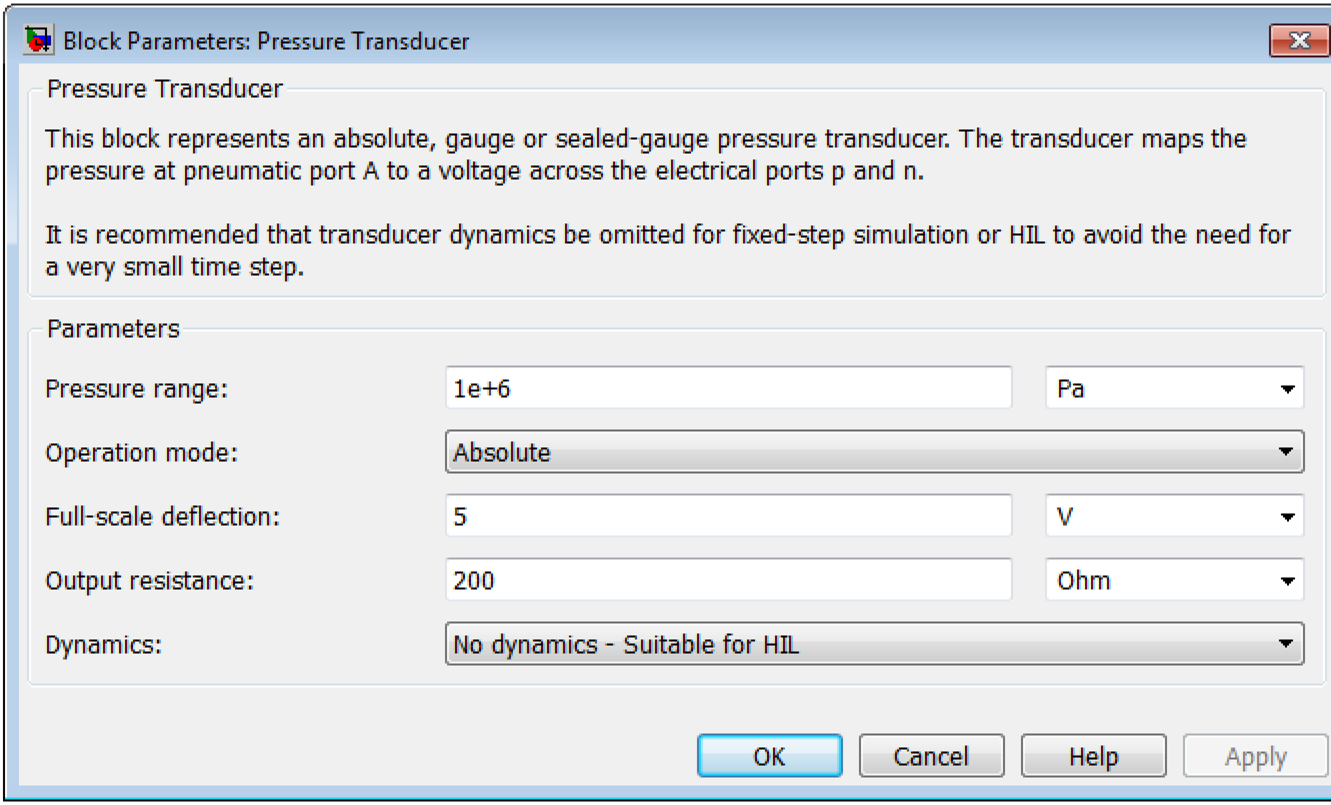
The Pressure Transducer block models a generic pressure transducer that turns a pressure measurement into a voltage. The output voltage is linearly proportional to the pressure, and the block outputs zero volts if the pressure is less than zero. An input pressure equal to the **Pressure range** parameter 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 **Initial pressure** parameter.

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.

Dialog Box and 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-scape 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` Ω.

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** and the **Initial pressure** parameter values.

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.

Initial pressure
Determines the initial condition for the lag. This parameter is only visible when you select `Model transducer bandwidth` for the **Dynamics** parameter. The default value is `0` Pa.

Ports

The block has the following ports:

`A`
Pneumatic port.

`+`
Positive electrical port.

`-`
Negative electrical port.

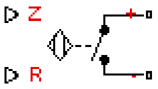
Purpose

Model simple distance sensor

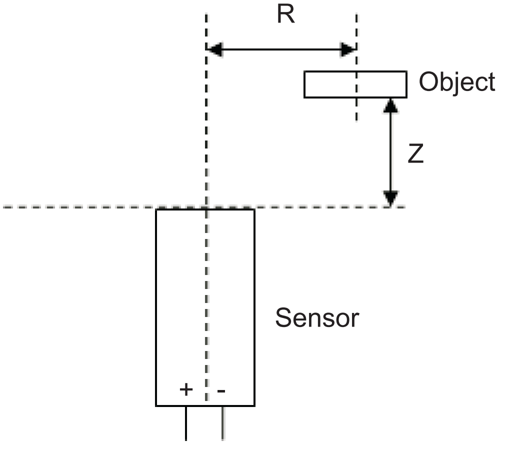
Library

Sensors

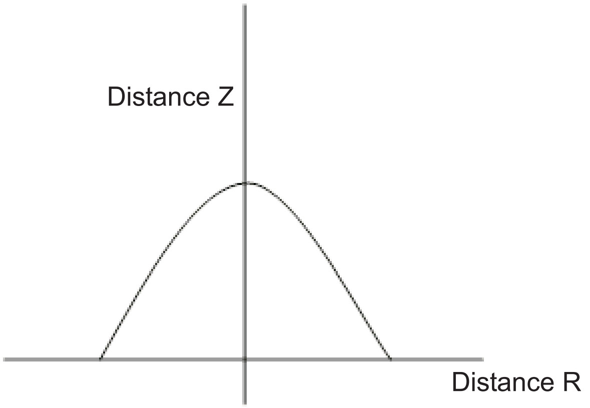
Description



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.

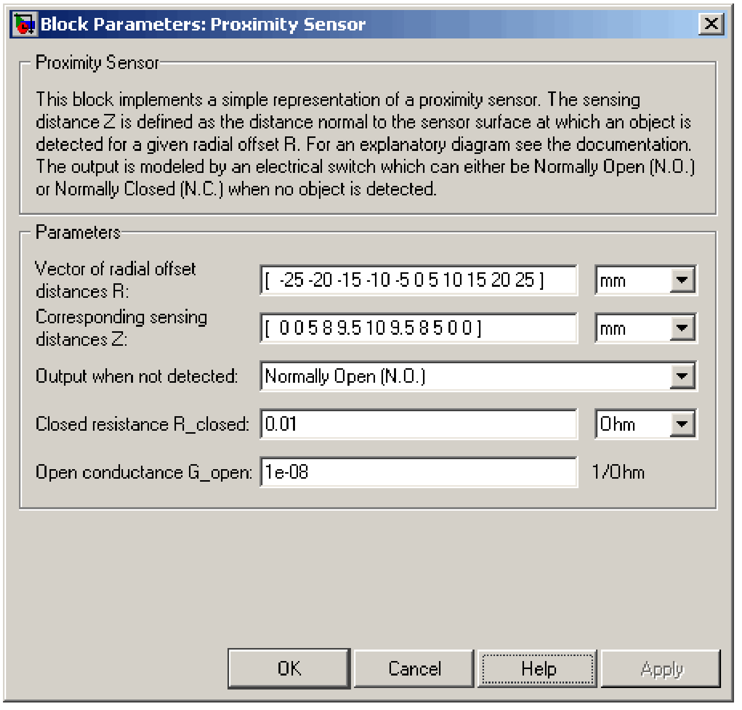


A typical sensing distance curve is shown in the following figure.



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.

Dialog Box and 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.

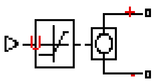
Purpose

Model generic linear sensor

Library

Sensors

Description

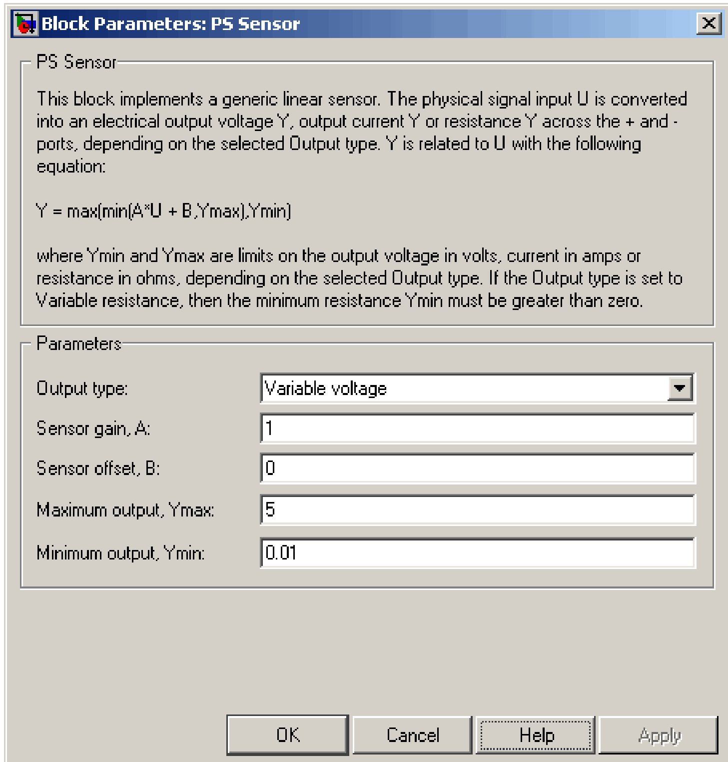


The PS Sensor block represents a generic linear sensor. The block converts the physical signal input U into an electrical output Y across the + and - ports. The **Output type** parameter value determines which of the following electrical outputs the block produces:

- Output voltage
- Output current
- Output resistance

Y is related to U as $Y = \max(\min(A*U + B, Y_{\max}), Y_{\min})$ where $Y_{\min}$ and $Y_{\max}$ are minimum and maximum limits on the output, respectively.

Dialog Box and 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`	Ω

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`	Ω

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

Simscape Controlled Voltage Source, Simscape Controlled Current Source, and Simscape Variable Resistor

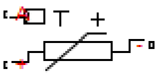
Purpose

Model switching type positive temperature coefficient (PTC) thermistor

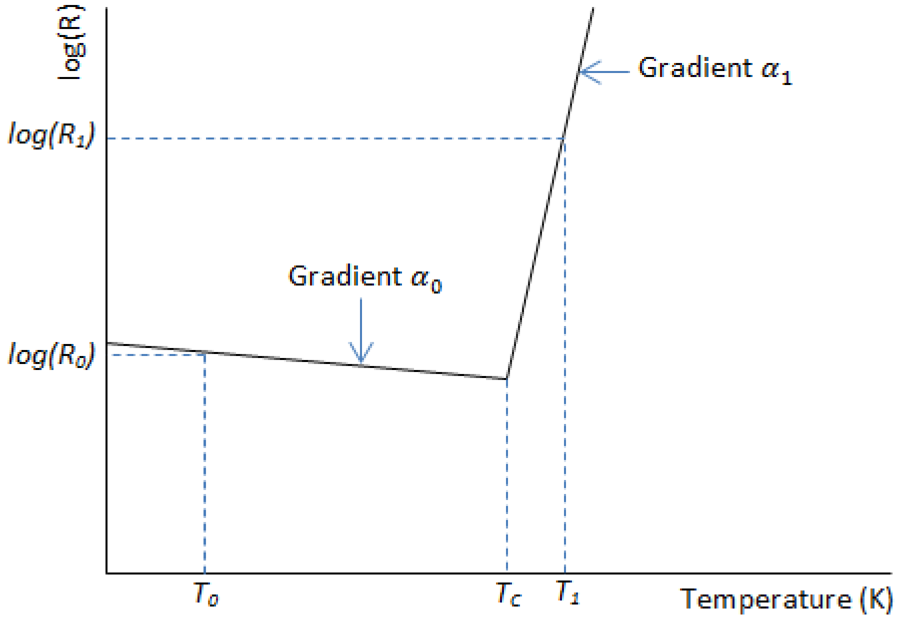
Library

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.
- a_0 is the temperature coefficient at nominal temperature T_0.
- a_1 is the temperature coefficient at reference temperature T_1.

The following equation describes the thermal behavior of the block:

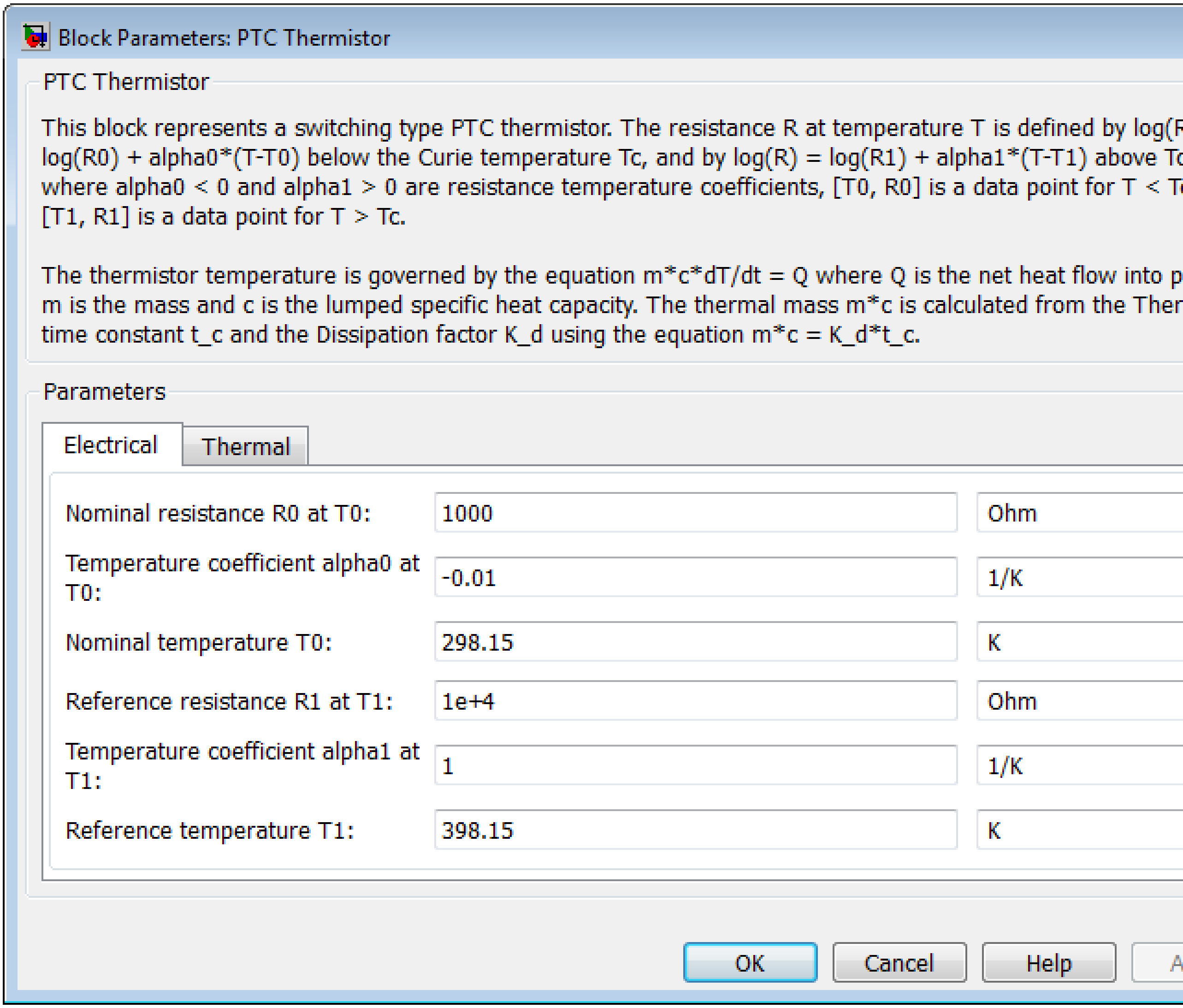
$$Q = K_d t_c \frac{dT}{dt}$$

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.

Dialog Box and Parameters

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.

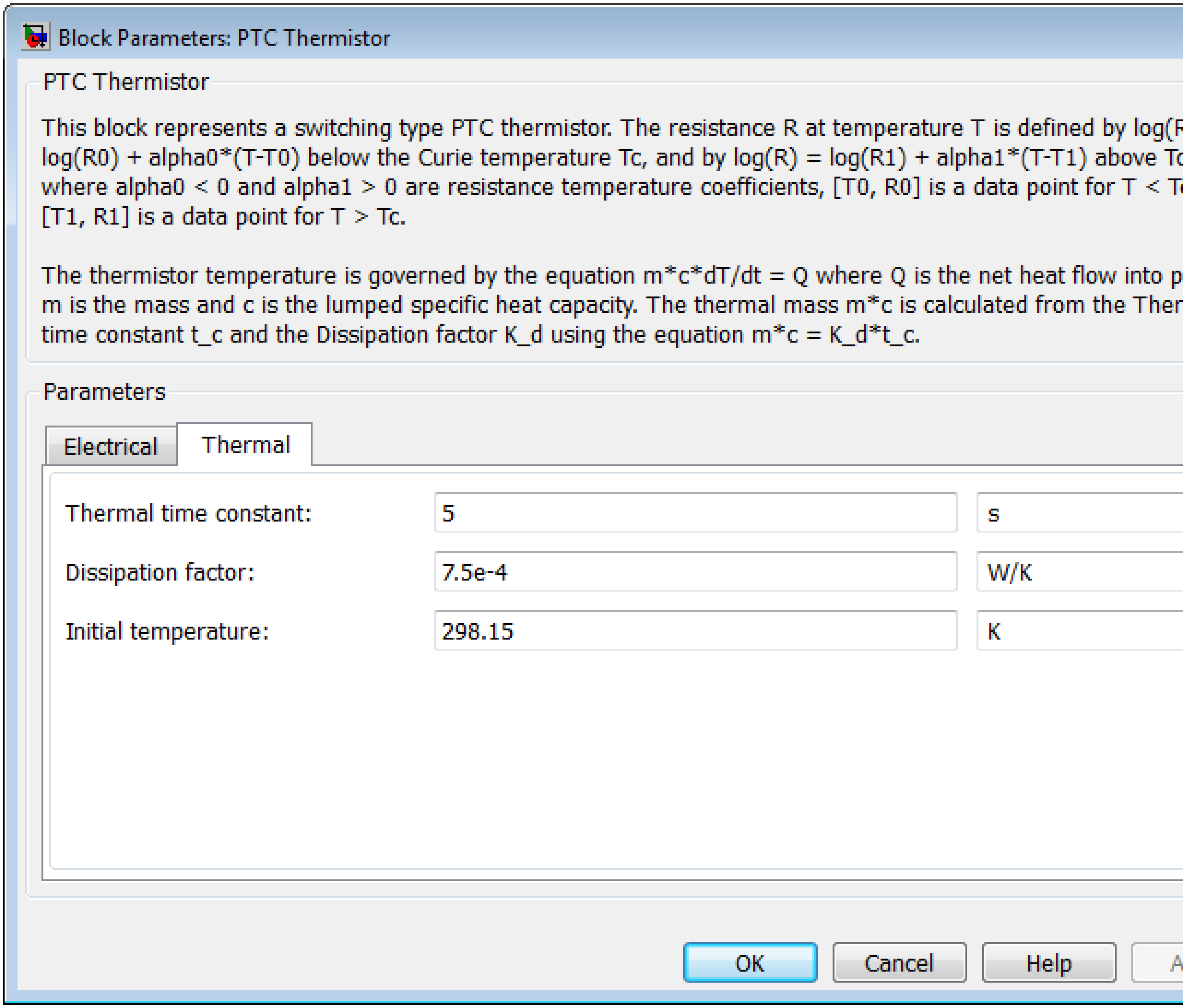
Nominal temperature T0
The temperature at which the nominal resistance is measured. The default value is `298.15` K.

Reference resistance R1 at T1
The reference resistance of the thermistor at the reference temperature. The default value is `10000` Ω.

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


Block Parameters: PTC Thermistor
PTC Thermistor
This block represents a switching type PTC thermistor. The resistance R at temperature T is defined by log(R
log(R0) + alpha0*(T-T0) below the Curie temperature Tc, and by log(R) = log(R1) + alpha1*(T-T1) above Tc
where alpha0 < 0 and alpha1 > 0 are resistance temperature coefficients, [T0, R0] is a data point for T < Tc
[T1, R1] is a data point for T > Tc.
The thermistor temperature is governed by the equation m*c*dT/dt = Q where Q is the net heat flow into p
m is the mass and c is the lumped specific heat capacity. The thermal mass m*c is calculated from the Ther
time constant t_c and the Dissipation factor K_d using the equation m*c = K_d*t_c.
Parameters
Electrical
Thermal
Thermal time constant:
5
s
Dissipation factor:
7.5e-4
W/K
Initial temperature:
298.15
K
OK
Cancel
Help

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.

Initial temperature
The temperature of the thermistor at the start of the simulation. The default value is `298.15` K.

Ports

The block has the following ports:

`A`
Thermal port.

`+`
Positive electrical port.

`-`
Negative electrical port.

See Also

Thermal Resistor

Thermistor

Pulse Current Source

Purpose

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.

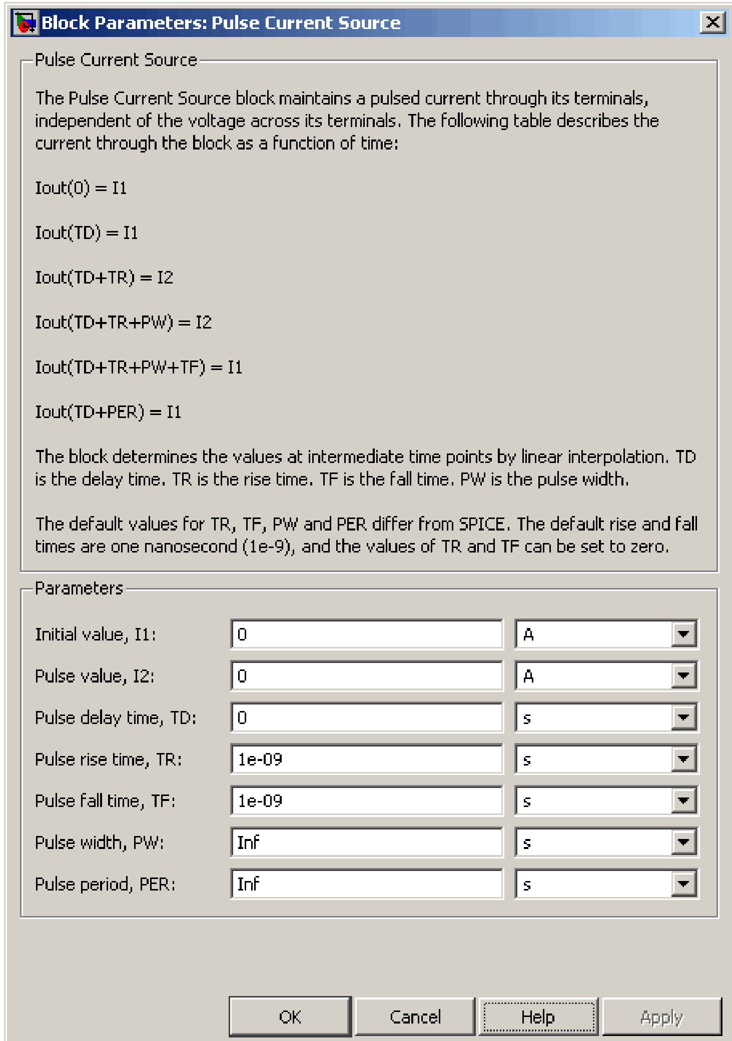
The specified values for *PW* and *PER* have the following effect on the block output:

- 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.

Dialog Box and 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

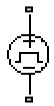
Purpose

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:

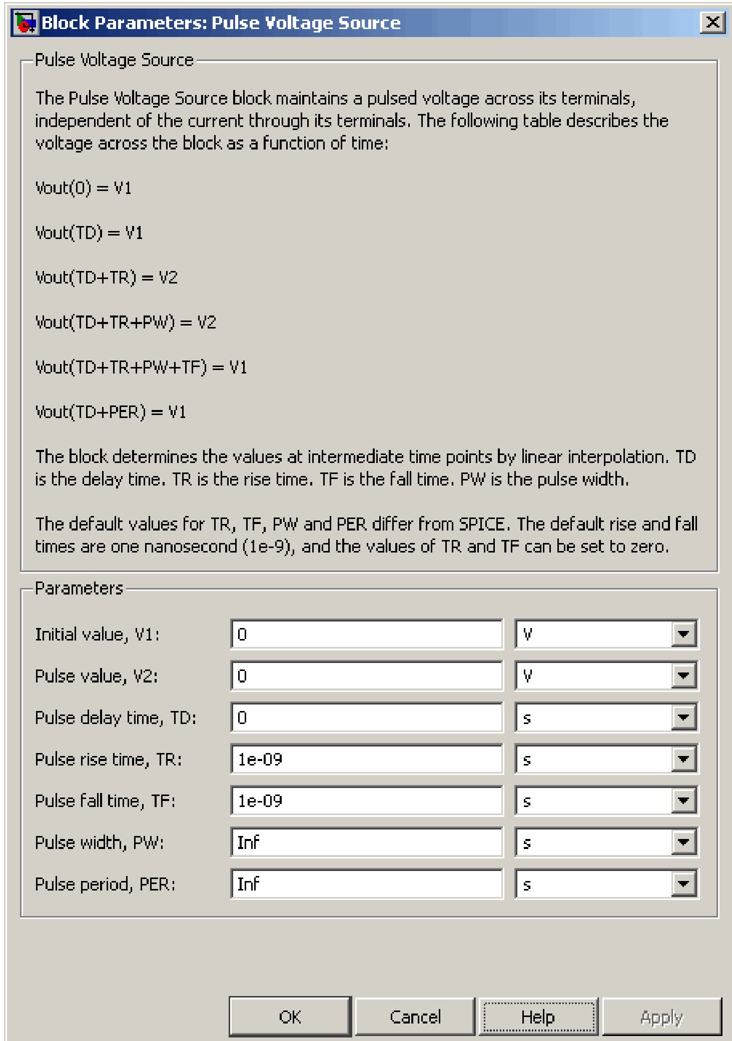
- *V1* is the **Initial value, V1** parameter value.
- *V2* is the **Pulse value, V2** parameter value.
- *TD* is the **Pulse delay time, TD** parameter value.
- *TR* is the **Pulse rise time, TR** parameter value.
- *TF* is the **Pulse fall time, TF** parameter value.
- *PW* is the **Pulse width, PW** 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.

Dialog Box and 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, I1** 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

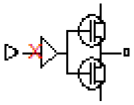
Purpose

Represent CMOS complementary output stage behaviorally

Library

Integrated Circuits

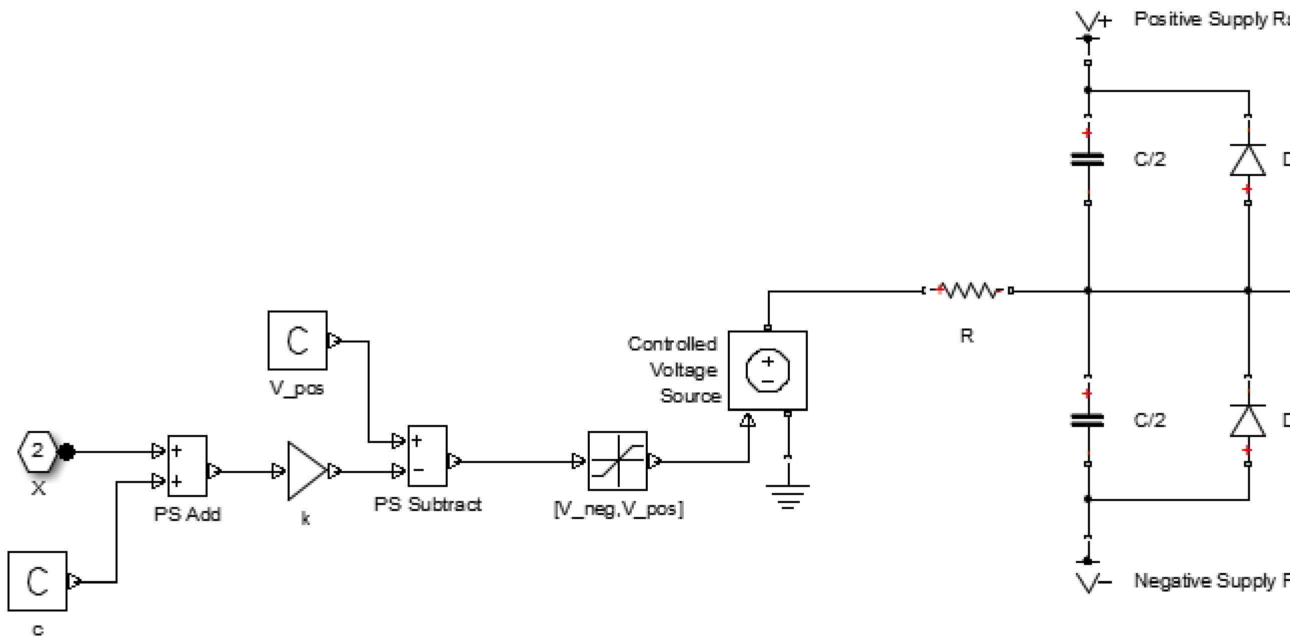
Description



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.

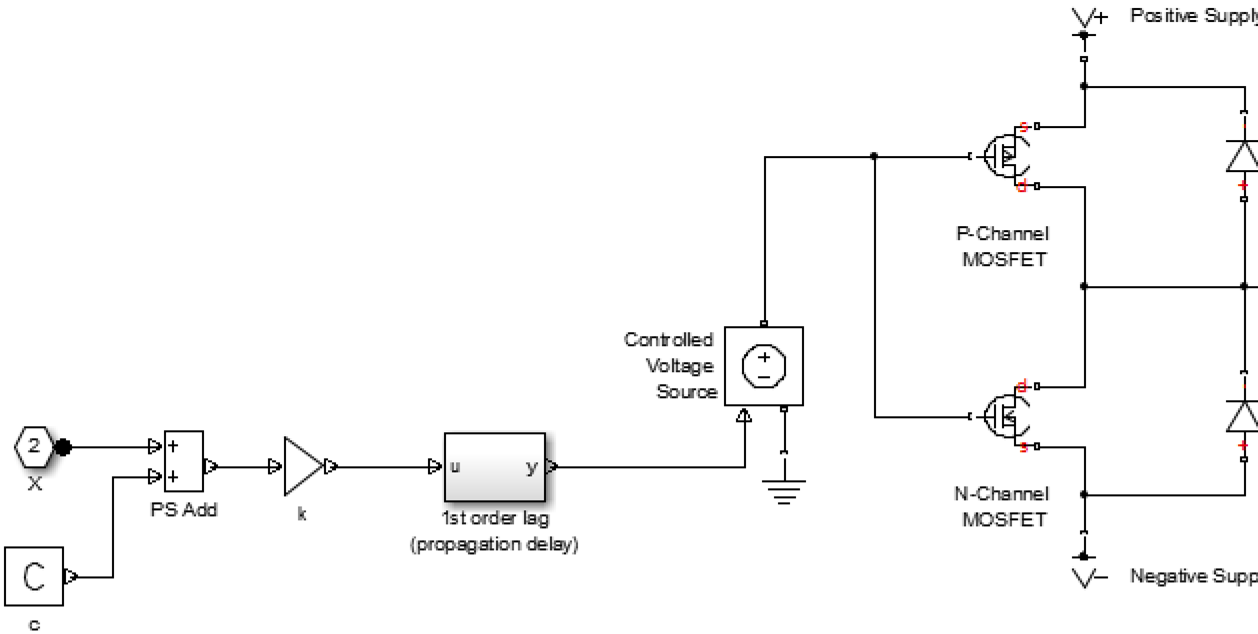
You can choose between are two output current-voltage relationships:

- `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:

$$Vg = k \cdot (X + c)$$

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.

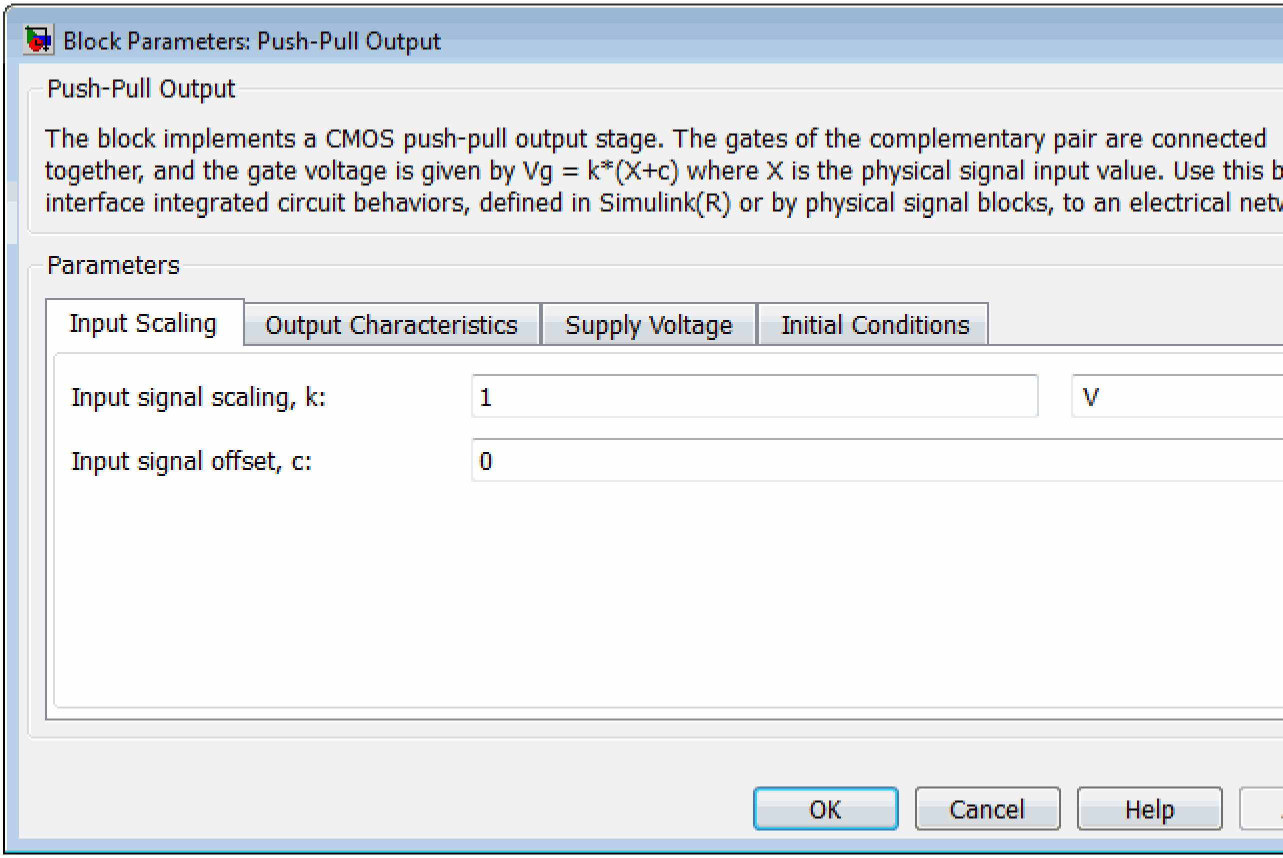
Basic Assumptions and Limitations

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.

Dialog Box and Parameters

Input Scaling Tab



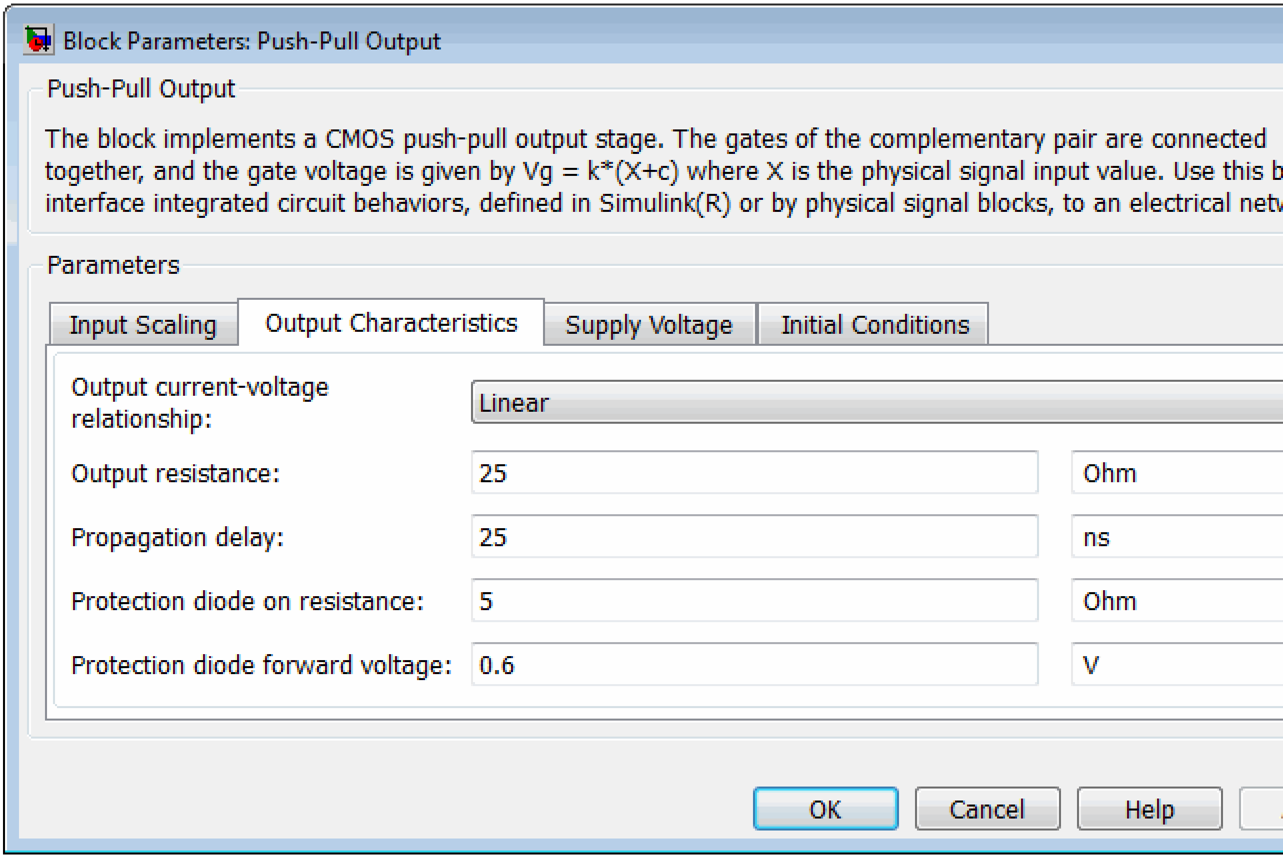
Input signal scaling, k

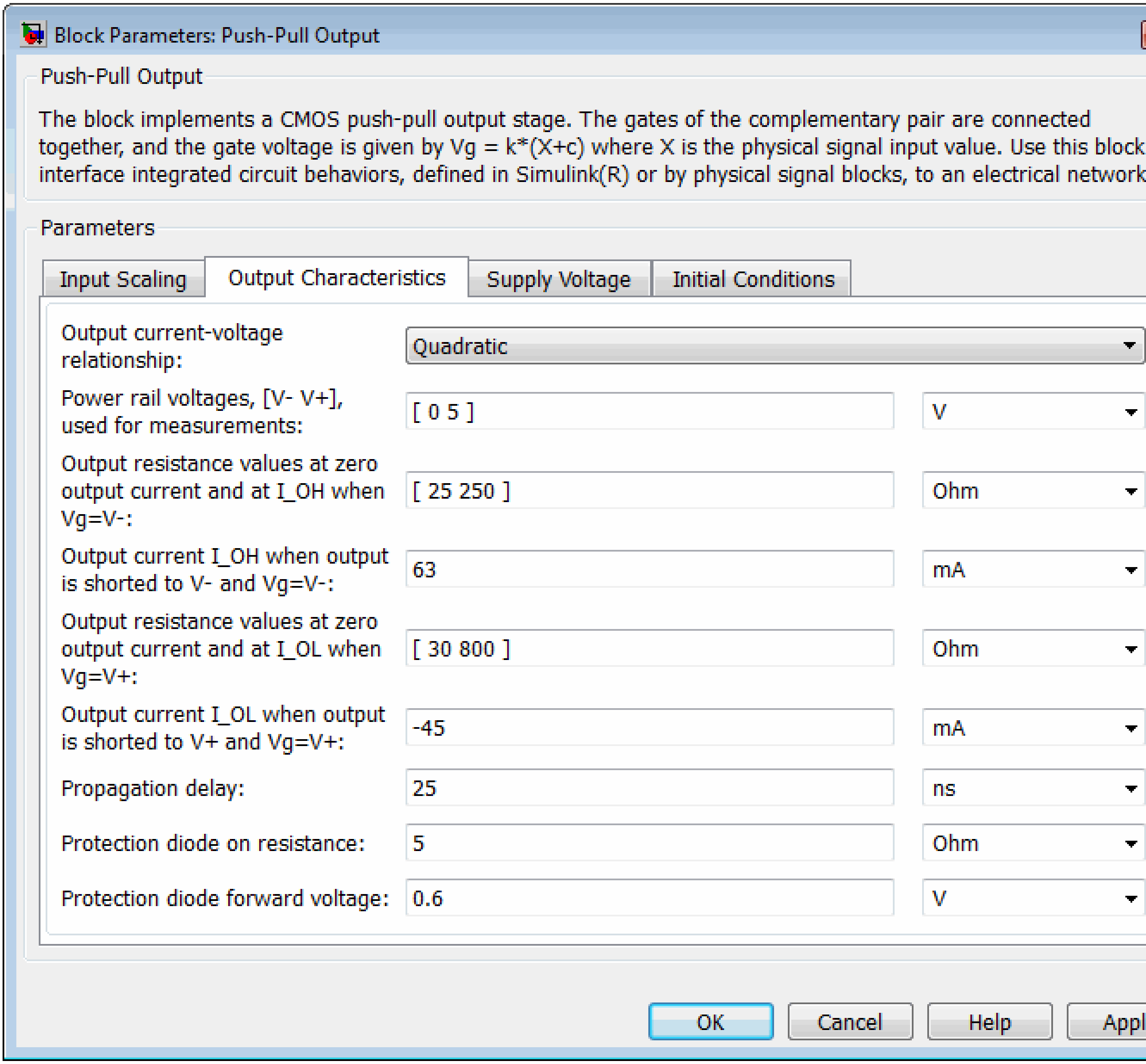
The input physical signal X is mapped to the gate voltage by $Vg = k \cdot (X + c)$, where k is the input signal scaling. Use this parameter in conjunction with the **Input signal offset, c** to map the range of X to the voltage range defined by the power supply. The default value is 1 V.

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




Block Parameters: Push-Pull Output
Push-Pull Output
The block implements a CMOS push-pull output stage. The gates of the complementary pair are connected together, and the gate voltage is given by Vg = k*(X+c) where X is the physical signal input value. Use this block interface integrated circuit behaviors, defined in Simulink(R) or by physical signal blocks, to an electrical network
Parameters
Input Scaling
Output Characteristics
Supply Voltage
Initial Conditions
Output current-voltage relationship:
Quadratic
Power rail voltages, [V- V+], used for measurements:
[0 5]
V
Output resistance values at zero output current and at I_OH when Vg=V-:
[25 250]
Ohm
Output current I_OH when output is shorted to V- and Vg=V-:
63
mA
Output resistance values at zero output current and at I_OL when Vg=V+:
[30 800]
Ohm
Output current I_OL when output is shorted to V+ and Vg=V+:
-45
mA
Propagation delay:
25
ns
Protection diode on resistance:
5
Ohm
Protection diode forward voltage:
0.6
V
OK
Cancel
Help
Appl

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` Ω.

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_OH when Vg=V-
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 complementary pair output is HIGH (Vg=V-) and there is no output current. The second value R_OH2 is the gradient of the output voltage-current relationship when the output is HIGH and the output current is I_OH. 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_OH when output is shorted to V- and Vg=V-
The resulting current when the output is HIGH (Vg=V-), 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_OL when Vg=V+

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 complementary pair output is LOW (Vg=V+) and there is no output current. The second value R_OL2 is the gradient of the output voltage-current relationship when the output is LOW and the output current is I_OL. 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_OL when output is shorted to V+ and Vg=V+

The resulting current when the output is LOW (Vg=V+), 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.

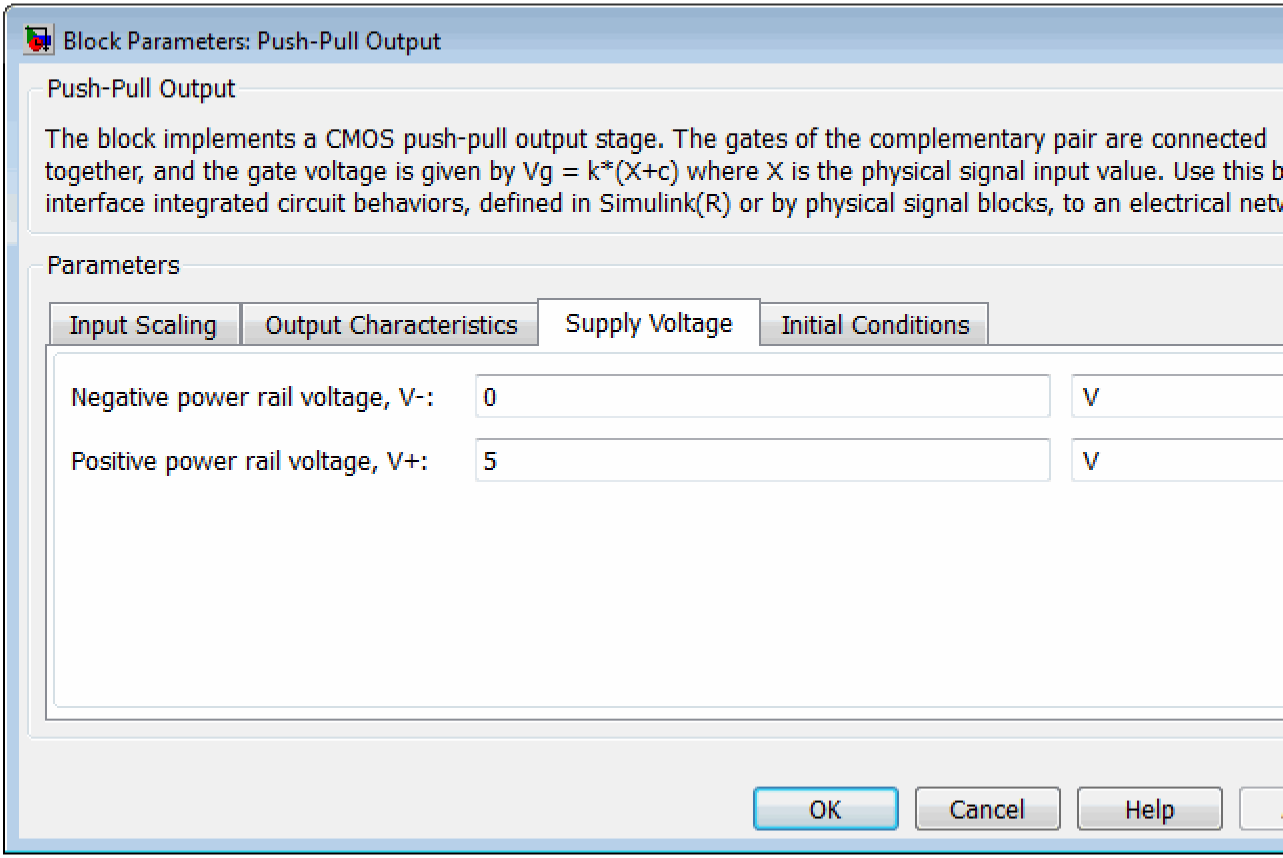
Protection diode on resistance

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



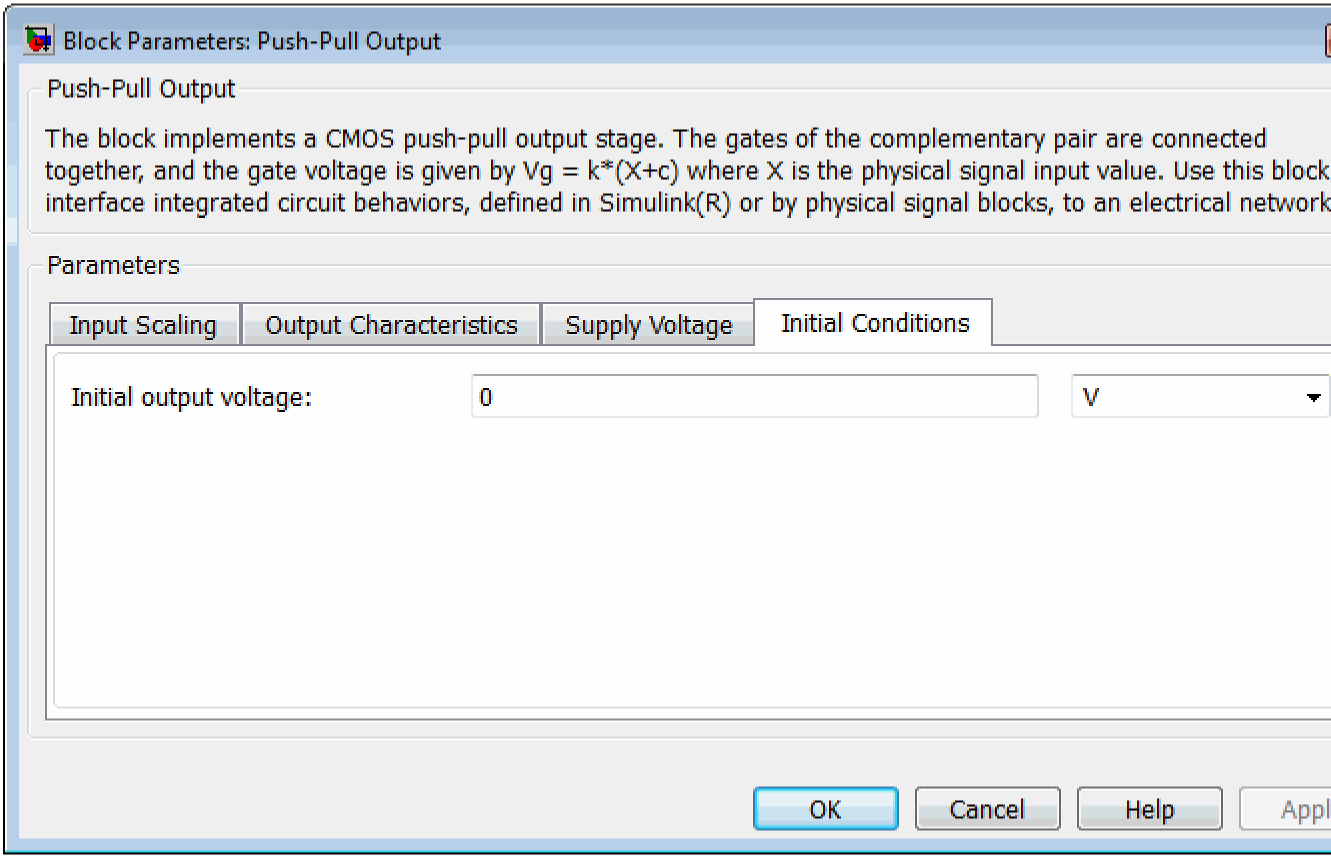
Negative power rail voltage, V-

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

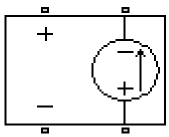
Purpose

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:

$$I_{out} = p(0) + p(1) * V_{in} + \ldots + p(n-1) * V_{in}^{n-1} + p(n) * V_{in}^{n}$$

- If you specify a scalar coefficient for the **Polynomial coefficients** parameter:

$$I_{out} = p * V_{in}$$

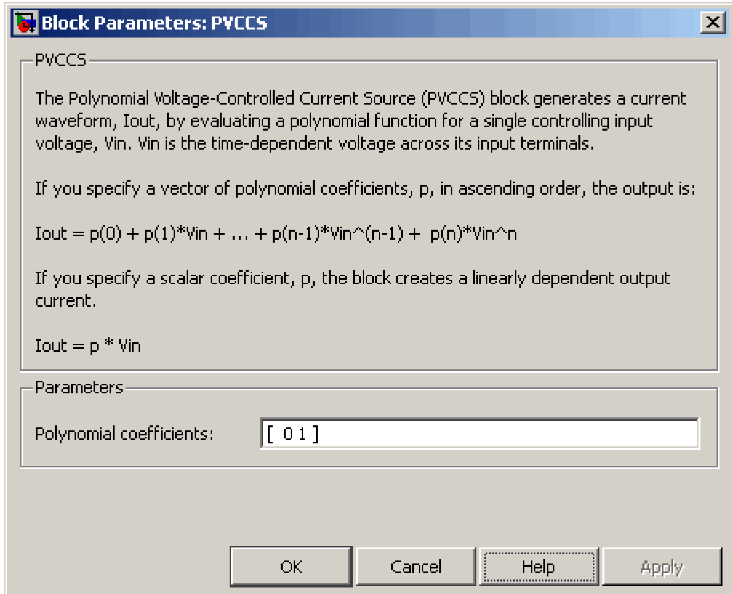
where:

- 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.

Dialog Box and 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

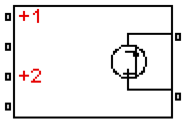
Purpose

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:

$$I_{out} = p_1 + p_2 * V_{in1} + p_3 * V_{in2} + p_4 * V_{in1}^2 + p_5 V_{in1} * V_{in2} + p_6 * V_{in2}^2 + \ldots$$

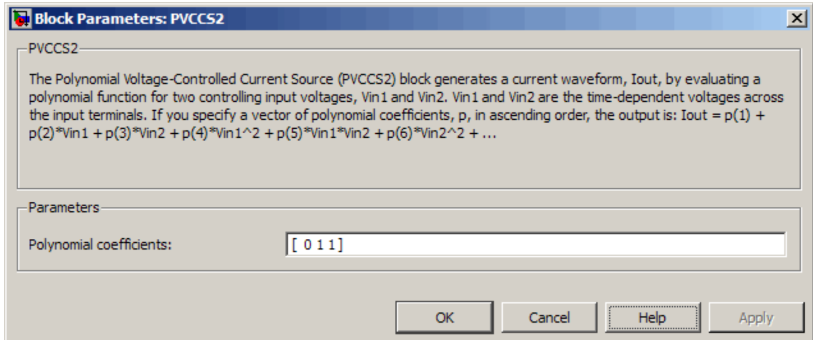
where:

- 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.

Dialog Box and 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

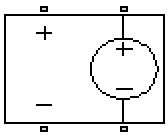
Purpose

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:

$$V_{out} = p(0) + p(1) * V_{in} + \ldots + p(n-1) * V_{in}^{n-1} + p(n) * V_{in}^{n}$$

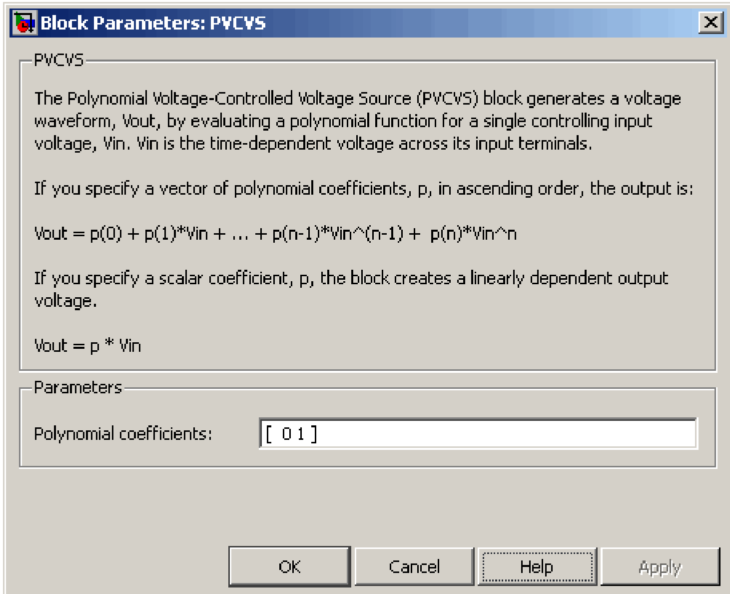
- If you specify a scalar coefficient for the **Polynomial coefficients** parameter:

$$V_{out} = p * V_{in}$$

where:

- V_{in} is the voltage across the input ports.
- p is the **Polynomial coefficients** parameter value.

Dialog Box and 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

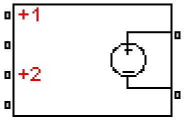
Purpose

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:

$$V_{out} = p_1 + p_2 * V_{in1} + p_3 * V_{in2} + p_4 * V_{in1}^2 + p_5 V_{in1} * V_{in2} + p_6 * V_{in2}^2 + \ldots$$

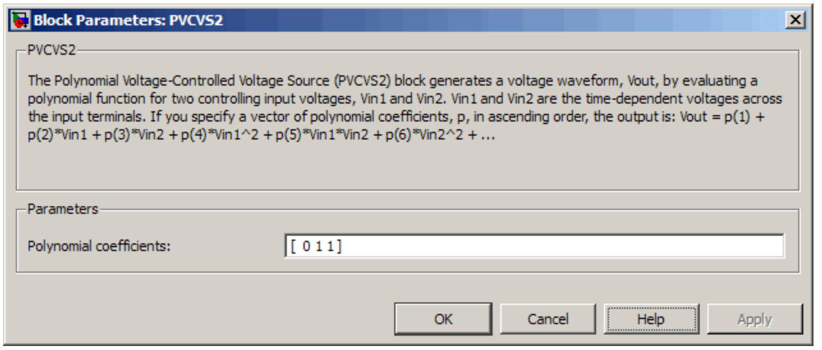
where:

- 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.

Dialog Box and 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

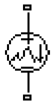
Purpose

Model lookup table current source

Library

SPICE-Compatible Components/Sources

Description

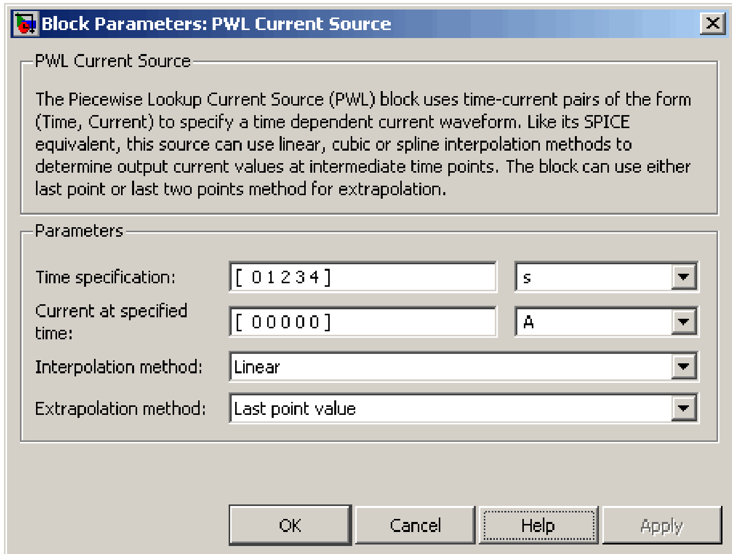


The PWL Current Source block represents a current source that you specify in lookup table form using a vector of time values and a vector of the corresponding current values. You must specify at least four time-current value pairs. The block generates a time-dependent current based on these time-current 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.

Dialog Box and 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 Polinomial (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

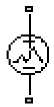
Purpose

Model lookup table voltage source

Library

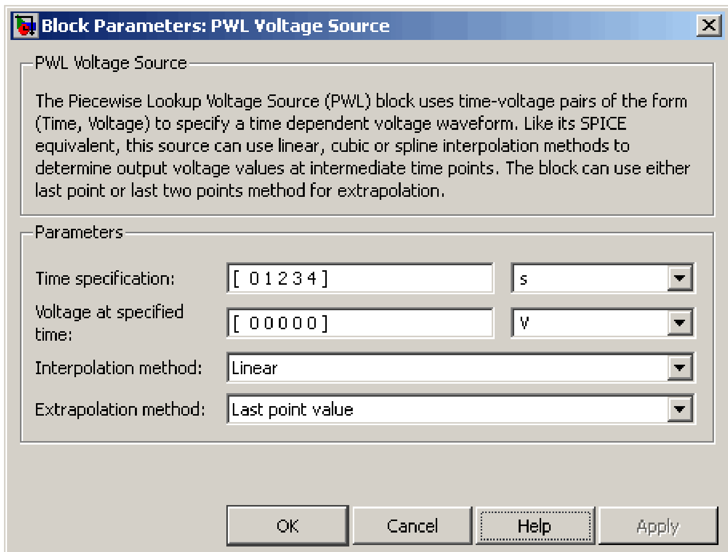
SPICE-Compatible Components/Sources

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.

Dialog Box and 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 Polinomial (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. Teulkolsky, and W.T. Wetterling *Numerical Recipes in C: The Art of Scientific Computing* Cambridge University Press, 1992.

See Also

PWL Current Source

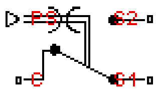
Purpose

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.

During switching, the relay behaves as follows:

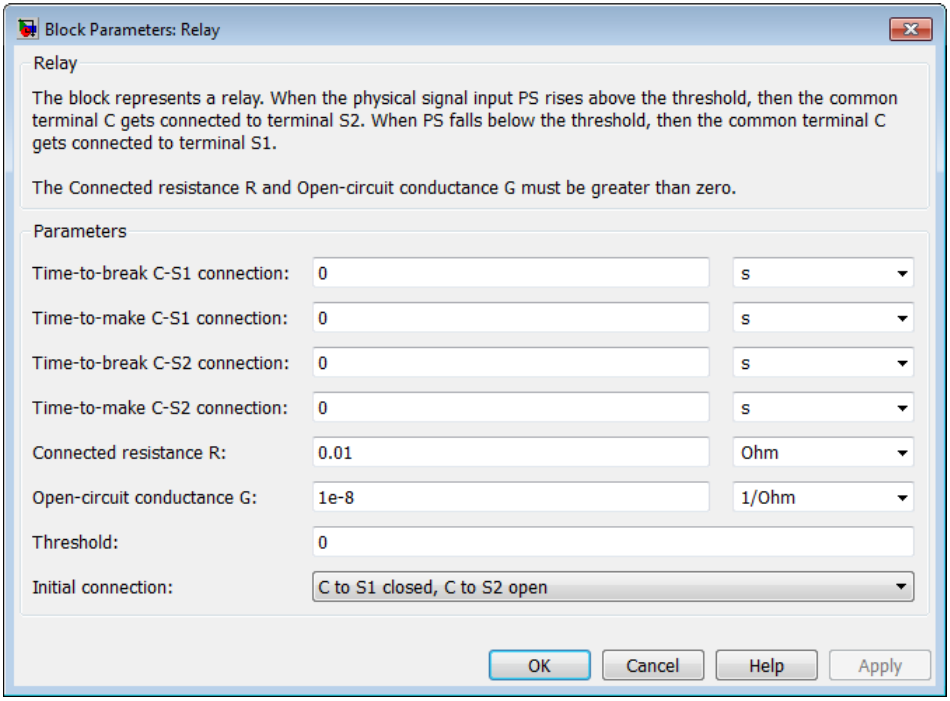
- When the relay closes, the C to S1 connection breaks open after delay **Time-to-break C-S1 connection**. The C to S2 connection closes after delay **Time-to-make C-S2 connection**.
- When the relay opens, the C to S2 connection breaks open after delay **Time-to-break C-S2 connection**. The C to S1 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.

Dialog Box and 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` Ω.

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`.

Initial connection
For the initial state of the relay, select one of the following options:

- `C to S1 closed, C to S2 open` — The common port C connects to the S1 contact. This is the default option.
- `C to S1 open, C to S2 closed` — The common port C connects to the S2 contact.

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

Simscape Switch

Resistor

Purpose

Model linear resistor with optional noise

Library

Passive Devices

Description

The Resistor block models a linear resistor, described with the following equation:

$$i = v/R$$

where:

- i is the current.
- v is the voltage.
- R is the resistance.

Optionally, 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 $4kT/R$).

If you set the **Noise mode** parameter to `Disabled`, then no noise is added, and the component behavior is identical to the Simscape Foundation library Resistor block.

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);
  ```

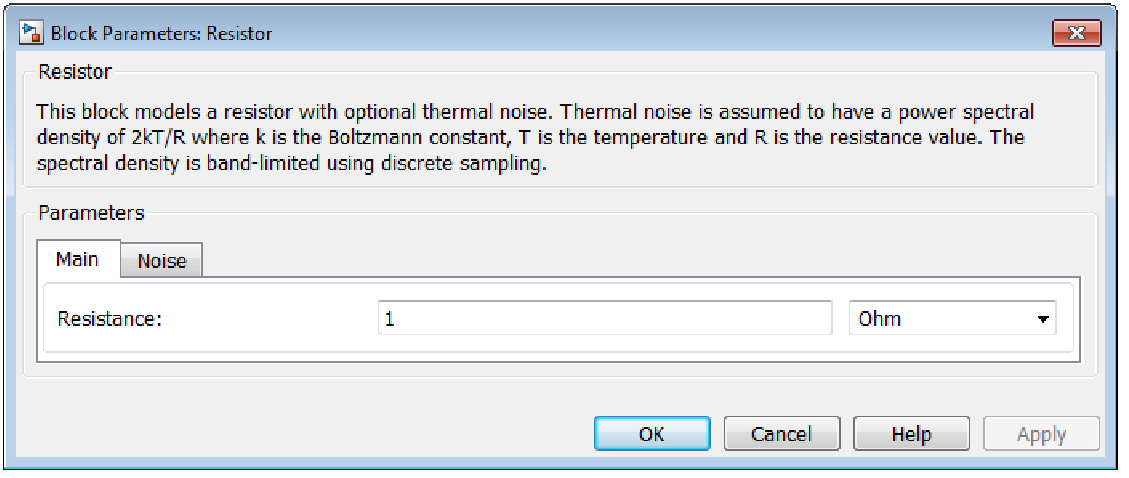
- `Repeatable` — The block uses a hidden parameter, called `auto_seed`, to always start the simulation with the same random number. The value of `auto_seed` is set whenever you copy the Resistor block from the block library to the model, or when you make a new copy of the Resistor block from an existing one in a model. 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.

Dialog Box and Parameters

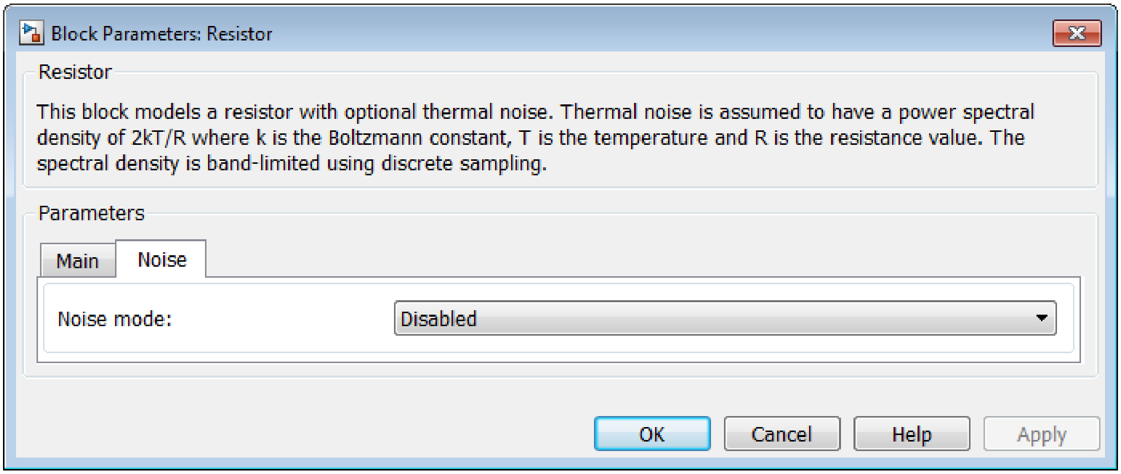
Main Tab



Resistance

The resistance value. The default value is 1 Ω.

Noise Tab



Noise mode

Select the noise option:

- `Disabled` — No noise is produced by the resistor. This is the default.
- `Enabled` — Resistor generates thermal noise current, and the associated parameters become visible on the **Noise** tab.

Device simulation temperature

The temperature of the thermal resistor at the start of the simulation. The default value is `25` °C.

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.

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, Voltage Source.

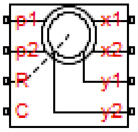
Purpose

Model rotary transformer that measures motor rotation angle

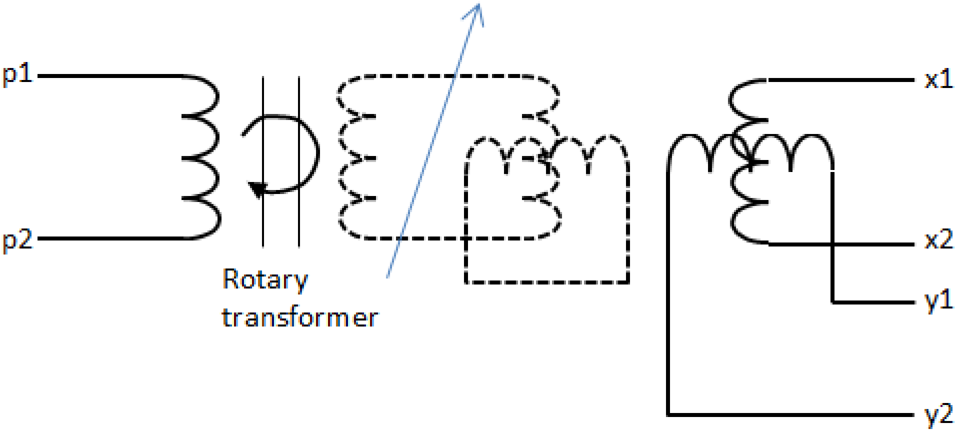
Library

Sensors

Description



The Resolver block models a generic resolver, which 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:

$$K_x = R\cos(N\Theta)$$

$$K_y = R\sin(N\Theta)$$

$$v_x = K_x v_p$$

$$v_y = K_y v_p$$

$$i_p = -K_x i_x - K_y i_y$$

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.
- Θ 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.
- Θ is the rotor angle.

It is assumed that coupling between the two secondary windings is zero.

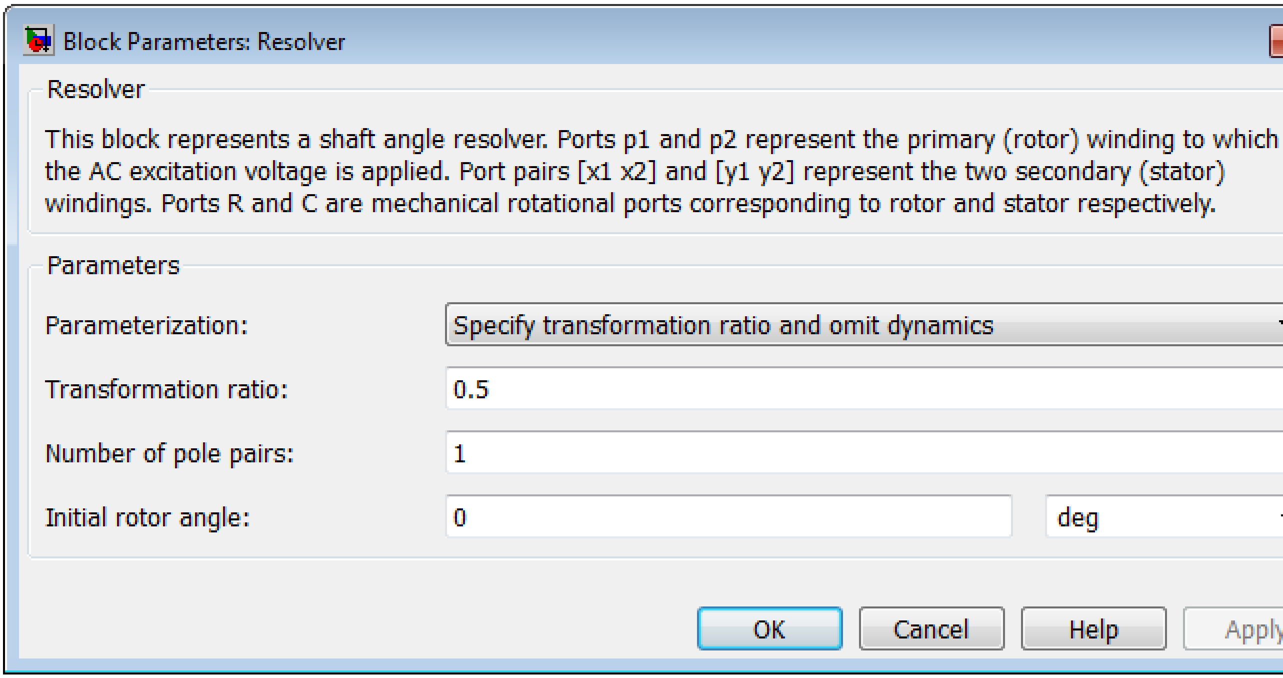
Datasheets typically do not quote the coefficient of coupling and inductance parameters, but instead give the transformation ratio R and measured impedances. If you select `Specify transformation ratio and measured impedances` for the **Parameterization** parameter, then the values you provide are used to determine values for the equation coefficients, as defined above.

Basic 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.

Dialog Box and Parameters



Parameterization

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 1-560. This is the default option.

- `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 1-561.
- `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 1-561. This model is valid for any input waveform.

Transformation ratio
The ratio between peak output voltage and peak input voltage assuming negligible secondary voltage drop due to resistance and inductance. This parameter is only visible when you select `Specify transformation ratio and omit dynamics` or `Specify transformation ratio and measured impedances` for the **Parameterization** parameter. If you select `Specify transformation ratio and measured impedances` for the **Parameterization** parameter, then the transformation ratio takes into account the voltage drop due to primary winding resistance. The default value is `0.5`.

Rotor resistance
This is the rotor (or equivalently the primary) ohmic resistance. This parameter is only visible when you select `Specify transformation ratio and measured impedances` or `Specify equation parameters directly` for the **Parameterization** parameter. The default value is `70` Ω.

Stator resistance

This is the secondary winding ohmic resistance. It is assumed that both secondaries have the same resistance. This parameter is only visible when you select `Specify transformation ratio and measured impedances` or `Specify equation parameters directly` for the **Parameterization** parameter. The default value is `180` Ω.

Rotor reactance

This is the rotor (or equivalently the primary) reactance with the secondary windings open-circuit. This parameter is only visible when you select `Specify transformation ratio and measured impedances` for the **Parameterization** parameter. The default value is `100` Ω.

Stator reactance

This is the stator (or equivalently the secondary) reactance with the primary winding open-circuit. This parameter is only visible when you select `Specify transformation ratio and measured impedances` for the **Parameterization** parameter. The default value is `300` Ω.

Frequency at which reactances and transformation ratio are specified

This is the frequency of the sinusoidal source used when measuring the reactances. This parameter is only visible when you select `Specify transformation ratio and measured impedances` for the **Parameterization** parameter. The default value is `10` kHz.

Rotor inductance

This is the rotor (or equivalently the primary) inductance L_p. This parameter is only visible when you select `Specify equation parameters directly` for the **Parameterization** parameter. The default value is `0.0016` H.

Stator inductance

This is the stator (or equivalently the secondary) inductance L_s. This parameter is only visible when you select `Specify equation` `parameters directly` for the **Parameterization** parameter. The default value is `0.0048` H.

Peak coefficient of coupling
: This is the peak coefficient of coupling between the primary and secondary windings. The parameter value should be greater than zero and less than one. This parameter is only visible when you select `Specify equation parameters directly` for the **Parameterization** parameter. The default value is `0.35`.

Number of pole pairs
: The number of pole pairs on the rotor. The default value is `1`.

Initial rotor angle
: The initial angle of the rotor, Θ. The default value is `0` degrees.

Ports

The block has the following ports:

`p1`, `p2`
: Electrical ports of the primary winding.

`x1`, `x2`
: Electrical ports of the first secondary winding.

`y1`, `y2`
: Electrical ports of the second secondary winding.

`R`, `C`
: Mechanical rotational ports.

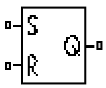
Purpose

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 1-568 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	0
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 Vcc (the **Supply voltage** parameter value), and output low is zero volts.

Basic Assumptions and Limitations

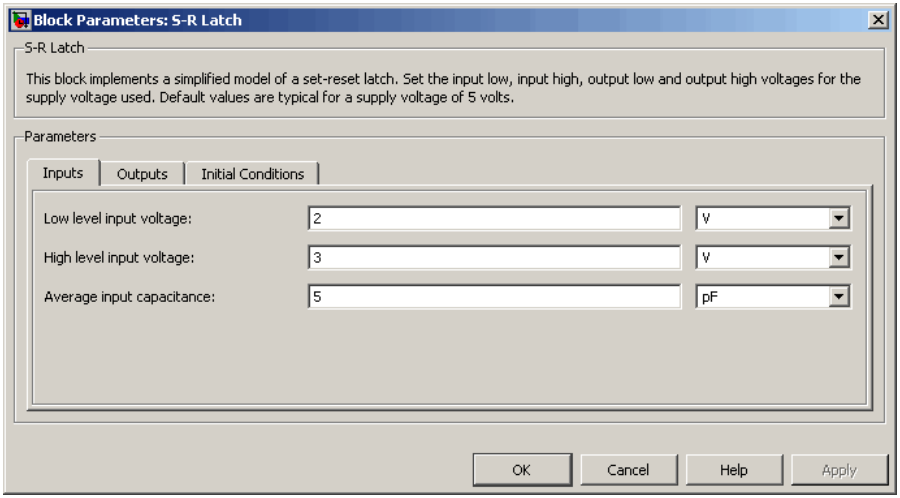
The block does not model the internal individual MOSFET devices that make up the gate (except for the final MOSFET pair if you select the `Quadratic` option for the **Output current-voltage relationship** parameter). This limitation has the following implications:

- 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.

Dialog Box and Parameters

Inputs Tab

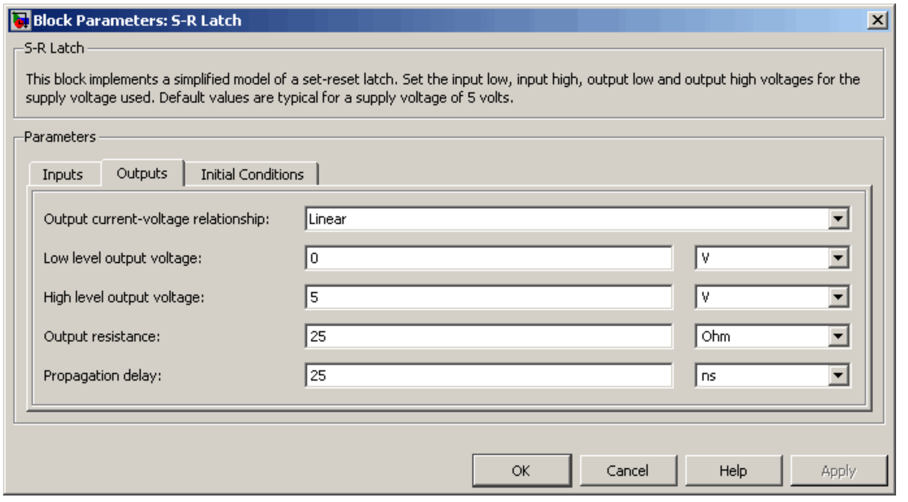


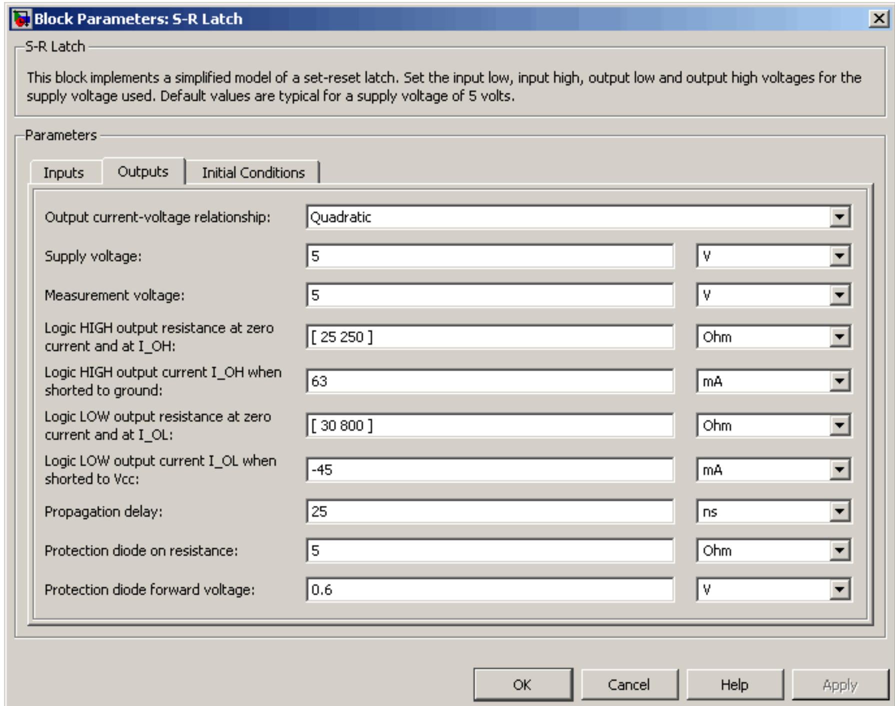
Low level input voltage
Voltage value less than which the block interprets the input voltage as `LOW`. The default value is `2` V.

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


Block Parameters: S-R Latch
S-R Latch
This block implements a simplified model of a set-reset latch. Set the input low, input high, output low and output high voltages for the supply voltage used. Default values are typical for a supply voltage of 5 volts.
Parameters
Inputs
Outputs
Initial Conditions
Output current-voltage relationship:
Linear
Low level output voltage:
0
V
High level output voltage:
5
V
Output resistance:
25
Ohm
Propagation delay:
25
ns
OK
Cancel
Help
Apply



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` Ω. 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 ]` Ω. 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 ]` Ω. 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.

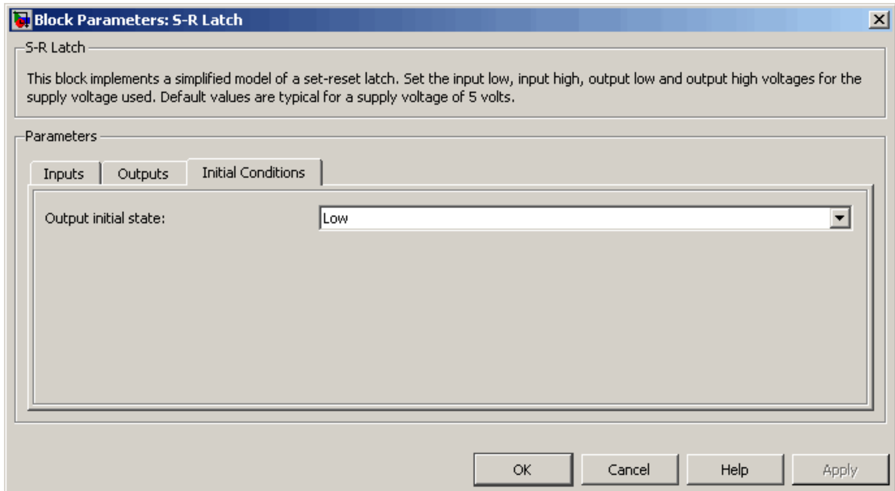
Protection diode on resistance

The gradient of the voltage-current relationship for the protection diodes when forward biased. The default value is `5` Ω. 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.

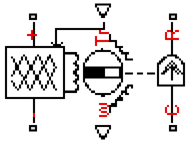
Purpose

Brushless motor model with closed-loop torque control

Library

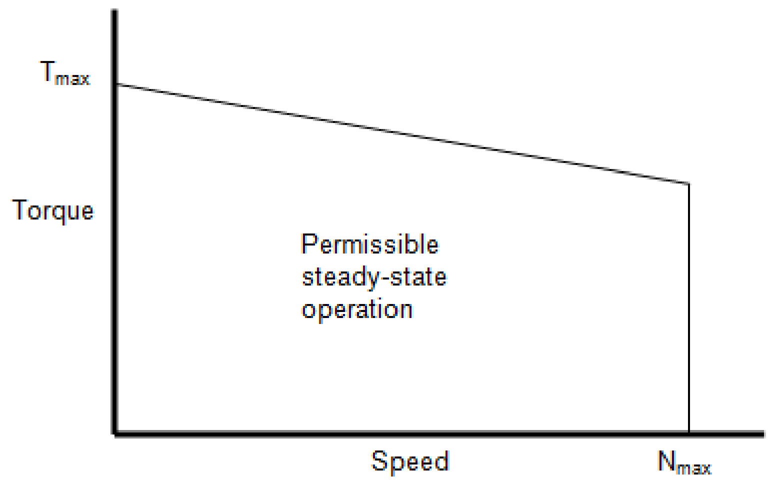
Rotational Actuators

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.

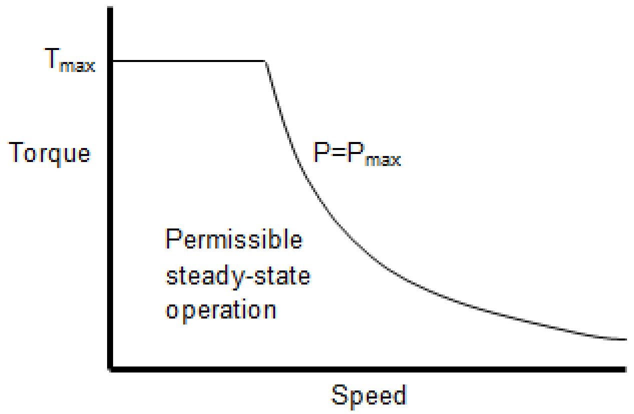
The block permits only the range of torques and speeds that the torque-speed envelope defines. In the default block configuration, you specify this data in the block dialog box as a set of speed data points and corresponding maximum torque values. The following figure shows a typical torque-speed envelope 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 models the electrical losses as the sum of 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 τ 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 ω is the speed and k_w is a constant. This represents iron losses due to eddy currents.

The block produces a positive torque acting 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".

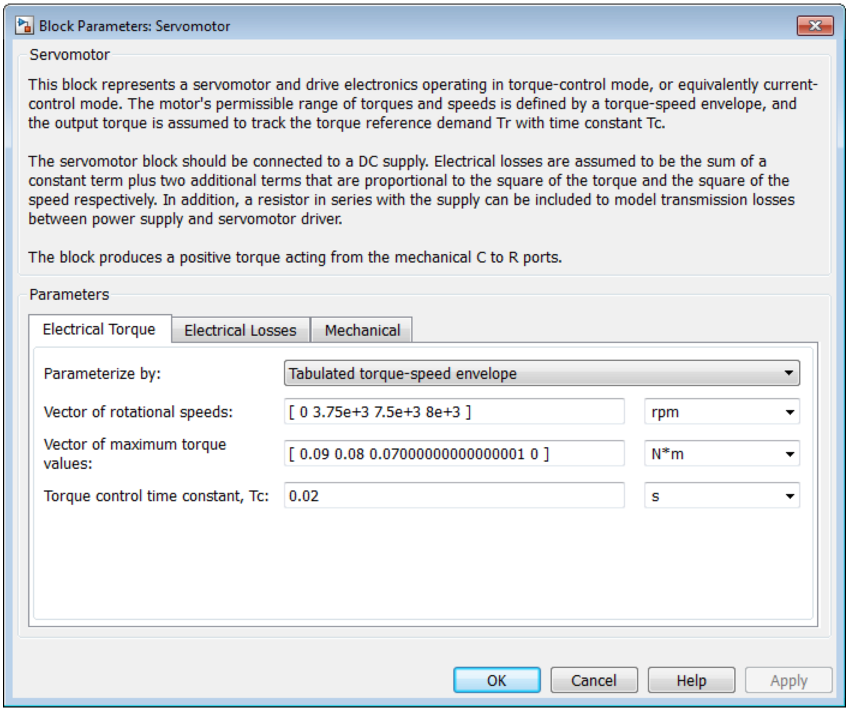
Basic Assumptions and Limitations

This model is based on the following assumptions:

- The motor driver tracks a torque demand with a time constant `Tc`.
- Motor speed fluctuations due to mechanical load do not affect the motor torque tracking.

Dialog Box and Parameters

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.

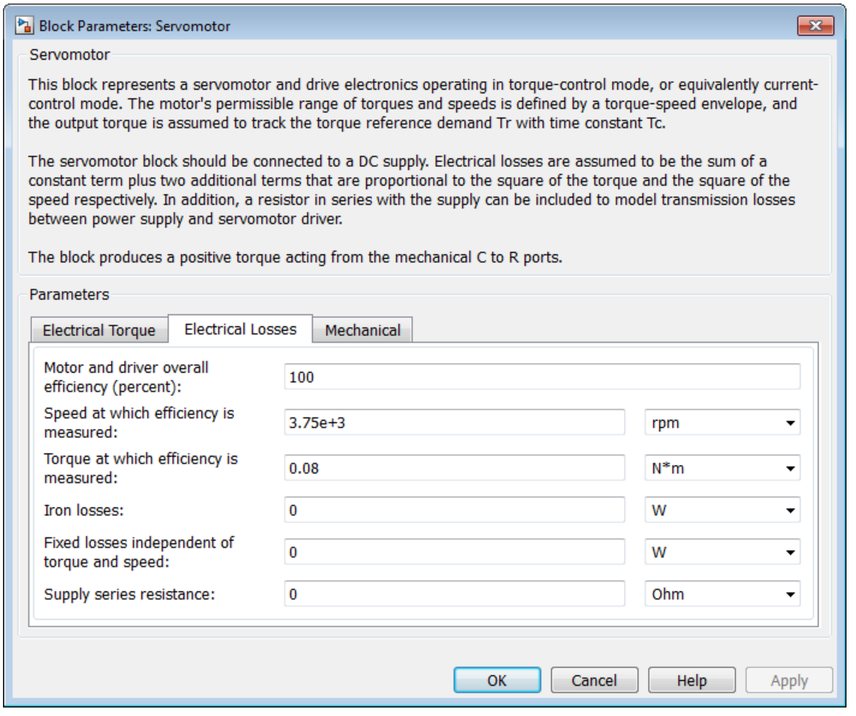
Vector of maximum torque values
Maximum torque values for permissible steady-state operation. This parameter is visible only if you select `Tabulated torque-speed envelope` for the **Parameterize by** parameter. These values correspond to the speeds in the **Vector of rotational speeds** parameter and 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



Motor and driver overall efficiency (percent)

The block defines overall efficiency as

$$\eta = 100 \frac{\tau_0 \omega_0}{\tau_0 \omega_0 + P_0 + k \tau_0^2 + k_w \omega^2}$$

where:

- τ_0 represents the **Torque at which efficiency is measured**.
- ω_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.
- $k_w\omega^2$ 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.

Speed at which efficiency is measured
Speed that the block uses to calculate torque-dependent electrical losses. The default value is `3.75e+03` rpm.

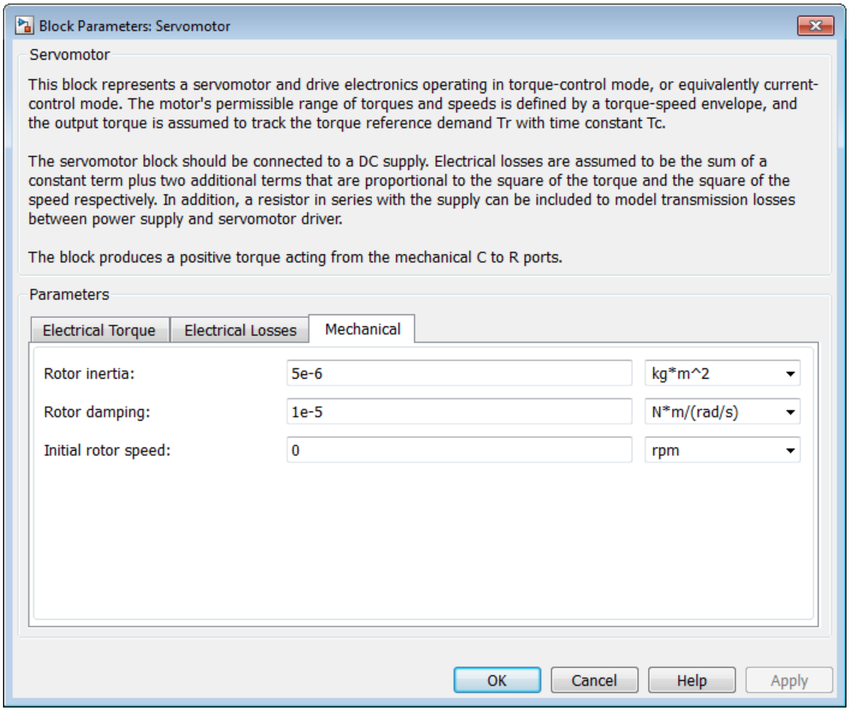
Torque at which efficiency is measured
Torque that the block uses to calculate torque-dependent electrical losses. The default value is `0.08` Nm.

Iron losses
Iron losses at the speed and torque at which efficiency is defined. 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. The default value is `0` W.

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` Ω.

Mechanical Tab



Rotor inertia
: Rotor resistance to change in motor motion. The default value is `5e-06` kg*m^2. The value can be zero.

Rotor damping
: Rotor damping. The default value is `1e-05` N*m/(rad/s). The value can be zero.

Initial rotor speed
: Rotor speed at the start of the simulation. The default value is `0` rpm.

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

Generic Rotary Actuator, DC Motor, Induction Motor, Shunt Motor, and Universal Motor.

SFFM Current Source

Purpose

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:

$$I_{out} = IO + IA * \sin\left(\left(2\pi * FC * Time\right) + MI * \sin\left(2\pi * FS * Time\right)\right)$$

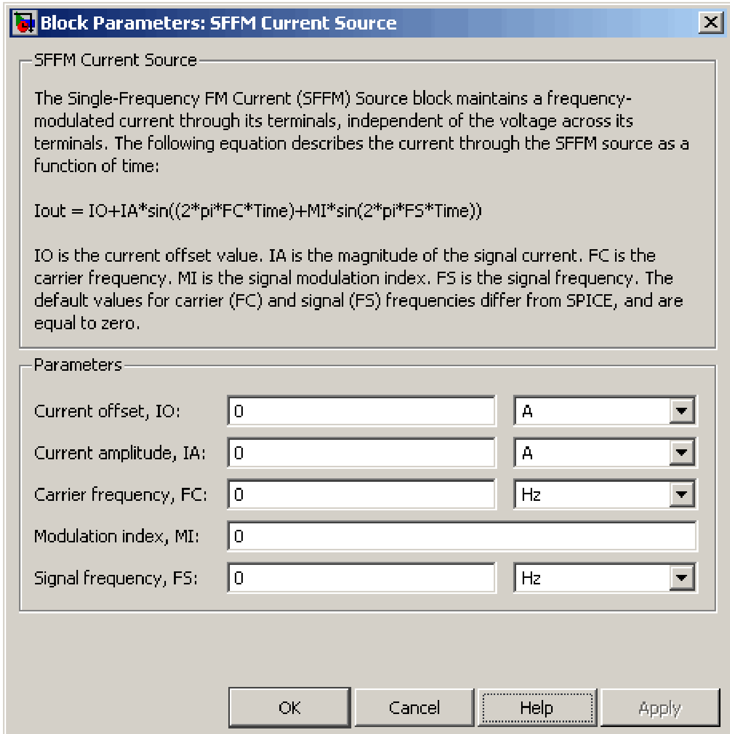
where:

- *I0* 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.

Dialog Box and 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

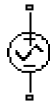
Purpose

Model single-frequency FM voltage source

Library

SPICE-Compatible Components/Sources

Description



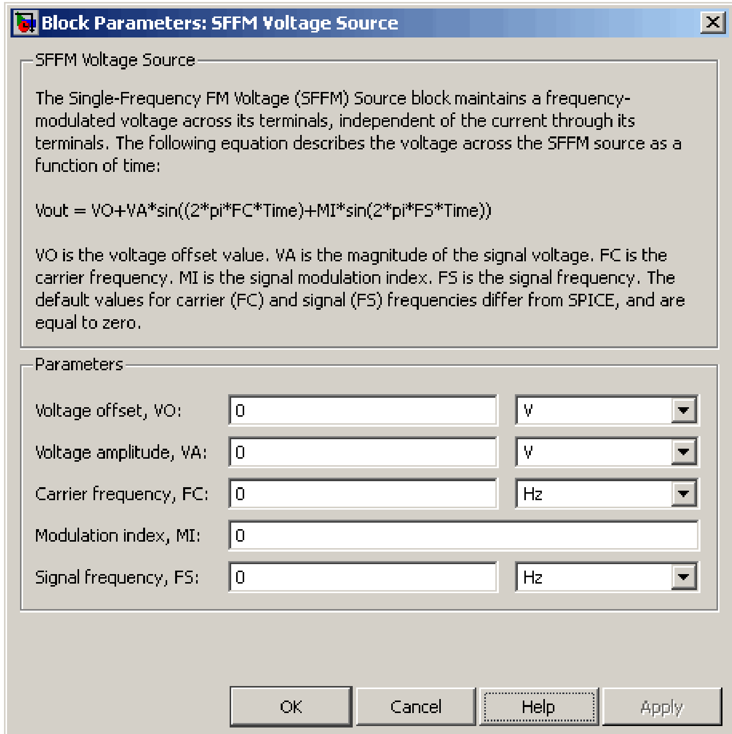
The SFFM Voltage Source block represents a single-frequency voltage source whose frequency-modulated output voltage value is independent of the current through the source. The following equation describes the output voltage as a function of time:

$$V_{out} = VO + VA * \sin\left(\left(2\pi * FC * Time\right) + MI * \sin\left(2\pi * FS * Time\right)\right)$$

where:

- *V0* 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.

Dialog Box and 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

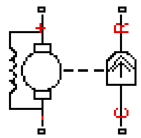
Purpose

Model electrical and torque characteristics of shunt motor

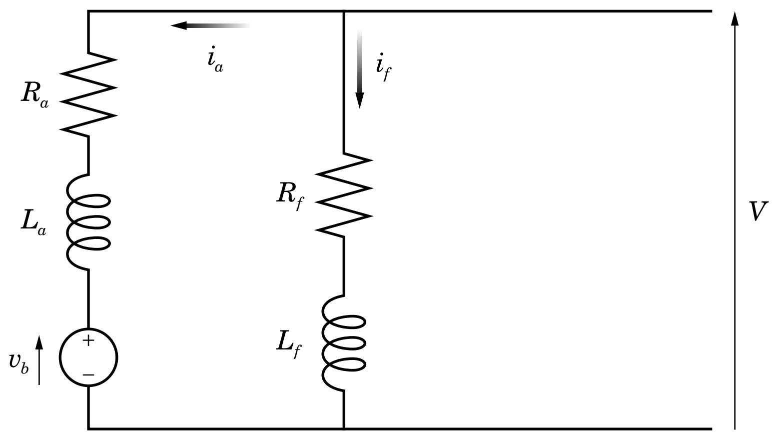
Library

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 ω 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:

$$i_f = \frac{V}{R_f}$$
$$i_a = \frac{V}{R_a}\left(1 - \frac{L_{af} w}{R_f}\right)$$

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

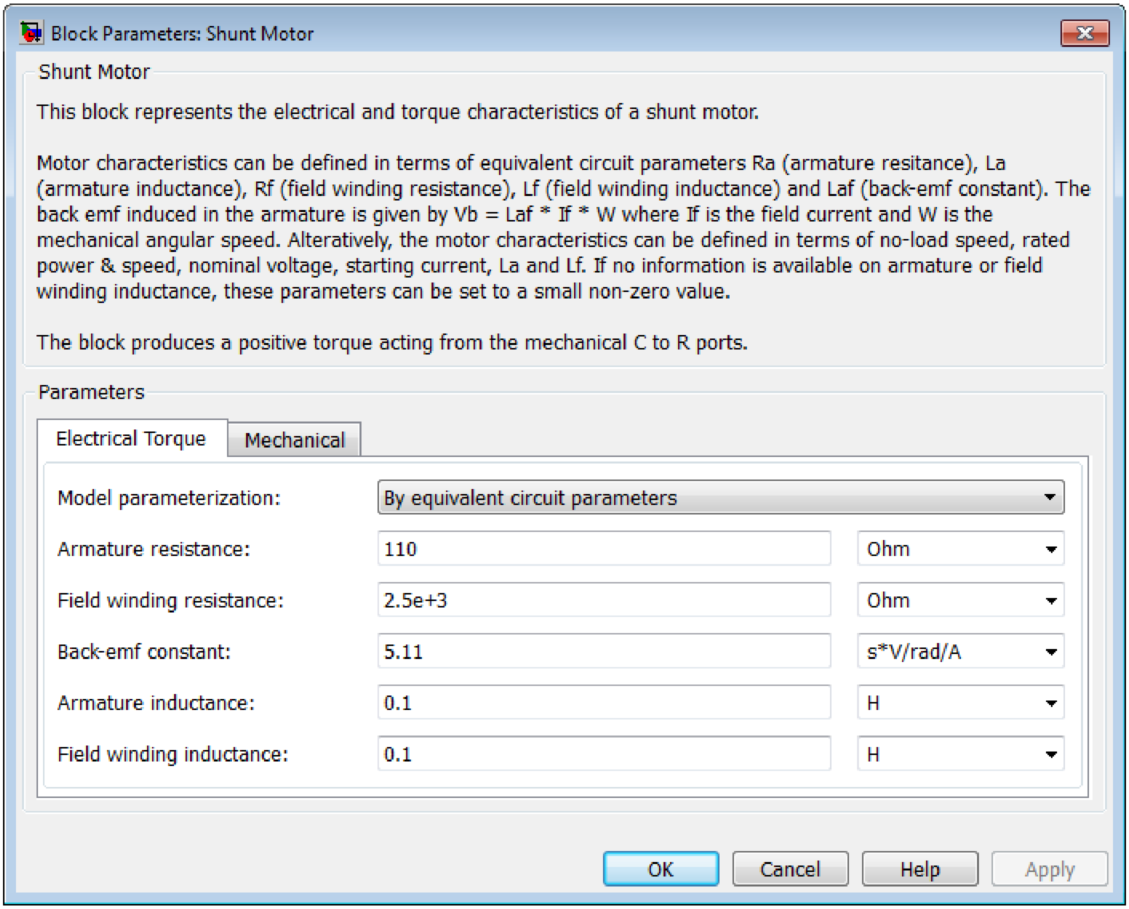
The block has two optional thermal ports, one per winding, hidden by default. To expose the thermal ports, right-click the block in your model, and then from the context menu select **Simscape block choices** > **Show thermal port**. This action displays the thermal ports on the block icon, and adds 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”.

Dialog Box and Parameters

- "Electrical Torque Tab" on page 1-593
- "Mechanical Tab" on page 1-596
- "Temperature Dependence Tab" on page 1-597
- "Thermal Port Tab" on page 1-598

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` Ω.

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` Ω.

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.

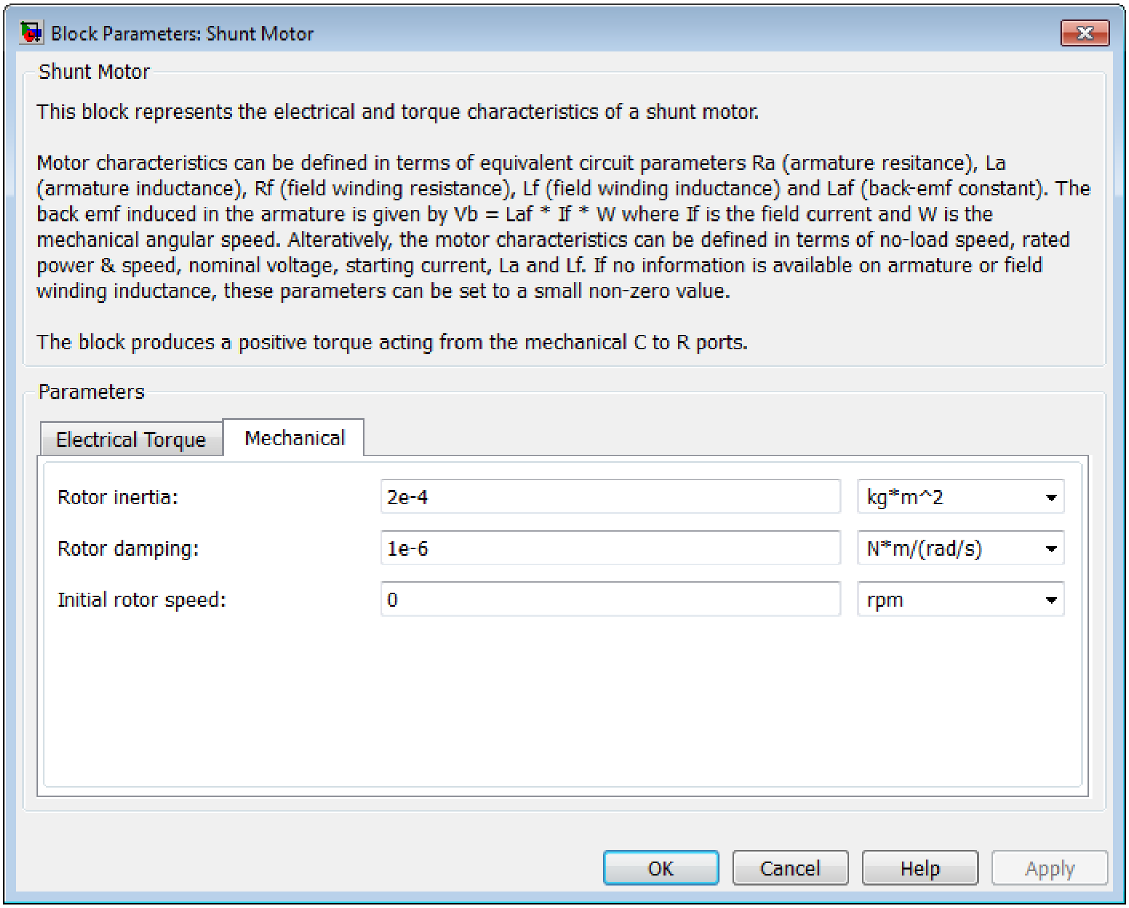
Rated speed (at rated load)
Motor speed at the rated load. This parameter is only visible when you select By rated power, rated speed & no-load speed for the **Model parameterization** parameter. The default value is 4e+03 rpm.

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

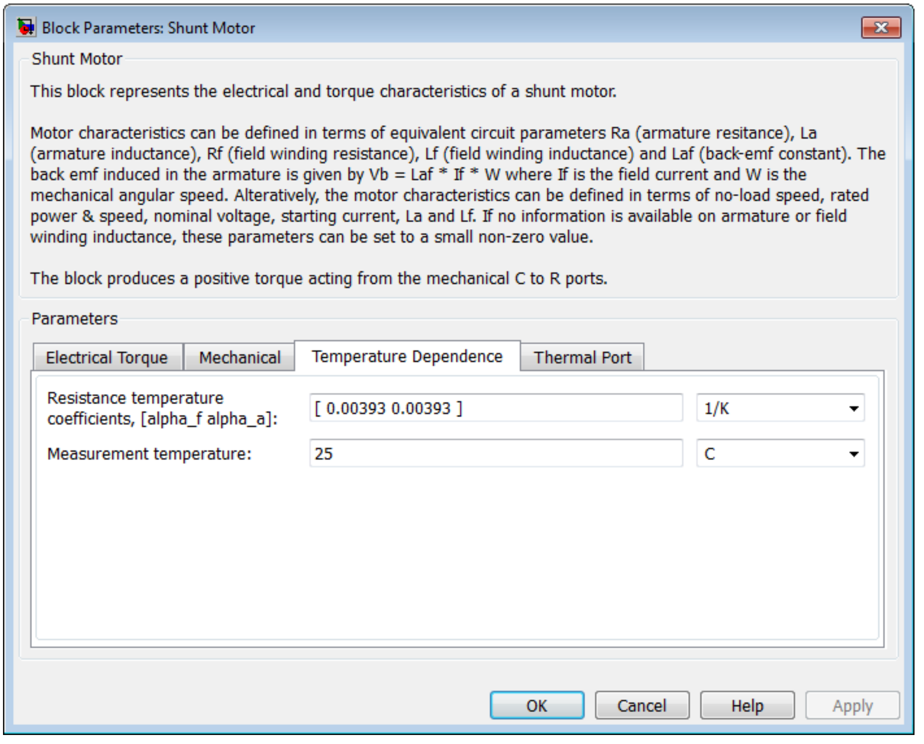
Rotor inertia. The default value is `2e-04` kg*m^2. The value can be zero.

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 1-897.



Resistance temperature coefficients, [alpha_f alpha_a]

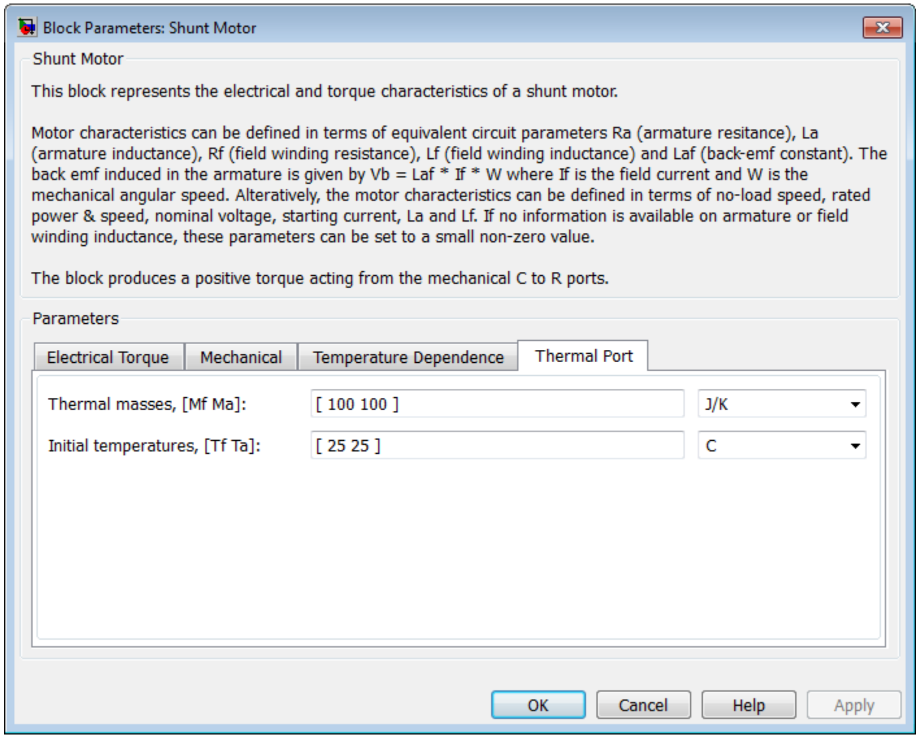
A 1 by 2 row vector defining the coefficient α in the equation relating resistance to temperature, as described in "Thermal Model for Actuator Blocks". The first element corresponds to the field winding, and the second to the armature. The default value is for copper, and is `[ 0.00393 0.00393 ]` 1/K.

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 1-897.



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 1-897.

`Ha`
: Armature winding thermal port. For more information, see “Thermal Ports” on page 1-897.

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, and Universal Motor.

Sinusoidal Current Source

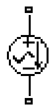
Purpose

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:

$$I_{out}\left(Time < TD\right) = IO$$

$$I_{out}\left(Time \geq TD\right) = IO + IA * e^{-(Time-TD)*DF} * \sin\left(2\pi * FREQ * \left(Time - TD\right)\right)$$

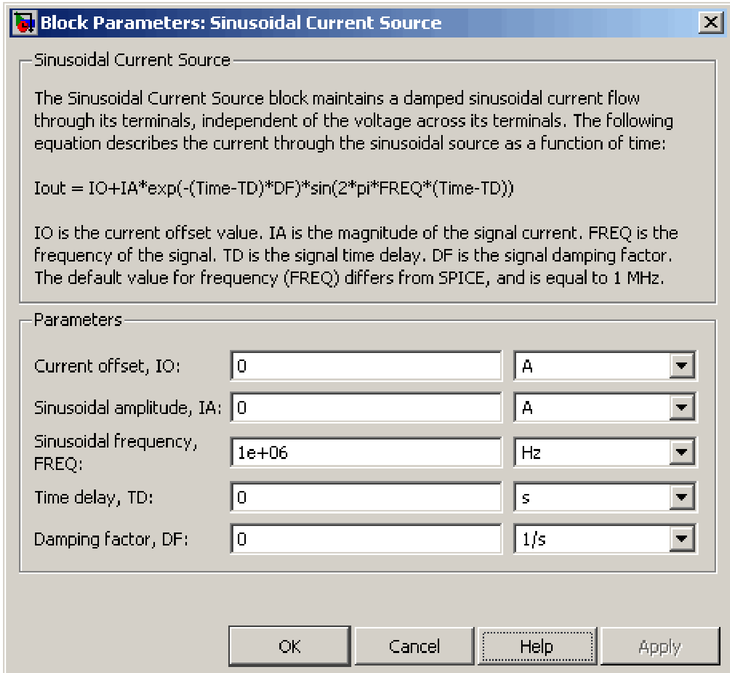
where:

- *I0* 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.

Dialog Box and 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 multiples 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

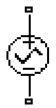
Purpose

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:

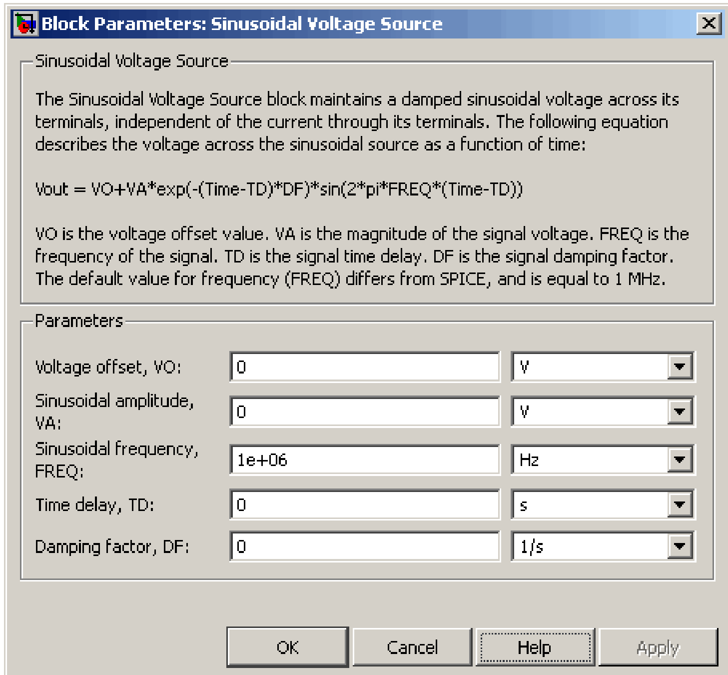
$$V_{out}\left(Time < TD\right) = VO$$

$$V_{out}\left(Time \geq TD\right) = VO + VA * e^{-(Time - TD) * DF} * \sin\left(2\pi * FREQ * \left(Time - TD\right)\right)$$

where:

- *V0* 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.

Dialog Box and Parameters



Voltage offset, V0

The magnitude of the time-independent part of the output voltage. The default value is `0` V.

Sinusoidal amplitude, VA

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 multiples 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.

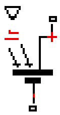
See Also

Sinusoidal Current Source

Purpose Solar cell model

Library Sources

Description The Solar Cell block represents a solar cell current source.



The solar cell model includes the following components:

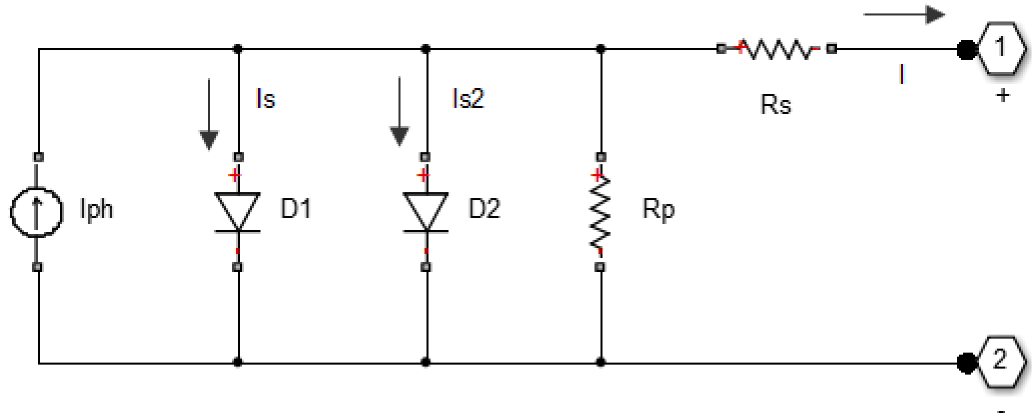
- "Solar-Induced Current" on page 1-607
- "Temperature Dependence" on page 1-610
- "Thermal Port" on page 1-611

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:

$$I = I_{ph} - I_s * \left(e^{(V+I*R_s)/(N*V_t)} - 1\right) - I_{s2} * (e^{(V+I*R_s)/(N_2*V_t)} - 1) - (V + I * R_s)/R_p$$

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:

$$I_{ph}(t) = I_{ph} * \left(1 + TIPH1 * \left(T - T_{meas}\right)\right)$$

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_s and the solar cell temperature T:

$$I_{s1}(T) = I_{s1} * \left(\frac{T}{T_{meas}}\right)^{\left(TXIS1/N\right)} * e^{\left(EG*\left(\frac{T}{T_{meas}}-1\right)/\left(N*V_t\right)\right)}$$

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:

$$I_{s2}(T) = I_{s2} * \left(\frac{T}{T_{meas}}\right)^{\left(TXIS2/N_2\right)} * e^{\left(EG*\left(\frac{T}{T_{meas}}-1\right)/\left(N_2*V_t\right)\right)}$$

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:

$$R_p(T) = R_p * \left(\frac{T}{T_{meas}} \right)^{TRP1}$$

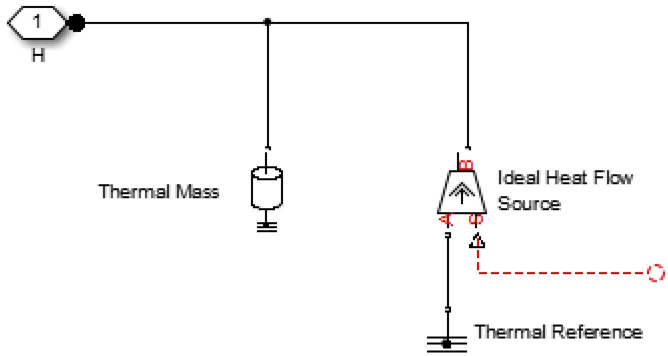
where *TRP1* is the **Temperature exponent for Rp, TRP1** parameter value.

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.

Solar Cell



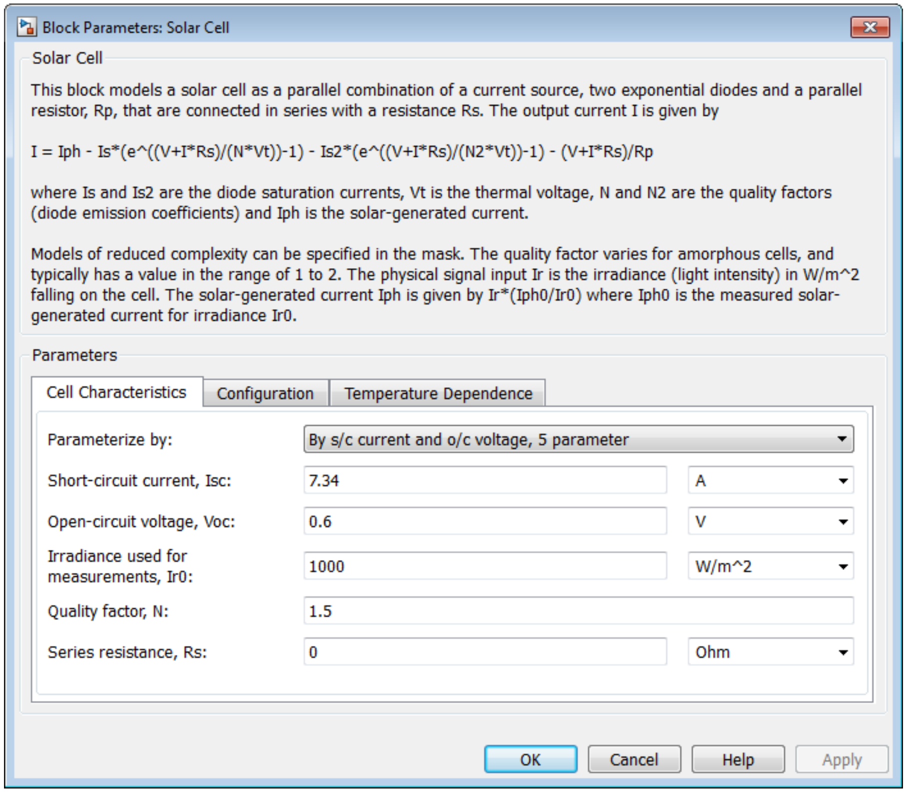
The internally generated heat in the solar cell is calculated according to the equivalent circuit diagram, shown at the beginning of the reference page, in the "Solar-Induced Current" on page 1-607 section. It is the sum of the i^2 ·R losses for each of the resistors plus the losses in each of the diodes.

The internally generated heat due to electrical losses is a separate heating effect to that of the solar irradation. 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.

Dialog Box and Parameters

- "Cell Characteristics Tab" on page 1-613
- "Configuration Tab" on page 1-616
- "Temperature Dependence Tab" on page 1-617
- "Thermal Port Tab" on page 1-618

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, Isc
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, Voc
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, Is
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^2.

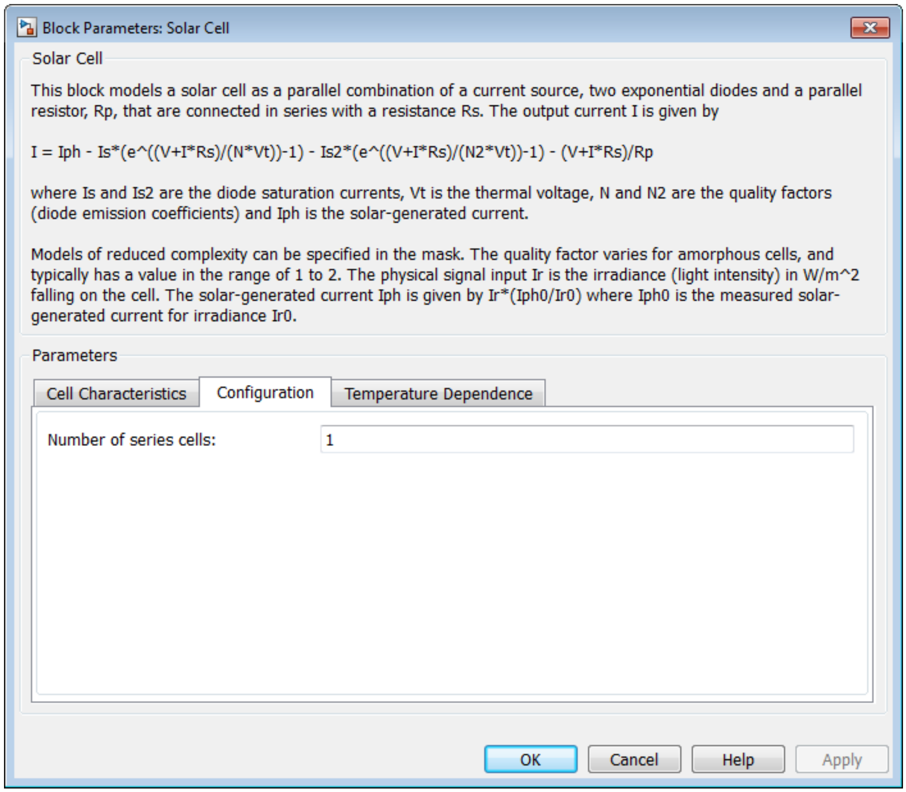
Quality factor, N
: The emission coefficient of the first diode. The default value is `1.5`.

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` Ω.

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 `inf` Ω.

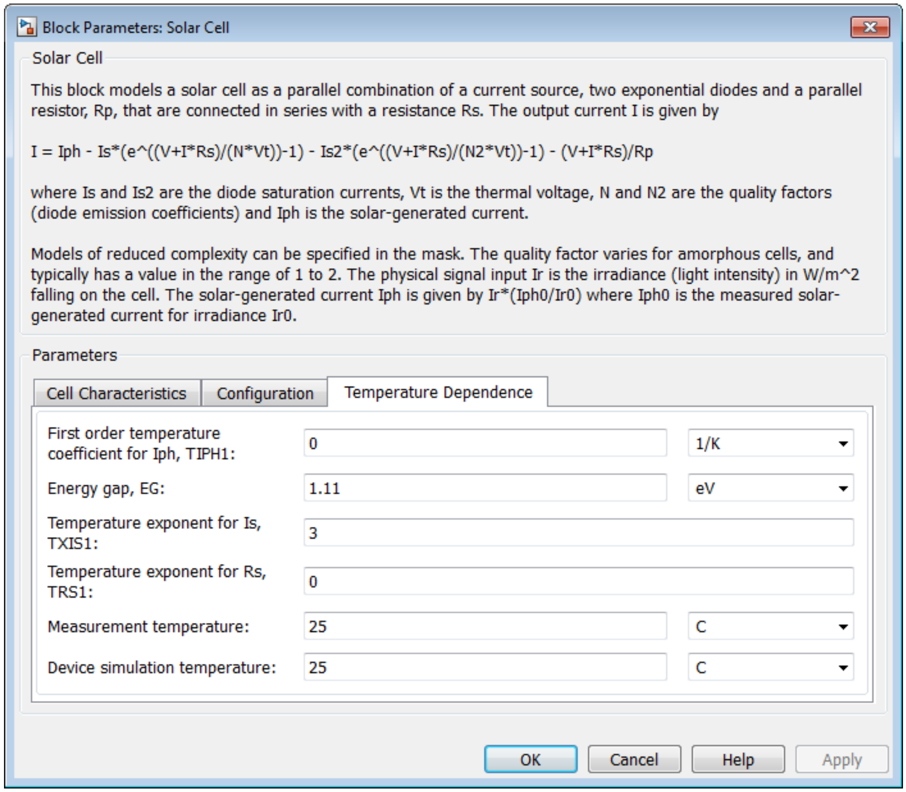
Configuration Tab



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 Iph, 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 Is, 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 Is2, 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 Rs, 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`.

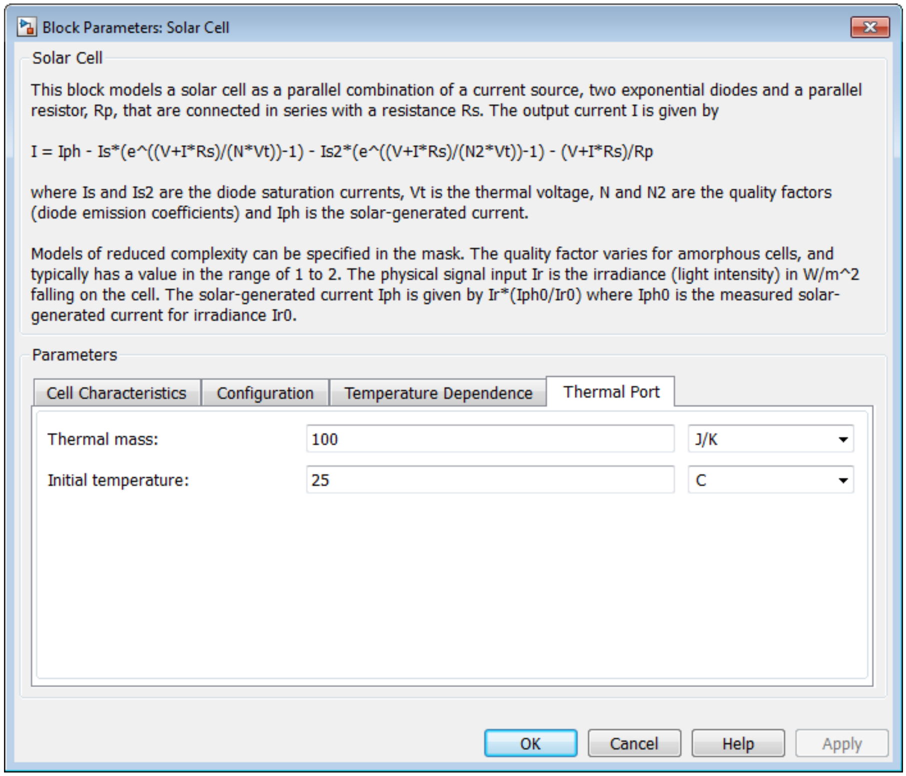
Temperature exponent for Rp, TRP1
The order of the exponential increase in the parallel resistance as temperature increases. This parameter is only visible when you select `By equivalent circuit parameters, 8 parameter` for the **Parameterize by** parameter. The default value is `0`. The value must be 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 1-611.



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:

`Ir`
: 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.

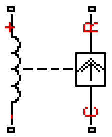
Purpose

Model electrical characteristics and generated force of solenoid

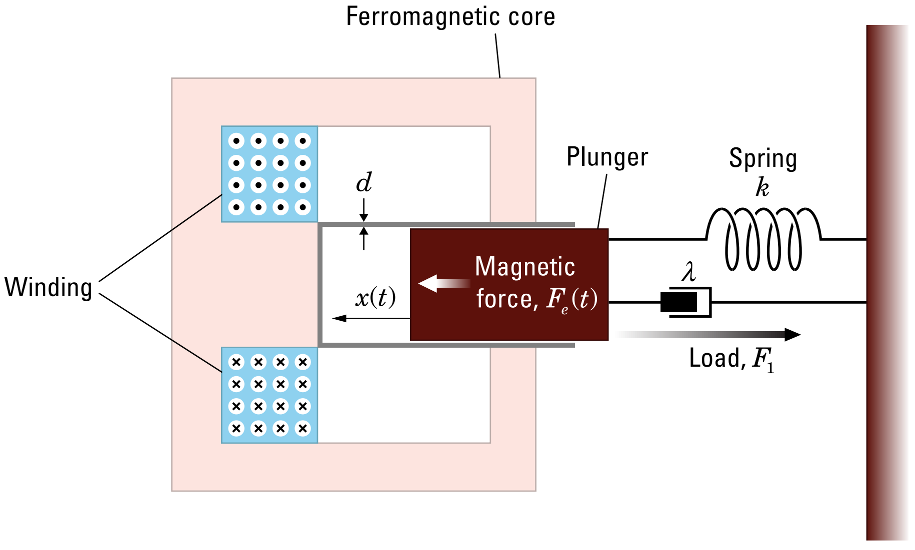
Library

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, λ 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 α and β 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 α and β by taking the two specified force and stroke measurements and substituting them into the preceding equation. It solves the resulting equations for α and β.

A positive current from the electrical + to - ports creates a negative force (i.e., a pulling force) from the mechanical C to R ports.

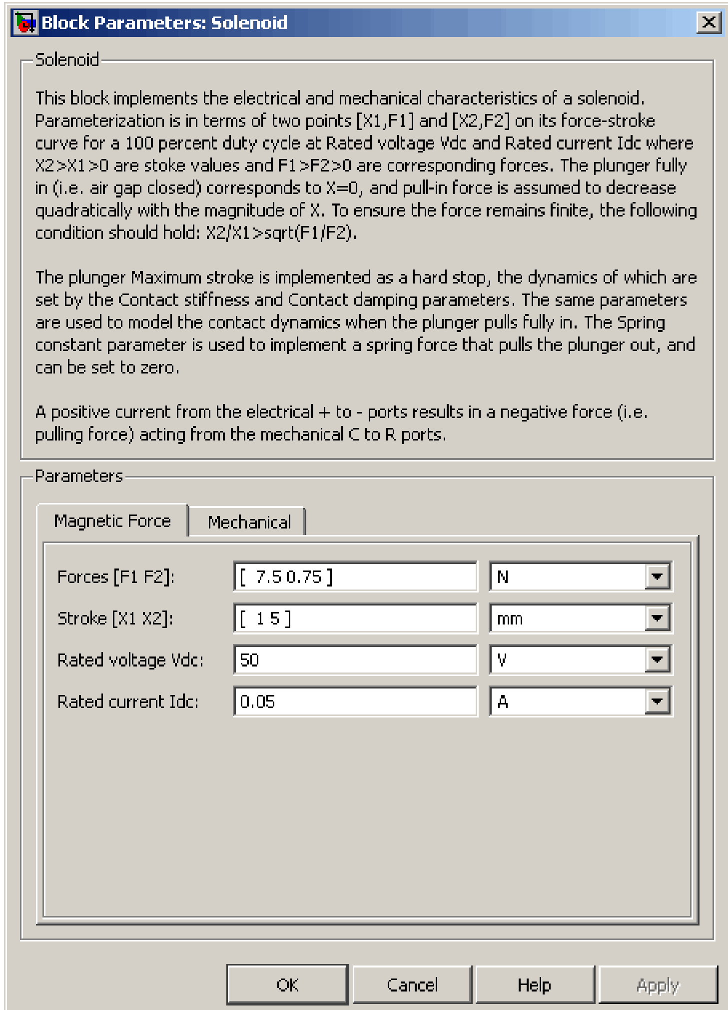
Thermal Port

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".

Dialog Box and Parameters

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 doesn't provide a force-stroke curve, set F1 to the holding torque (when X1 = 0) and F2 to the pull-in torque 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.

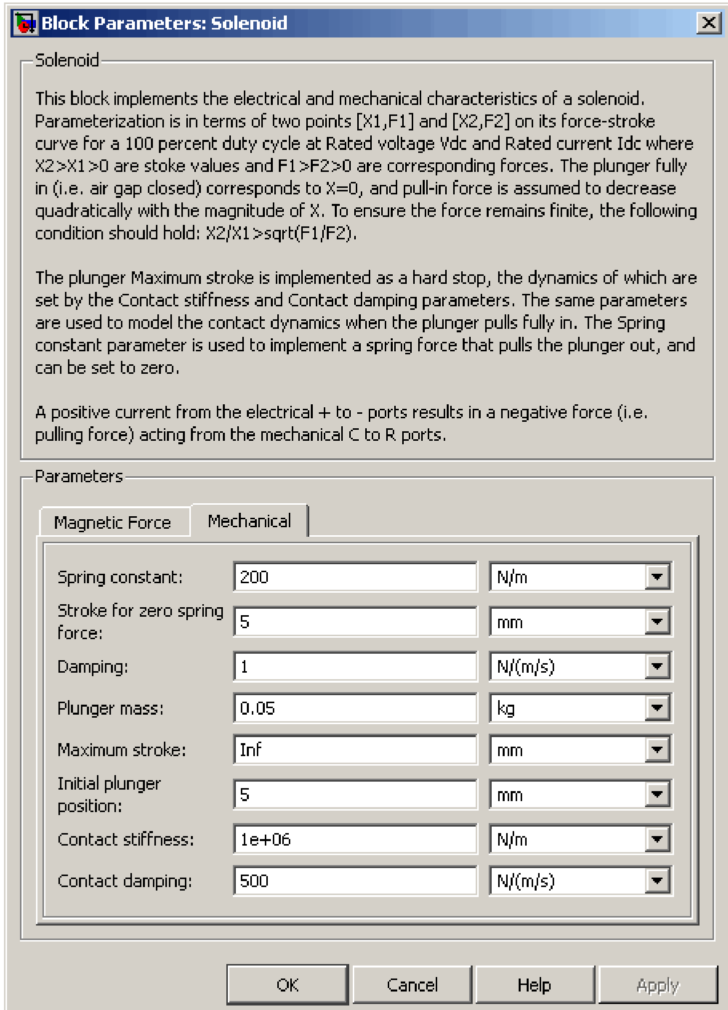
Rated voltage Vdc

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 λ 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.

Initial plunger position
The amount by which the plunger is displaced at the start of the simulation. The default value is `0` m.

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.

SPDT Switch

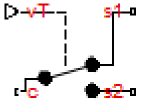
Purpose

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.

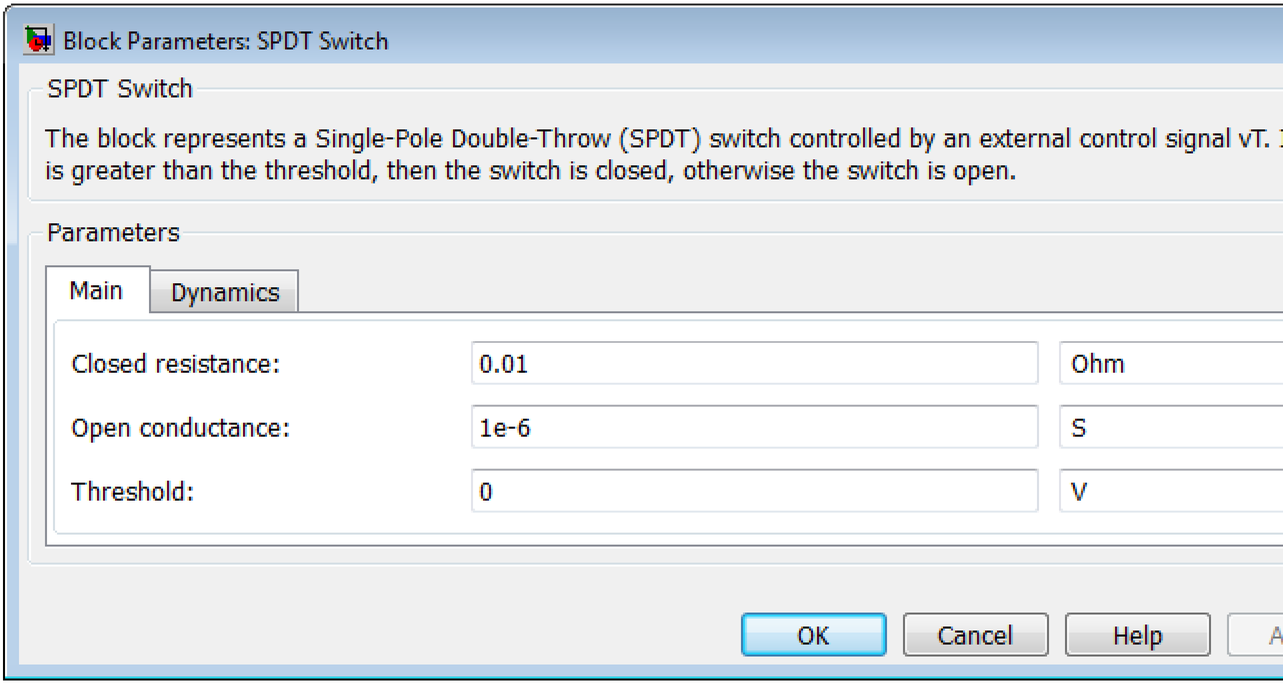
The switch is closed if the voltage presented at the `vT` control port exceeds the value of the **Threshold** parameter.

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`.

Dialog Box and Parameters

- "Main Tab" on page 1-629
- "Dynamics Tab" on page 1-630

Main Tab



Closed resistance

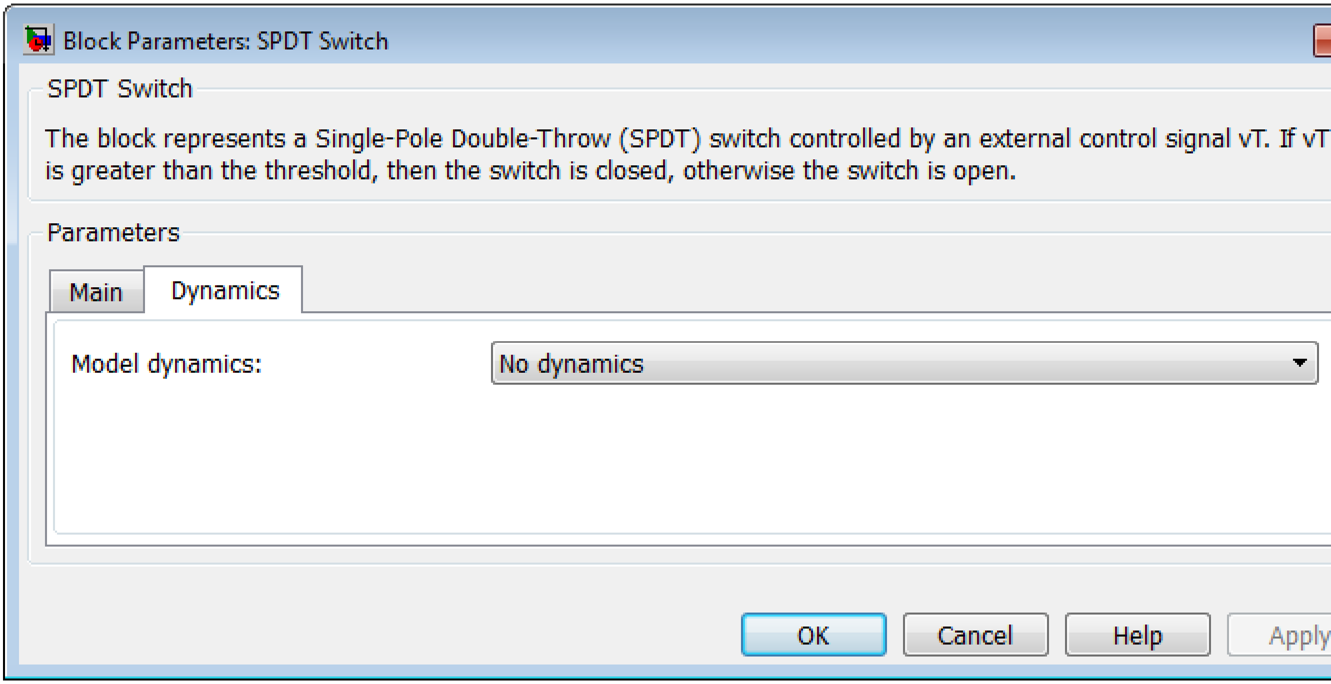
Resistance between the `c` and `s` electrical ports when the switch is closed. The value must be greater than zero. The default value is `0.01` Ω.

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` 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 vT 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, vT
: The value of the physical signal input `vT` 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:

`vT`
: Physical signal that opens and closes the switch.

`c`, `s1`, `s2`
: Electrical conserving ports.

See Also

DPDT Switch

DPST Switch

SPST Switch

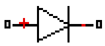
Simscape Switch

SPICE Diode

Purpose 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 1-632
- “Junction Charge Model” on page 1-634
- “Temperature Dependence” on page 1-635

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
$V_d > 80 * V_t$	$I_d = IS\left(\left(\frac{V_d}{V_t} - 79\right)e^{80} - 1\right) + V_d * G\min$
$80 * V_t \geq V_d \geq -3 * V_t$	$I_d = IS * (e^{V_d/V_t} - 1) + V_d * G\min$

Applicable Range of V_d Values	Corresponding I_d Equation
$-3*V_t > V_d \geq -BV$	$I_d = -IS\left(1+\frac{27}{\left(V_d/V_t\right)^3 e^3}\right)+V_d*G\min$
$V_d < -BV$	$I_d = -IBV*\left(e^{(-(BV+V_d)/V_t}-1\right)- IS*\left(1-\left(\frac{3}{e*BV/V_t}\right)^3\right)+V_d*G\min$

Where:

- IS is the **Saturation current, IS** parameter value.
- $V_t = N*k*T/q$
- N is the **Emission coefficient, ND** 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
$V_d < FC*VJ$	$Q_d = TT*I_d + CJO*VJ*\dfrac{1-\left(1-\dfrac{V_d}{VJ}\right)^{1-MG}}{1-MG}$
$V_d \geq FC*VJ$	$Q_d = TT*I_d + CJO*\left(F1+\dfrac{F3*(V_d - FC*VJ)+\left(\dfrac{MG}{2*VJ}\right)*\left(V_d^2 - (FC*VJ)^2\right)}{F2}\right)$

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.
- *CJ0* is the **Zero-bias junction capacitance CJ0** parameter value.
- *MG* is the **Grading coefficient MG** 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*:

$$IS(T) = IS * (T/T_{meas})^{\frac{XTI}{ND}} * e^{\left(\frac{T}{T_{meas}}-1\right)*\frac{EG}{V_t}}$$

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, ND** 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.
- $EG_{T_{meas}} = 1.16eV - \left(7.02e-4 * T_{meas}{}^{2}\right) / \left(T_{meas} + 1108\right)$
- $EG_T = 1.16eV - \left(7.02e-4 * T^2\right) / \left(T + 1108\right)$

The block provides the following relationship between the junction capacitance CJO and the diode temperature T:

$$CJO(T) = CJO * \left[1 + MG * \left(400e - 6 * \left(T - T_{meas} \right) - \frac{VJ(T) - VJ}{VJ} \right) \right]$$

where *CJO* is the **Zero-bias junction capacitance CJ0** parameter value.

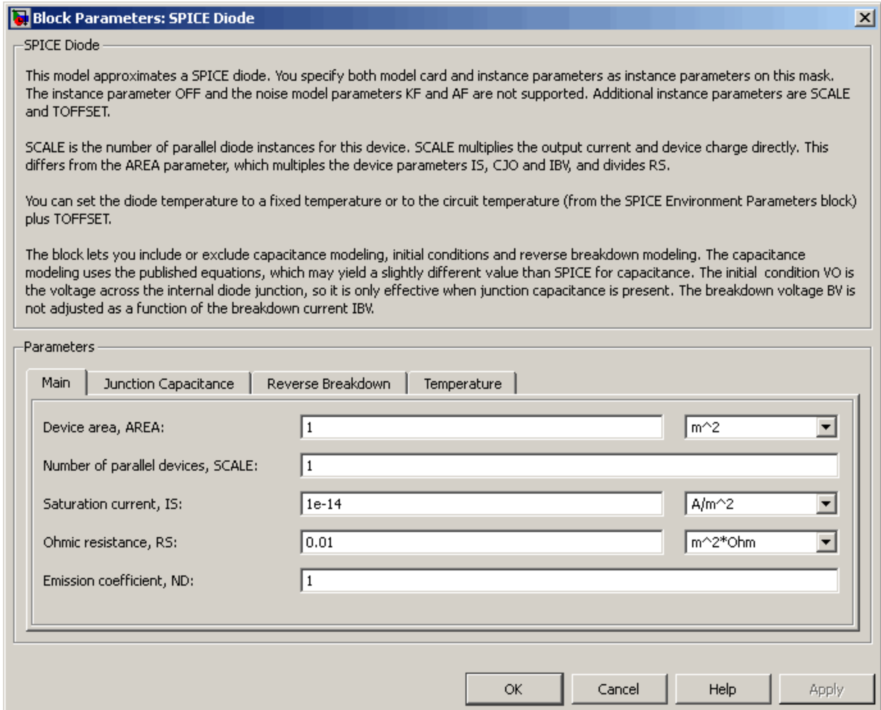
Basic Assumptions and Limitations

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.

Dialog Box and Parameters

Main Tab



Device area, AREA

The diode area. This value multiplies the **Saturation current, IS**, **Zero-bias junction capacitance CJ0**, 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^2. The value must be greater than or equal to `0`.

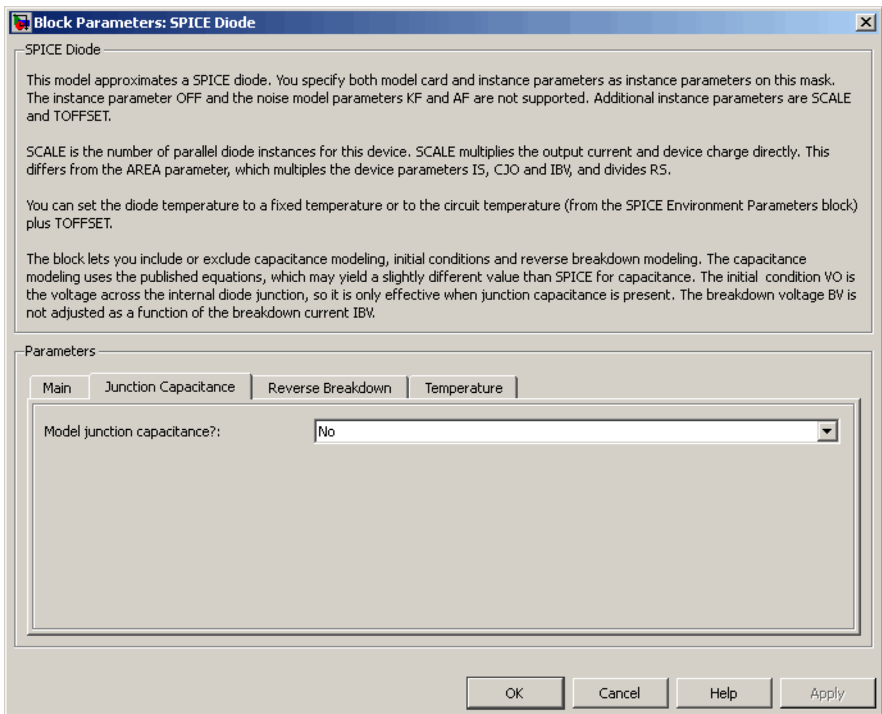
Ohmic resistance, RS

The series diode connection resistance. The default value is `0.01` m^2*Ω. The value must be greater than or equal to `0`.

Emission coefficient, ND

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 CJ0
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^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` V. The value must be greater than `0.01` V.

Grading coefficient MG
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.

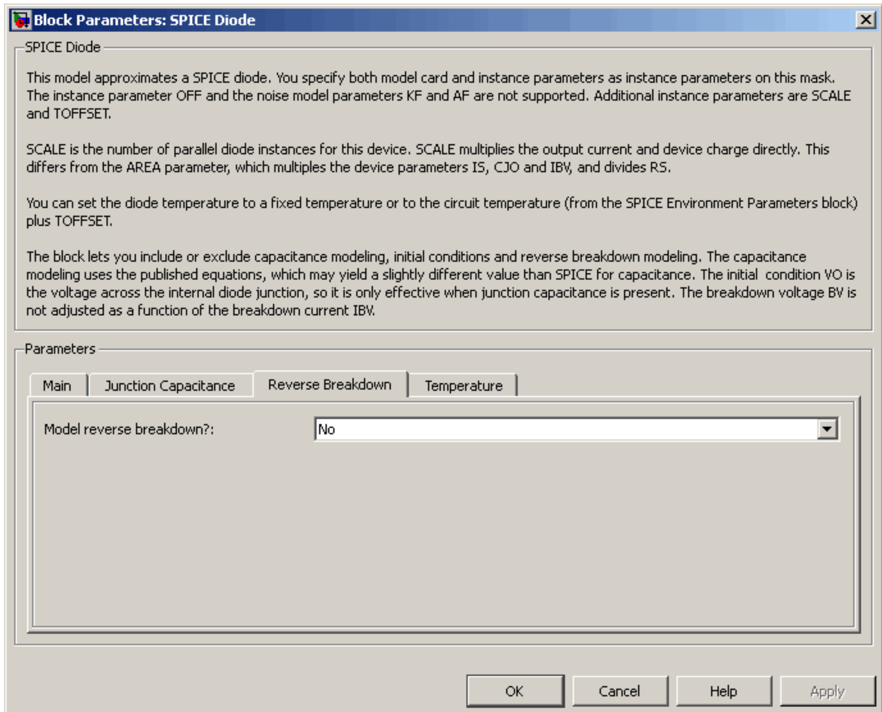
Note The SPICE Diode block applies the initial diode voltage across the junction capacitors and not across the ports.

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 CJ0** 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

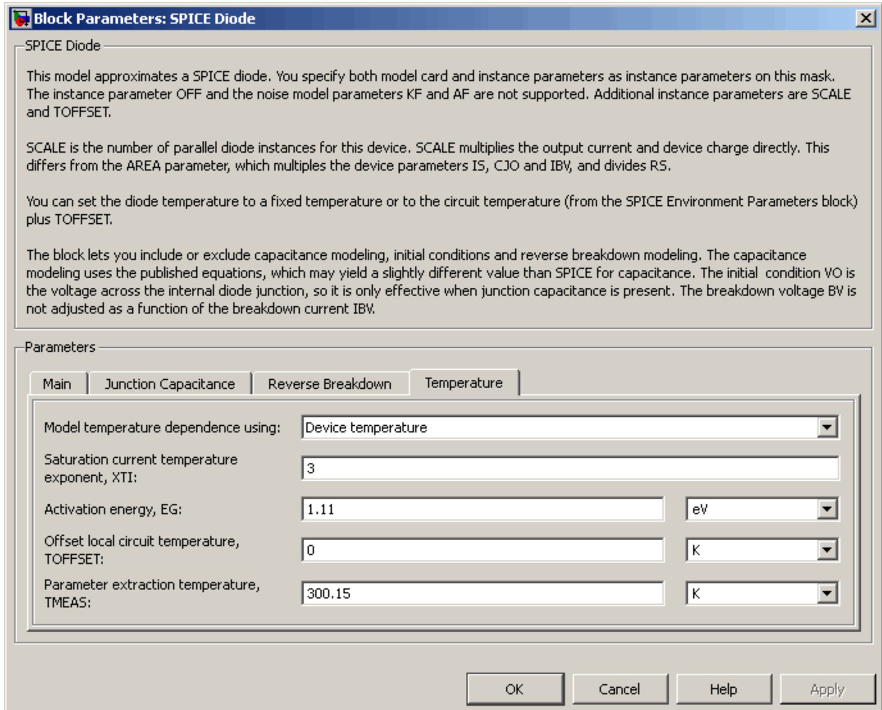
The diode current that corresponds to the **Reverse breakdown voltage, BV** value. This parameter is only visible when you select Yes for the **Model reverse breakdown** parameter. The default value is 0.001 A/m_2. The value must be greater than 0.

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 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` 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` 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

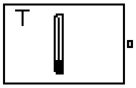
Purpose

Set parameters that apply to all connected SPICE-compatible blocks

Library

SPICE-Compatible Components/Utilities

Description



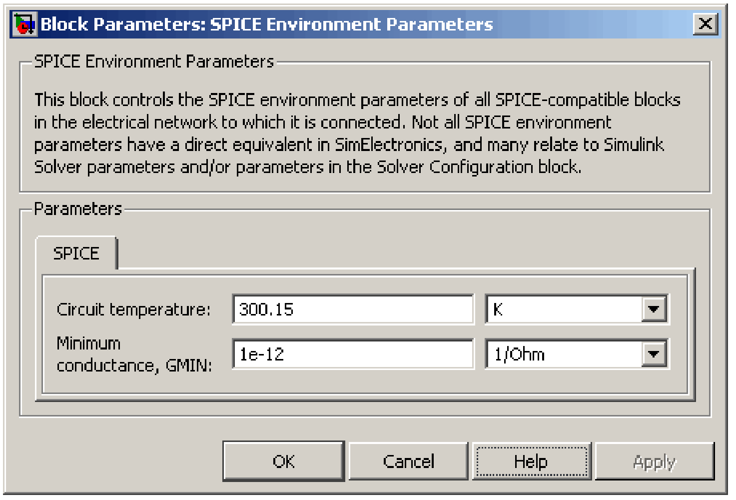
The SPICE Environment Parameters block lets you set parameters that apply to all SPICE-compatible blocks in an electrical network:

- 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.

Dialog Box and 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/Ω.

Ports

The block has the following ports:

`OUT`

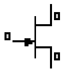
Electrical output.

SPICE NJFET

Purpose 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 1-648
- "Gate-Drain Current-Voltage Model" on page 1-649
- "Drain-Source Current-Voltage Model" on page 1-650
- "Junction Charge Model" on page 1-651
- "Temperature Dependence" on page 1-653

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
$V_{gs} > 80 * V_t$	$I_{gs} = IS * \left(\left(\frac{V_{gs}}{V_t} - 79 \right) e^{80} - 1 \right) + V_{gs} * G\min$
$80 * V_t \geq V_{gs}$	$I_{gs} = IS * \left(e^{V_{gs}/V_t} - 1 \right) + V_{gs} * G\min$

Where:

- IS is the **Saturation current, IS** parameter value.
- $V_t = ND * k * T/q$
- ND is the **Emission coefficient, ND** 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
$V_{gd} > 80 * V_t$	$I_{gd} = IS * \left(\left(\frac{V_{gd}}{V_t} - 79 \right) e^{80} - 1 \right) + V_{gd} * G\min$
$80 * V_t \geq V_{gd}$	$I_{gd} = IS * \left(e^{V_{gd}/V_t} - 1 \right) + V_{gd} * G\min$

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
$V_{gs} - V_{to} \leq 0$	$I_{ds} = 0$
$0 < V_{gs} - V_{to} \leq V_{ds}$	$I_{ds} = \beta \left(V_{gs} - V_{to} \right)^2 \left(1 + \lambda V_{ds} \right)$
$0 < V_{ds} < V_{gs} - V_{to}$	$I_{ds} = \beta V_{ds} \left(2 \left(V_{gs} - V_{to} \right) - V_{ds} \right) \left(1 + \lambda V_{ds} \right)$

Where:

- V_{to} is the **Threshold voltage, VTO** parameter value.
- β is the **Transconductance, BETA** parameter value.
- λ 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
$V_{gd} - V_{to} \le 0$	$I_{ds} = 0$
$0 < V_{gd} - V_{to} \le -V_{ds}$	$I_{ds} = -\beta \left(V_{gd} - V_{to}\right)^2 \left(1 - \lambda V_{ds}\right)$
$0 < -V_{ds} < V_{gs} - V_{to}$	$I_{ds} = \beta V_{ds} \left(2\left(V_{gd} - V_{to}\right) + V_{ds}\right)\left(1 - \lambda V_{ds}\right)$

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
$V_{gs} < FC * VJ$	$Q_{gs} = \dfrac{CGS * VJ * \left(1 - \left(1 - \dfrac{V_{gs}}{VJ}\right)^{1-MG}\right)}{1 - MG}$
$V_{gs} \ge FC * VJ$	$Q_{gs} = CGS * \left(F1 + \dfrac{F3 * \left(V_{gs} - FC * VJ\right) + \dfrac{MG * \left(V_{gs}^2 - \left(FC * VJ\right)^2\right)}{2 * VJ}}{F2}\right)$

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, MG** parameter value.

- $F1 = \frac{VJ * \left(1 - (1 - FC)^{1-MG}\right)}{1 - MG}$
- $F2 = (1 - FC)^{1+MG}$
- $F3 = 1 - FC * (1 + MG)$

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
$V_{gd} < FC * VJ$	$Q_{gd} = \frac{CGD * VJ * \left(1 - \left(1 - \frac{V_{gd}}{VJ}\right)^{1-MG}\right)}{1 - MG}$
$V_{gd} \geq FC * VJ$	$Q_{gd} = CGD * \left(F1 + \frac{F3 * (V_{gd} - FC * VJ) + \frac{MG * \left(V_{gd}^2 - (FC * VJ)^2\right)}{2 * VJ}}{F2}\right)$

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, ND** 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.
- $EG_{T_{meas}} = 1.16eV - \left(7.02e - 4 * T_{meas}{}^2\right) / \left(T_{meas} + 1108\right)$
- $EG_T = 1.16eV - \left(7.02e - 4 * T^2\right) / \left(T + 1108\right)$

The block provides the following relationship between the gate-source junction capacitance *CGS* and the transistor temperature *T*:

$$CGS(T) = CGS * \left[1 + MG * \left(400e - 6 * \left(T - T_{meas}\right) - \frac{VJ(T) - VJ}{VJ}\right)\right]$$

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 β is the **Transconductance, BETA** parameter value.

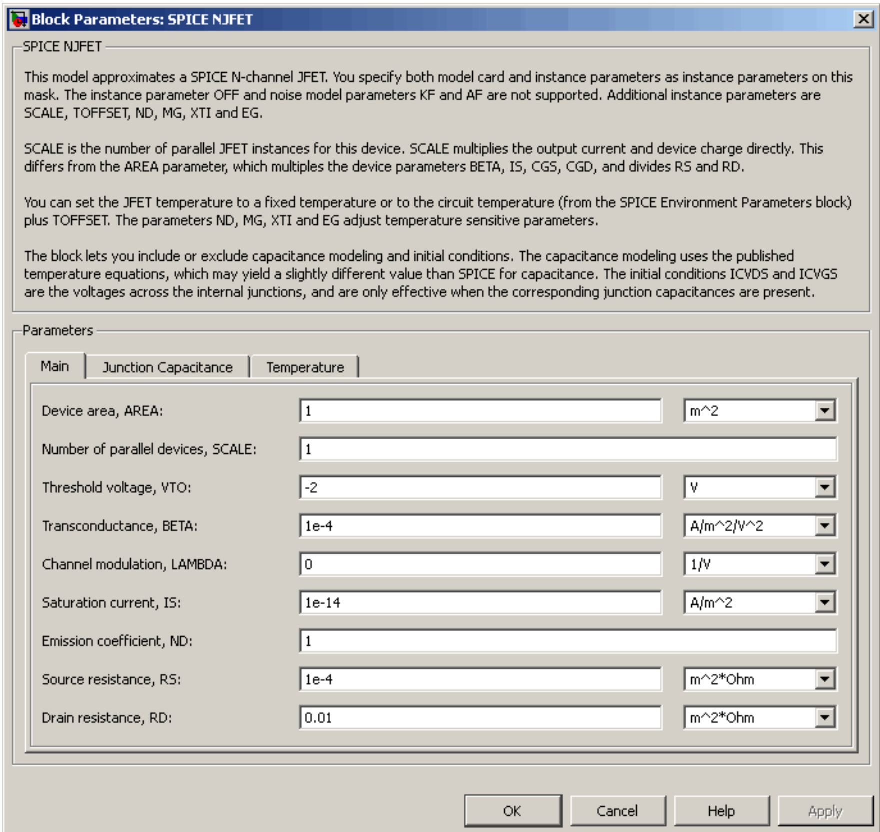
Basic Assumptions and Limitations

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.

Dialog Box and 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 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 V.

Transconductance, BETA

The derivative of drain current with respect to gate voltage. The default value is 1e-04 $A/m^2/V^2$. The value must be greater than or equal to 0.

Channel modulation, LAMBDA

The channel-length modulation. The default value is 0 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 1e-14 A/m^2. The value must be greater than or equal to 0.

Emission coefficient, ND

The transistor emission coefficient or ideality factor. The default value is 1. The value must be greater than 0.

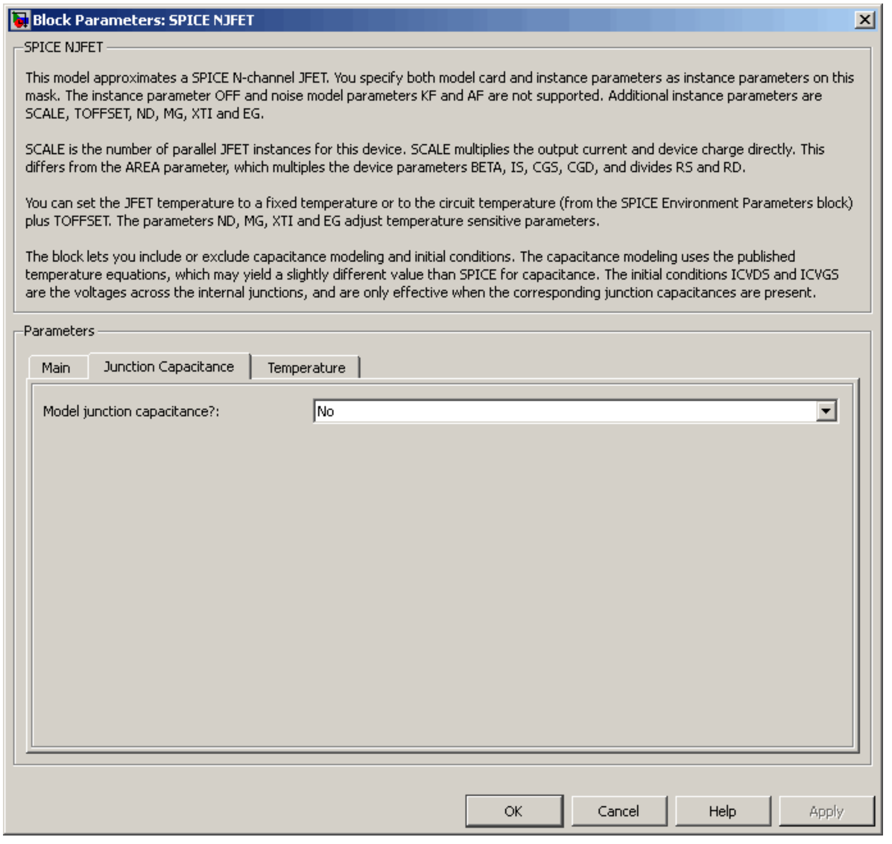
Source resistance, RS

The transistor source resistance. The default value is 1e-4 $m^2*\Omega$. The value must be greater than or equal to 0.

Drain resistance, RD

The transistor drain resistance. The default value is 0.01 $m^2*\Omega$. 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` 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` 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` V. The value must be greater than `0.01` V.

Grading coefficient, MG

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:

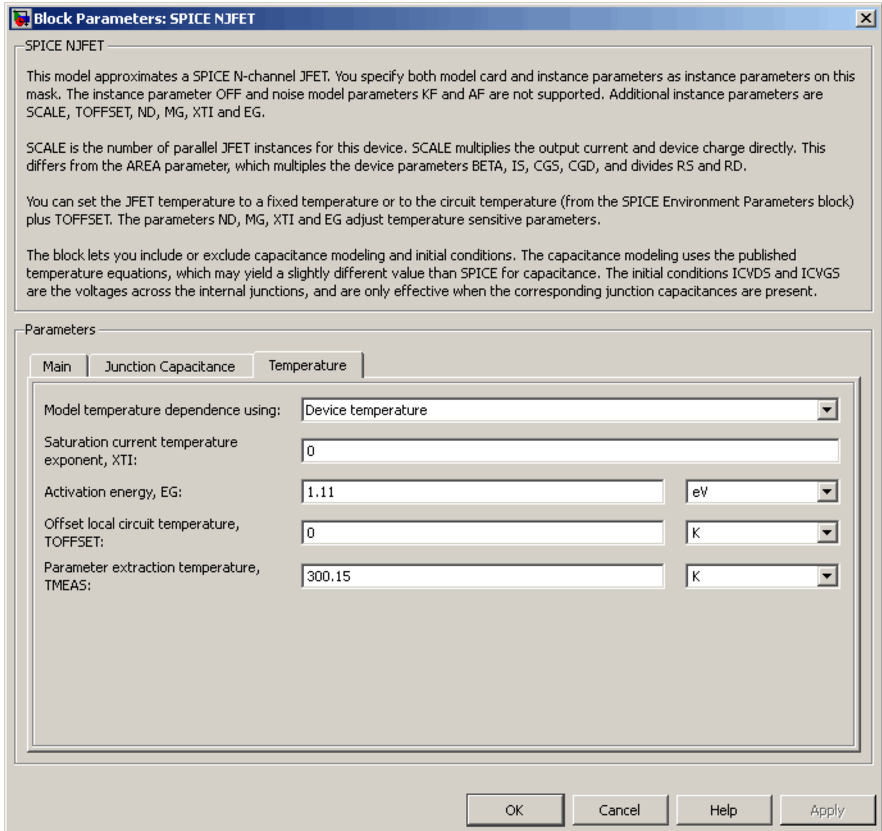
- `No` — Do not specify an initial condition for the model. This is the default option.
- `Yes` — Specify the initial diode voltage.

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

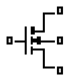
Purpose

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 1-664
- "Bulk-Source Diode Model" on page 1-665
- "Bulk-Drain Diode Model" on page 1-666
- "Level 1 Drain Current Model" on page 1-667
- "Level 3 Drain Current Model" on page 1-670
- "Junction Charge Model" on page 1-676
- "Temperature Dependence" on page 1-681

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
$V_{bs} > 80 * V_{tn}$	$I_{bs} = IS_{bs} * \left(\left(\frac{V_{bs}}{V_{tn}} - 79 \right) e^{80} - 1 \right) + V_{bs} * G\min$
$80V_{tn} \geq V_{bs}$	$I_{bs} = IS_{bs} * \left(e^{V_{bs}/V_{tn}} - 1 \right) + V_{bs} * G\min$

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, ND** 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
$V_{bd} > 80 * V_{tn}$	$I_{bd} = IS_{bd} * \left(\left(\frac{V_{bd}}{V_{tn}} - 79 \right) e^{80} - 1 \right) + V_{bd} * G\min$
$80V_{tn} \geq V_{bd}$	$I_{bd} = IS_{bd} * \left(e^{V_{bd}/V_{tn}} - 1 \right) + V_{bd} * G\min$

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
$V_{gs} - V_{on} \le 0$	$I_d = 0$
$0 < V_{gs} - V_{on} \le V_{ds}$	$I_d = BETA*\left(V_{gs} - V_{on}\right)^2 \dfrac{\left(1 + LAMBDA*V_{ds}\right)}{2}$
$0 < V_{ds} < V_{gs} - V_{on}$	$I_d = BETA* V_{ds}\left(\left(V_{gs} - V_{on}\right) - \dfrac{V_{ds}}{2}\right)\left(1 + LAMBDA*V_{ds}\right)$

Where:

- V_{on} is:
  - $MTYPE*VBI + GAMMA\sqrt{PHI - V_{bs}}$ if $V_{bs} \le 0$.
  - $MTYPE*VBI + GAMMA\left(\sqrt{PHI} - \dfrac{V_{bs}}{2\sqrt{PHI}}\right)$ if $0 < V_{bs} \le 2*PHI$.
  - $MTYPE*VBI$ 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.

- $U0*3.9*\varepsilon_0/TOX$, 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 1-681.
- *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.
  - $TOX*\sqrt{2*11.7*\varepsilon_0*q*NSUB}/(3.9*\varepsilon_0)$, if **Bulk threshold, GAMMA** is `NaN` and you specify values for both the **Oxide thickness, TOX** and **Substrate doping, NSUB** parameters.
- ε_0 is the permittivity of free space, 8.854214871e-12 F/m.
- n_i is the carrier concentration of intrinsic silicon, 1.45e10 cm^{-3}.

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
$V_{gd} - V_{on} \leq 0$	$I_d = 0$
$0 < V_{gd} - V_{on} \leq -V_{ds}$	$I_d = -BETA\left(V_{gd} - V_{on}\right)^2 (1 - LAMBDA * V_{ds})/2$
$0 < V_{ds} < V_{gd} - V_{on}$	$I_d = BETA *$ $V_{ds}\left(\left(V_{gd} - V_{on}\right) + V_{ds}/2\right)(1 - LAMBDA * V_{ds})$

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.

- $Scale_{VMAX}$ is the Velocity Saturation Scaling.
- $Scale_{LChan}$ is the Channel Length Modulation Scaling.
- $Scale_{INV}$ is the Weak Inversion Scaling.

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}

Basic Drain Current Model

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 1-667.
- The block calculates F_{GATE} using the following equation:

$$F_{gate} = \frac{1}{1 + THETA * \left(V_{gsx} - V_{TH} \right)}$$

- *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 $V_{BS} \le 0$, 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:

$$V_{TH} = V_{BI} - \frac{8.15e^{-22} * ETA}{COX * \left(LENGTH - 2 * LD\right)^3} * V_{DS} + GAMMA * F_s * \sqrt{V_{bulk}} + F_n * V_{bulk}$$

- For information about how the block calculates V_{BI}, see “Temperature Dependence” on page 1-681.
- *ETA* is the **Vds dependence threshold volt, ETA** parameter value.
- $COX = \varepsilon_{ox}/TOX$, where ε_{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:

$$\alpha = \frac{2\varepsilon_{si}}{qNSUB}$$

$$XD = \sqrt{\alpha}$$

$$wc = .0631353 + .8013292 * \frac{XD * \sqrt{V_{bulk}}}{XJ} - .01110777 * \left(\frac{XD * \sqrt{V_{bulk}}}{XJ}\right)^2 + \frac{LD}{XJ}$$

$$F_s = 1 - \left(wc * \sqrt{1 - \left(\frac{XD * \sqrt{V_{bulk}}}{XJ + XD * \sqrt{V_{bulk}}}\right)^2} - \frac{LD}{XJ} \right)$$

where ε_{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:

$$F_n = \frac{DELTA * \pi * \varepsilon_{si}}{2 * COX * WIDTH}$$

- *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:

$$V_{dsat} = \frac{V_{gsx} - V_{TH}}{1 + F_B}$$

Velocity Saturation Scaling

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, $Scale_{VMAX} = 1$.

Channel Length Modulation Scaling

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 a 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 Δl:

1 The block first calculates the intermediate value Δl_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 Δl_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 α.

- Otherwise, the block uses the following equation to calculate the intermediate value Δl_0:

$$\Delta l = \sqrt{KA*\left(V_{DS} - V_{dsat}\right)}$$

2 The block checks for punch through and calculates Δl.

- If Δl_0 is greater than ($LENGTH$-2*LD)/2, the block calculates Δl using the following equation:

$$\Delta l = \left(1 - \frac{\left(LENGTH - 2*LD\right)}{4*\Delta l_0}\right)*\left(LENGTH - 2*LD\right)$$

- Otherwise, $\Delta l = \Delta l_0$.

Weak Inversion Scaling

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

Junction Overlap 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.

Bulk Junction Charges

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
$V_{bd} < FC * PB$	$Q_{bottom} = \frac{CBD * PB * \left(1 - \left(1 - \frac{V_{bd}}{PB}\right)^{1-MJ}\right)}{1-MJ}$ if $CBD > 0$. $Q_{bottom} = \frac{CJ * AD * PB * \left(1 - \left(1 - \frac{V_{bd}}{PB}\right)^{1-MJ}\right)}{1-MJ}$ otherwise.
$V_{bd} \geq FC * PB$	$Q_{bottom} = CBD * \left(F1 + \frac{F3 * (V_{bd} - FC * PB) + \frac{MJ * \left(V_{bd}^2 - (FC * PB)^2\right)}{2 * PB}}{F2}\right)$ if $CBD > 0$. $Q_{bottom} = CJ * AD * \left(F1 + \frac{F3 * (V_{bd} - FC * PB) + \frac{MJ * \left(V_{bd}^2 - (FC * PB)^2\right)}{2 * PB}}{F2}\right)$ otherwise.

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.

- $F1 = \dfrac{PB * \left(1 - \left(1 - FC\right)^{1-MJ}\right)}{1 - MJ}$
- $F2 = \left(1 - FC\right)^{1+MJ}$
- $F3 = 1 - FC * \left(1 + MJ\right)$

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
$V_{bd} < FC * PB$	$Q_{sidewall} = \dfrac{CJSW * PD * PB * \left(1 - \left(1 - \dfrac{V_{bd}}{PB}\right)^{1-MGSW}\right)}{1-MGSW}$
$V_{bd} \geq FC * PB$	$Q_{sidewall} = CJSW * PD * \left(F1 + \dfrac{F3 * \left(V_{bd} - FC * PB\right) + \dfrac{MGSW * \left(V_{bd}^2 - \left(FC * PB\right)^2\right)}{2 * PB}}{F2}\right)$

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.

- $F1 = \dfrac{PB * \left(1 - \left(1 - FC\right)^{1-MJSW}\right)}{1-MJSW}$
- $F2 = \left(1 - FC\right)^{1+MJSW}$
- $F3 = 1 - FC * \left(1 + MJSW\right)$

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*:

$$KP(T) = \frac{KP}{\left(T / T_{meas} \right)^{3/2}}$$

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:

- $EG_{T_{meas}} = 1.16eV - \left(7.02e-4 * T_{meas}{}^2\right) / \left(T_{meas} + 1108\right)$
- $EG_T = 1.16eV - \left(7.02e-4 * T^2\right) / \left(T + 1108\right)$

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.

- $\Phi - 3.25 + EG_{T_{meas}}/2 + MTYPE * PHI/2 - NSS * q * TOX / (3.9 * \varepsilon_0)$ $+MTYPE * (GAMMA * \sqrt{PHI} + PHI)$, if **Threshold voltage, VTO** is `NaN` and you specify values for both the **Oxide thickness, TOX** and **Substrate doping, NSUB** parameters.

- Φ 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.
  - $TOX * \sqrt{2 * 11.7 * \varepsilon_0 * q * NSUB} / (3.9 * \varepsilon_0)$, 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, ND** 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*:

$$CBD(T) = CBD\frac{pbo + MJ*\left(4*10^4*(T-300.15)*pbo - (PB(T)-pbo)\right)}{pbo + MJ*\left(4*10^4*(T_{meas}-300.15)*pbo - (PB-pbo)\right)}$$

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*:

$$CJSW(T) = CJSW \frac{pbo + MJSW * \left(4 * 10^4 * (T - 300.15) * pbo - (PB(T) - pbo)\right)}{pbo + MJSW * \left(4 * 10^4 * (T_{meas} - 300.15) * pbo - (PB - pbo)\right)}$$

where:

- *MJSW* is the **Side grading coefficient, MJSW** parameter value.

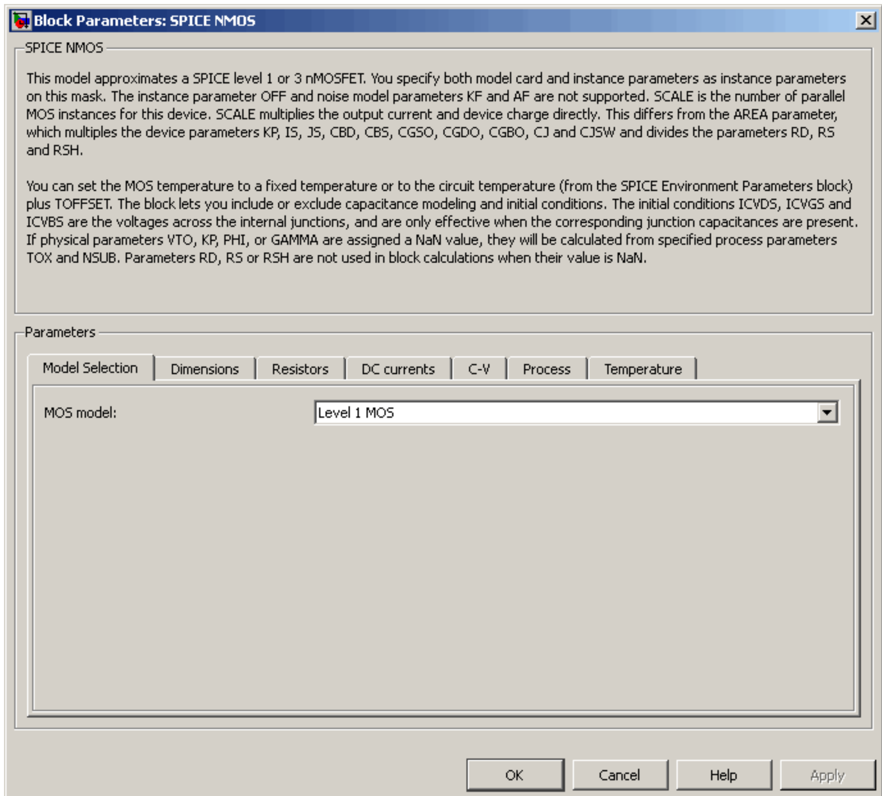
Basic Assumptions and Limitations

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.

Dialog Box and Parameters

Model Selection Tab

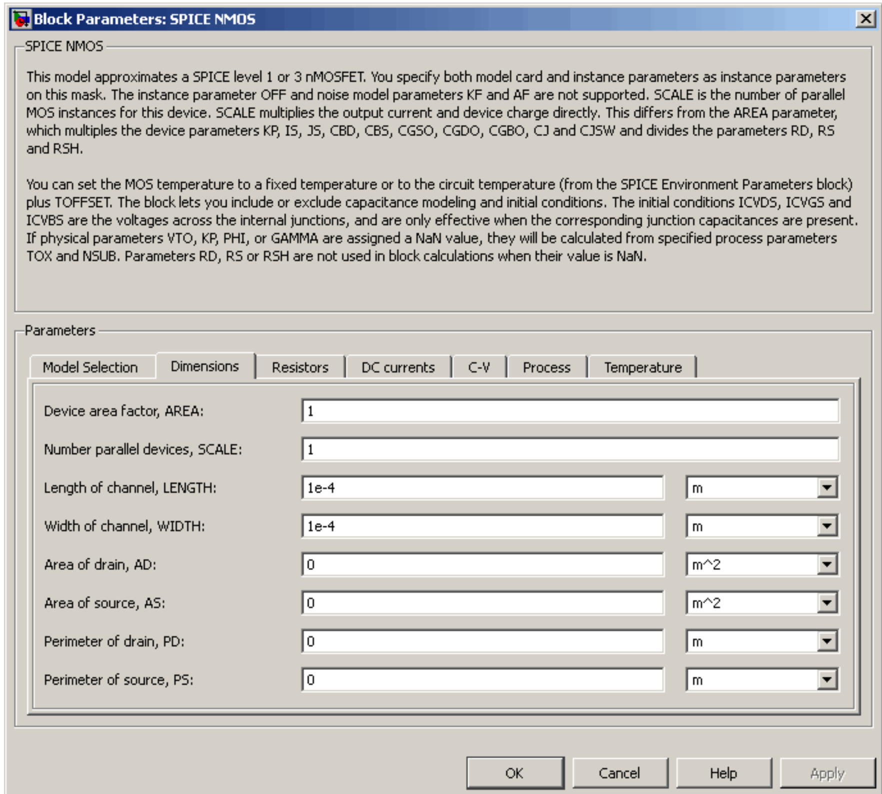


MOS model

Select one of the following MOSFET model options:

- `Level 1 MOS` — Use the “Level 1 Drain Current Model” on page 1-667. This is the default option.
- `Level 3 MOS` — Use the “Level 3 Drain Current Model” on page 1-670.

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^2. 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 m^2. The value must be greater than or equal to 0.

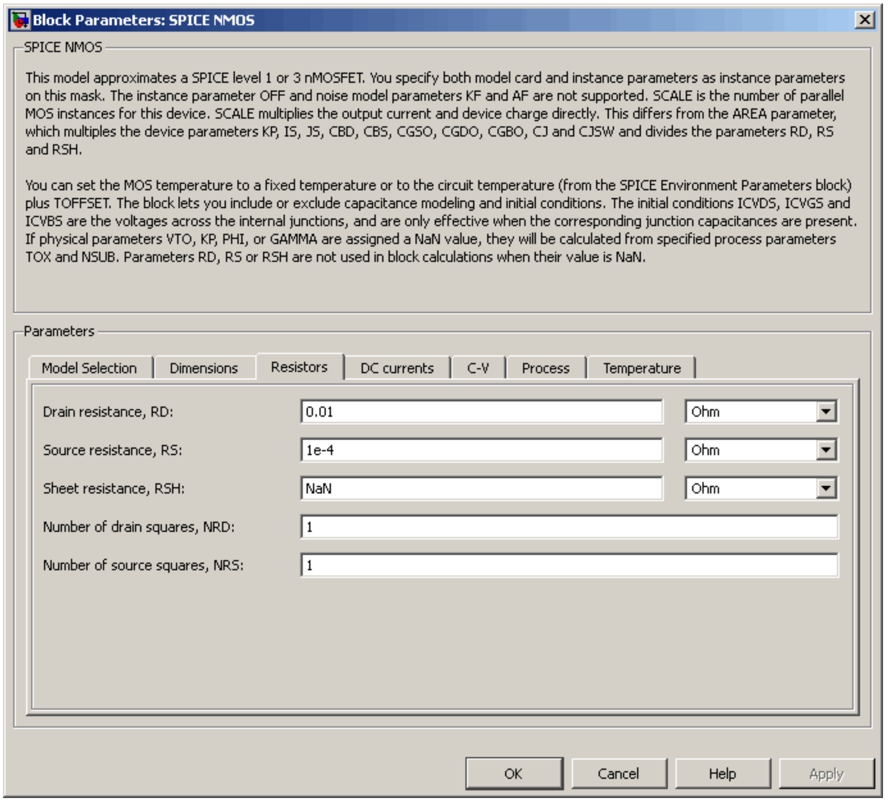
Perimeter of drain, PD

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 Ω. If you set this parameter to NaN Ω, this value means the parameter is unspecified, so the block calculates the drain resistance as described in "Resistance Calculations" on page 1-664. The value must be equal to 0 or greater than or equal to Rmin. Rmin is a built-in model constant whose value is 1e-12.

Source resistance, RS

The transistor source ohmic resistance. The default value is 1e-4 Ω. If you set this parameter to NaN Ω, this value means the parameter is unspecified, so the block calculates the drain resistance as described in "Resistance Calculations" on page 1-664. The value must be equal to 0 or greater than or equal to Rmin. Rmin is a built-in model constant whose value is 1e-12.

Sheet resistance, RSH

Resistance per square of the transistor source and drain. The default value is NaN Ω. 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 1-664. The value must be greater than or equal to 0.

Number of drain squares, NRD

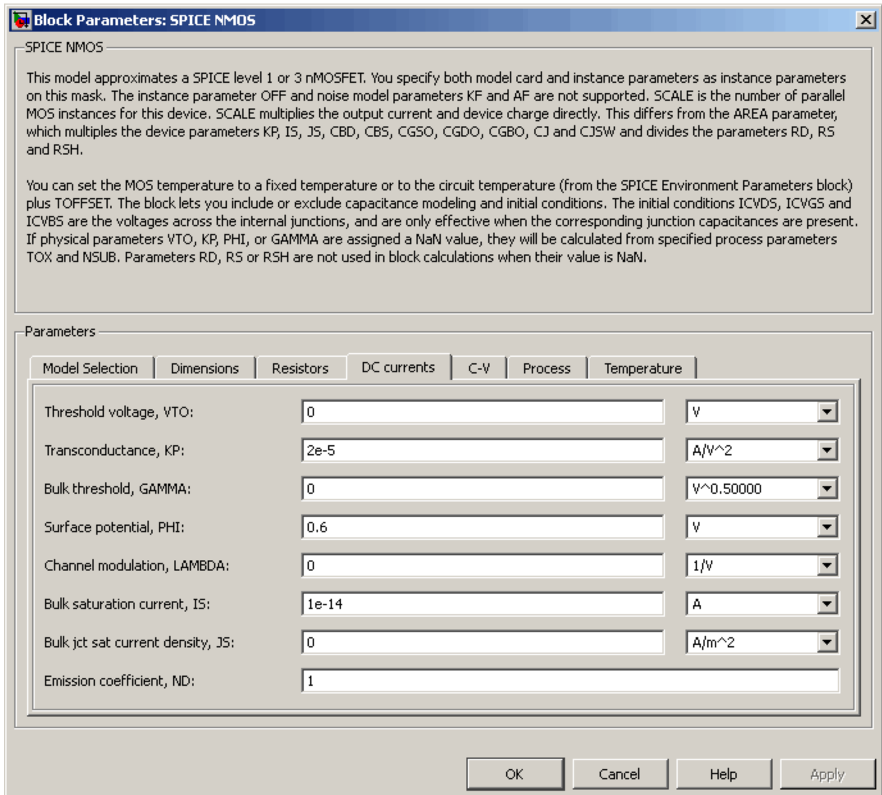
Number of squares of resistance that make up the transistor drain diffusion. The default value is 1 . The value must be greater than or equal to 0. The block only uses this parameter value if you do not specify one or both of the **Drain resistance, RD** and **Source resistance, RS** parameter values, as described in "Resistance Calculations" on page 1-664.

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 1-664.

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 1-681.

Transconductance, KP

The derivative of drain current with respect to gate voltage. The default value is `2e-05` 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 1-667 or "Level 3 Drain Current Model" on page 1-670 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 1-667 and "Level 3 Drain Current Model" on page 1-670. 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 1-667 or "Level 3 Drain Current Model" on page 1-670 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 1-667 or "Level 3 Drain Current Model" on page

1-670 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^2. The value must be greater than or equal to `0`.

Emission coefficient, ND
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` $1/cm^2$.

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`.

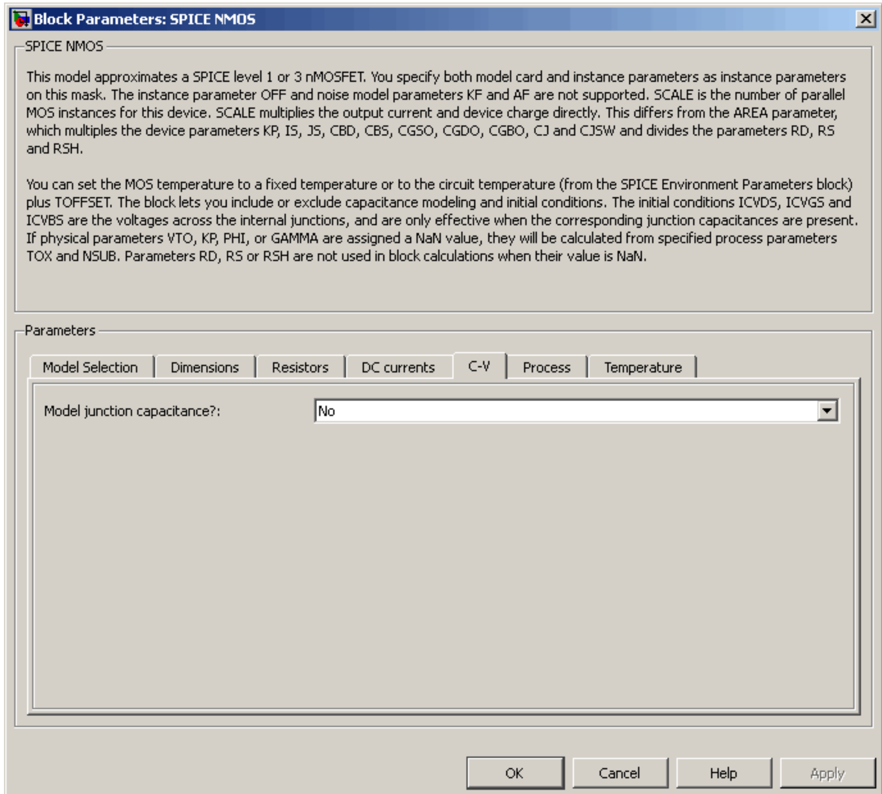
Vgs dependence on mobility, THETA

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 `0` 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 `0.2`.

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 `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`.

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 `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`.

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 `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`.

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^2. 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.

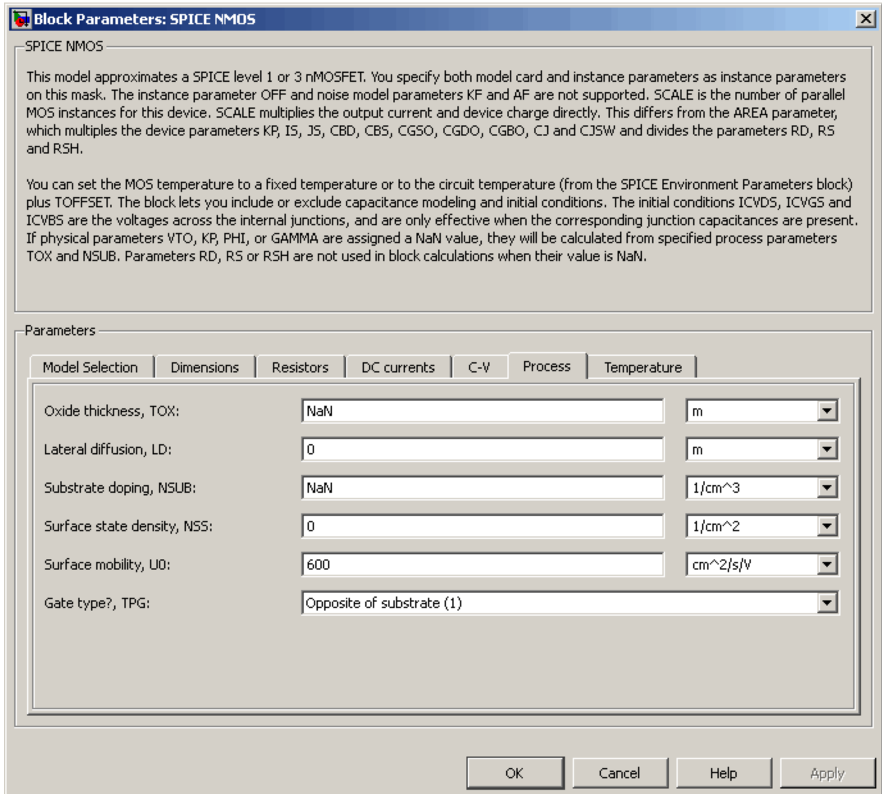
Initial condition voltage ICVGS

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^3. 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^2.

Surface mobility, U0

Zero-bias surface mobility coefficient. The default value is `600` cm^2/V/s.

Junction depth, XJ

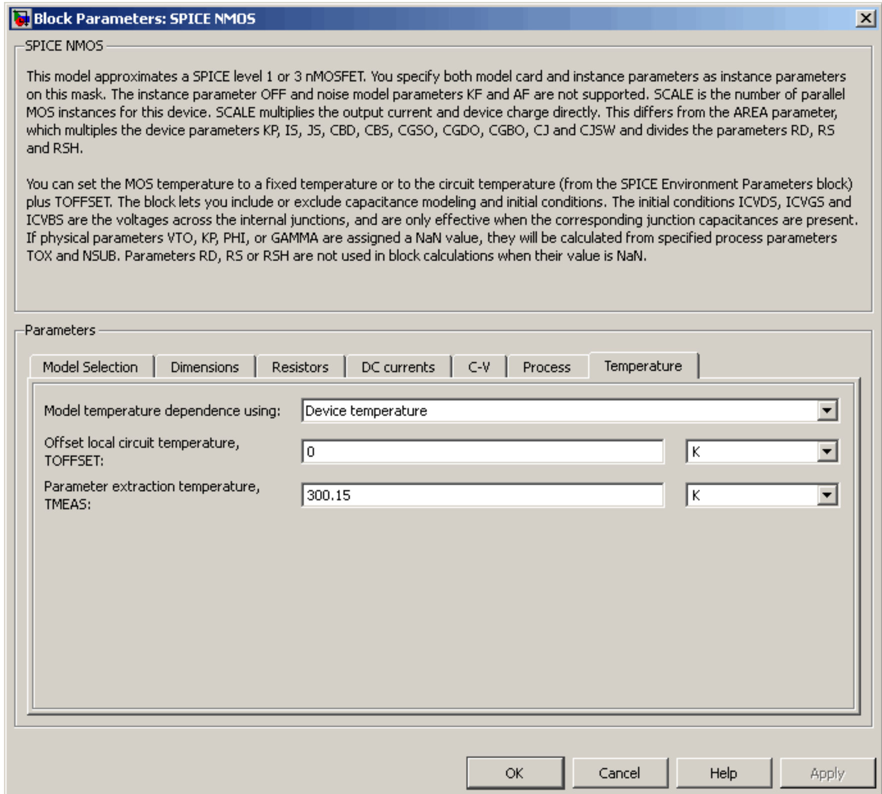
Junction depth. This parameter is only visible when you select `Level 3 MOS` for the **MOS model** parameter. The default value is `0` m.

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

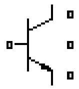
Purpose

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 1-704
- "Base Resistance Model" on page 1-708
- "Transit Charge Modulation Model" on page 1-708
- "Junction Charge Model" on page 1-709
- "Temperature Dependence" on page 1-711

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 704
- Terminal Currents on page 707
- Base Charge Model on page 707

Base-Emitter and Base-Collector Junction Currents

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 $V_{BE} \le 80 * V_{TF}$

$$I_{bef} = IS * \left(e^{(V_{BE}/V_{TF})} - 1 \right) + G_{min} * V_{BE}$$

$$I_{bee} = ISE * \left(e^{(V_{BE}/V_{TE})} - 1 \right)$$

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 $V_{BC} \le 80 * V_{TR}$

$$I_{bcr} = IS * \left(e^{(V_{BC}/V_{TR})} - 1 \right) + G_{min} * V_{BC}$$

$$I_{bcc} = ISC * \left(e^{(V_{BC}/V_{TC})} - 1 \right)$$

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.

Terminal Currents

The terminal currents, I_B and I_C are the base and collector currents, defined as positive into the device. They are calculated as:

$$I_B = -\left(\frac{I_{ebf}}{BF} + I_{ebe} + \frac{I_{cbr}}{BR} + I_{cbc}\right)$$

$$I_C = -\left(\frac{I_{ebf} - I_{cbr}}{q_b} - \frac{I_{cbr}}{BR} - I_{\mathrm{cbc}}\right)$$

where BF and BR are the **Forward beta, BF** and **Reverse beta, BR** parameter values, respectively.

Base Charge Model

The base charge, q_b, 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_{bb} 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

$$r_{bb} = RBM + 3*\left(RB - RBM\right)*\left(\frac{\tan z - z}{z*\tan^2 z}\right)$$

  where

$$z = \frac{\sqrt{1 + 144 I_B / \left(\pi^2 IRB\right)} - 1}{\left(24/\pi^2\right)\sqrt{\left(I_B / IRB\right)}}$$

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:

$$Q_{be} = TF_{\text{mod}} * I_{be} + Q_{dep}$$

- For the internal base-collector junctions:

$$Q_{bc} = TR * I_{bc} + XCJC * Q_{dep}$$

- For the external base-collector junctions:

$$Q_{b_{ext}c} = (1 - XCJC) * Q_{dep}$$

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
$V_{jct} < FC*VJ$	$Q_{dep} = C_{jct} * VJ * \frac{1 - \left(1 - V_{jct}/VJ\right)^{(1-MJ)}}{1-MJ}$
$V_{jct} \geq FC*VJ$	$Q_{dep} = C_{jct} * \left[F1 + \frac{F3*\left(V_{jct} - FC*VJ\right) + \frac{MJ*\left[V_{jct}^{2} - \left(FC*VJ\right)^{2}\right]}{2*VJ}}{F2} \right]$

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.

- $F1 = VJ * \left(1 - (1 - FC)^{(1-MJ)}\right) / (1 - MJ)$
- $F2 = (1 - FC)^{(1+MJ)}$
- $F3 = 1 - FC * (1 + MJ)$

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.
- *VJS* is the **Substrate built-in potential, VJS** parameter value.
- *MJS* 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*:

$$IS(T) = IS * \left(T/T_{meas}\right)^{XTI} * e^{\left(\frac{T}{T_{meas}}-1\right)*\frac{EG}{V_t}}$$

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.
- $EG_{T_{meas}} = 1.16eV - \left(7.02e - 4 * T_{meas}{}^2\right) / \left(T_{meas} + 1108\right)$
- $EG_T = 1.16eV - \left(7.02e - 4 * T^2\right) / \left(T + 1108\right)$

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*:

$$CJE(T) = CJE * \left[1 + MJE * \left(400e - 6 * \left(T - T_{meas}\right) - \frac{VJE(T) - VJE}{VJE} \right) \right]$$

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:

- β 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*:

$$ISE(T) = ISE * \left(\frac{T}{T_{meas}} \right)^{\text{-XTB}} * \left(\frac{\text{IS(T)}}{\text{IS}} \right)^{1/NE}$$

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*.

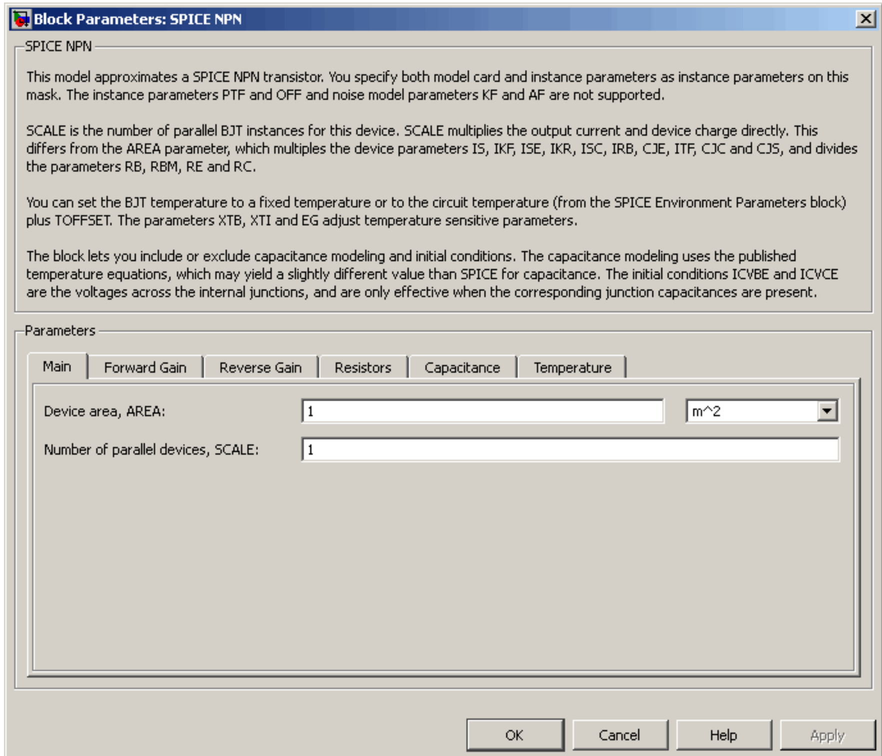
Basic Assumptions and Limitations

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.

Dialog Box and 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**

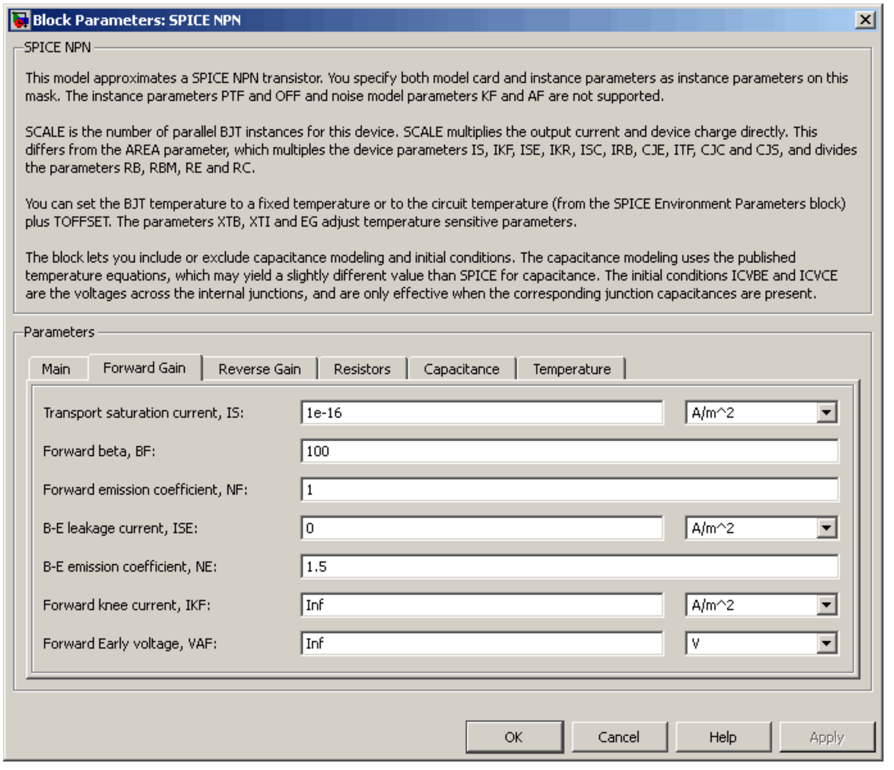
It divides the following parameter values:

- **Zero-bias base resistance, RB**
- **Minimum base resistance, RBM**
- **Emitter resistance, RE**
- **Collector resistance, RC**

The default value is 1 m^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.

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^2. 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 `0` A/m^2. The value must be greater than or equal to `0`.

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 `0`.

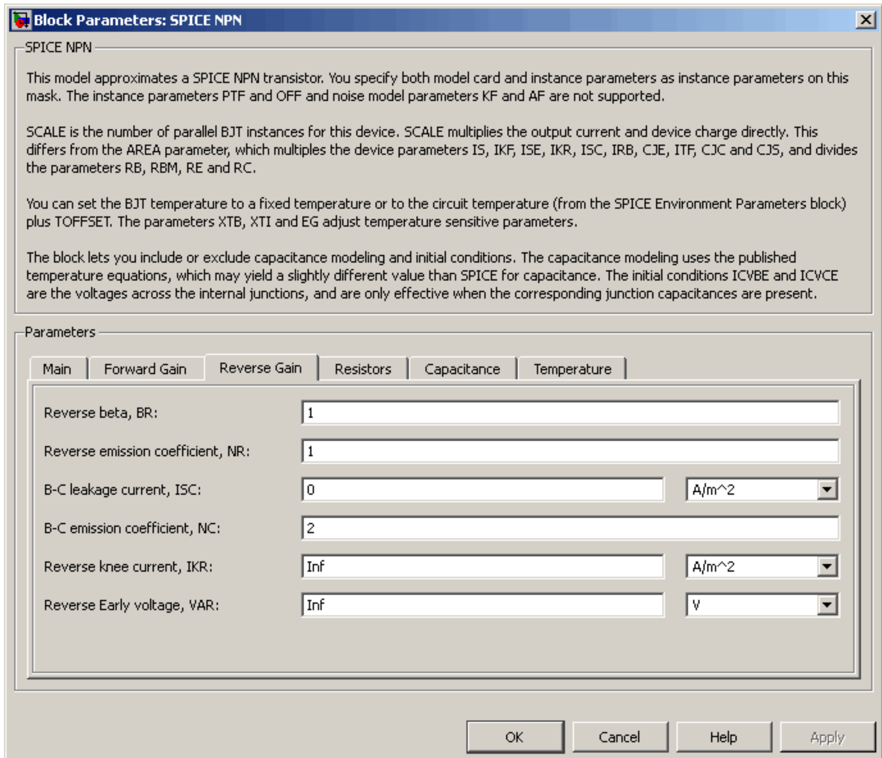
Forward knee current, IKF

The current value at which forward-beta high-current roll-off occurs. The default value is `Inf` A/m^2. The value must be greater than or equal to `0`.

Forward Early voltage, VAF

The forward Early voltage. The default value is `Inf` V. The value must be greater than or equal to `0`.

Reverse Gain Tab



Reverse beta, BR

The ideal maximum reverse beta. The default value is 1. The value must be greater than 0.

Reverse emission coefficient, NR

The reverse emission coefficient or ideality factor. The default value is 1. The value must be greater than 0.

B-C leakage current, ISC

The base-collector leakage current. The default value is 0 A/m^2. The value must be greater than or equal to 0.

B-C emission coefficient, NC

The base-collector emission coefficient or ideality factor. The default value is `2`. The value must be greater than `0`.

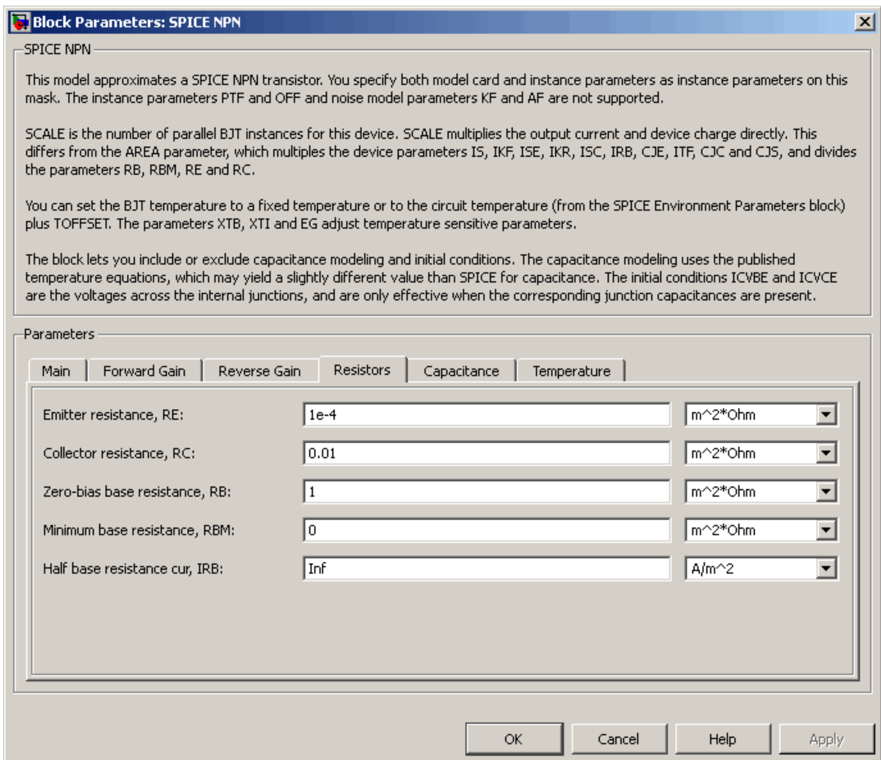
Reverse knee current, IKR

The current value at which reverse-beta high-current roll-off occurs. The default value is `Inf` A/m^2. The value must be greater than or equal to `0`.

Reverse Early voltage, VAR

The reverse Early voltage. The default value is `Inf` V. The value must be greater than or equal to `0`.

Resistors Tab



Emitter resistance, RE
The resistance of the emitter. The default value is `1e-4` $m^2*\Omega$. The value must be greater than or equal to `0`.

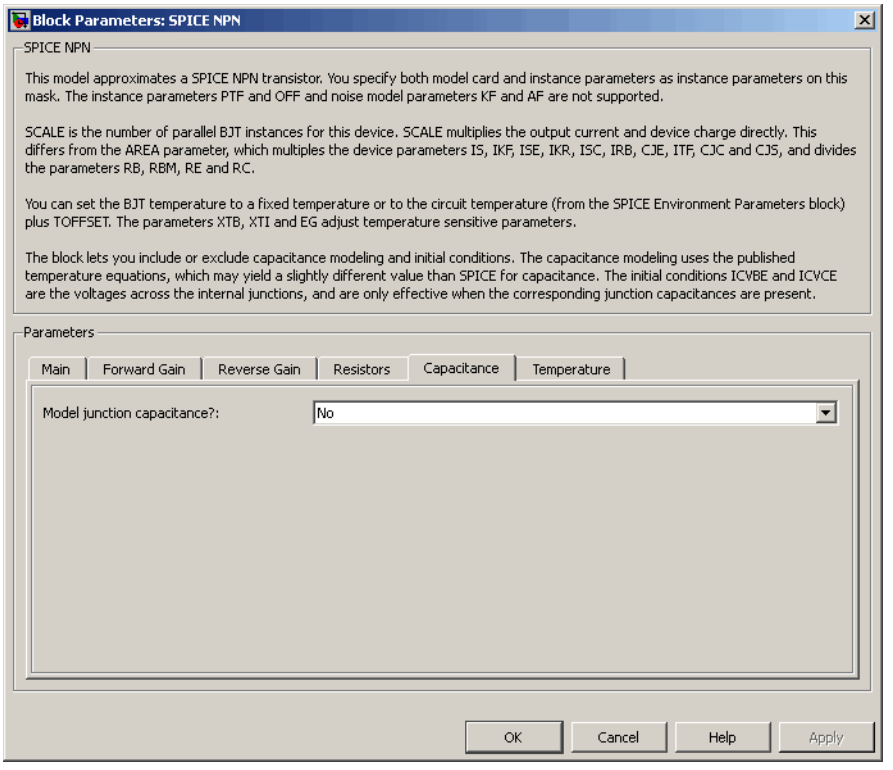
Collector resistance, RC
The resistance of the collector. The default value is `0.01` $m^2*\Omega$. The value must be greater than or equal to `0`.

Zero-bias base resistance, RB
The resistance of the base. The default value is `1` $m^2*\Omega$. 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` $m^2*\Omega$. 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 `Inf` A/m^2. The value must be greater than or equal to `0`. Use the default value of `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` 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 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^2. 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^2. 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.

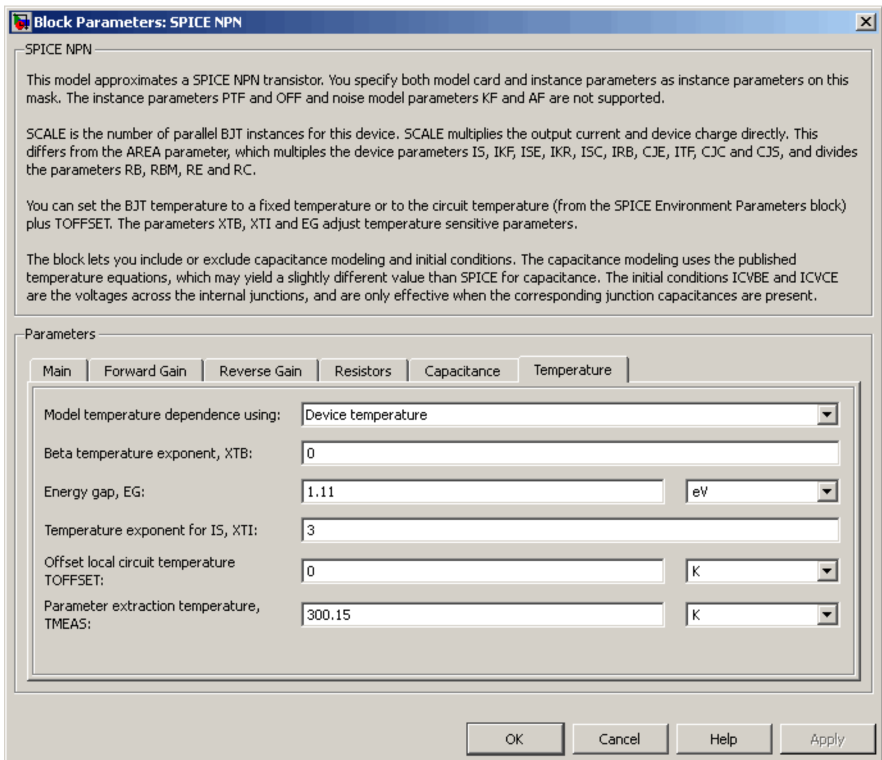
C-S junction capacitance, CJS

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^2. 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

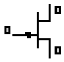
Purpose

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 1-730
- "Drain-Gate Current-Voltage Model" on page 1-731
- "Source-Drain Current-Voltage Model" on page 1-732
- "Junction Charge Model" on page 1-733
- "Temperature Dependence" on page 1-735

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
$V_{sg} > 80 * V_t$	$I_{sg} = IS * \left(\left(\frac{V_{sg}}{V_t} - 79 \right) e^{80} - 1 \right) + V_{sg} * G\min$
$80 * V_t \geq V_{sg}$	$I_{sg} = IS * \left(e^{V_{sg}/V_t} - 1 \right) + V_{sg} * G\min$

Where:

- IS is the **Saturation current, IS** parameter value.
- $V_t = ND * k * T/q$
- ND is the **Emission coefficient, ND** 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
$V_{dg} > 80 * V_t$	$I_{dg} = IS * \left(\left(\frac{V_{dg}}{V_t} - 79 \right) e^{80} - 1 \right) + V_{dg} * G\min$
$80 * V_t \geq V_{dg}$	$I_{dg} = IS * \left(e^{V_{dg}/V_t} - 1 \right) + V_{dg} * G\min$

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**
$V_{sg} - V_{to} \leq 0$	$I_{sd} = 0$
$0 < V_{sg} - V_{to} \leq V_{sd}$	$I_{sd} = -\beta * \left(V_{sg} - V_{to}\right)^2 * \left(1 + \lambda * V_{sd}\right)$
$0 < V_{sd} < V_{sg} - V_{to}$	$I_{sd} = \beta * V_{sd} * \left(2 * \left(V_{sg} - V_{to}\right) - V_{sd}\right) * \left(1 + \lambda * V_{sd}\right)$

Where:

- V_{to} is the **Threshold voltage, VTO** parameter value.
- β is the **Transconductance, BETA** parameter value.
- λ 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
$V_{dg} - V_{to} \leq 0$	$I_{sd} = 0$
$0 < V_{dg} - V_{to} \leq -V_{sd}$	$I_{sd} = \beta * \left(V_{dg} - V_{to}\right)^2 * \left(1 - \lambda * V_{sd}\right)$
$0 < -V_{sd} < V_{dg} - V_{to}$	$I_{sd} = \beta * V_{sd} * \left(2 * \left(V_{dg} - V_{to}\right) + V_{sd}\right) * \left(1 - \lambda * V_{sd}\right)$

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
$V_{sg} < FC * VJ$	$Q_{sg} = \dfrac{CGS * VJ * \left(1 - \left(1 - \dfrac{V_{sg}}{VJ}\right)^{1-MG}\right)}{1 - MG}$
$V_{sg} \geq FC * VJ$	$Q_{sg} = CGS * \left(F1 + \dfrac{F3 * \left(V_{sg} - FC * VJ\right) + \dfrac{MG * \left(V_{sg}^2 - \left(FC * VJ\right)^2\right)}{2 * VJ}}{F2}\right)$

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, MG** parameter value.

- $F1 = \frac{VJ * \left(1 - (1 - FC)^{1-MG}\right)}{1 - MG}$
- $F2 = (1 - FC)^{1+MG}$
- $F3 = 1 - FC * (1 + MG)$

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
$V_{dg} < FC * VJ$	$Q_{dg} = \frac{CGD * VJ * \left(1 - \left(1 - \frac{V_{dg}}{VJ}\right)^{1-MG}\right)}{1 - MG}$
$V_{dg} \geq FC * VJ$	$Q_{dg} = CGD * \left(F1 + \frac{F3 * (V_{dg} - FC * VJ) + \frac{MG * \left(V_{dg}^2 - (FC * VJ)^2\right)}{2 * VJ}}{F2}\right)$

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, ND** 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.
- $EG_{T_{meas}} = 1.16eV - \left(7.02e - 4 * T_{meas}{}^2\right) / \left(T_{meas} + 1108\right)$
- $EG_T = 1.16eV - \left(7.02e - 4 * T^2\right) / \left(T + 1108\right)$

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 β is the **Transconductance, BETA** parameter value.

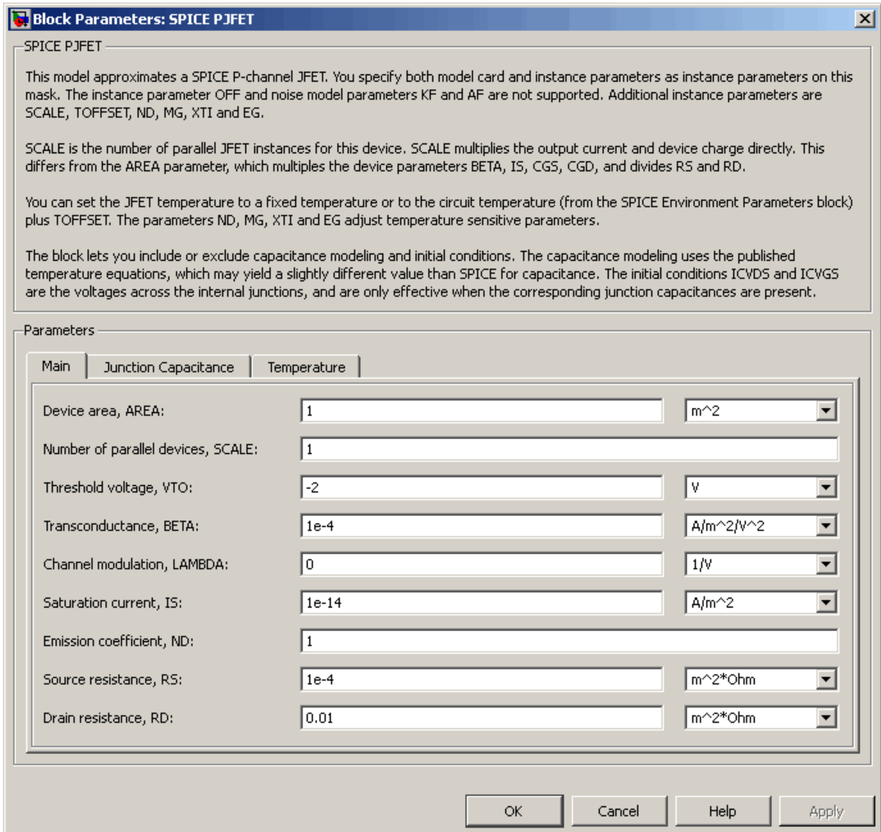
Basic Assumptions and Limitations

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.

Dialog Box and 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` 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 V.

Transconductance, BETA
The derivative of drain current with respect to gate voltage. The default value is 1e-04 $A/m^2/V^2$. The value must be greater than or equal to 0.

Channel modulation, LAMBDA
The channel-length modulation. The default value is 0 1/V.

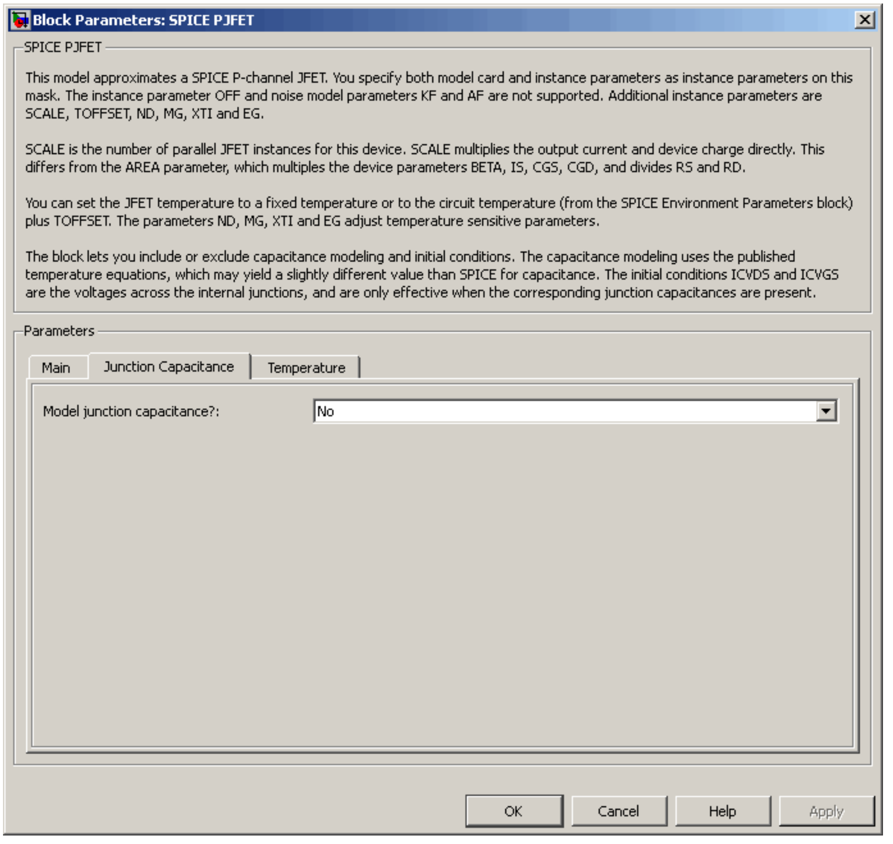
Saturation current, IS
The magnitude of the current that the ideal diode equation approaches asymptotically for very large reverse bias levels. The default value is 1e-14 A/m^2. The value must be greater than or equal to 0.

Emission coefficient, ND
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 1e-4 $m^2*\Omega$. The value must be greater than or equal to 0.

Drain resistance, RD
The transistor drain resistance. The default value is 0.01 $m^2*\Omega$. 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 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 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 V. The value must be greater than 0.01 V.

Grading coefficient, MG

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:

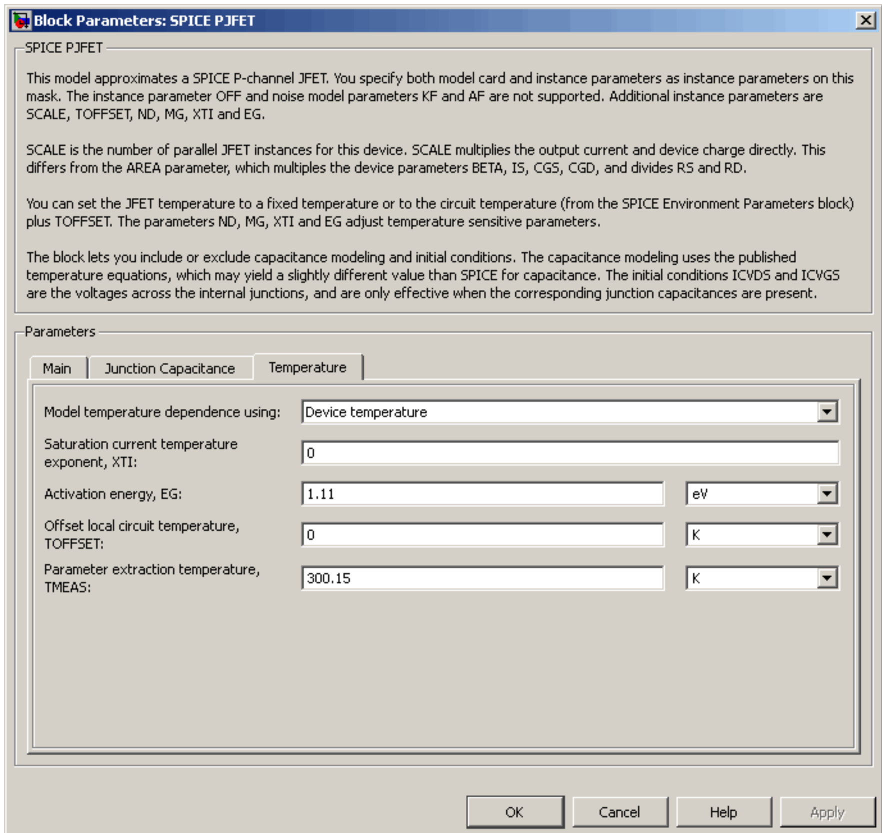
- No — Do not specify an initial condition for the model. This is the default option.
- Yes — Specify the initial diode voltage.

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` eVi.

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

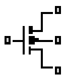
Purpose

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 1-746
- "Bulk-Source Diode Model" on page 1-747
- "Bulk-Drain Diode Model" on page 1-748
- "Level 1 Drain Current Model" on page 1-749
- "Level 3 Drain Current Model" on page 1-752
- "Junction Charge Model" on page 1-758
- "Temperature Dependence" on page 1-763

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
$V_{sb} > 80 * V_{tn}$	$I_{sb} = IS_{sb} * \left(\left(\frac{V_{sb}}{V_{tn}} - 79\right)e^{80} - 1\right) + V_{sb} * G\min$
$80V_{tn} \geq V_{sb}$	$I_{sb} = IS_{sb} * \left(e^{V_{sb}/V_{tn}} - 1\right) + V_{sb} * G\min$

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, ND** 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
$V_{db} > 80 * V_{tn}$	$I_{db} = IS_{db} * \left(\left(\frac{V_{db}}{V_{tn}} - 79 \right) e^{80} - 1 \right) + V_{db} * G\min$
$80V_{tn} \geq V_{db}$	$I_{db} = IS_{db} * \left(e^{V_{db}/V_{tn}} - 1 \right) + V_{db} * G\min$

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
$V_{sg} - V_{on} \le 0$	$I_{sd} = 0$
$0 < V_{sg} - V_{on} \le V_{sd}$	$I_{sd} = BETA*\left(V_{sg} - V_{on}\right)^2 \dfrac{\left(1 + LAMBDA*V_{sd}\right)}{2}$
$0 < V_{sd} < V_{sg} - V_{on}$	$I_{sd} = BETA*$ $V_{sd}\left(\left(V_{sg} - V_{on}\right) - \dfrac{V_{sd}}{2}\right)\left(1 + LAMBDA*V_{sd}\right)$

Where:

- V_{on} is:
  - $MTYPE * VBI + GAMMA\sqrt{PHI - V_{sb}}$ if $V_{sb} \le 0$.
  - $MTYPE * VBI + GAMMA\left(\sqrt{PHI} - \dfrac{V_{sb}}{2\sqrt{PHI}}\right)$ if $0 < V_{sb} \le 2 * PHI$.
  - $MTYPE*VBI$ 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.

- $U0*3.9*\varepsilon_0/TOX$, 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 1-681.
- *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.
  - $TOX*\sqrt{2*11.7*\varepsilon_0*q*NSUB}/(3.9*\varepsilon_0)$, if **Bulk threshold, GAMMA** is `NaN` and you specify values for both the **Oxide thickness, TOX** and **Substrate doping, NSUB** parameters.
- ε_0 is the permittivity of free space, 8.854214871e-12 F/m.
- n_i is the carrier concentration of intrinsic silicon, 1.45e10 cm^{-3}.

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
$V_{dg} - V_{on} \leq 0$	$I_{sd} = 0$
$0 < V_{dg} - V_{on} \leq -V_{sd}$	$I_{sd} = -BETA\left(V_{dg} - V_{on}\right)^2 \left(1 - LAMBDA * V_{sd}\right)/2$
$0 < V_{sd} < V_{dg} - V_{on}$	$I_{sd} = BETA* V_{sd}\left(\left(V_{dg} - V_{on}\right) + V_{sd}/2\right)\left(1 - LAMBDA * V_{sd}\right)$

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.
- $Scale_{VMAX}$ is the Velocity Saturation Scaling.
- $Scale_{LChan}$ is the Channel Length Modulation Scaling.
- $Scale_{INV}$ is the Weak Inversion Scaling.

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}

Basic Drain Current Model

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 1-749.
- The block calculates F_{GATE} using the following equation:

$$F_{gate} = \frac{1}{1 + THETA * \left(V_{sgx} - V_{TH} \right)}$$

- *THETA* is the **Vgs dependence on mobility, THETA** parameter value.

- $V_{sgx} = \max(V_{SG}, V_{on})$
- 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 $V_{SB} \le 0$, $V_{bulk} = PHI - V_{BS}$.
  - 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 1-763.

- *ETA* is the **Vds dependence threshold volt, ETA** parameter value.
- $COX = \varepsilon_{ox}/TOX$, where ε_{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:

$$\alpha = \frac{2\varepsilon_{si}}{qNSUB}$$

$$XD = \sqrt{\alpha}$$

$$wc = .0631353 + .8013292 * \frac{XD * \sqrt{V_{bulk}}}{XJ}$$

$$-.01110777 * \left(\frac{XD * \sqrt{V_{bulk}}}{XJ} \right)^2 + \frac{LD}{XJ}$$

$$F_s = 1 - \left(wc * \sqrt{1 - \left(\frac{XD * \sqrt{V_{bulk}}}{XJ + XD * \sqrt{V_{bulk}}} \right)^2} - \frac{LD}{XJ} \right)$$

where ε_{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:

$$F_n = \frac{DELTA * \pi * \varepsilon_{si}}{2 * COX * WIDTH}$$

- *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:

$$V_{dsat} = \frac{V_{sgx} - V_{TH}}{1 + F_B}$$

Velocity Saturation Scaling

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, $Scale_{VMAX} = 1$.

Channel Length Modulation Scaling

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 a 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 Δl:

1 The block first calculates the intermediate value Δl_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 Δl_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 α.

- Otherwise, the block uses the following equation to calculate the intermediate value Δl_0:

$$\Delta l = \sqrt{KA*(V_{SD}-V_{dsat})}$$

2 The block checks for punch through and calculates Δl.

- If Δl_0 is greater than $(LENGTH\text{-}2*LD)/2$, the block calculates Δl using the following equation:

$$\Delta l = \left(1-\frac{(LENGTH-2*LD)}{4*\Delta l_0}\right)*(LENGTH-2*LD)$$

- Otherwise, $\Delta l = \Delta l_0$.

Weak Inversion Scaling

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

Junction Overlap 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.

Bulk Junction Charges

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
$V_{db} < FC * PB$	$Q_{bottom} = \dfrac{CBD * PB * \left(1 - \left(1 - \dfrac{V_{db}}{PB}\right)^{1-MJ}\right)}{1-MJ}$ if $CBD > 0$. $Q_{bottom} = \dfrac{CJ * AD * PB * \left(1 - \left(1 - \dfrac{V_{db}}{PB}\right)^{1-MJ}\right)}{1-MJ}$ otherwise.
$V_{db} \geq FC * PB$	$Q_{bottom} = CBD * \left(F1 + \dfrac{F3 * (V_{db} - FC * PB) + \dfrac{MJ * \left(V_{db}^2 - (FC * PB)^2\right)}{2 * PB}}{F2}\right)$ if $CBD > 0$. $Q_{bottom} = CJ * AD * \left(F1 + \dfrac{F3 * (V_{db} - FC * PB) + \dfrac{MJ * \left(V_{db}^2 - (FC * PB)^2\right)}{2 * PB}}{F2}\right)$ otherwise.

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.

- $F1 = \dfrac{PB*\left(1 - \left(1 - FC\right)^{1-MJ}\right)}{1-MJ}$
- $F2 = \left(1 - FC\right)^{1+MJ}$
- $F3 = 1 - FC*\left(1 + MJ\right)$

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
$V_{db} < FC * PB$	$Q_{sidewall} = \dfrac{CJSW * PD * PB * \left(1 - \left(1 - \dfrac{V_{db}}{PB}\right)^{1-MJSW}\right)}{1 - MJSW}$
$V_{db} \geq FC * PB$	$Q_{sidewall} = CJSW * PD * \left(F1 + \dfrac{F3 * \left(V_{db} - FC * PB\right) + \dfrac{MJSW * \left(V_{db}^2 - \left(FC * PB\right)^2\right)}{2 * PB}}{F2} \right)$

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.

- $F1 = \dfrac{PB * \left(1 - \left(1 - FC\right)^{1-MJSW}\right)}{1 - MJSW}$
- $F2 = \left(1 - FC\right)^{1+MJSW}$
- $F3 = 1 - FC * \left(1 + MJSW\right)$

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*:

$$KP(T) = \frac{KP}{\left(T/T_{meas}\right)^{3/2}}$$

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:

- $EG_{T_{meas}} = 1.16eV - \left(7.02e-4 * T_{meas}{}^2\right) / \left(T_{meas} + 1108\right)$
- $EG_T = 1.16eV - \left(7.02e-4 * T^2\right) / \left(T + 1108\right)$

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.

- $\Phi - 3.25 + EG_{T_{meas}}/2 + MTYPE * PHI/2 - NSS * q * TOX / (3.9 * \varepsilon_0)$

  $+MTYPE * (GAMMA * \sqrt{PHI} + PHI)$, if **Threshold voltage, VTO** is `NaN` and you specify values for both the **Oxide thickness, TOX** and **Substrate doping, NSUB** parameters.

- *Φ* 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.
  - $TOX * \sqrt{2 * 11.7 * \varepsilon_0 * q * NSUB} / (3.9 * \varepsilon_0)$, 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, ND** 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*:

$$CBD(T) = CBD\frac{pbo + MJ * \left(4 * 10^4 * (T - 300.15) * pbo - (PB(T) - pbo)\right)}{pbo + MJ * \left(4 * 10^4 * (T_{meas} - 300.15) * pbo - (PB - pbo)\right)}$$

where:

- *CBD* is the **Zero-bias BD capacitance, CBD** parameter value.
- *MJ* is the **Bottom grading coefficient, MJ** parameter value.
- $pbo = \dfrac{PB + \frac{kT_{meas}}{q}\left(\log\left(\frac{T_{meas}}{300.15}\right)^3 + \frac{q}{k}\left(\frac{1.115}{300.15} - \frac{EG_{T_{meas}}}{T}\right)\right)}{T_{meas}/300.15}$

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*:

$$CJSW(T) = CJSW \frac{pbo + MJSW * \left(4 * 10^4 * (T - 300.15) * pbo - (PB(T) - pbo)\right)}{pbo + MJSW * \left(4 * 10^4 * (T_{meas} - 300.15) * pbo - (PB - pbo)\right)}$$

where:

- *MJSW* is the **Side grading coefficient, MJSW** parameter value.

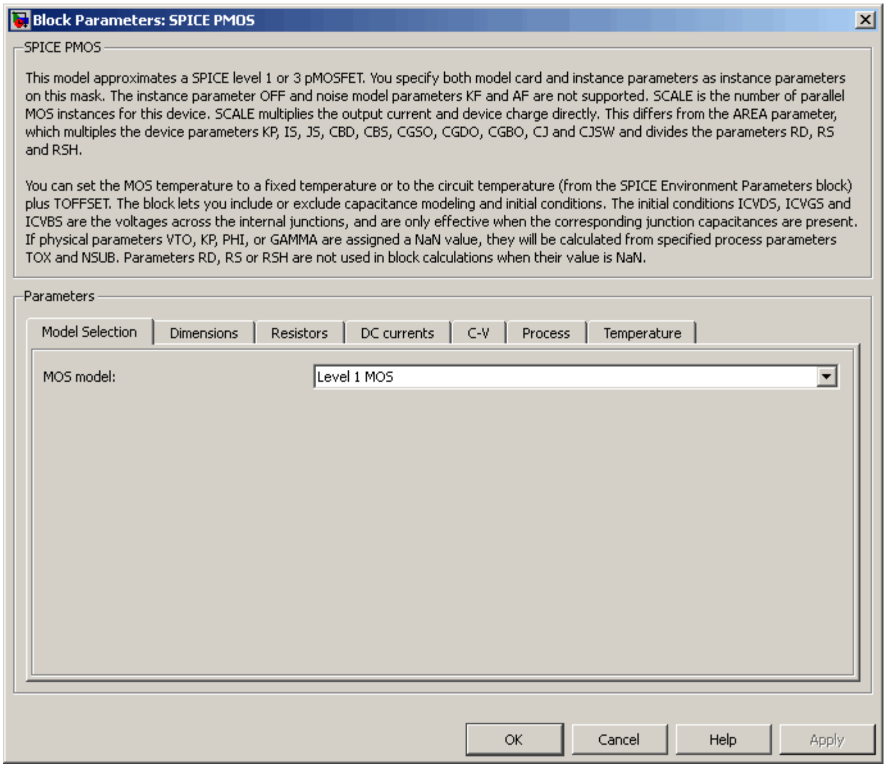
Basic Assumptions and Limitations

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.

Dialog Box and Parameters

Model Selection Tab

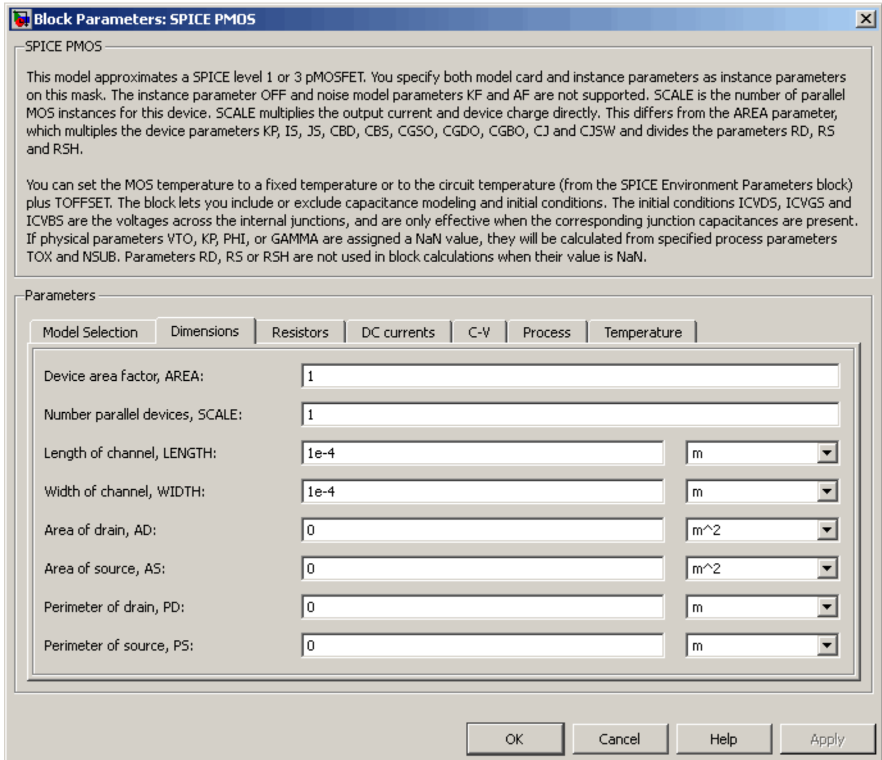


MOS model

Select one of the following MOSFET model options:

- `Level 1 MOS` — Use the “Level 1 Drain Current Model” on page 1-749. This is the default option.
- `Level 3 MOS` — Use the “Level 3 Drain Current Model” on page 1-752.

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^2. 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 m^2. The value must be greater than or equal to 0.

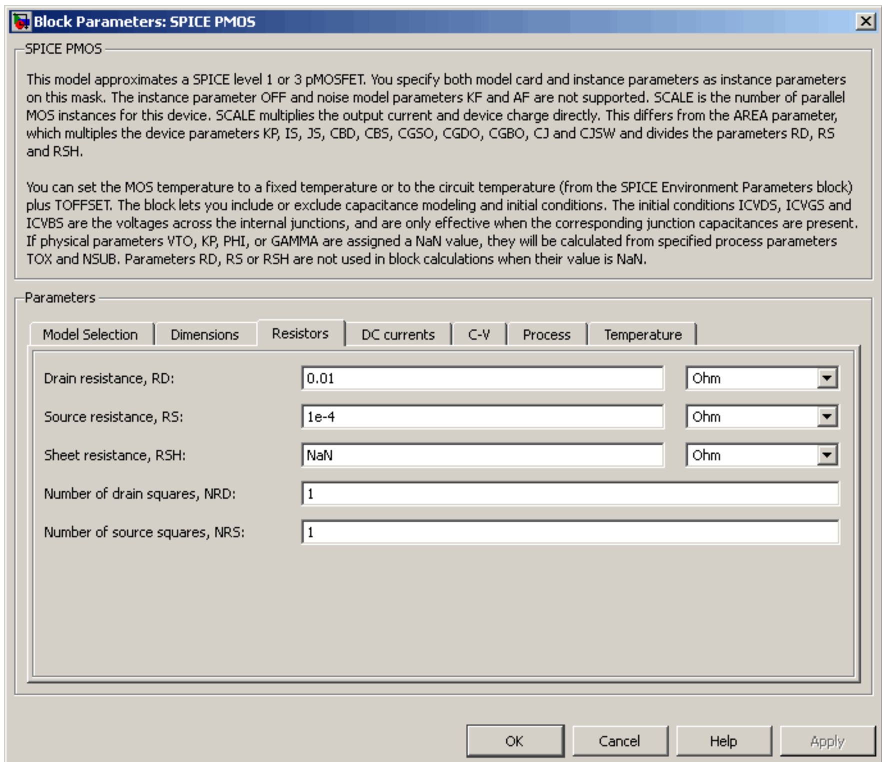
Perimeter of drain, PD

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` Ω. If you set this parameter to `NaN` Ω, this value means the parameter is unspecified, so the block calculates the drain resistance as described in “Resistance Calculations” on page 1-746. The value must be equal to `0` or greater than or equal to `Rmin`. `Rmin` is a built-in model constant whose value is `1e-12`.

Source resistance, RS

The transistor source ohmic resistance. The default value is `1e-4` Ω. If you set this parameter to `NaN` Ω, this value means the parameter is unspecified, so the block calculates the drain resistance as described in “Resistance Calculations” on page 1-746. The value must be equal to `0` or greater than or equal to `Rmin`. `Rmin` is a built-in model constant whose value is `1e-12`.

Sheet resistance, RSH

Resistance per square of the transistor source and drain. The default value is `Nan` Ω. 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 1-746. The value must be greater than or equal to `0`.

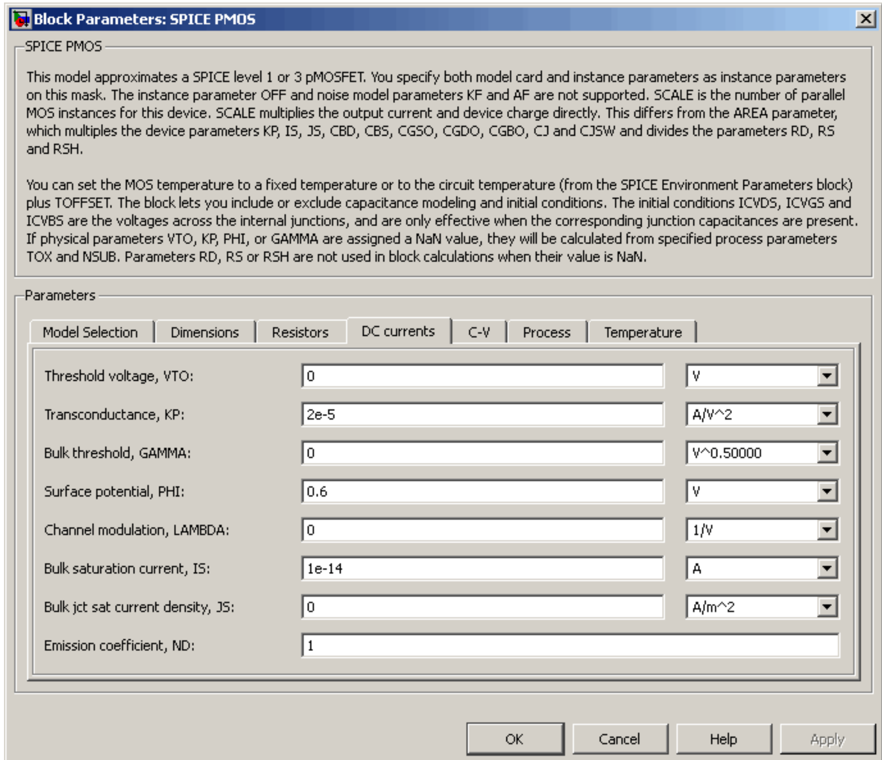
Number of drain squares, NRD

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 1-746.

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 1-746.

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 1-681.

Transconductance, KP

The derivative of drain current with respect to gate voltage. The default value is `2e-05` 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 1-667 or "Level 3 Drain Current Model" on page 1-752 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 1-667. 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 1-667 or "Level 3 Drain Current Model" on page 1-752 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 1-667 or "Level 3 Drain Current Model" on page 1-752 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^2. The value must be greater than or equal to `0`.

Emission coefficient, ND

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^2.

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`.

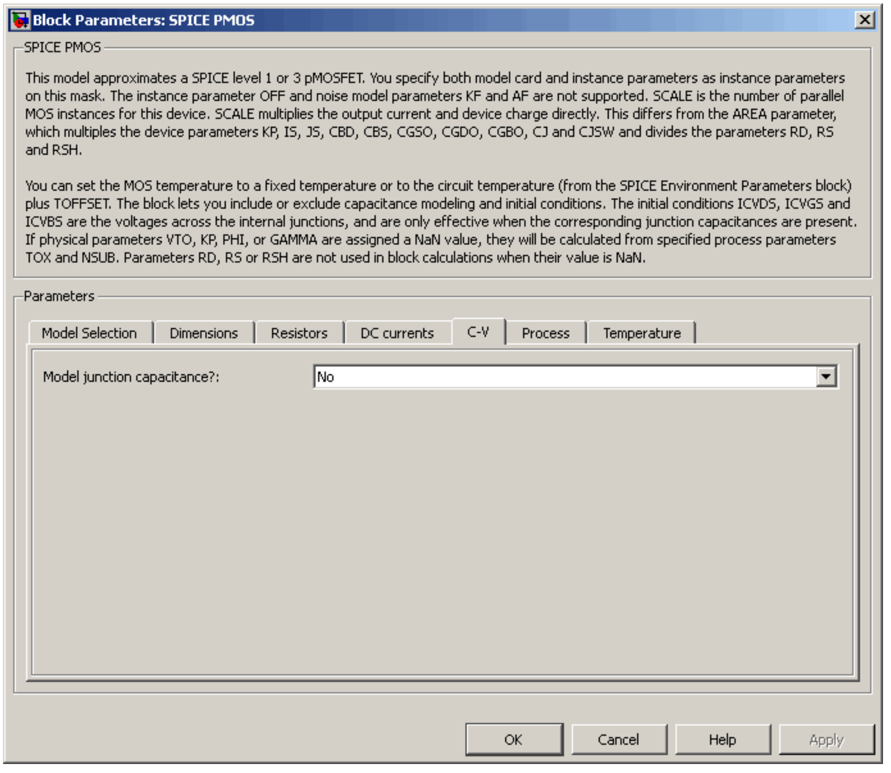
Vgs dependence on mobility, THETA

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 `0` 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 `0.2`.

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 `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`.

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 `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`.

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 `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`.

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^2. 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.

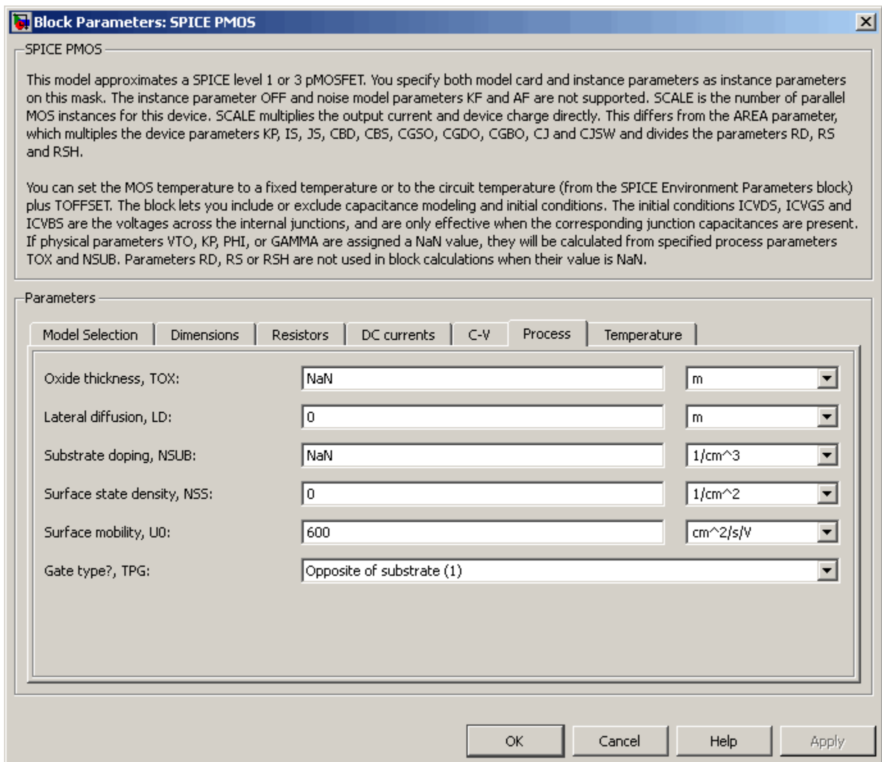
Initial condition voltage ICVGS
Gate-source voltage at the start of the simulation. This parameter is only visible when you select `Yes` for the **Model junction**

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^3. 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^2.

Surface mobility, U0
: Zero-bias surface mobility coefficient. The default value is `600` cm^2/V/s.

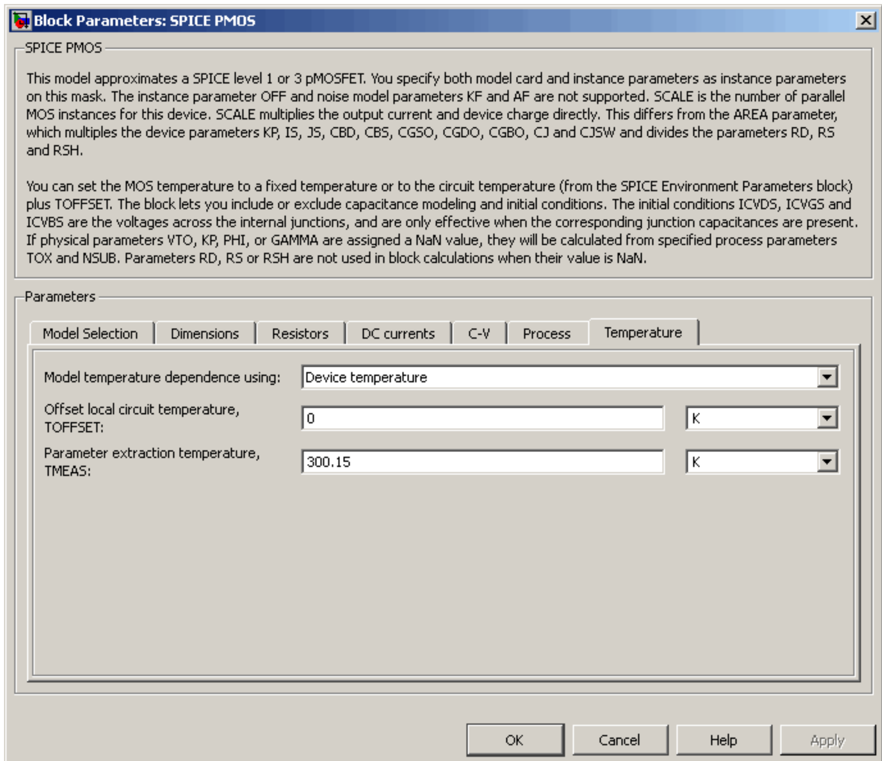
Junction depth, XJ
: Junction depth. This parameter is only visible when you select `Level 3 MOS` for the **MOS model** parameter. The default value is `0` m.

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

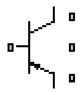
Purpose

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 1-785
- "Base Resistance Model" on page 1-789
- "Transit Charge Modulation Model" on page 1-789
- "Junction Charge Model" on page 1-790
- "Temperature Dependence" on page 1-792

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 785
- Terminal Currents on page 788
- Base Charge Model on page 788

Emitter-Base and Collector-Base Junction Currents

The base-emitter junction current is calculated using the following equations:

- When $V_{EB} > 80 * V_{TF}$:

$$I_{ebf} = IS * \left(\left(\frac{V_{EB}}{V_{TF}} - 79 \right) * e^{80} - 1 \right) + G_{\min} * V_{EB}$$

$$I_{ebe} = ISE * \left(\left(V_{EB} - 80 * V_{TF} + V_{TE} \right) * \frac{e^{(80*V_{TF}/V_{TE})}}{V_{TE}} - 1 \right)$$

- 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 $V_{CB} > 80 * V_{TR}$:

$$I_{cbr} = IS * \left(\left(\frac{V_{CB}}{V_{TR}} - 79 \right) * e^{80} - 1 \right) + G_{\min} * V_{CB}$$

$$I_{cbc} = ISC * \left(\left(V_{CB} - 80 * V_{TR} + V_{TC} \right) * \frac{e^{(80*V_{TR}/V_{TC})}}{V_{TC}} - 1 \right)$$

- 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.

Terminal Currents

The terminal currents, I_B and I_C are the base and collector currents, defined as positive into the device. They are calculated as:

$$I_B = -\left(\frac{I_{ebf}}{BF} + I_{ebe} + \frac{I_{cbr}}{BR} + I_{cbc} \right)$$

$$I_C = -\left(\frac{I_{ebf} - I_{cbr}}{q_b} - \frac{I_{cbr}}{BR} - I_{\text{cbc}} \right)$$

where BF and BR are the **Forward beta, BF** and **Reverse beta, BR** parameter values, respectively.

Base Charge Model

The base charge, q_b, 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_{bb} 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

$$r_{bb} = RBM + 3*\left(RB - RBM\right)*\left(\frac{\tan z - z}{z*\tan^2 z}\right)$$

  where:

$$z = \frac{\sqrt{1 + 144 I_B / \left(\pi^2 IRB\right)} - 1}{\left(24/\pi^2\right)\sqrt{\left(I_B / IRB\right)}}$$

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:

$$Q_{eb} = TF_{\text{mod}} * I_{eb} + Q_{dep}$$

- 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
$V_{jct} < FC * VJ$	$Q_{dep} = C_{jct} * VJ * \frac{1 - \left(1 - V_{jct}/VJ\right)^{(1-MJ)}}{1 - MJ}$
$V_{jct} \geq FC * VJ$	$Q_{dep} = C_{jct} * \left[F1 + \frac{F3 * \left(V_{jct} - FC * VJ\right) + \frac{MJ * \left[V_{jct}^2 - \left(FC * VJ\right)^2\right]}{2 * VJ}}{F2} \right]$

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.

- $F1 = VJ*\left(1-\left(1-FC\right)^{(1-MJ)}\right)/\left(1-MJ\right)$
- $F2 = \left(1-FC\right)^{(1+MJ)}$
- $F3 = 1-FC*\left(1+MJ\right)$

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
$V_{sc} < 0$	$Q_{sc} = CJS*VJS*\left(\frac{1-\left(1-V_{sc}/VJS\right)^{(1-MJS)}}{1-MJS}\right)$
$V_{sc} \geq 0$	$Q_{sc} = CJS*\left(1+MJS*V_{sc}/(2*VJS)\right)*V_{sc}$

where:

- *CJS* is the **C-S junction capacitance, CJS** parameter value.
- *VJS* is the **Substrate built-in potential, VJS** parameter value.
- *MJS* 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*:

$$IS(T) = IS * \left(T/T_{meas}\right)^{XTI} * e^{\left(\frac{T}{T_{meas}} - 1\right) * \frac{EG}{V_t}}$$

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.
- $EG_{T_{meas}} = 1.16eV - \left(7.02e-4 * T_{meas}^{\ 2}\right) / \left(T_{meas} + 1108\right)$
- $EG_T = 1.16eV - \left(7.02e-4 * T^2\right) / \left(T + 1108\right)$

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*:

$$CJE(T) = CJE * \left[1 + MJE * \left(400e-6 * \left(T - T_{meas}\right) - \frac{VJE(T) - VJE}{VJE}\right)\right]$$

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:

- β 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*:

$$ISE(T) = ISE * \left(\frac{T}{T_{meas}} \right)^{\text{-XTB}} * \left(\frac{\text{IS(T)}}{\text{IS}} \right)^{1/NE}$$

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*.

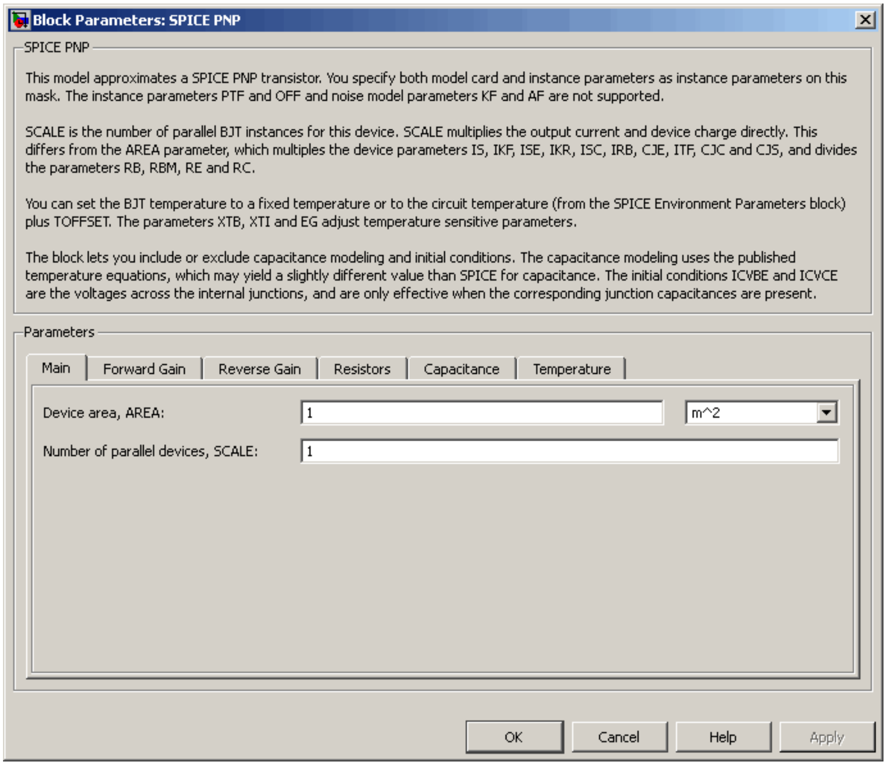
Basic Assumptions and Limitations

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.

Dialog Box and 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**

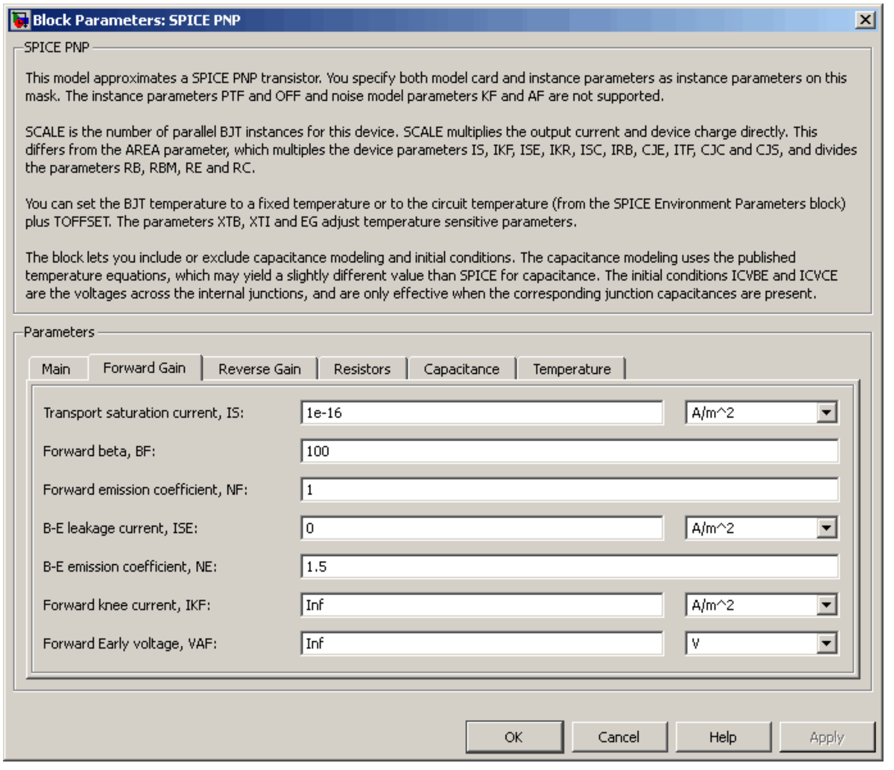
It divides the following parameter values:

- **Zero-bias base resistance, RB**
- **Minimum base resistance, RBM**
- **Emitter resistance, RE**
- **Collector resistance, RC**

The default value is 1 m^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.

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^2. 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 0 A/m^2. The value must be greater than or equal to 0.

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 0.

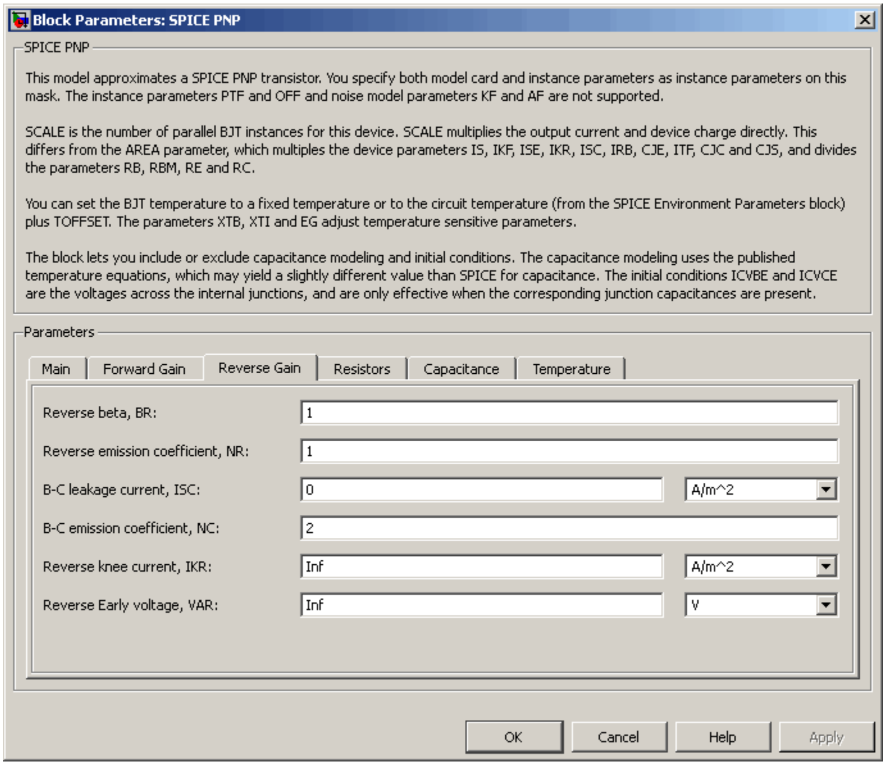
Forward knee current, IKF

The current value at which forward-beta high-current roll-off occurs. The default value is 0 A/m^2. The value must be greater than or equal to 0. For this parameter, the software interprets a value of 0 as infinity.

Forward Early voltage, VAF

The forward Early voltage. 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.

Reverse Gain Tab



Reverse beta, BR

The ideal maximum reverse beta. The default value is 1. The value must be greater than 0.

Reverse emission coefficient, NR

The reverse emission coefficient or ideality factor. The default value is 1. The value must be greater than 0.

B-C leakage current, ISC

The base-collector leakage current. The default value is 0 A/m^2. The value must be greater than or equal to 0.

B-C emission coefficient, NC

The base-collector emission coefficient or ideality factor. The default value is 2. The value must be greater than 0.

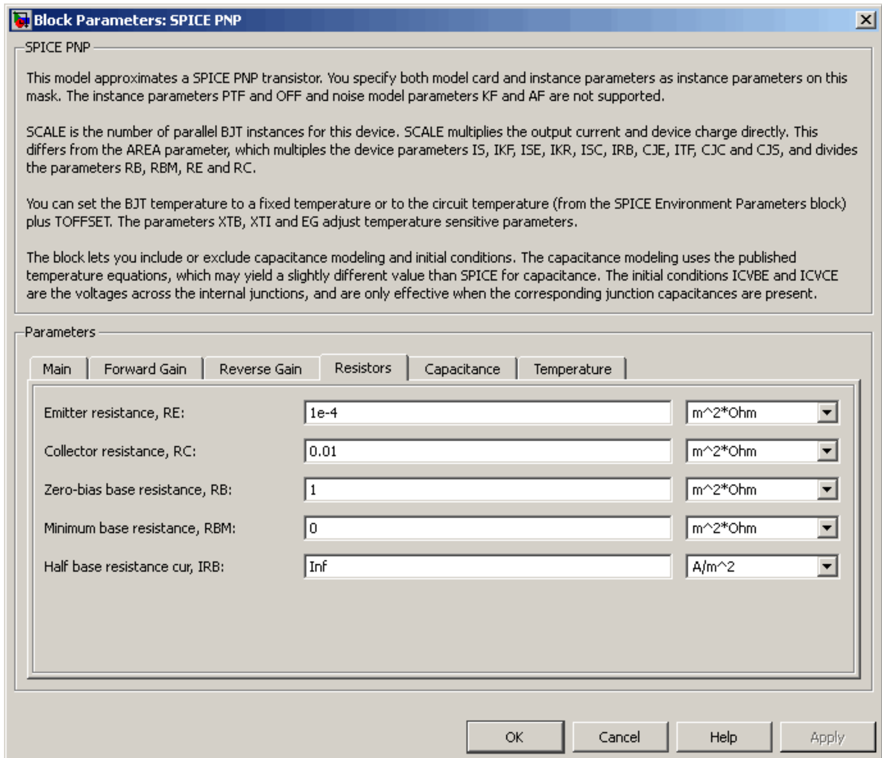
Reverse knee current, IKR

The current value at which reverse-beta high-current roll-off occurs. The default value is 0 A/m^2. The value must be greater than or equal to 0. For this parameter, the software interprets a value of 0 as infinity.

Reverse Early voltage, VAR

The reverse Early voltage. 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.

Resistors Tab



Emitter resistance, RE

The resistance of the emitter. The default value is `1e-4` m^2*Ω. The value must be greater than or equal to `0`.

Collector resistance, RC

The resistance of the collector. The default value is `0.01` m^2*Ω. 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` m^2*Ω. The value must be greater than or equal to `0`.

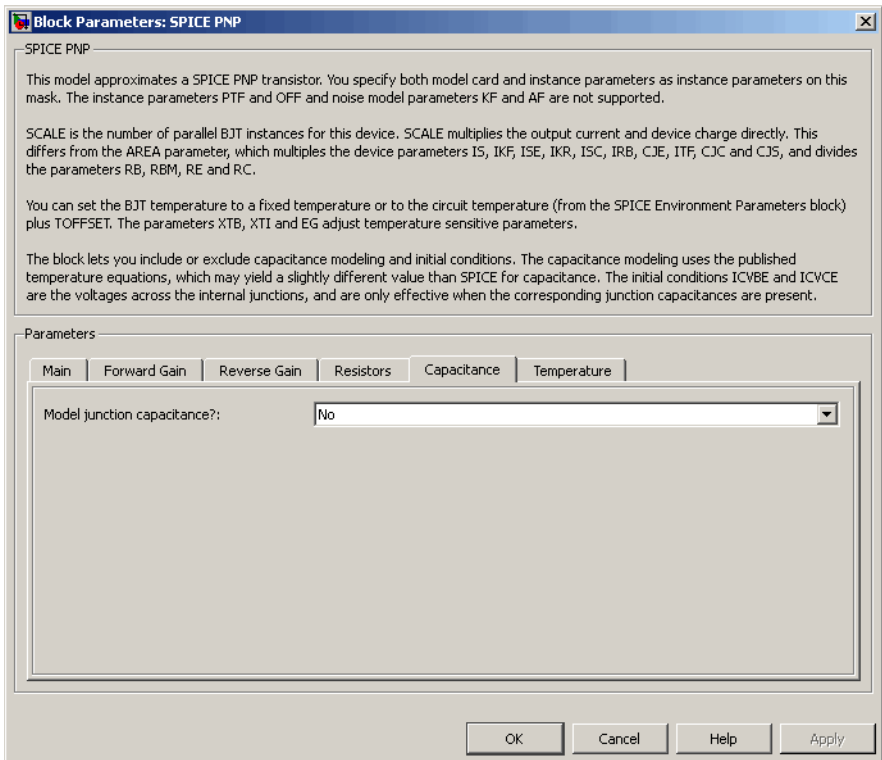
Minimum base resistance, RBM

The minimum resistance of the base. The default value is `0` $m^2 * \Omega$. 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 `Inf` A/m^2. The value must be greater than or equal to `0`. Use the default value of `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` 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^2. 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^2. 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^2. The value must be greater than or equal to `0`.

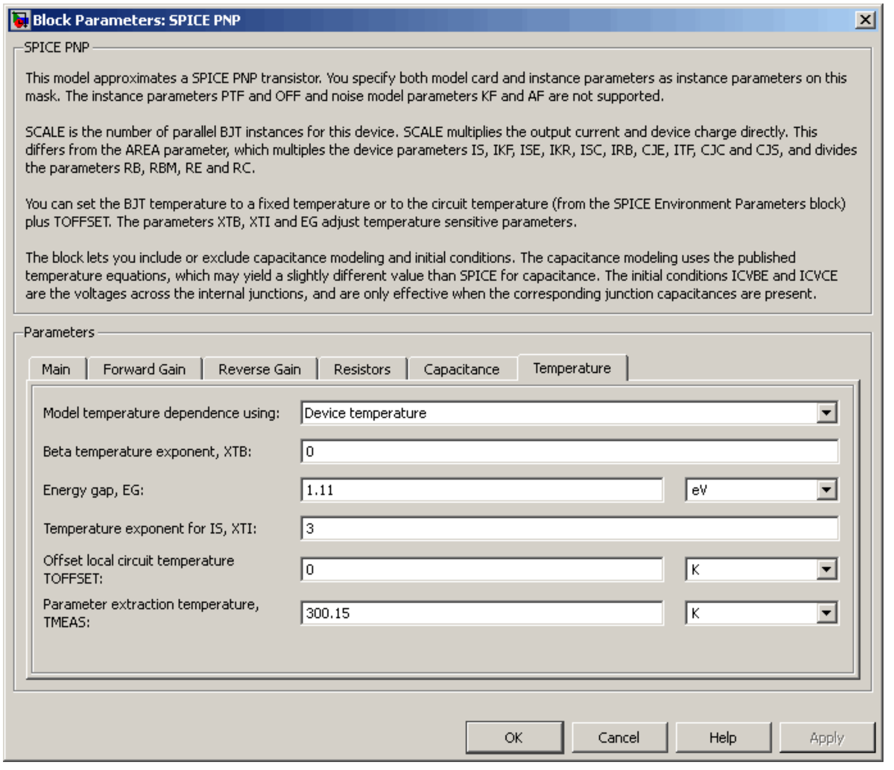
Substrate built-in potential, VJS

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

Purpose

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:

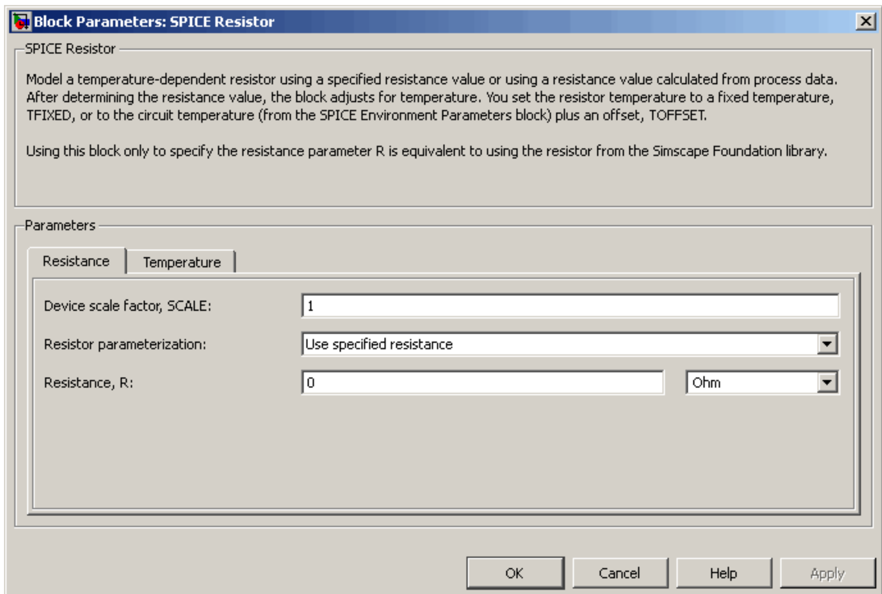
$$R = R_0(1 + TC1(T - T_{nom}) + TC2(T - T_{nom})^2)$$

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.

Dialog Box and 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` Ω.

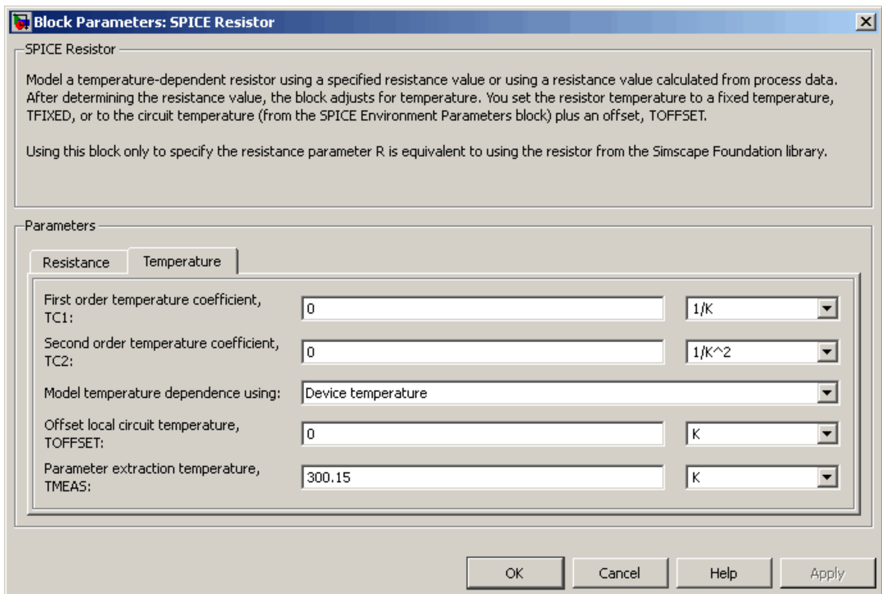
Sheet resistance, RSH
Resistance per square of the resistor. This parameter is only visible when you select `Calculate from process information` for the **Resistor parameterization** parameter. The default value is `0` Ω.

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` $1/K^2$.

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

SPST Switch

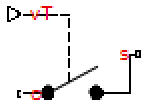
Purpose

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.

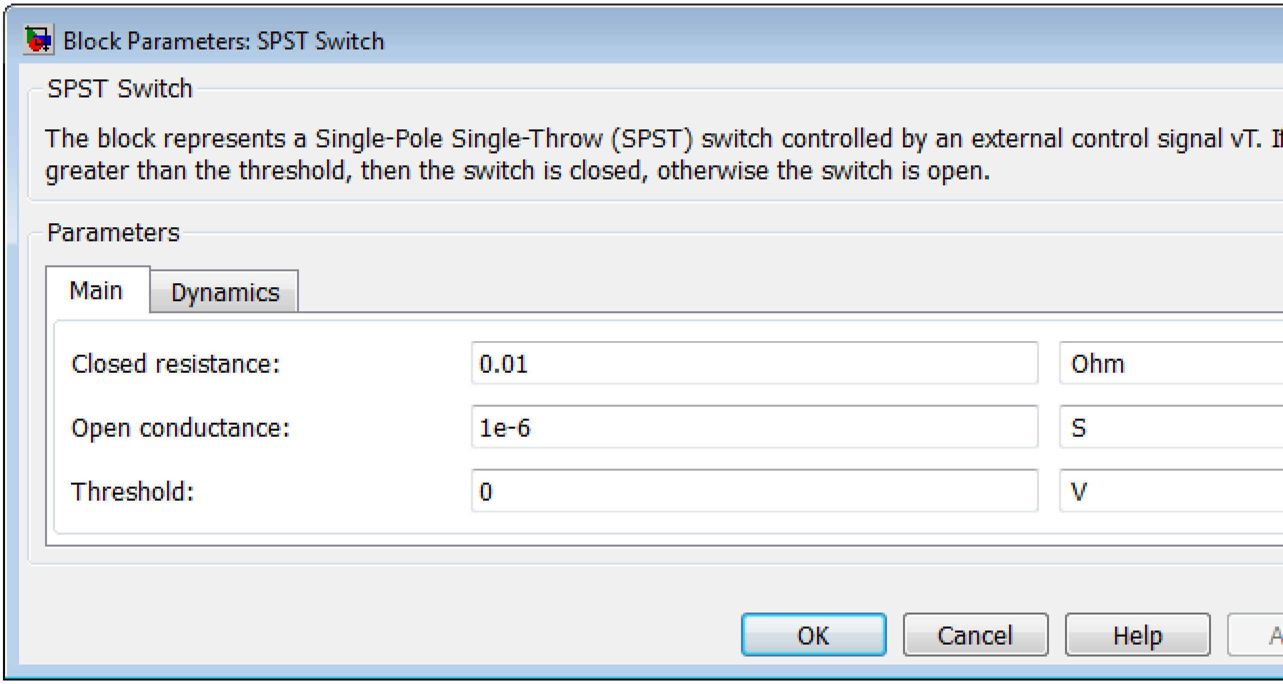
The switch is closed if the voltage presented at the `vT` control port exceeds the value of the **Threshold** parameter.

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`.

Dialog Box and Parameters

- "Main Tab" on page 1-819
- "Dynamics Tab" on page 1-820

Main Tab



Closed resistance

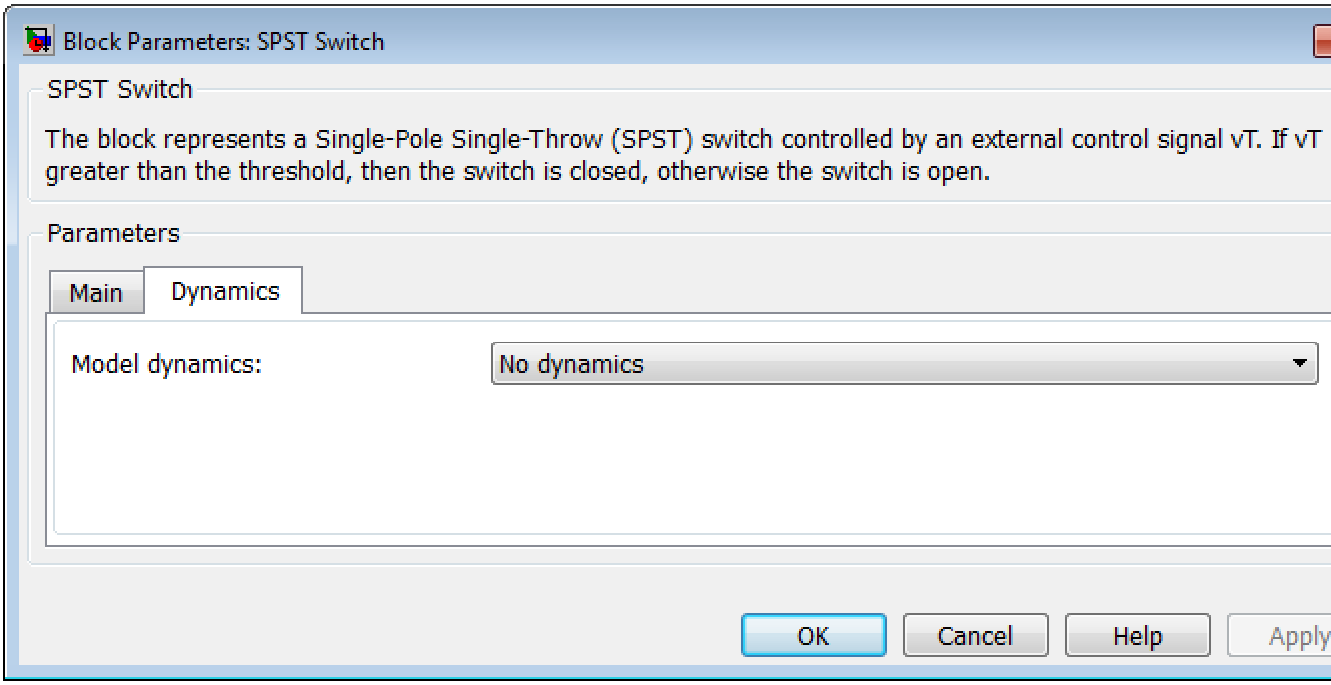
Resistance between the `c` and `s` electrical ports when the switch is closed. The value must be greater than zero. The default value is `0.01` Ω.

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` 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 vT 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, vT
The value of the physical signal input `vT` 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:

`vT`
Physical signal that opens and closes the switch.

`c`, `s`
Electrical conserving ports.

See Also

DPDT Switch

DPST Switch

SPDT Switch

Simscape Switch

Stepper Motor

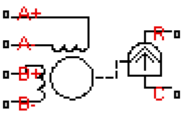
Purpose

Model stepper motor

Library

Rotational Actuators

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:

$$e_A = -K_m \omega \sin(N_r \theta)$$

$$e_B = K_m \omega \cos(N_r \theta)$$

$$\frac{di_A}{dt} = \left(v_A - Ri_A - e_A\right) / L$$

$$\frac{di_B}{dt} = \left(v_B - Ri_B - e_B\right) / L$$

$$\frac{d\omega}{dt} = \left(-K_m \left(i_A - e_A / R_m\right)\sin\left(N_r \theta\right) + K_m \left(i_B - e_B / R_m\right)\cos\left(N_r \theta\right) - B\omega\right) / J$$

$$\frac{d\theta}{dt} = \omega$$

where:

- e_A and e_B are the back 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.
- ω is the rotor speed.
- Θ is the rotor angle.

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.

Thermal Ports

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. For more information on using thermal ports in actuator blocks, see "Simulating Thermal Effects in Rotational and Translational Actuators".

Basic Assumptions and Limitations

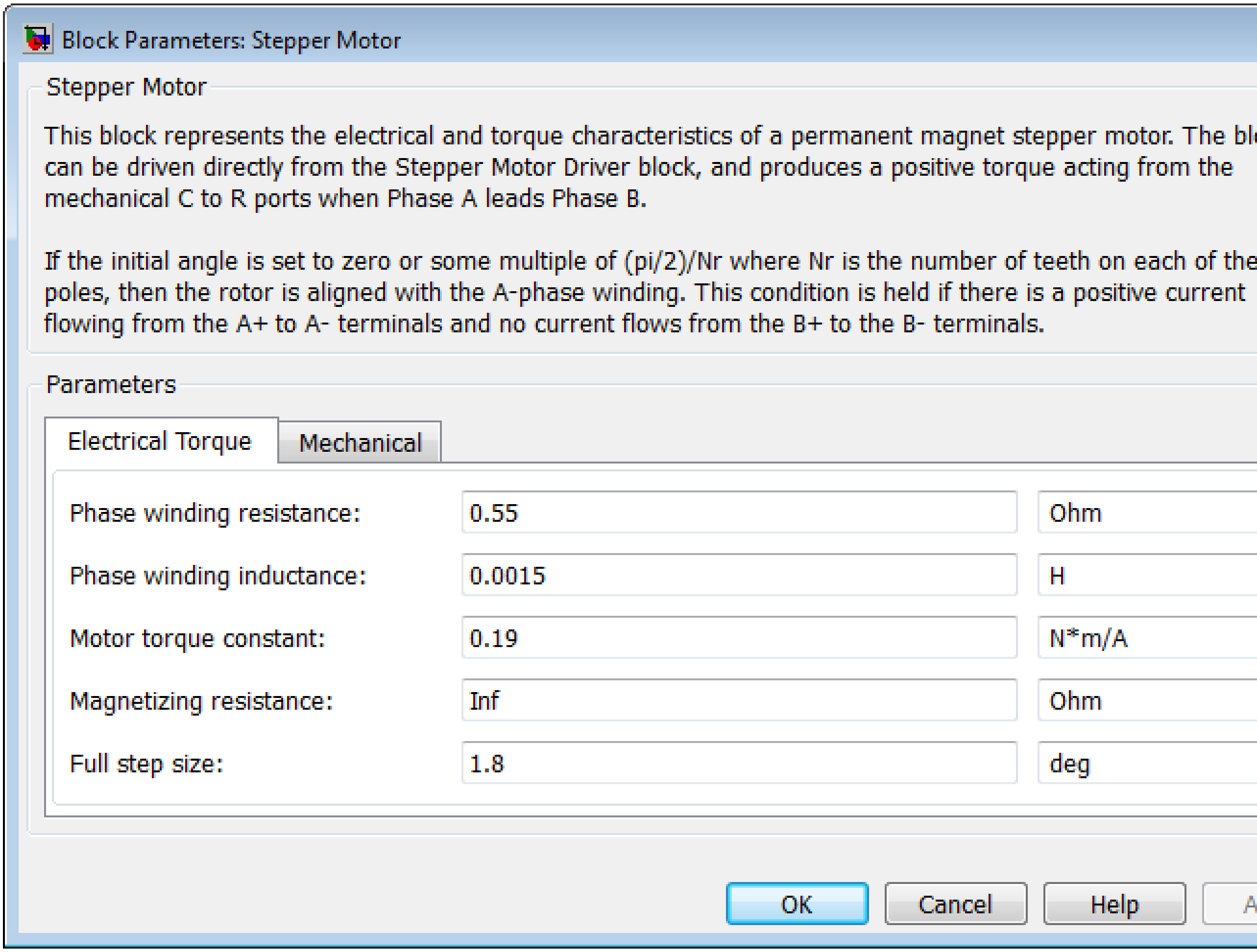
The model is based on the following assumptions:

- This model neglects magnetic saturation effects, detent torque, 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 –π and π.

Dialog Box and Parameters

- "Electrical Torque Tab" on page 1-825
- "Mechanical Tab" on page 1-827
- "Temperature Dependence Tab" on page 1-828
- "Thermal Port Tab" on page 1-830

Electrical Torque Tab



Phase winding resistance

Resistance of the A and B phase windings. The default value is `0.55` Ω.

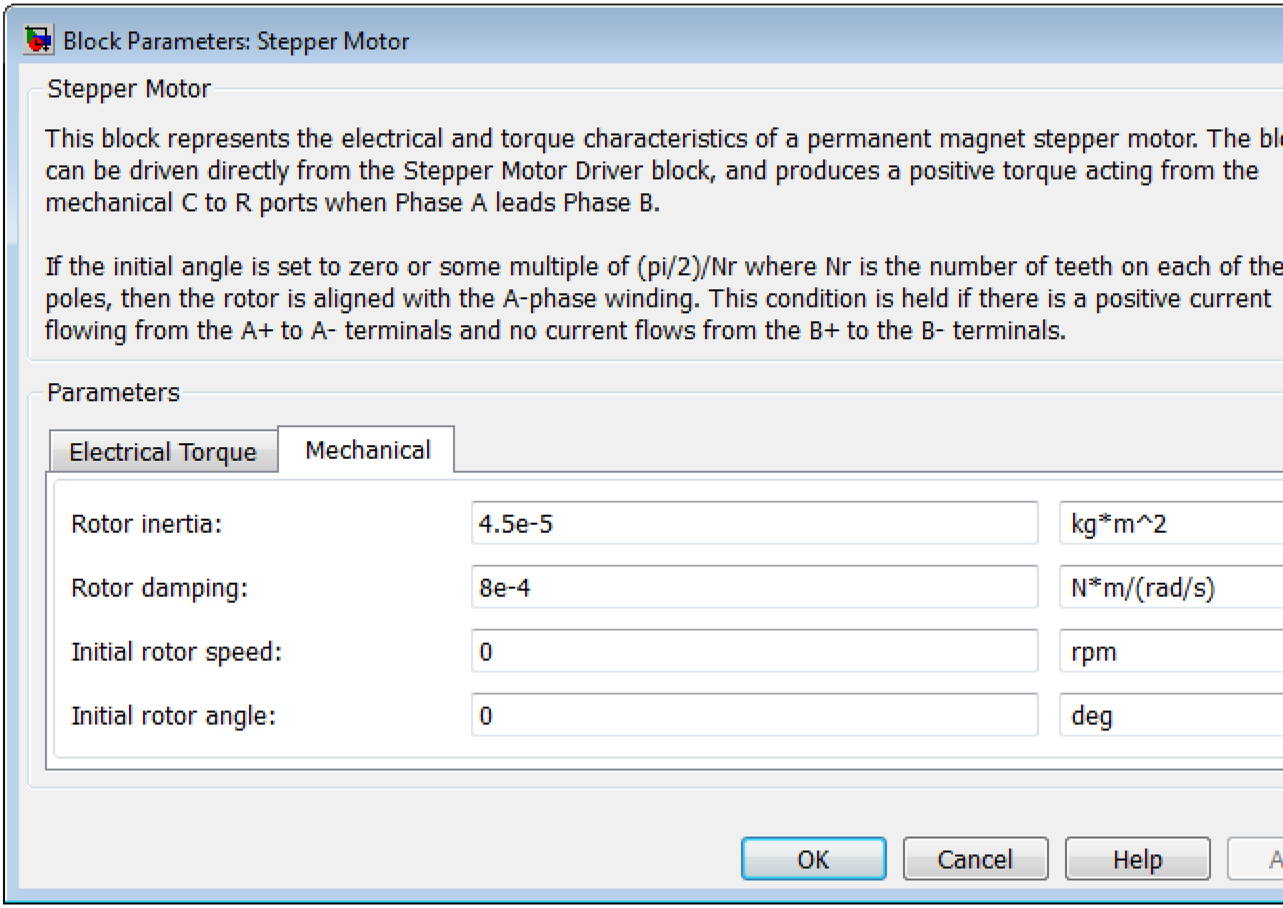
Phase winding inductance
Inductance of the A and B phase windings. The default value is `0.0015` H.

Motor torque constant
Motor torque constant K_m. The default value is `0.19` N*m/A.

Magnetizing resistance
The 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.

Full step size
Step size when changing the polarity of either the A or B phase current. The default value is `1.8°`.

Mechanical Tab



Rotor inertia

Resistance of the rotor to change in motor motion. The default value is `4.5e-05` kg*m^2. The value can be zero.

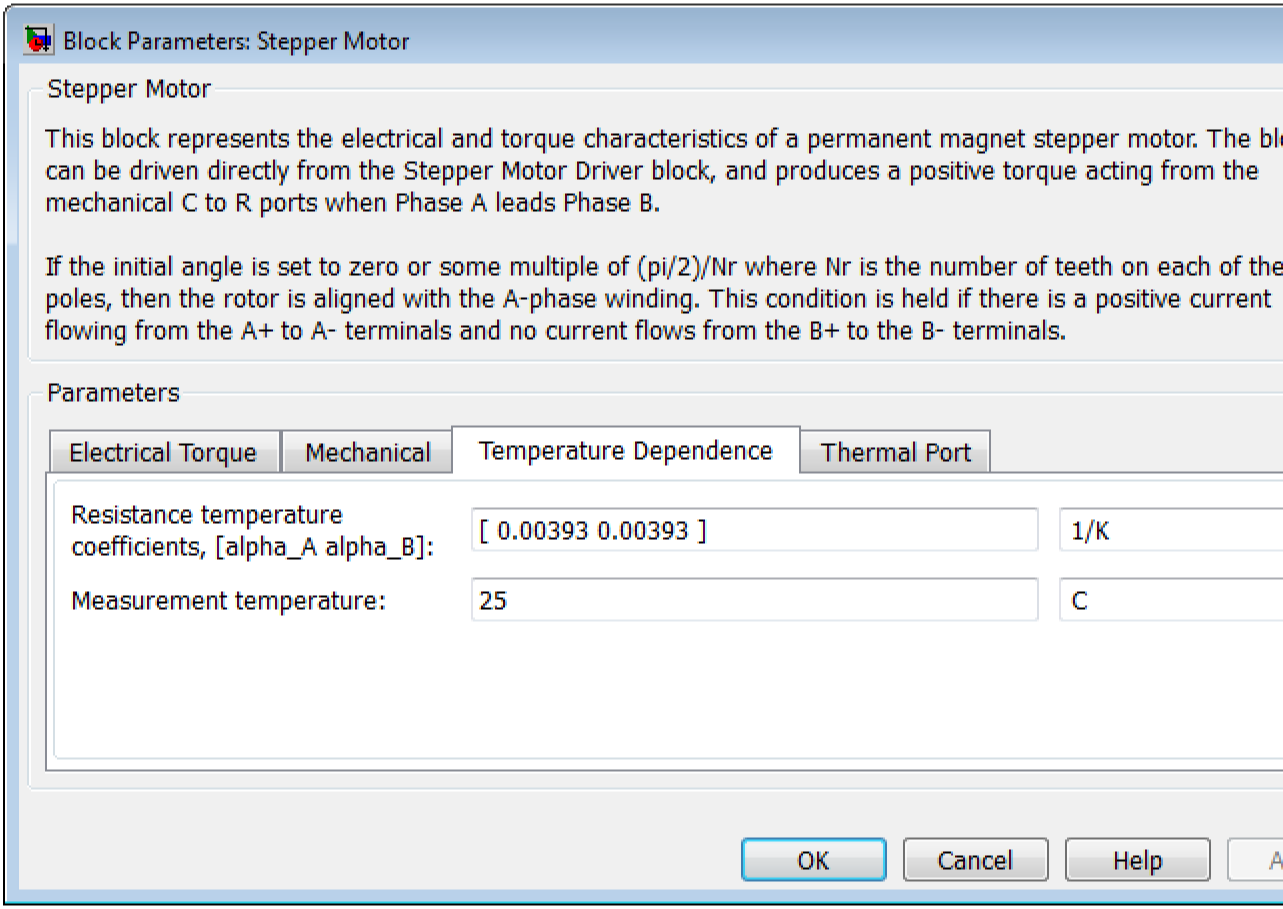
Rotor damping
Energy dissipated by the rotor. The default value is 8e-04 N*m/(rad/s). The value can be zero.

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 1-823.



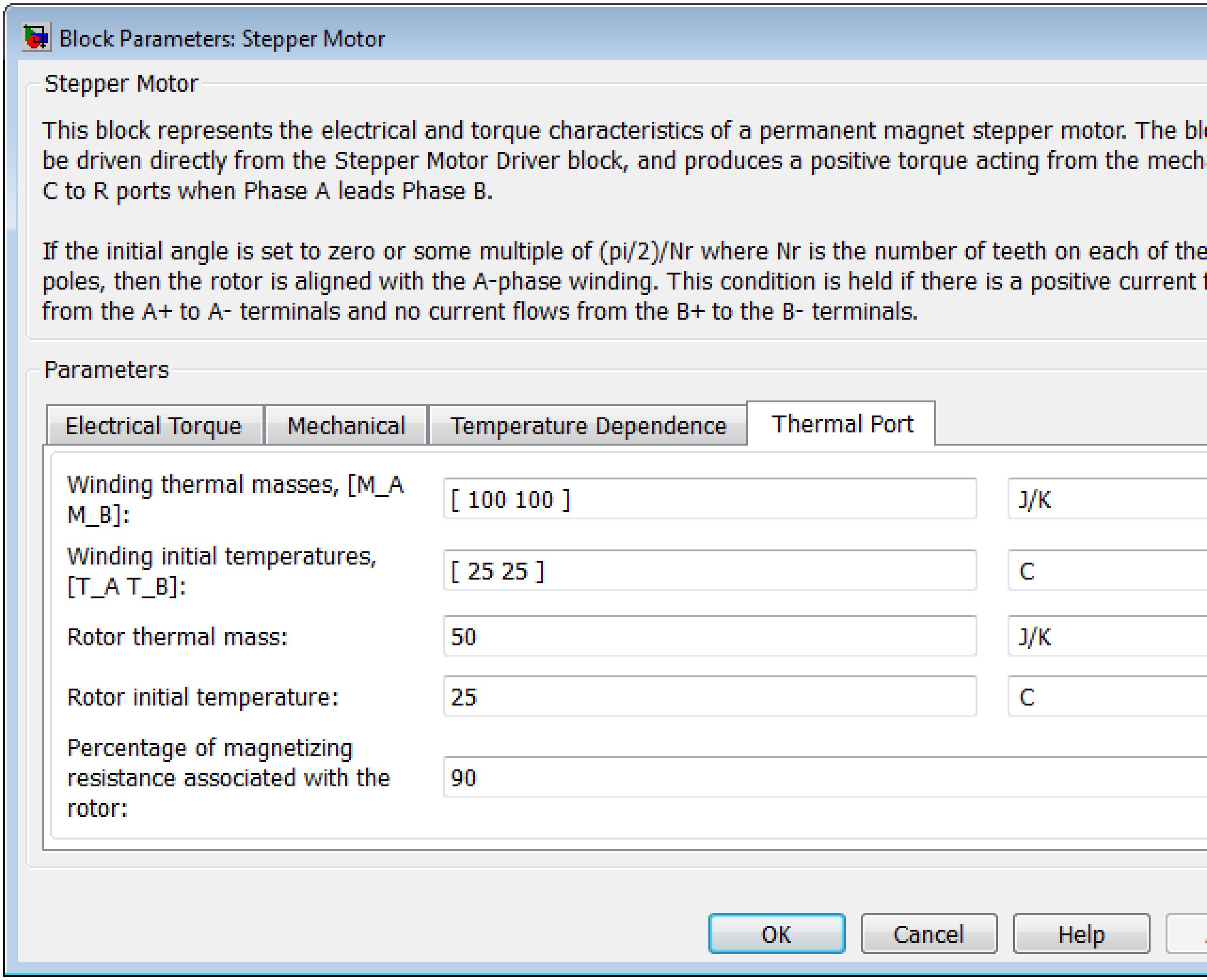
Resistance temperature coefficients, [alpha_A alpha_B]

A 1 by 2 row vector defining the coefficient α in the equation relating resistance to temperature, as described in "Thermal Model for Actuator Blocks". The first element corresponds to winding A, and the second to winding B. 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 1-823.


Block Parameters: Stepper Motor
Stepper Motor
This block represents the electrical and torque characteristics of a permanent magnet stepper motor. The bl
be driven directly from the Stepper Motor Driver block, and produces a positive torque acting from the mech
C to R ports when Phase A leads Phase B.
If the initial angle is set to zero or some multiple of (pi/2)/Nr where Nr is the number of teeth on each of the
poles, then the rotor is aligned with the A-phase winding. This condition is held if there is a positive current f
from the A+ to A- terminals and no current flows from the B+ to the B- terminals.
Parameters
Electrical Torque
Mechanical
Temperature Dependence
Thermal Port
Winding thermal masses, [M_A M_B]:
[100 100]
J/K
Winding initial temperatures, [T_A T_B]:
[25 25]
C
Rotor thermal mass:
50
J/K
Rotor initial temperature:
25
C
Percentage of magnetizing resistance associated with the rotor:
90
OK
Cancel
Help

Winding thermal masses, [M_A M_B]
A 1 by 2 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. The default value is `[ 100 100 ]` J/K.

Winding initial temperatures, [T_A T_B]
A 1 by 2 row vector defining the temperature of the A and B thermal ports at the start of simulation. The default value is `[ 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 two winding thermal ports HA and HB. The default value is `90`%.

Ports

The block has the following ports:

`A+`
Top A-phase electrical connection.

`A-`
Lower A-phase electrical connection

`B+`
Top B-phase electrical connection.

`B-`
Lower B-phase electrical connection.

C

Mechanical rotational conserving port.

R

Mechanical rotational conserving port.

HA

Winding A thermal port. For more information, see "Thermal Ports" on page 1-823.

HB

Winding B thermal port. For more information, see "Thermal Ports" on page 1-823.

HR

Rotor thermal port. For more information, see "Thermal Ports" on page 1-823.

Examples

See the Controlled Stepper Motor example.

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

Stepper Motor Driver

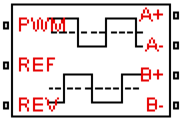
Purpose

Model stepper motor driver

Library

Drivers

Description



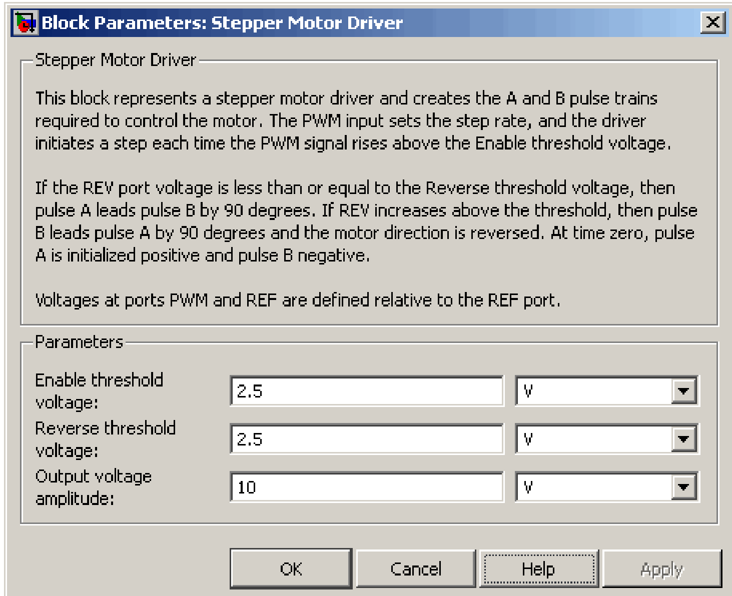
The Stepper Motor Driver block represents a stepper motor driver. It creates the pulse trains, A and B, required to control the motor. This block initiates a step each time the voltage at the PWM port rises above the **Enable threshold voltage**.

If the voltage at the REV port is less than or equal to the **Reverse threshold voltage**, pulse A leads pulse B by 90 degrees. If the voltage at the REV port is greater than the **Reverse threshold voltage**, 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.

Use the Controlled PWM Voltage block to create the voltage at the PWM port. This block creates a network engine event every time the PWM signal goes high. The network engine event triggers a simulation time point when the PWM signal goes high, which ensures good simulation accuracy. If you instead use the Controlled Voltage Source block from the Foundation library, you need to set a suitably small time step for the simulation. For information about specifying the Simulink step size, see "Choose a Solver" in the Simulink documentation.

Dialog Box and Parameters



Enable threshold voltage

When the voltage at the PWM port rises above this threshold, the Stepper Motor Driver block initiates a step. The default value is `2.5` V.

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.

Ports

The block has the following ports:

A+

Positive electrical output of pulse A.

`A-`
: Negative electrical output of pulse A

`B+`
: Positive electrical output of pulse B.

`B-`
: Negative electrical output of pulse B.

`PWM`
: Triggering input step voltage.

`REF`
: Input floating reference voltage.

`REV`
: Input voltage that controls motor direction.

Examples

See the Controlled Stepper Motor example.

See Also

Controlled PWM Voltage and Stepper Motor.

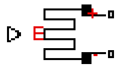
Purpose

Model deformation sensor

Library

Sensors

Description



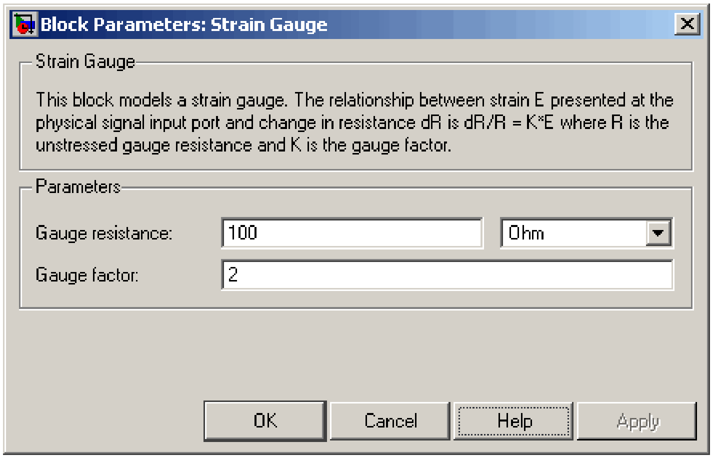
The Strain Gauge block represents a sensor that generates a change in resistance as a function of strain using the following equation:

$$\frac{\Delta R}{R} = K\varepsilon$$

where:

- $\Delta R/R$ is the fractional change in resistance.
- ε is the strain at port B.
- K is the **Gauge factor** parameter value.

Dialog Box and Parameters



Gauge resistance

The unstressed gauge resistance. The default value is 100 Ω.

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:

B

Strain input.

+

Positive electrical port.

-

Negative electrical port.

Thermal Resistor

Purpose

Model resistor with thermal port

Library

Passive Devices

Description

The Thermal Resistor block represents a temperature-dependent resistor. The resistance when the temperature at the thermal port is T is

$$R = R_0(1+\alpha(T-T_0))$$

where:

- R_0 is the nominal resistance at the reference temperature T_0.
- α is the temperature coefficient.

The following equation describes the thermal behavior of the block:

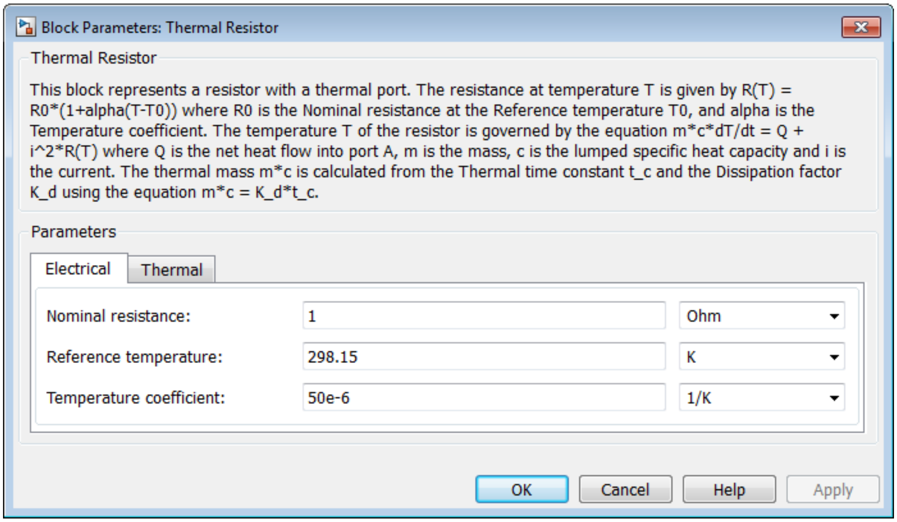
$$Q = K_d t_c \frac{dT}{dt} - i^2 R$$

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.
- i is the current through the resistor.

Dialog Box and Parameters

Electrical Tab



Nominal resistance

The nominal resistance of the thermistor at the reference temperature. Many datasheets quote the nominal resistance at 25°C (298.15 K) and list it as R25. The default value is `1` Ω.

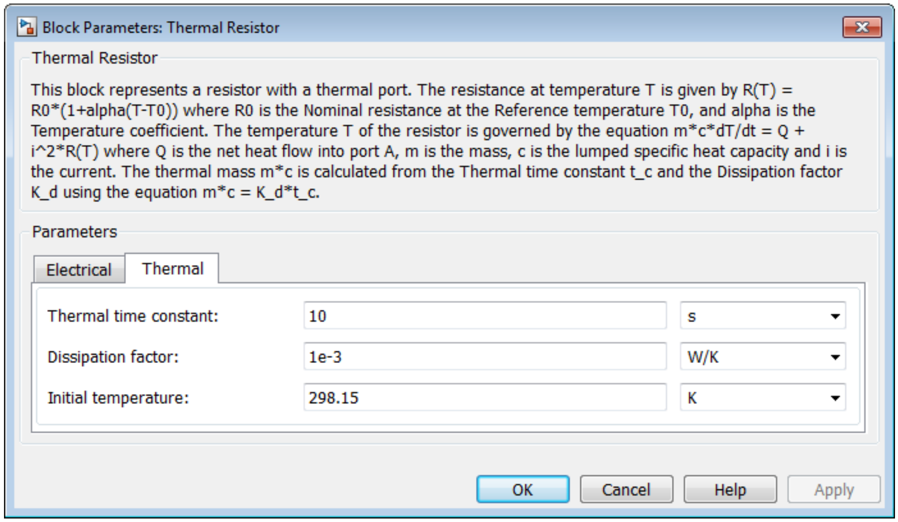
Reference temperature

The temperature at which the nominal resistance was measured. The default value is `298.15` K.

Temperature coefficient

The coefficient α in the equation that describes resistance as a function of temperature. The default value is `5e-05` 1/K.

Thermal Tab



Thermal time constant
: The time it takes the resistor temperature to reach 63% of the final temperature change when a step change in ambient temperature occurs. The default value is `10` s.

Dissipation factor
: The thermal power required to raise the thermal resistor temperature by one K. The default value is `0.001` W/K.

Initial temperature
: The temperature of the thermal resistor at the start of the simulation. The default value is `298.15` K.

Ports

The block has the following ports:

A
: Resistor thermal port

+
: Positive electrical port

-
: Negative electrical port

See Also

Thermistor, Thermocouple.

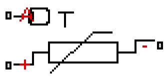
Purpose

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

$$R = R_0 (e^{B(1/T - 1/T_0)} - 1)$$

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.

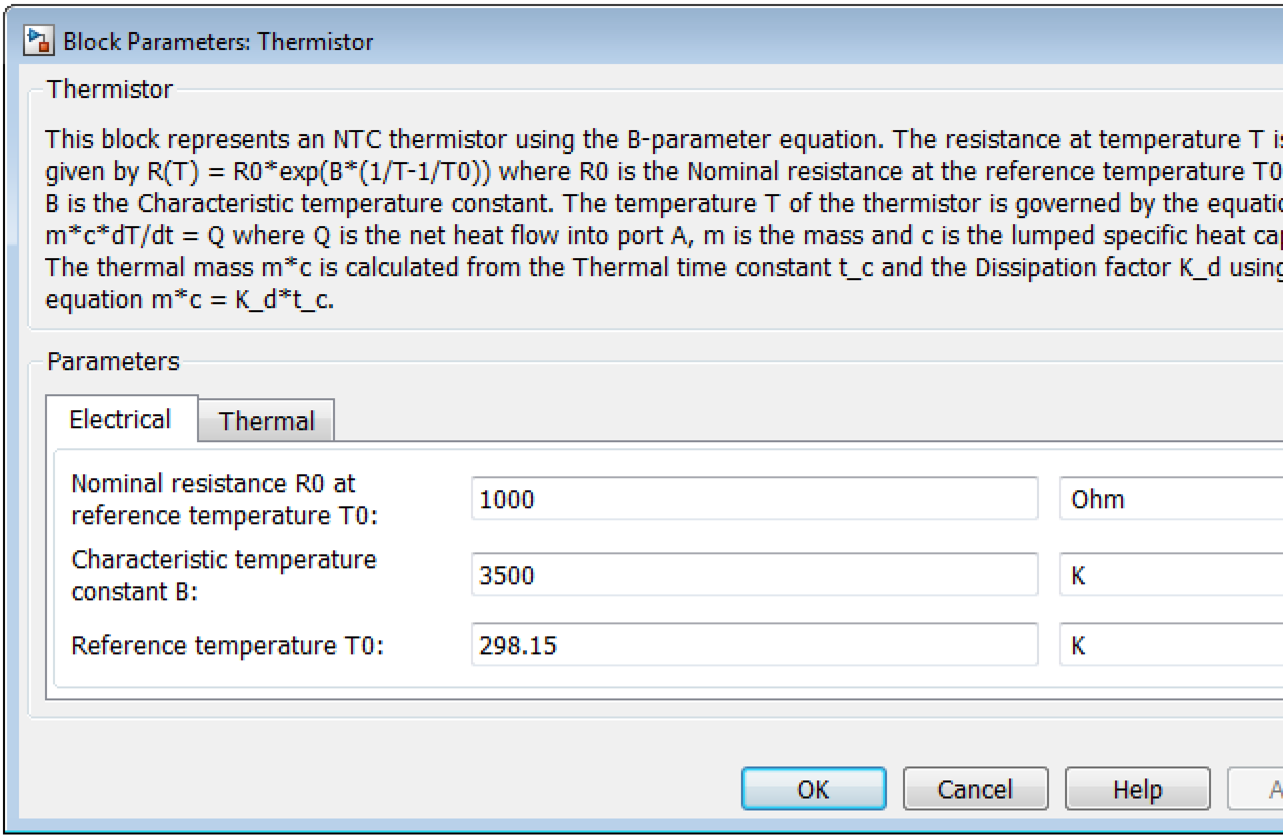
2. Connect the A port of the Convective Heat Transfer block to a Simscape Ideal Temperature Source block whose temperature is set to the ambient temperature.

3 Set the **Area** parameter of the Convective Heat Transfer block to an approximate area A_{nom}.

4 Set the **Heat transfer coefficient** parameter of the Convective Heat Transfer block to K_d/A_{nom}.

Dialog Box and Parameters

Electrical Tab



Nominal resistance R0 at reference temperature T0

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 Ω.

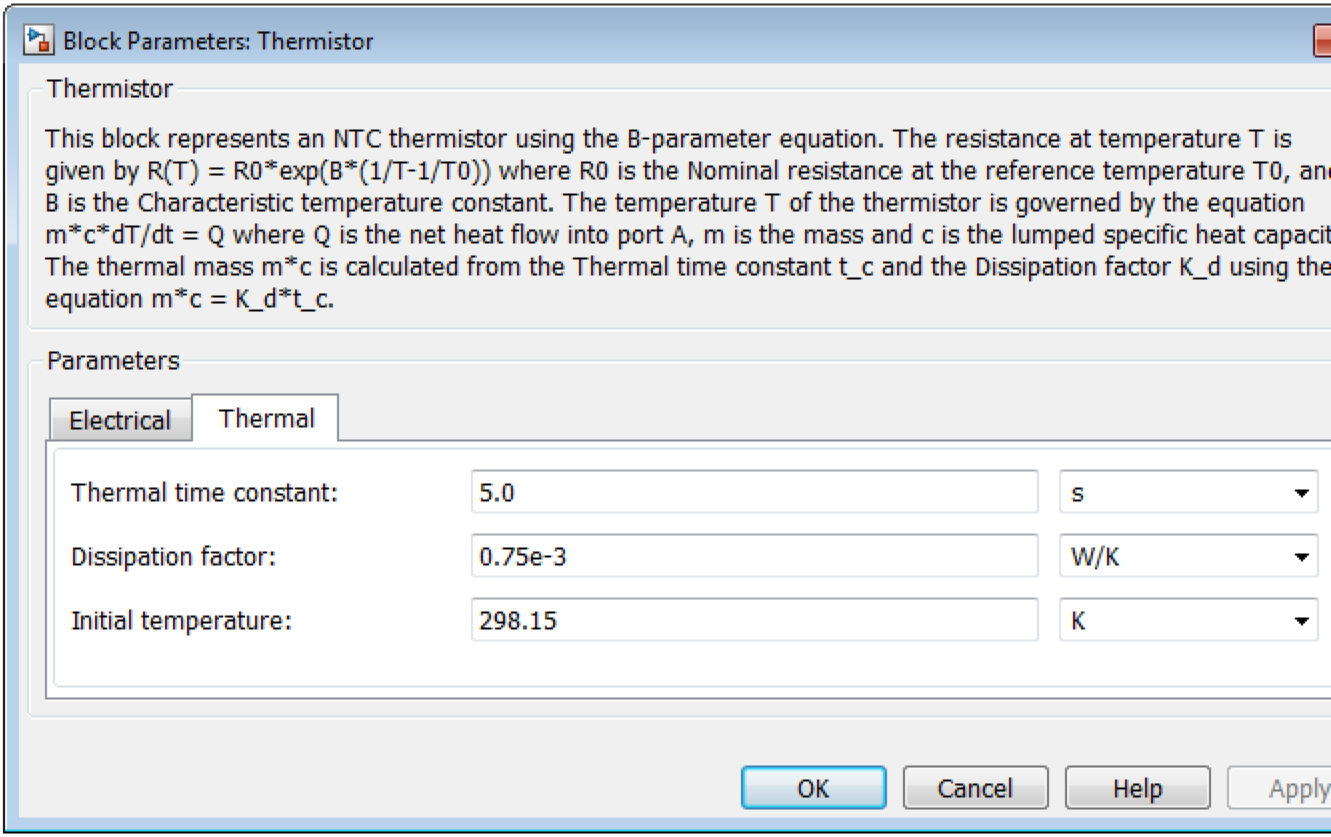
Characteristic temperature constant B

The coefficient *B* in the equation that describes resistance as a function of temperature. The default value is `3500` K.

Reference temperature T0

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 `0.75e-4` W/K.

Initial temperature
The temperature of the thermistor at the start of the simulation. The default value is `298.15` K.

Ports

The block has the following ports:

`A`
Thermal port.

`+`
Positive electrical port.

`-`
Negative electrical port.

See Also

PTC Thermistor

Thermal Resistor

Thermocouple

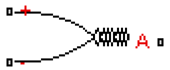
Purpose

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]. The voltage E across the device in mV is

$$E(mV) = c0 + c1*t + \ldots + cn*t^n$$

where:

- ci is the i^{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.

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}.

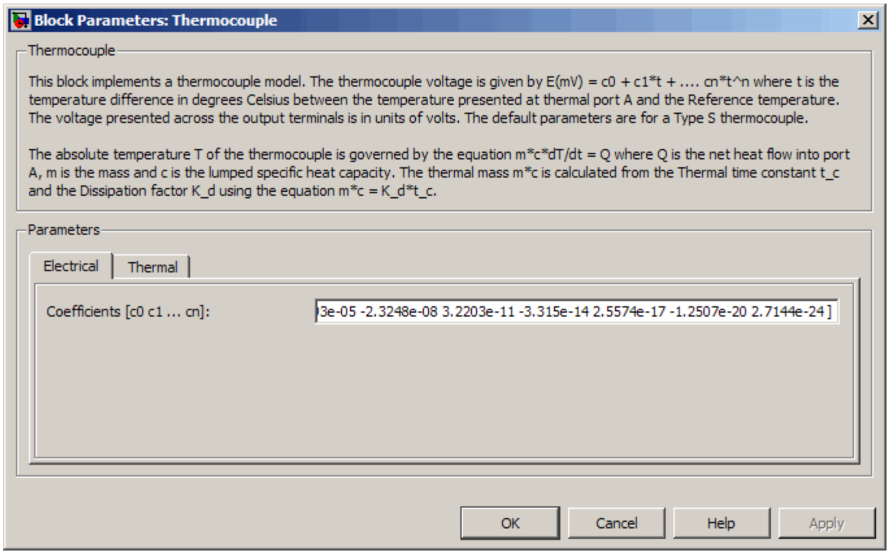
Basic Assumptions and Limitations

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.
- This block does not include the additional exponential term that Type K thermocouples use when parameterized for $t > 0$.

Dialog Box and Parameters

Electrical Tab

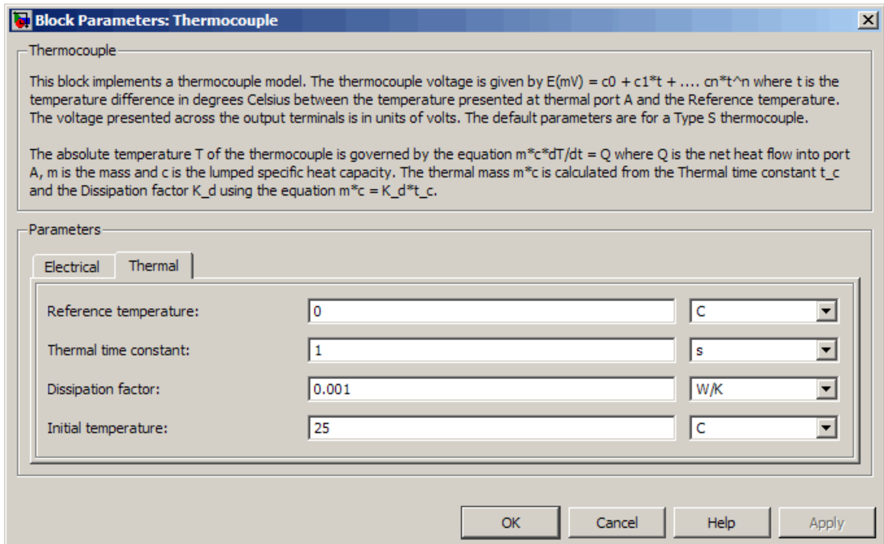


Coefficients [c0 c1 ... cn]

The vector of coefficients *c* in the equation that describes voltage as a function of temperature. The default value is `[ 0 0.0054031 1.2593e-05 -2.3248e-08 3.2203e-11 -3.315e-14 2.5574e-17 -1.2507e-20 2.7144e-24 ]`. This value specifies a Type S thermocouple, which is valid in the range -50 to 1064 degrees C.

Note You can download parameters for other 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.

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 http://srdata.nist.gov/its90/main

See Also

Thermal Resistor.

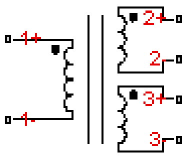
Purpose

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

$$V_1 = L_1 \frac{dI_1}{dt} + M_{12} \frac{dI_2}{dt} + M_{13} \frac{dI_3}{dt}$$

$$V_2 = M_{12} \frac{dI_1}{dt} + L_2 \frac{dI_2}{dt} + M_{23} \frac{dI_3}{dt}$$

$$V_3 = M_{13} \frac{dI_1}{dt} + M_{23} \frac{dI_2}{dt} + L_3 \frac{dI_3}{dt}$$

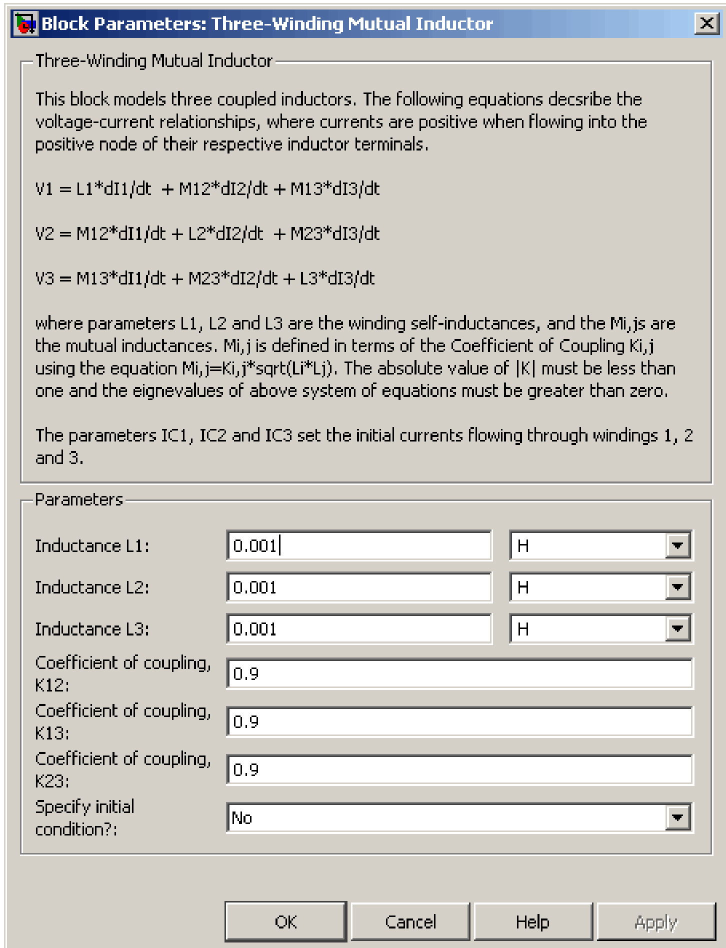
where:

- V_i is voltage across the ith winding.
- I_i is current through the ith winding.
- L_i is self inductance of the ith winding.
- M_{ij} is mutual inductance of the ith and jth 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.

Dialog Box and 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-

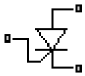
Negative electrical voltage of the third mutual inductor.

Purpose Model thyristor using NPN and PNP transistors

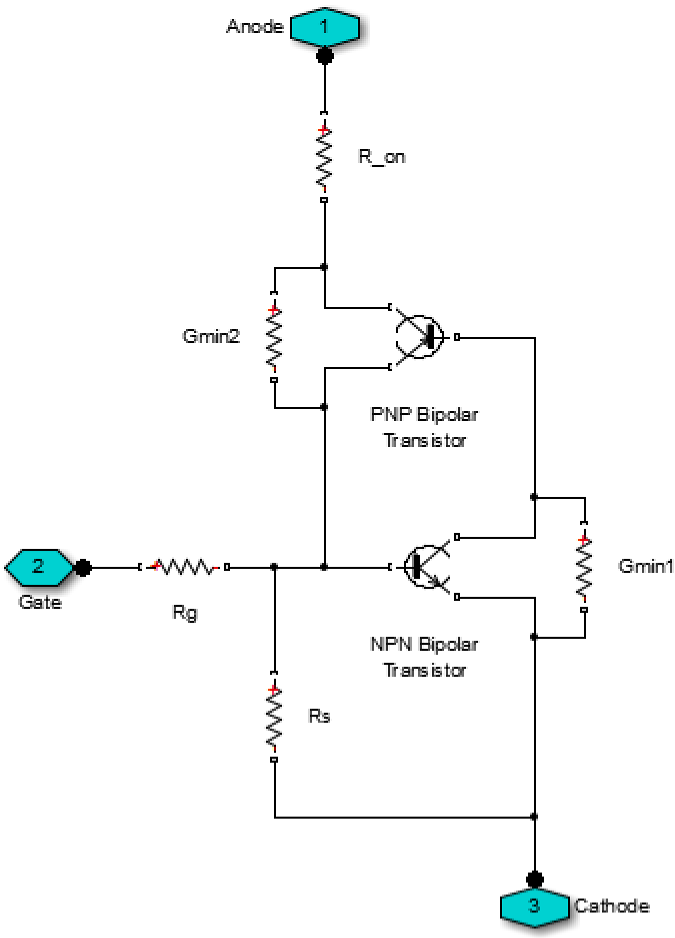
Library Semiconductor Devices

Thyristor

Description



The Thyristor block represents a system-level thyristor model that can be parameterized from a typical datasheet. The thyristor model is built from 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.

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 Rs 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 Rs 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 Gmin1 and Gmin2 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.

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

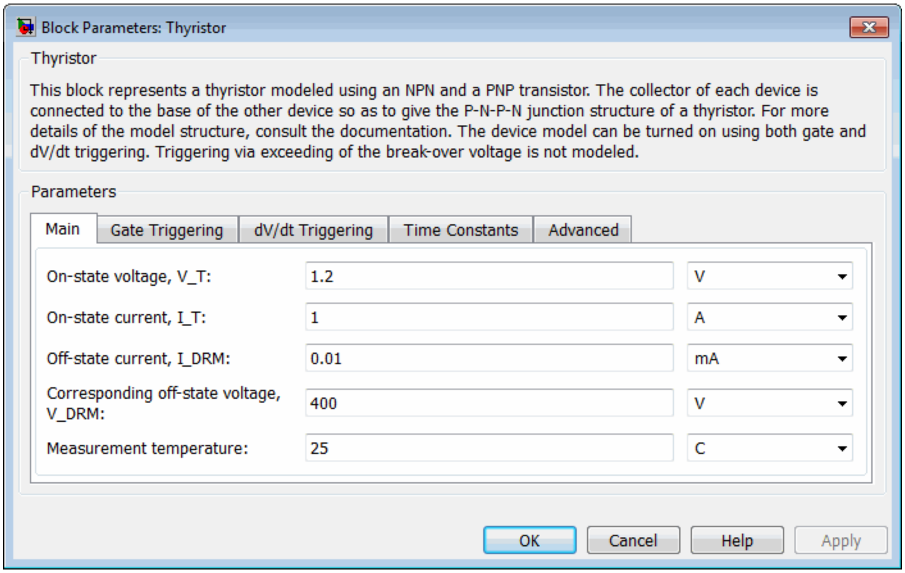
The Thyristor block has the following limitations:

- This block does not model temperature-dependent effects. SimElectronics 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.
- 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, Rs** 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.

Dialog Box and Parameters

Main Tab



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.

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.

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.

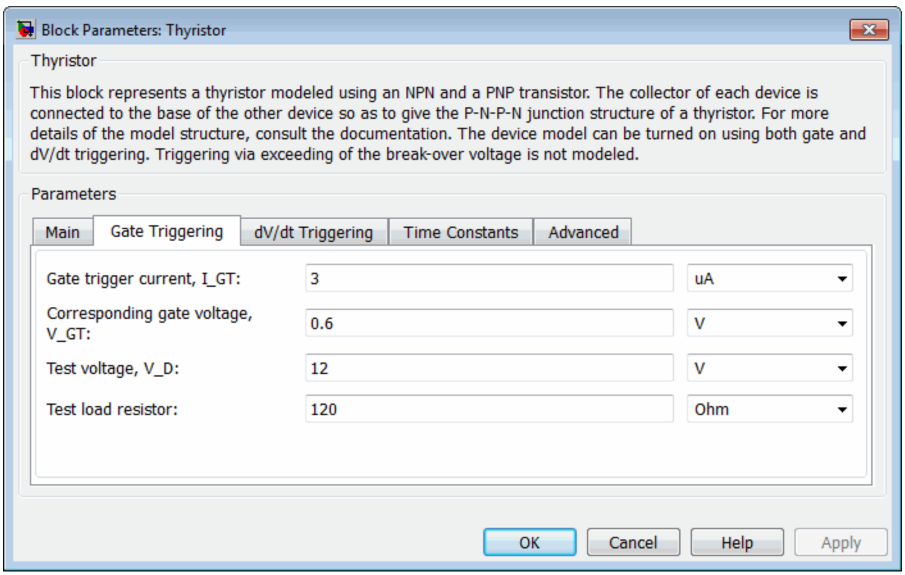
Corresponding off-state voltage, V_DRM

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.

Measurement temperature

The device simulation temperature. You must specify all block parameter values for this temperature. The default value is `25` °C.

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}.

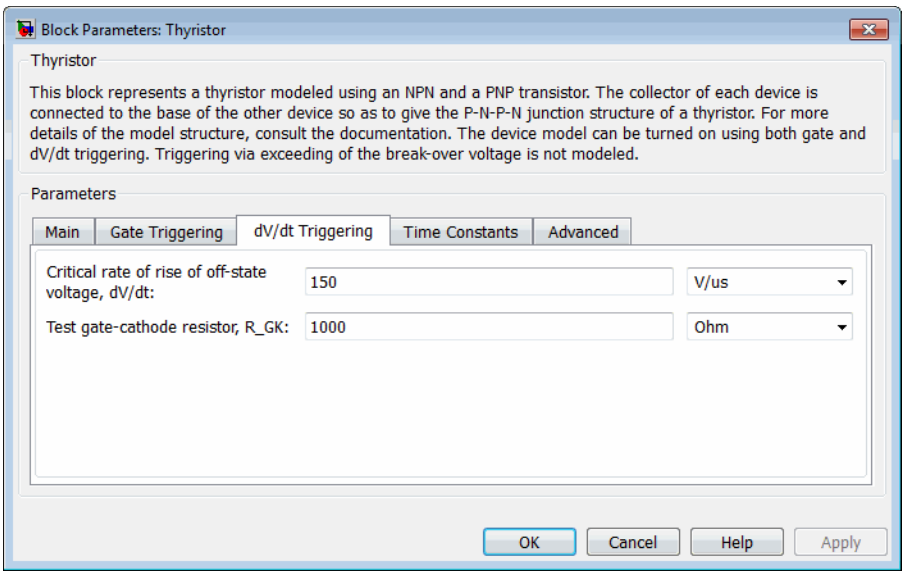
You must set the value of the **Internal shunt resistor, Rs** parameter on the **Advanced** tab to ensure that the gate triggers at i_{GT}, and not for currents less that i_{GT}. The default value is 3 µA.

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.

Test load resistor
Load resistor used when quoting values for v_{GT} and i_{GT}. The default value is 120 Ω.

dV/dt Triggering Tab



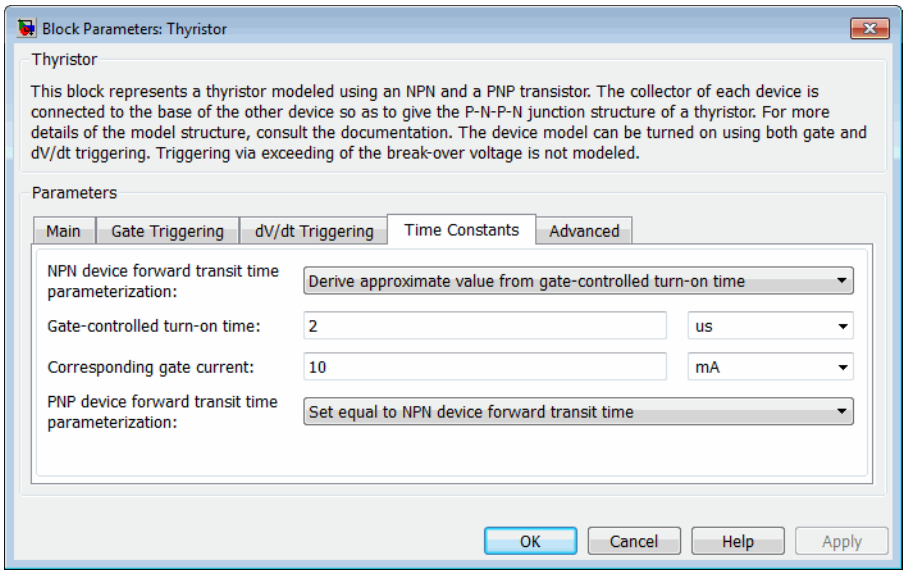
Critical rate of rise of off-state voltage, dV/dt

Rate at which the off-state anode-cathode voltage must be increased for the thyristor to turn on. The default value is 150 V/µs.

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 KΩ.

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.

Gate-controlled turn-on time

Time for the gate to turn from the off to the on state when a gate current is applied. This parameter is visible only when you select Derive approximate value from gate-controlled turn-on time for **NPN device forward transit time parameterization**. The default value is 2 μs.

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}. This parameter is visible only when you select Derive approximate value from gate-controlled turn-on time for **NPN device forward transit time parameterization**. The default value is 10 mA.

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]. This parameter is visible only when you select Specify directly for **NPN device forward transit time parameterization**. The default value is 0.3 μs.

PNP device forward transit time parameterization

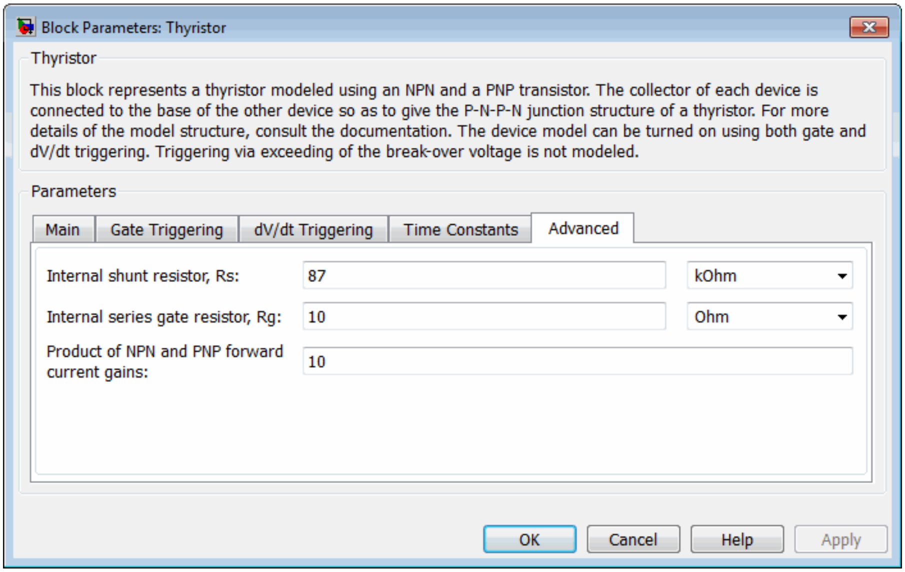
Select one of the following options:

- 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.

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]. This parameter is visible only when you select Specify directly for **PNP device forward transit time parameterization**. The default value is `0.3` μs.

Advanced Tab



Internal shunt resistor, Rs

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 that 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 Rs is small, and it can be set to `inf`. The default value is `87` kΩ.

Internal series gate resistor, Rg

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 Ω.

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.

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

Purpose

Model timer integrated circuit behaviorally

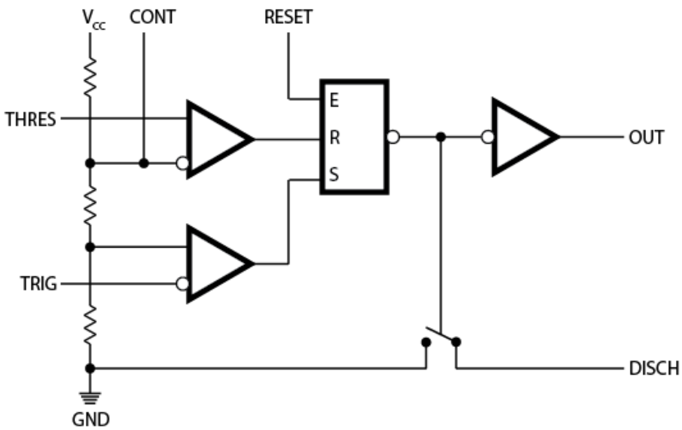
Library

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 SimElectronics 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.

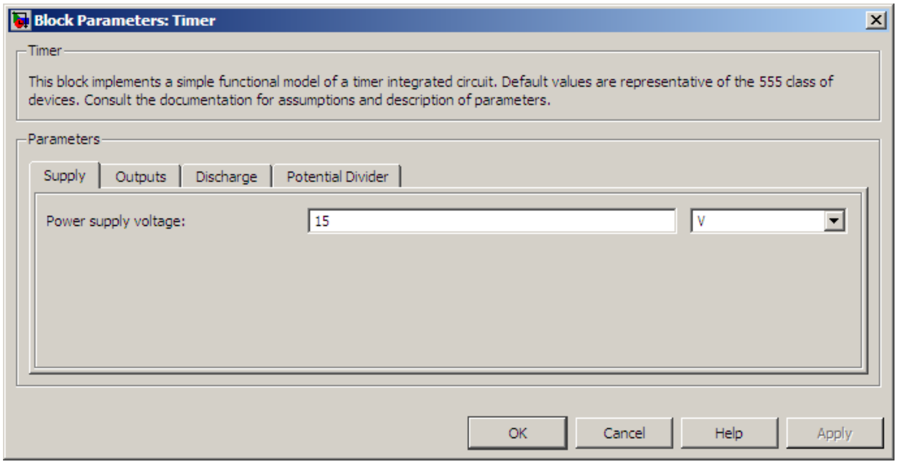
Basic Assumptions and Limitations

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.

Dialog Box and Parameters

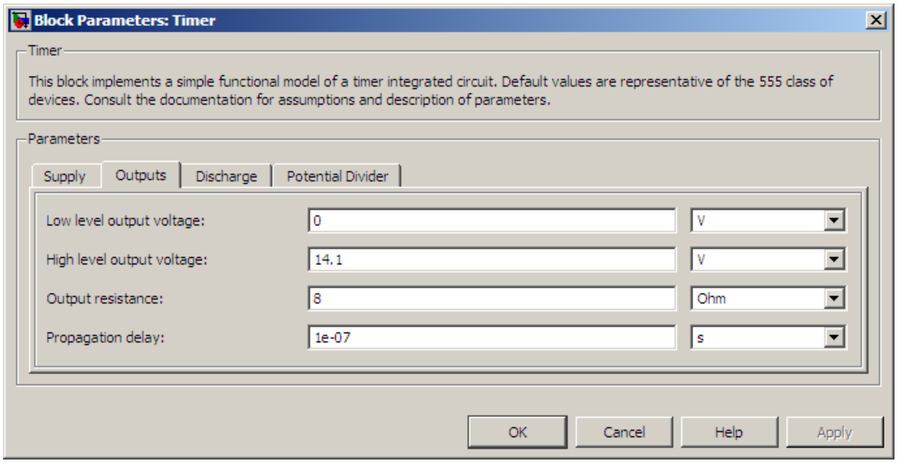
Supply Tab



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

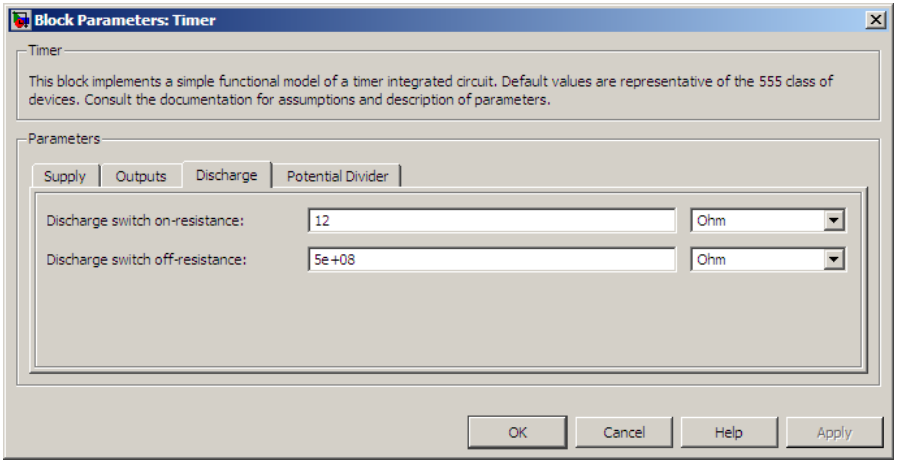
The output voltage V_{OH} when the timer output is high and no current is drawn. The default value is 14.1 V.

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 Ω.

Propagation delay
Set this value to the input-pulse or output-pulse rise time. The default value is `1e-07` s.

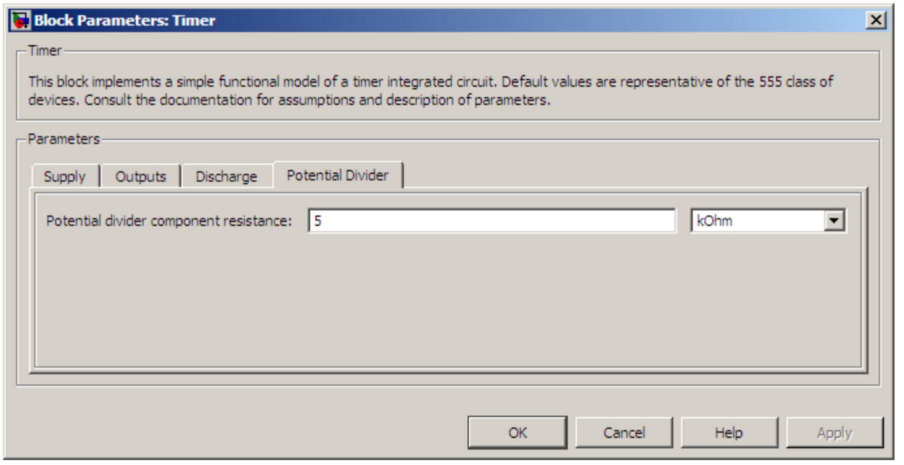
Discharge Tab



Discharge switch on-resistance
A representative value is the discharge pin saturation voltage divided by the corresponding current. The default value is `12` Ω.

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` Ω.

Potential Divider Tab



Potential divider component resistance

A typical value for a 555-type timer is 5 kΩ. 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 kΩ.

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

S-R Latch and Comparator.

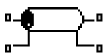
Purpose

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:

$$v_1(t) - i_1(t)\,Z_0 = v_2(t-\tau) + i_2(t-\tau)\,Z_0$$

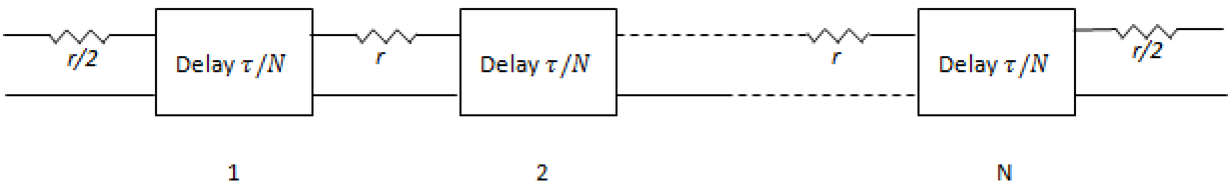
$$v_2(t) - i_2(t)\,Z_0 = v_1(t-\tau) + i_1(t-\tau)\,Z_0$$

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.
- τ is the transmission line delay.
- Z_0 is the line characteristic impedance.

Delay-Based and Lossy

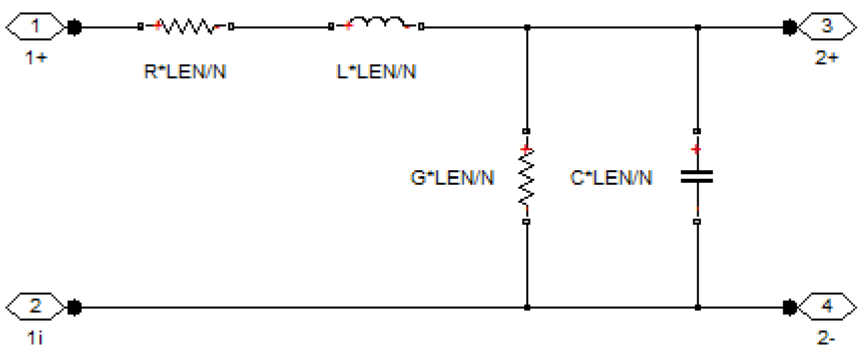
To introduce losses, the second option, `Delay-based and lossy`, connects *N* delay-based components, each defined by the above equations, in series via a set of resistors, as shown in the following illustration.



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.



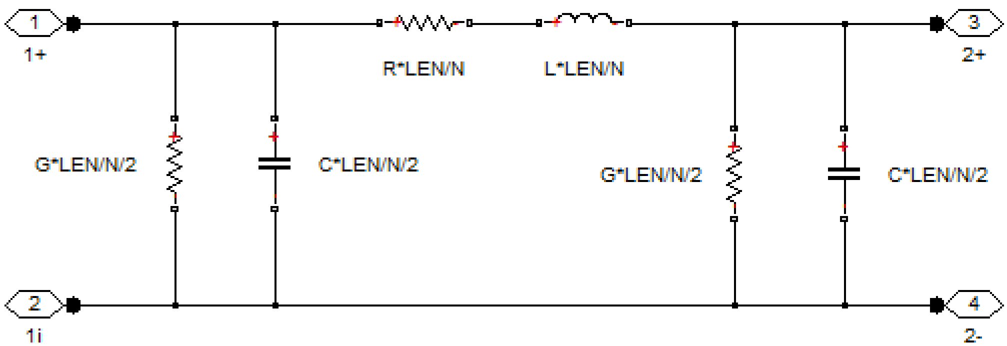
The lumped parameter parameterization uses *N* copies of the above segment model connected in series.

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].

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:

$$L = C \cdot Z_0^2$$

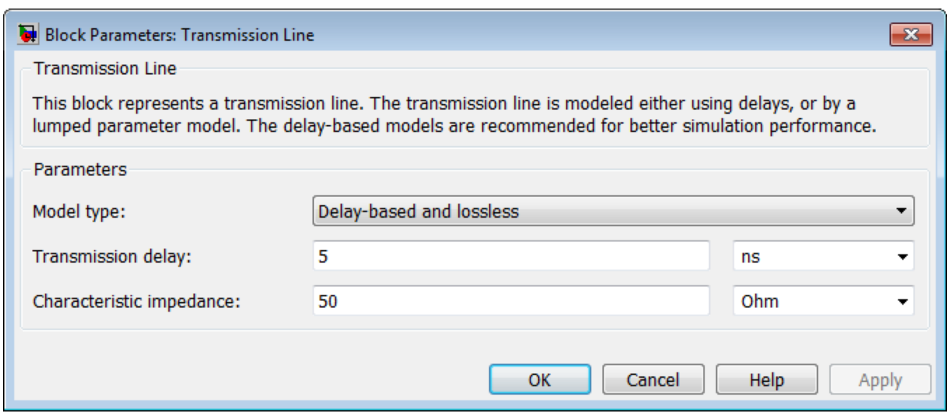
The block lets you specify either L or Z_0 when using the lumped parameter model.

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.

Dialog Box and 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 1-875. 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 1-876.
- `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 1-876.
- `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 1-877.

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` Ω.

Parameterization

This parameter appears for lumped parameter models only. Select the model parameterization method, as described in "Lumped Parameter Line Model Parameterization" on page 1-877:

- `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` µH/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` pF/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` Ω/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 `5e-6` S/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

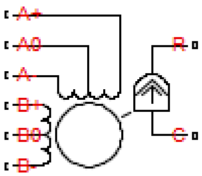
[1] Sussman-Fort, S.E. and J.C. Hantgan. "SPICE Implementation of Lossy Transmission Line and Schottky Diode Models." *IEEE Transactions on Microwave Theory and Techniques*. Vol. 36, No. 1, January, 1988.

Unipolar Stepper Motor

Purpose 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:

$$e_{A+} = -K_m \omega \sin(N_r \theta)$$

$$e_{A-} = K_m \omega \sin(N_r \theta)$$

$$e_{B+} = K_m \omega \sin(N_r \theta)$$

$$e_{B-} = -K_m \omega \sin(N_r \theta)$$

$$\frac{di_{A+}}{dt} = \left(v_{A+} - Ri_{A+} - e_{A+}\right) / L$$

$$\frac{di_{A-}}{dt} = \left(v_{A-} - Ri_{A-} + e_{A-}\right) / L$$

$$\frac{di_{B+}}{dt} = \left(v_{B+} - Ri_{B+} - e_{B+}\right) / L$$

$$\frac{di_{B-}}{dt} = \left(v_{B-} - Ri_{B-} + e_{B-}\right) / L$$

$$\frac{d\omega}{dt} = \left(-K_m\left(i_{A+} - i_{A-} - \left(e_{A+} - e_{A-}\right) / R_m\right)\sin\left(N_r\theta\right) + K_m\left(i_{B+} - i_{B-} - \left(e_{B+} - e_{B-}\right) / R_m\right)\cos\left(N_r\theta\right) - B\omega\right) / J$$

$$\frac{d\theta}{dt} = \omega$$

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.

- ω is the rotor speed.
- Θ is the rotor angle.

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-

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

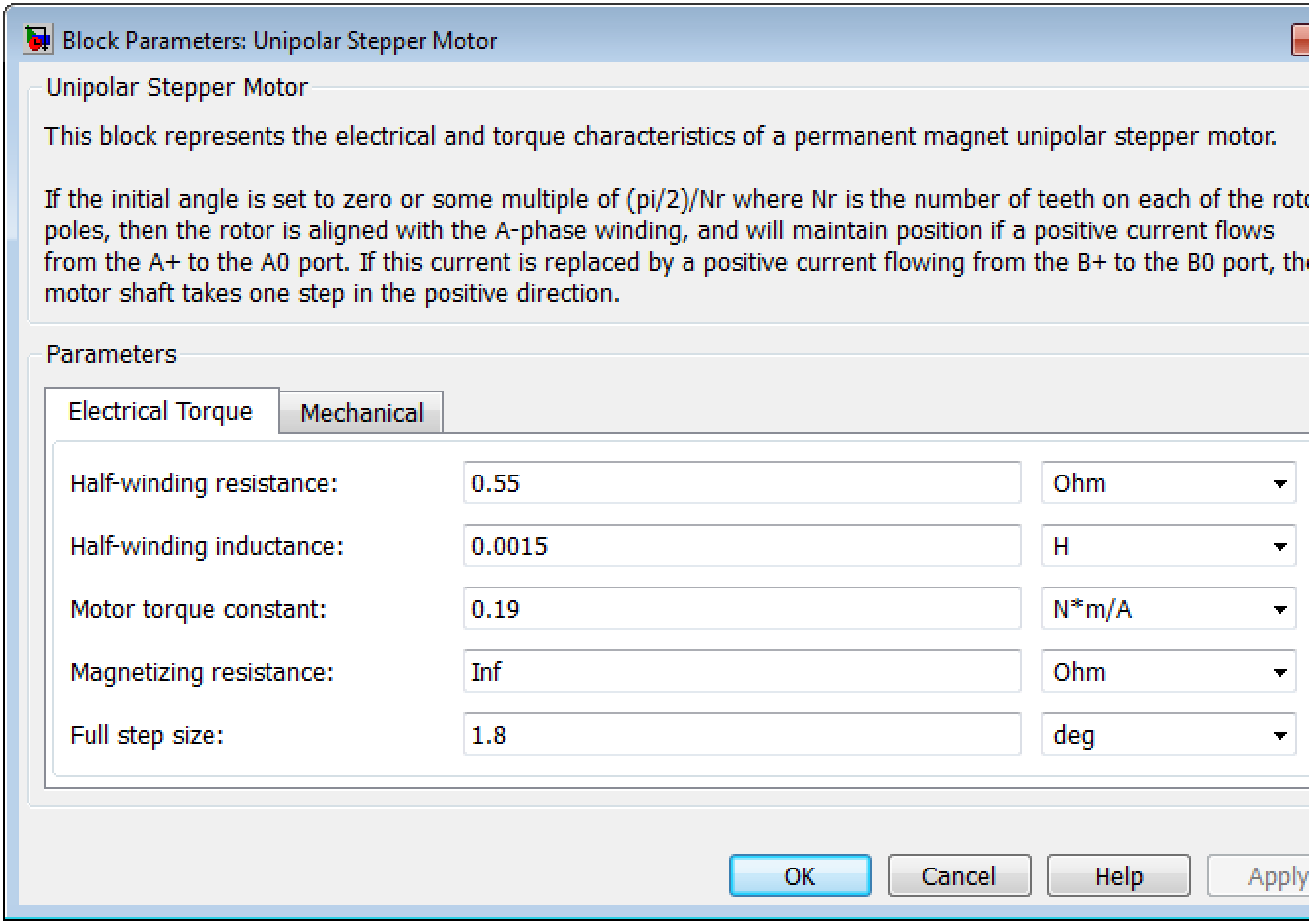
The model is based on the following assumptions:

- This model neglects magnetic saturation effects, detent torque, 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 π.
- All four half-windings are assumed to be identical, and therefore have the same resistance temperature coefficient, alpha, and the same thermal mass.

Dialog Box and Parameters

- "Electrical Torque Tab" on page 1-886
- "Mechanical Tab" on page 1-888
- "Temperature Dependence Tab" on page 1-889
- "Thermal Port Tab" on page 1-891

Electrical Torque Tab



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. The default value is `0.55` Ω.

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. The default value is `0.0015` H.

Motor torque constant

Motor torque constant K_m. The default value is `0.19` N*m/A.

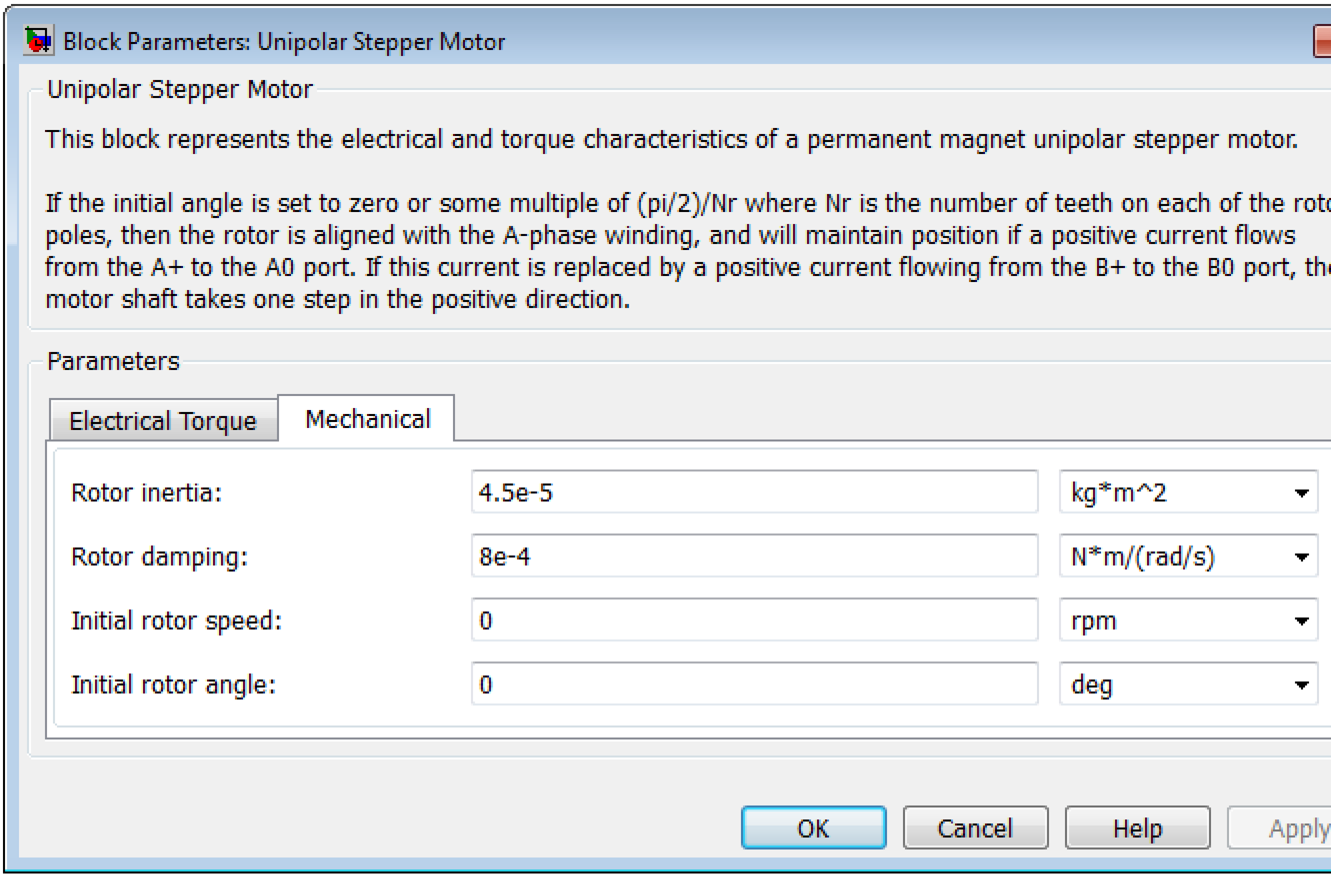
Magnetizing resistance

The total magnetizing resistance seen from each of the phase windings, for example across A+ and A0. The value must be greater than zero. The default value is `Inf`, which implies that there are no iron losses.

Full step size

Step size when changing the polarity of either the A or B phase current. The default value is `1.8°`.

Mechanical Tab



Rotor inertia
: Resistance of the rotor to change in motor motion. The default value is `4.5e-05` kg*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

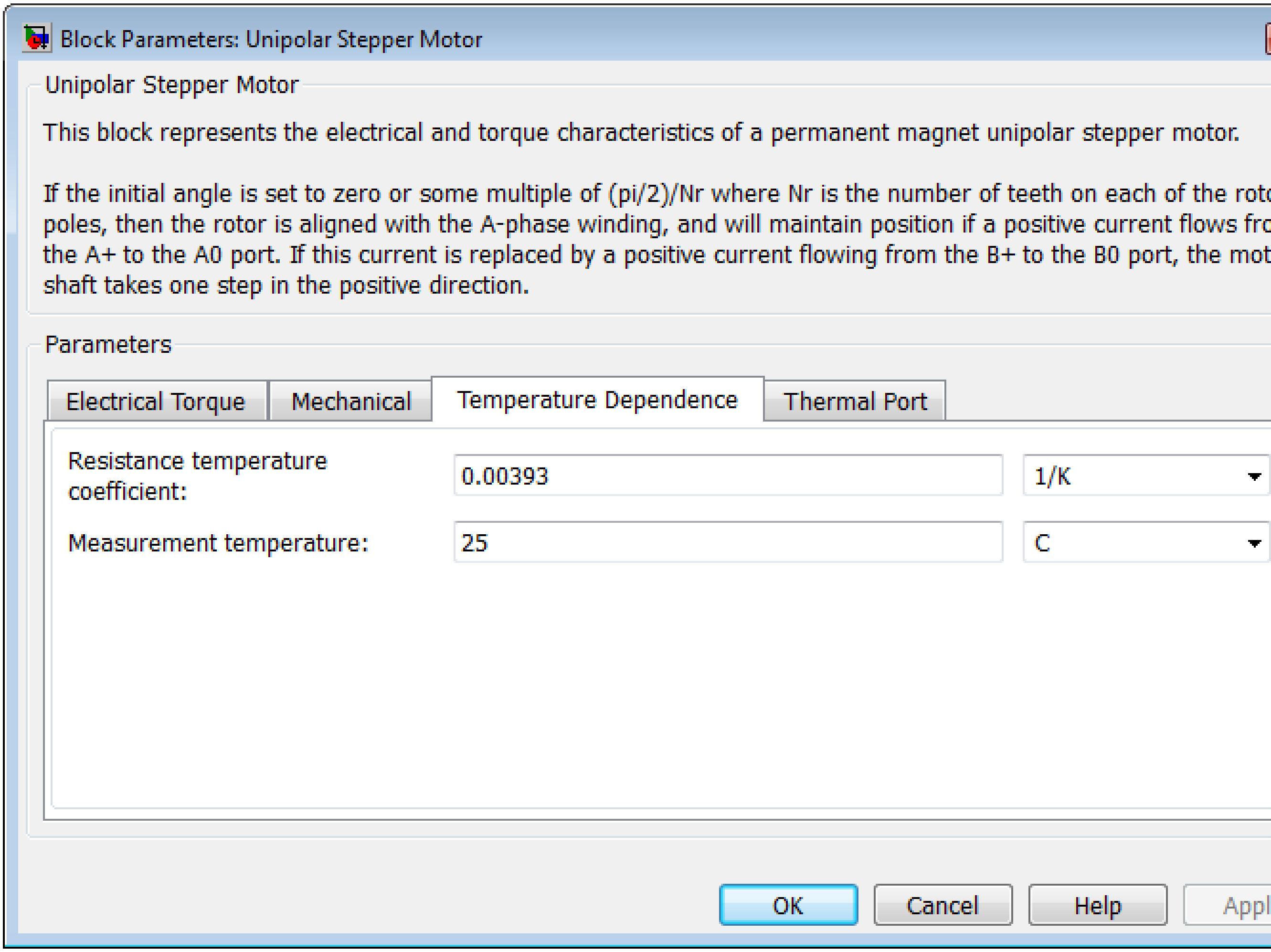
Speed of the rotor at the start of the simulation. The default value is `0` rpm.

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 1-884.



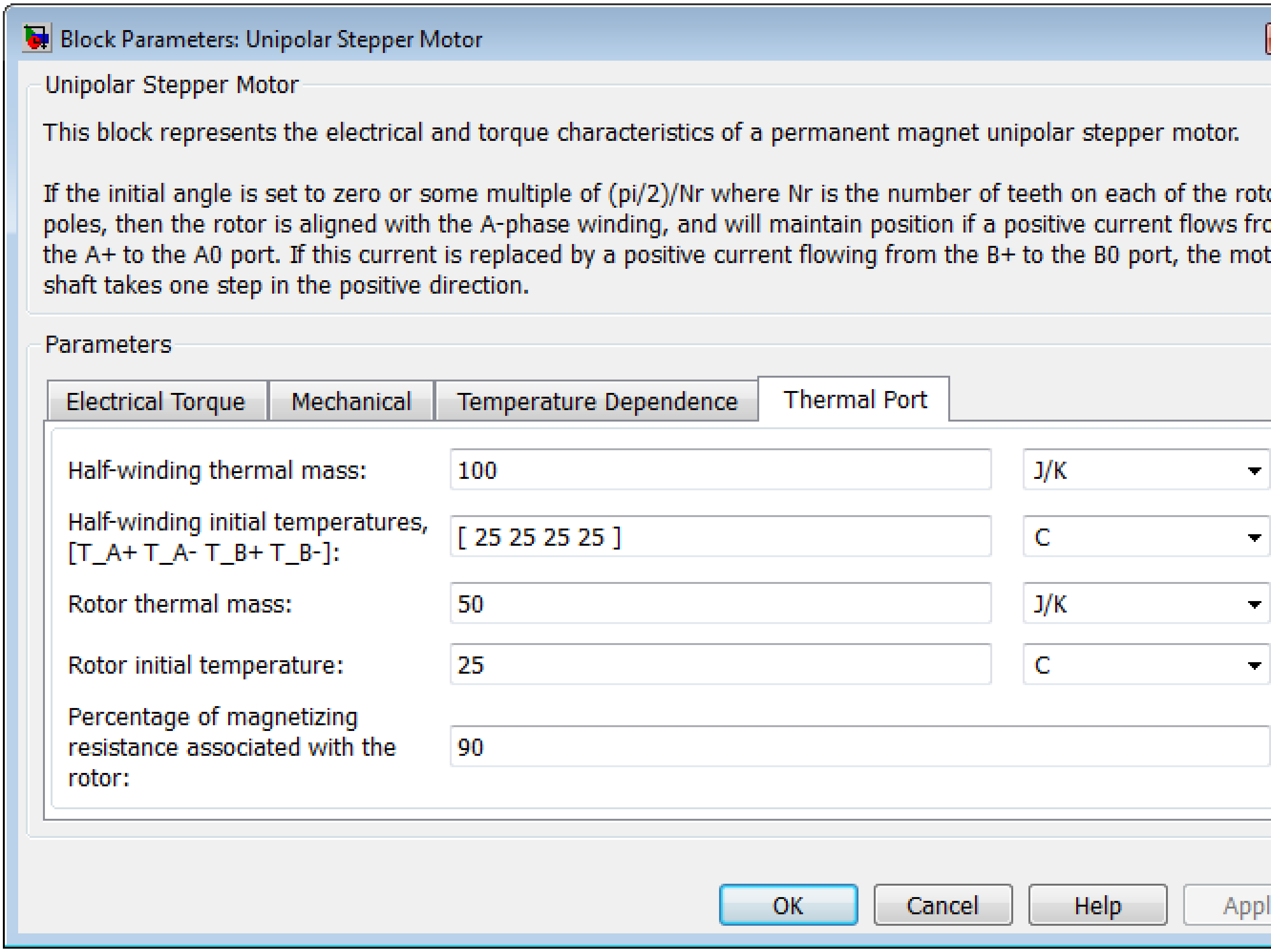
Resistance temperature coefficient

Parameter α 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 1-884.



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 1-884.

HA-

Thermal port for winding between A- and A0. For more information, see "Thermal Ports" on page 1-884.

HB+

Thermal port for winding between B+ and B0. For more information, see "Thermal Ports" on page 1-884.

HB-

Thermal port for winding between B- and B0. For more information, see "Thermal Ports" on page 1-884.

HR

Thermal port for rotor. For more information, see "Thermal Ports" on page 1-884.

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

Stepper Motor Driver

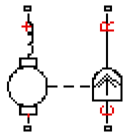
Purpose

Model electrical and torque characteristics of a universal (or series) motor

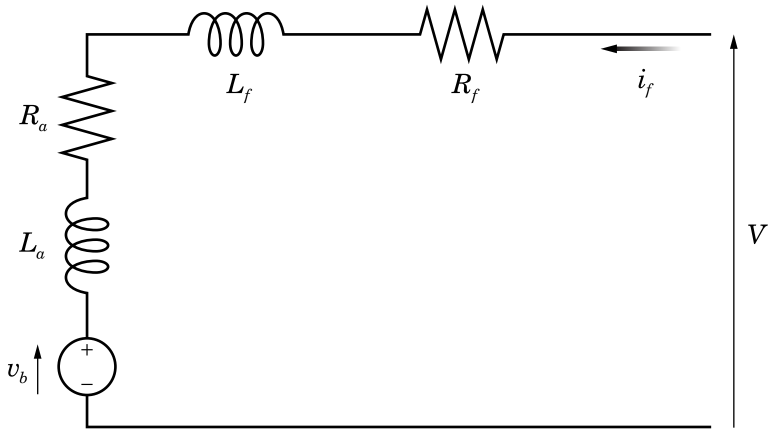
Library

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 ω 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 Shunt 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

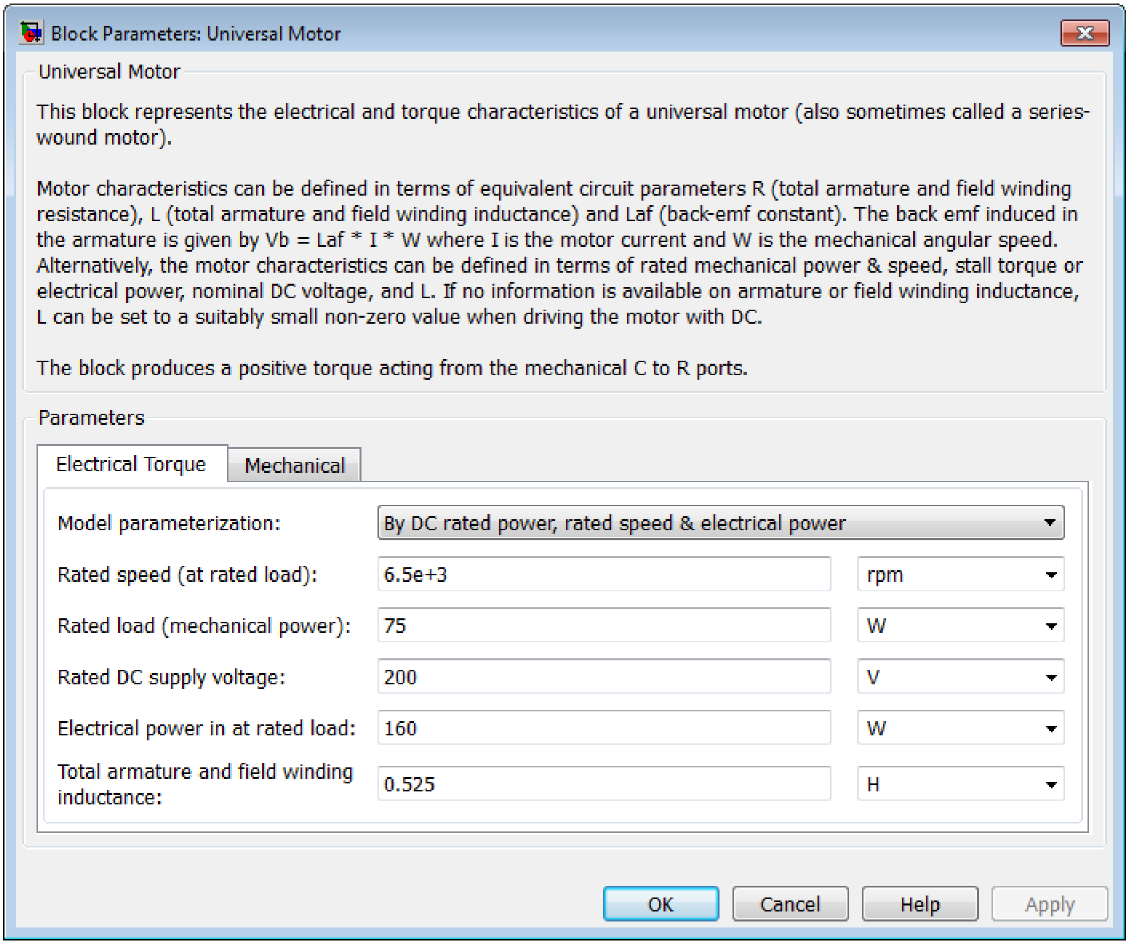
The block has two optional thermal ports, one per winding, hidden by default. To expose the thermal ports, right-click the block in your model, and then from the context menu select **Simscape block choices** > **Show thermal port**. This action displays the thermal ports on the block icon, and adds 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".

Dialog Box and Parameters

- "Electrical Torque Tab" on page 1-899
- "Mechanical Tab" on page 1-903
- "Temperature Dependence Tab" on page 1-904
- "Thermal Port Tab" on page 1-905

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` Ω.

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.

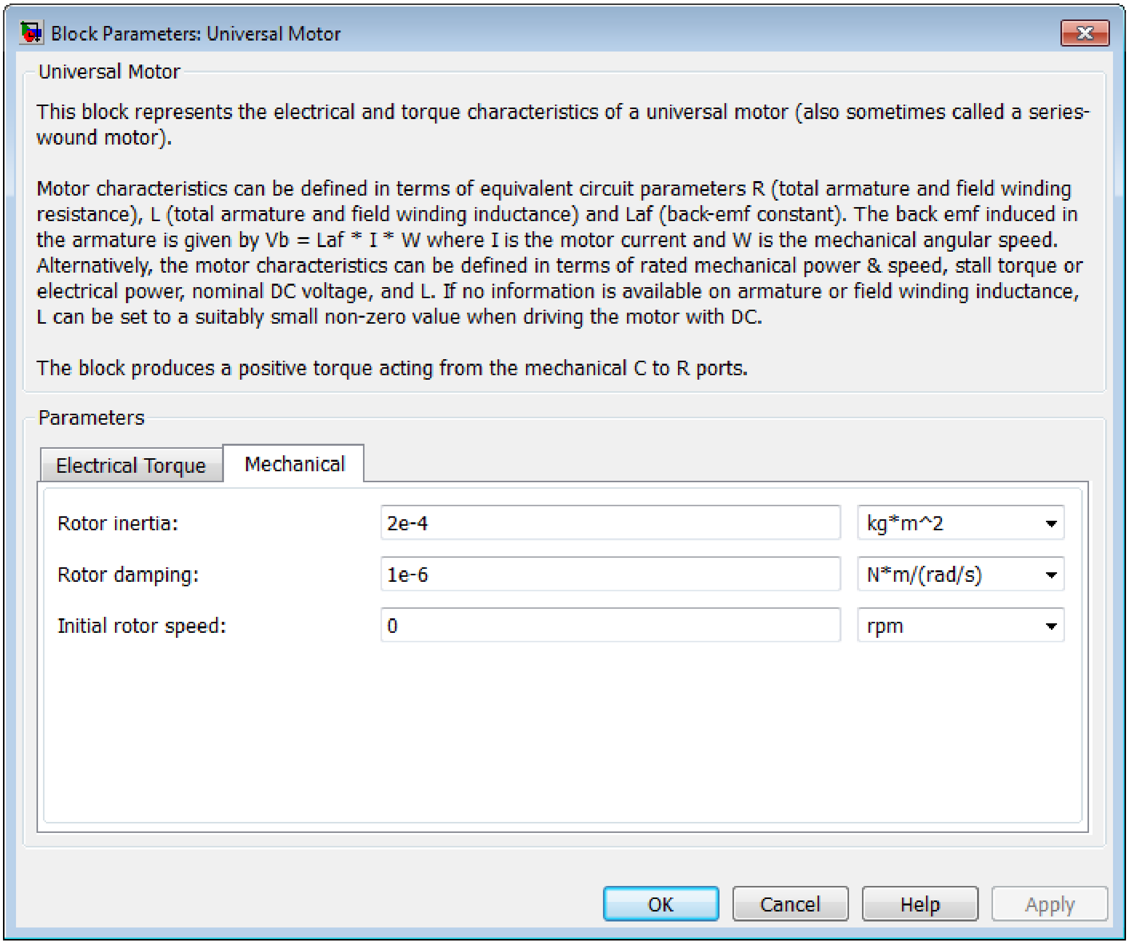
RMS current at rated load

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^2. The value can be zero.

Rotor damping

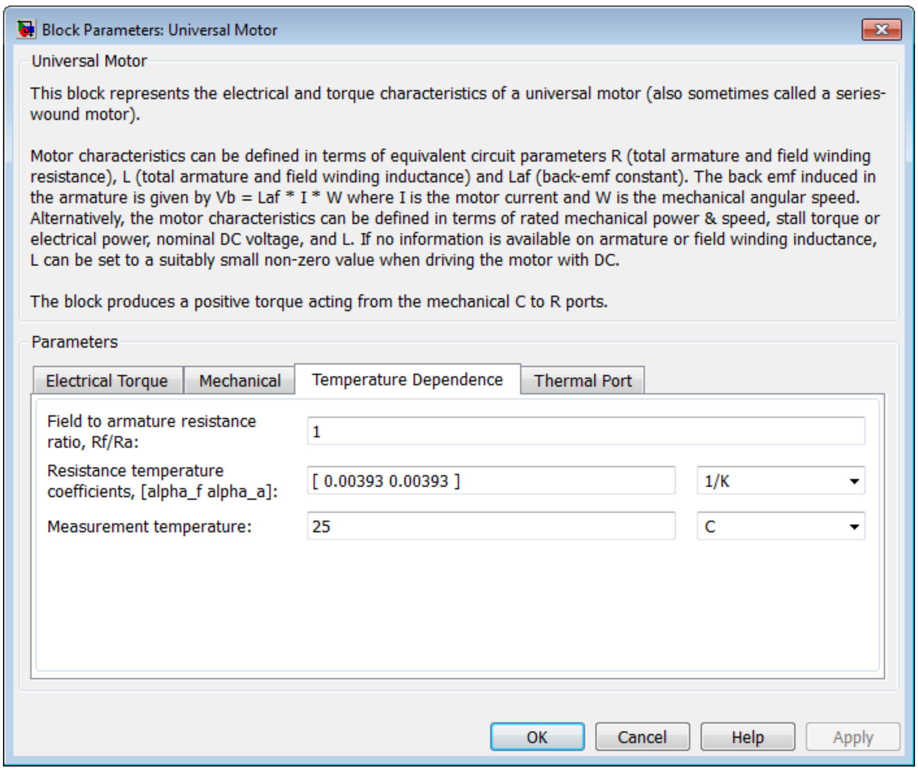
Rotor damping. The default value is `1e-06` N*m/(rad/s). The value can be zero.

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 1-897.



Field to armature resistance ratio, Rf/Ra

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]

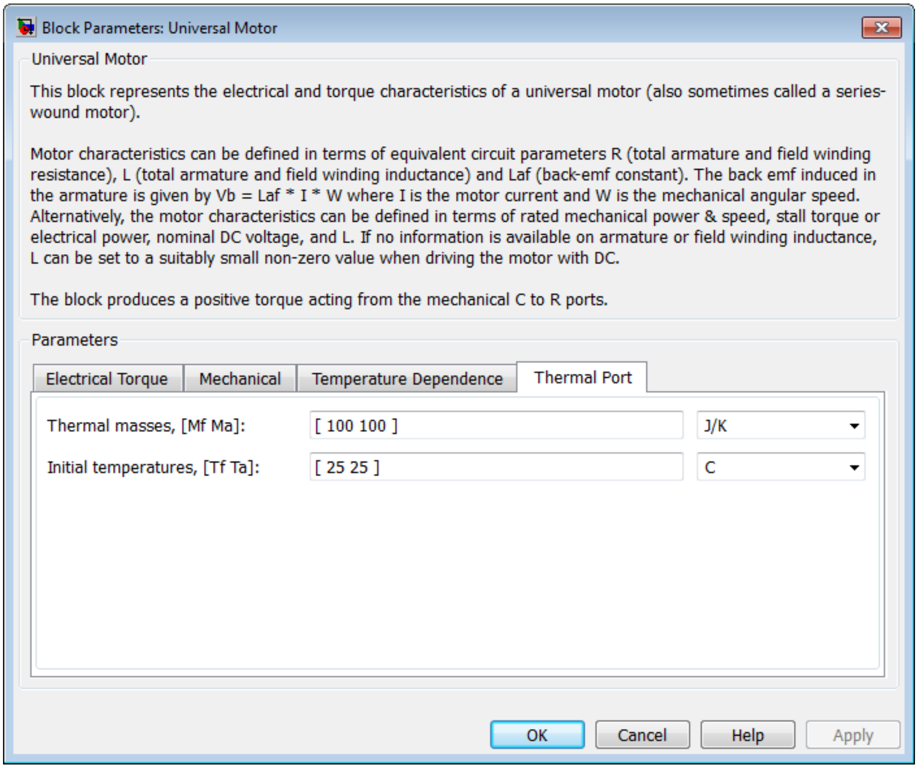
A 1 by 2 row vector defining the coefficient α in the equation relating resistance to temperature, as described in "Thermal Model for Actuator Blocks". The first element corresponds to the field winding, and the second to the armature. The default value is for copper, and is `[ 0.00393 0.00393 ]` 1/K.

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 1-897.



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 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 1-897.

`Ha`
: Armature winding thermal port. For more information, see "Thermal Ports" on page 1-897.

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, and Shunt Motor.

Variable Capacitor

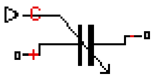
Purpose

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:

- $i = \frac{dC}{dt}v + C\frac{dv}{dt}$

  Use the preceding equation when the capacitance is defined as the ratio of the charge Q to the steady-state voltage:

  $$C(v) = \frac{Q(v)}{v}$$

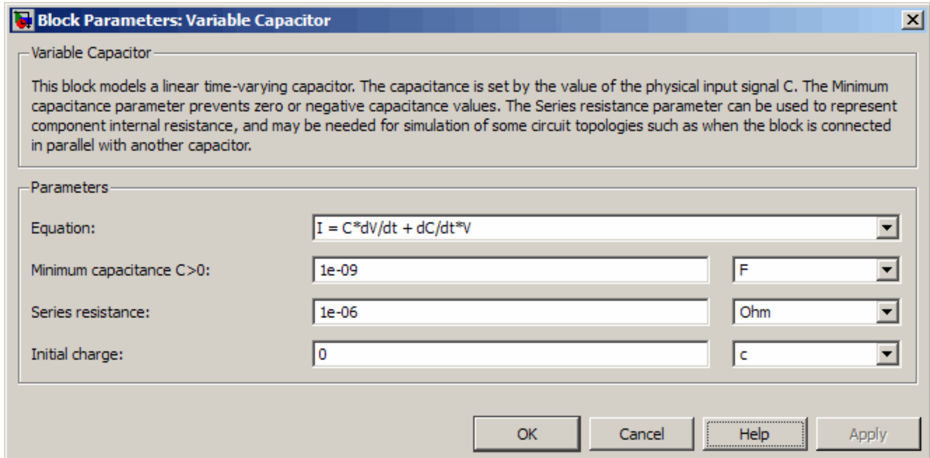
- $i = C\frac{dv}{dt}$

  Use the preceding equation when the capacitance is defined as the local gradient of the charge-voltage curve for a given voltage:

  $$C(v) = \frac{dQ(v)}{dv}$$

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.

Dialog Box and Parameters



Equation

Select one of the following options for block capacitance:

- `I = C*dV/dt + dC/dt*V` — This equation assumes the capacitance is defined as the ratio of the charge to the steady-state voltage. This option is the default.
- `I = C*dV/dt` — This equation assumes the capacitance is defined as the local gradient of the charge-voltage curve for a given 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` Ω.

Initial charge

The charge at the start of the simulation. This parameter is only visible when you select `I = C*dV/dt + dC/dt*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*dV/dt` for the **Equation** parameter. The default value is `0` V.

Ports

The block has the following ports:

`C`
Capacitance. C must be finite and greater than zero.

`+`
Positive electrical port.

`-`
Negative electrical port.

See Also

Variable Inductor, Simscape Variable Resistor

Purpose

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 Φ to the steady-state current:

  $$L(i) = \frac{\Phi(i)}{i}$$

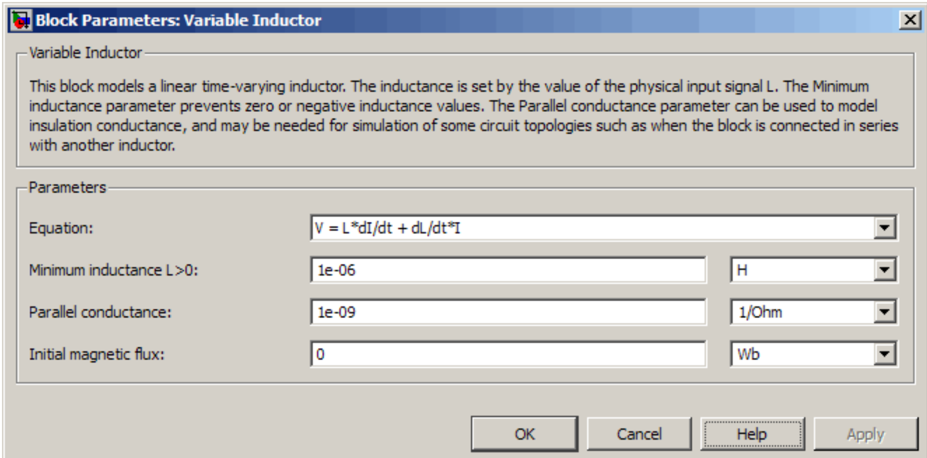
- $v = L \frac{di}{dt}$

  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.

Dialog Box and Parameters



Equation

Select one of the following options for block inductance:

- `V = L*dI/dt + dL/dt*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*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*dI/dt + dL/dt*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*dI/dt` for the **Equation** parameter. The default value is `0` A.

Ports

The block has the following ports:

`L`

Inductance. L must be finite and greater than zero.

`+`

Positive electrical port.

`-`

Negative electrical port.

See Also

Variable Capacitor, Simscape Variable Resistor

Voltage-Controlled Oscillator

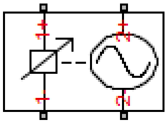
Purpose

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:

$$v_{\text{lim}} = \begin{cases} v_{\text{min}} & \text{for } v_{\text{in}} < v_{m\,\text{in}} \\ v_{\text{in}} & \text{for } v_{m\,\text{in}} \le v_{\text{in}} \le v_{\text{max}} \\ v_{\text{max}} & \text{for } v_{\text{in}} > v_{\text{max}} \end{cases}$$

$$\dot{\Phi} = 2\pi F\left(v_{\text{lim}}\right)$$

$$v_{out} = A\sin\left(2\pi f_{nom}t + \Phi\right) - i_{out}R_{out}$$

where:

- v_{in} is the voltage applied across the 1+ and 1– ports.
- v_{out} is the voltage across the 2+ and 2– ports.
- f_{nom} is the oscillator frequency when the input control voltage is v_{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:

$$F = f_{nom} + k\left(v_{lim} - v_{nom}\right)$$

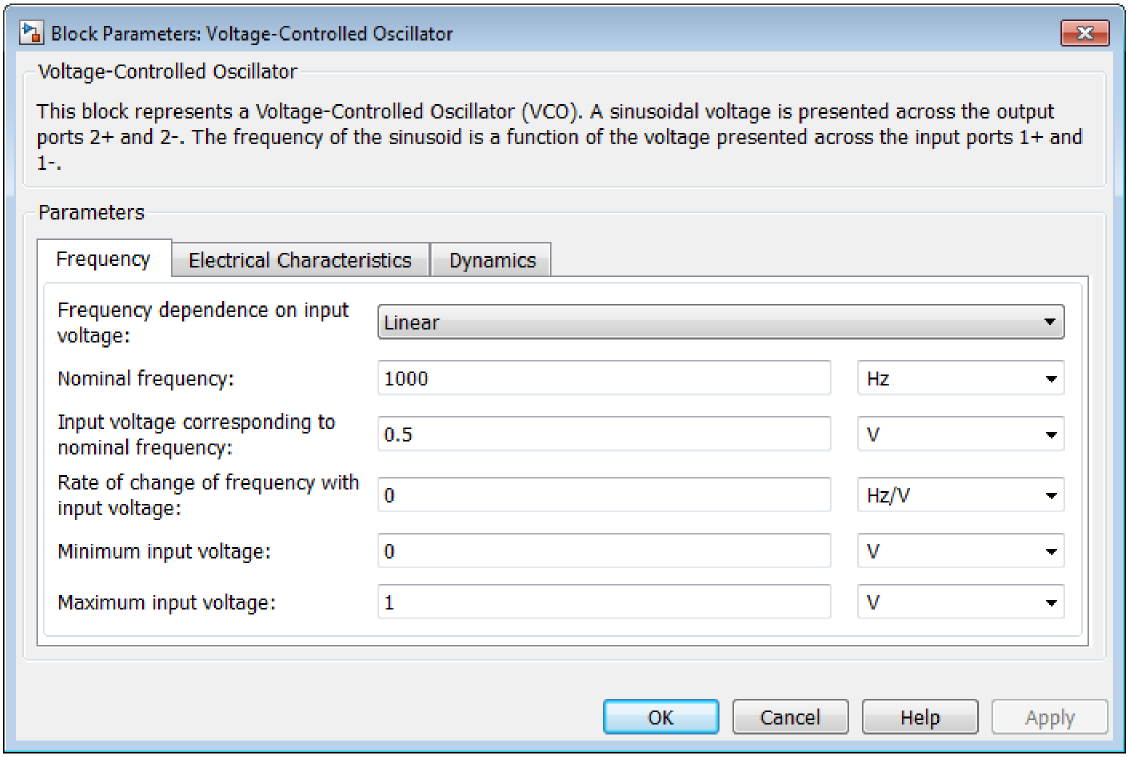
where k is the rate of change of frequency with input voltage.

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.

Dialog Box and Parameters

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 `0` 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.

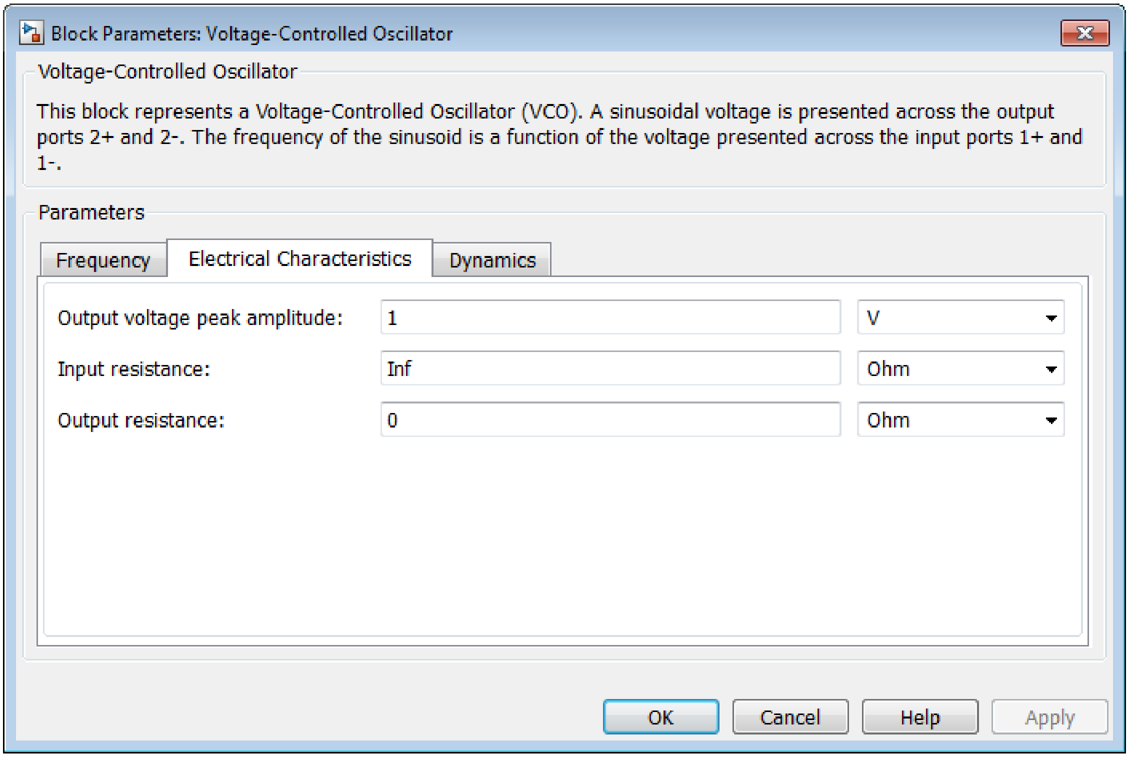
Maximum input voltage
The maximum input voltage that affects VCO frequency. This parameter is visible only if you select `Linear` for the **Frequency dependence on input voltage** parameter. The default value is `1` V.

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

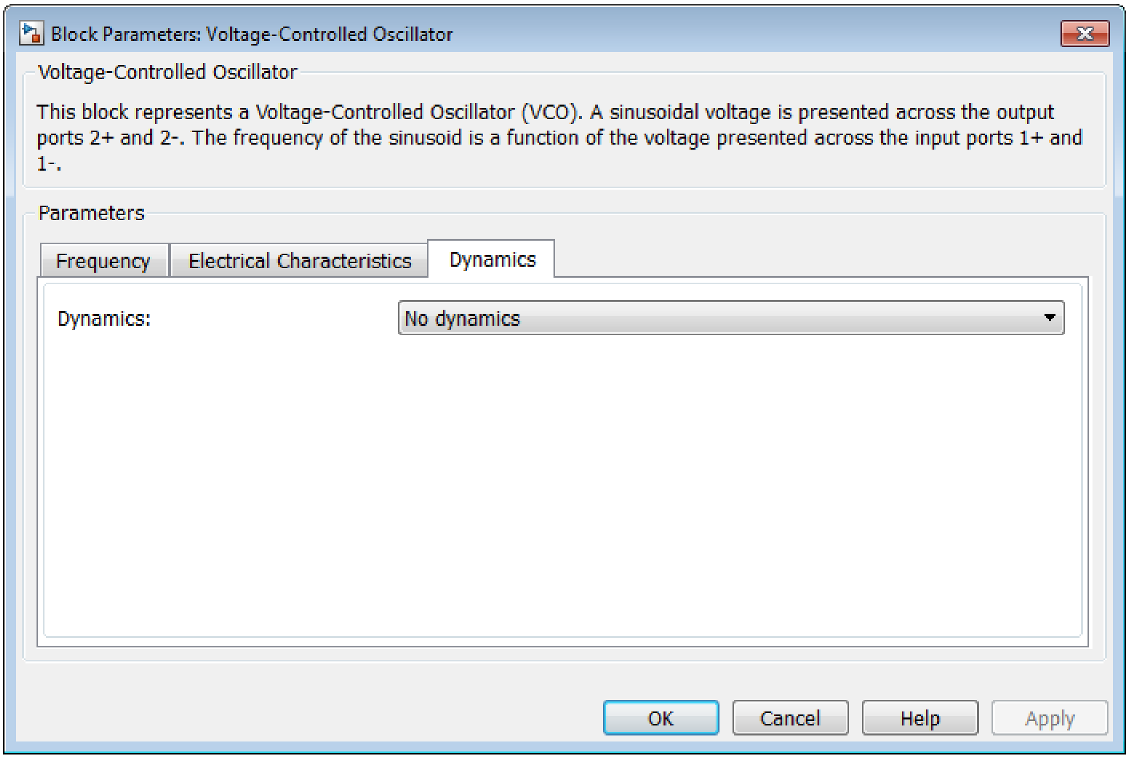


Output voltage peak amplitude
: The peak amplitude of the voltage across the 2+ and 2– terminals. The default value is 1 V.

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

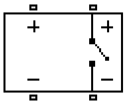
Purpose

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.

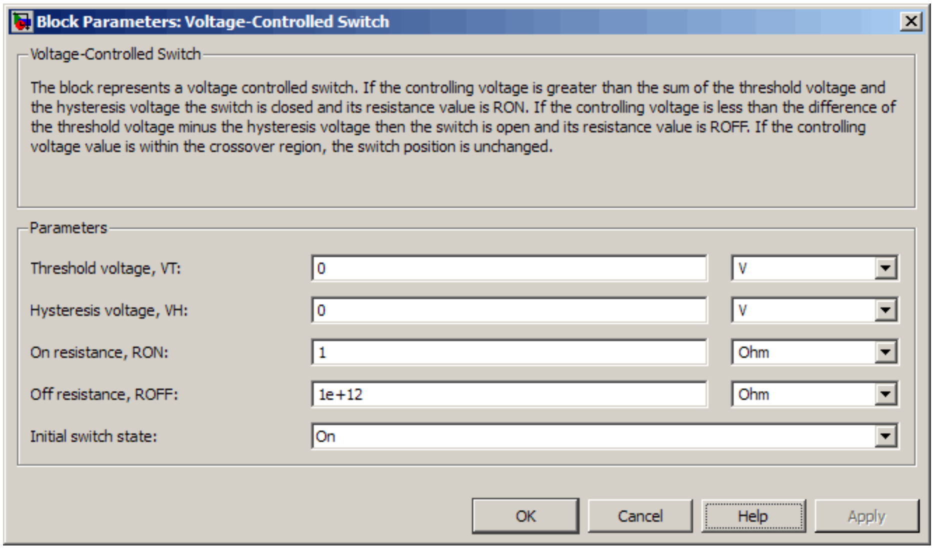
Basic Assumptions and Limitations

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.

Dialog Box and 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` Ω.

Off resistance, ROFF
The resistance of the switch when it is open. The default value is `1e+12` Ω.

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

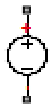
Purpose

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:

$$v = v_{DC} + v_{AC} \sin(2\pi f t + \phi) + v_N$$

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.
- φ 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);
  ```

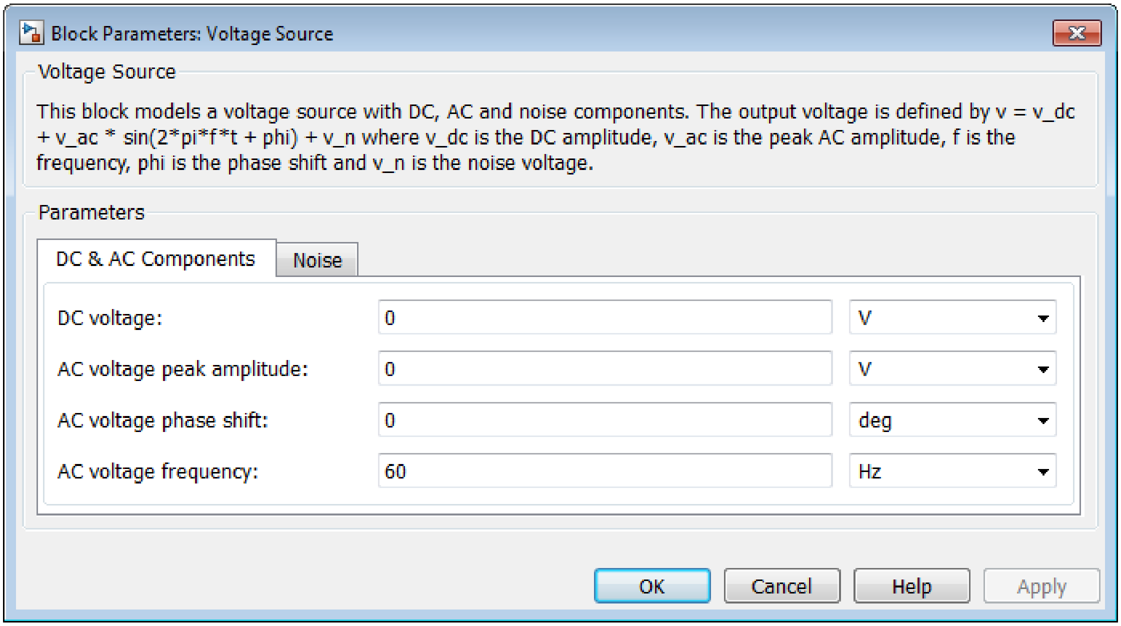
- `Repeatable` — The block uses a hidden parameter, called `auto_seed`, to always start the simulation with the same random number. The value of `auto_seed` is set whenever you copy the Resistor block from the block library to the model, or when you make a new copy of the Resistor block from an existing one in a model. 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.

Dialog Box and Parameters

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

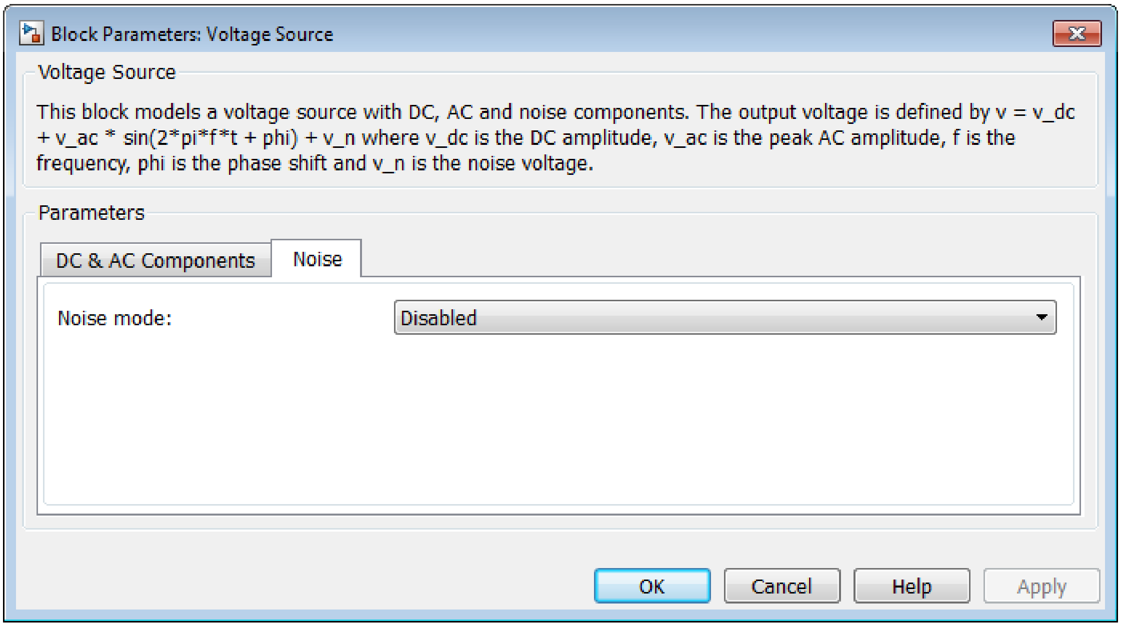
Amplitude of the AC component of the output voltage. The default value is `0` V. Enter a nonzero value to add an AC component to the voltage source.

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/Hz. To avoid this unit ambiguity, some datasheets quote noise voltage as a noise density with units of V/√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/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.

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.

Index

A

Accelerometer block 1-2

B

Band-Limited Op-Amp block 1-7

C

CMOS AND block 1-11
CMOS Buffer block 1-19
CMOS NAND block 1-27
CMOS NOR block 1-35
CMOS NOT block 1-43
CMOS OR block 1-51
CMOS XOR block 1-59
Comparator block 1-67
Controlled PWM Voltage block 1-72
Crystal block 1-79
Current Source block 1-88
Current-Controlled Switch block 1-85

D

DC Current Source block 1-93
DC Motor block 1-95
DC Voltage Source block 1-105
DC-DC Converter block 1-106
Diode block
    Diode 1-115
DPDT Switch block 1-142
DPST Switch block 1-146

E

Exponential Current Source block 1-150
Exponential Voltage Source block 1-153

F

FEM-Parameterized Linear Actuator block 1-156
FEM-Parameterized Rotary Actuator block 1-165
Finite-Gain Op-Amp block 1-174
Fully Differential Op-Amp block 1-177
Fuse block 1-186

G

Generic Battery block 1-188
Generic Linear Actuator block 1-192
Generic Rotary Actuator block 1-198
Gyro block 1-204

H

H-Bridge block 1-208

I

Incandescent Lamp block 1-220
Incremental Shaft Encoder block 1-227
Induction Motor block 1-230

L

Light-Emitting Diode block 1-246

M

Multiplier block 1-261

N

N-Channel IGBT block 1-267
N-Channel JFET block 1-285
N-Channel MOSFET block 1-302
Negative Supply Rail block 1-317
Nonlinear Inductor block 1-319
Nonlinear Transformer block 1-330
NPN Bipolar Transistor block 1-345

O

Operational Transconductance Amplifier block 1-361
Optocoupler block 1-371

P

P-Channel JFET block 1-383
P-Channel MOSFET block 1-400
Photodiode block 1-425
Piezo Linear Motor block 1-442
Piezo Rotary Motor block 1-451
Piezo Stack block 1-461
PNP Bipolar Transistor block 1-470
Polynomial Current-Controlled Current Source block 1-415
Polynomial Current-Controlled Voltage Source block 1-420
Polynomial Voltage-Controlled Current Source block 1-534
Polynomial Voltage-Controlled Voltage Source block 1-539
Positive Supply Rail block 1-486
Potentiometer block 1-488
Power Sensor block 1-492
Pressure Transducer block 1-495
Proximity Sensor block 1-499
PS Sensor block 1-503
PTC Thermistor block 1-507
Pulse Current Source block 1-513
Pulse Voltage Source block 1-517
Push-Pull Output block 1-521
PWL Current Source block 1-544
PWL Voltage Source block 1-548

R

Relay block 1-551
Resistor block 1-554
Resolver block 1-559

S

S-R Latch block 1-567
Servomotor block 1-575
SFFM Current Source block 1-584
SFFM Voltage Source block 1-587
Shunt Motor block 1-590
Sinusoidal Current Source block 1-601
Sinusoidal Voltage Source block 1-604
Solar Cell block 1-607
Solenoid block 1-621
SPDT Switch block 1-628
SPICE Diode block 1-632
SPICE Environment Parameters block 1-646
SPICE NJFET block 1-648
SPICE NMOS block 1-664
SPICE NPN block 1-704
SPICE PJFET block 1-730
SPICE PMOS block 1-746
SPICE PNP block 1-785
SPICE Resistor block 1-812
SPST Switch block 1-818
Stepper Motor block 1-822
Stepper Motor Driver block 1-834
Strain Gauge block 1-837

T

Thermal Resistor block 1-839
Thermistor block 1-843
Thermocouple block 1-848
Three-Winding Mutual Inductor block 1-853
Thyristor block 1-857
Timer block 1-869
Transmission Line block 1-875
Two-Input Polynomial Current-Controlled Current Source block 1-418
Two-Input Polynomial Current-Controlled Voltage Source block 1-423
Two-Input Polynomial Voltage-Controlled Current Source block 1-537

Two-Input Polynomial Voltage-Controlled
Voltage Source block 1-542

U

Unipolar Stepper Motor block 1-882
Universal Motor block 1-895

V

Variable Capacitor block 1-908
Variable Inductor block 1-911
Voltage Source block 1-924
Voltage-Controlled Oscillator block 1-914
Voltage-Controlled Switch block 1-921