

Simscape™ Release Notes



MATLAB® & SIMULINK®

How to Contact MathWorks



Latest news: www.mathworks.com
Sales and services: www.mathworks.com/sales_and_services
User community: www.mathworks.com/matlabcentral
Technical support: www.mathworks.com/support/contact_us



Phone: 508-647-7000



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

Simscape™ Release Notes

© COPYRIGHT 2007–2016 by The MathWorks, Inc.

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

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

Trademarks

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

Patents

MathWorks products are protected by one or more U.S. patents. Please see www.mathworks.com/patents for more information.

Simscape Language	1-2
Lookup Table Improvements: Define 3D tables, interpolate using Akima splines	1-2
Event Variables and edge Operator: Capture values at events	1-3
Integral Operator, integ: Specify time integration of an expression	1-4
Improved attributes for member visibility and access	1-5
Simscape language improved ease of use	1-6
Additional MATLAB operators allowed in Simscape language	1-8
Improved strictness and accuracy	1-9
Change to Thermal Liquid domain definition	1-12
New base classes for Thermal Liquid components	1-12
Renamed Thermal Liquid component source files	1-12
 Foundation Library and Simulation	 1-14
Run-Time Parameters: Speed up simulation tasks and modify component parameter values without regenerating C code	1-14
Simulation Speed Improvements: Up to 5x speed improvement for simulations using generated code	1-15
Stream Logging Data to Disk: Increase capacity for Simscape simulation results	1-15
Simscape Component Block: Efficiently create custom components by selecting Simscape language file directly from block	1-16
Simscape Results Explorer Unit Selection: Select y-axis units directly in plot pane	1-16
PS Lookup Table (3D) Block: Graphically define implicit equations that require lookup tables with three independent variables	1-17

Validation of Signal Units: Ensure consistent units specification on Simulink signals connected to Simscape physical networks	1-18
Thermal Resistor block in the Electrical Elements library . .	1-20
Mass Flow Rate & Thermal Flux Sensor (TL) block renamed	1-20
Thermal Liquid volumetric flow rate source and sensor blocks	1-20
Custom environment pressure in Thermal Liquid converter blocks	1-21
Numerical improvements in Thermal Liquid blocks for greater model fidelity	1-21
Sign reversal of thermal expansion coefficient	1-22
Functionality being removed or changed	1-23

R2015aSP1

Bug Fixes

R2015b

Simscape Language	3-2
Units handling refinements in Simscape language	3-2
Foundation Library and Simulation	3-3
Two-Phase Fluid domain and block library	3-3
Variable priority option None for initialization	3-3
Periodic Operators library	3-4
Speed and efficiency improvements for simulation of switched linear systems	3-4
Domain-specific line style propagation through block icons . .	3-4
Rotational Inerter and Translational Inerter blocks	3-5
Real-Time Performance Advisor checks for physical models . .	3-5
Real-Time Simulation Documentation Enhancements	3-5
Variable Viewer link to block diagram renamed	3-5

Naming schema change within the data logging variable structure	3-6
---	-----

R2015a

Foundation Library and Simulation	4-2
Variable Viewer link to block diagram	4-2
Improved solver efficiency for model initialization	4-2
Sparkline plots for logged data	4-2
Data logging for models created using ssc_new	4-3
Pressure setting and name change for Thermal Liquid reservoir blocks	4-4
New examples	4-4

R2014b

Simscape Language	5-2
Refrigeration example modeling two-phase fluid flow	5-2
Foundation Library and Simulation	5-3
Domain-specific line styles for representing physical connections	5-3
Improved interface for connecting Simscape blocks	5-3
Option to configure logging of Simscape simulation results per block	5-3
Flat view option and improved diagnostics for Variable Viewer	5-4
Simscape Results Explorer	5-4
Simscape logging into Simulink single output	5-5
Simscape logging integration with Simulation Data Inspector	5-6
Infinite resistance and port terminator blocks	5-6

Performance improvements for linear systems in Normal and Accelerator modes	5-7
---	-----

R2014a

Simscape Language	6-2
ssc_build library build process that no longer requires C compiler	6-2
priority attribute for setting relative priority of variable target values during initialization	6-2
Script for replacing through and across statements with branches and equations	6-3
Foundation Library and Simulation	6-5
Variables tab for specifying target value and priority for new initialization process	6-5
Variable Viewer for analyzing results of new initialization process	6-8
Statistics Viewer that displays variable source and number of eliminated variables	6-8
Fundamental Reluctance block	6-9
Hydro-mechanical converter blocks with fluid compressibility option	6-9
Handling of pressure or temperature below absolute zero during simulation	6-9
Input filtering options for 1-D/3-D connections	6-10
Software-in-the-loop simulation for physical models	6-10
Change in default settings for ssc_new	6-10
Functionality being removed or changed	6-10

R2013b

Simscape Language	7-2
--------------------------------	------------

branches section for defining the relationship between component Through variables and nodes	7-2
import statement enabling simplified access to other component classes	7-4
connect statement support for vector and matrix physical signals	7-4
Foundation Library and Simulation	7-5
Thermal Liquid domain and block library	7-5
Simscape model statistics viewer	7-5
Removal of laminar-turbulent zero-crossings in hydraulic blocks	7-5
New examples	7-6
Functionality being removed or changed	7-6

R2013a

Simscape Language	8-2
Variable-size domain parameters	8-2
Vector and matrix physical signals	8-2
Foundation Library and Simulation	8-3
Random Number and Uniform Random Number blocks	8-3
Perfect Insulator block for thermal domain	8-3
Initialization and diagnostic improvements	8-3
Model Advisor checks for outdated blocks and for physical unit consistency	8-4
Unit definition of Hz now consistent with SI	8-5
New format for saving simulation data log objects	8-6
New examples	8-6
Functionality being removed or changed	8-7

Simscape Language	9-2
Connection of components within a Simscape file	9-2
floor, ceil, fix, and round functions	9-3
Foundation Library and Simulation	9-5
Speed and efficiency improvements for simulation of switched linear systems	9-5
Zero-crossing statistics for Simscape logging	9-5
Counter and Repeating Sequence blocks that facilitate discrete sampling	9-5
Open-circuit terminator blocks for electrical, hydraulic, and mechanical domains in Foundation library	9-6
Viewable and customizable source files for additional Foundation library blocks	9-6
New examples	9-7

Modeling Delays in Simscape Language	10-2
New Blocks for Modeling Delays	10-2
1D and 2D Interpolation Available in Simscape Language .	10-2
Input Filtering Usability Enhancements	10-2
Zero Damping Allowed for Resistive Elements	10-4
Changes to Simscape Demos	10-4

R2011b

Simscape Language Enables User-Defined Diagnostics During Simulation	11-2
New Block for Modeling Discrete Delays	11-3
Specialized Simulator for Linear Systems	11-3
Rebuilding of Custom Block Libraries Now Required	11-3

R2011a

Additional Hydraulic Sources	12-2
Improved Simulation Diagnostics	12-2
Improved Scalability	12-2
Improved Algorithms for Algebraic Loop Detection and Zero- Crossing Robustness	12-3
Change in Evaluating Unit Expressions	12-3

R2010b

Simulation Data Logging Enhancements	13-2
Zero-Crossing Handling Enhancements	13-2
New Implementation of Relational Functions Without Creating Simulink Zero-Crossing Conditions	13-2
Improved Zero-Crossing Detection Diagnostics	13-3

C++ Code Generation Support	13-3
Sparse Solver Enhancement	13-3
Component Descriptor Is No Longer Inherited from the Base Class	13-3
Documentation Enhancements	13-4
Changes to Simscape Demos	13-4

R2010a

Magnetic Blocks Added to Foundation Library	14-2
Simulation Data Logging Now Available	14-2
Simscape Diagnostics Improvements	14-2
Sparse Jacobian Support	14-2
Ability to Generate Simscape Language Equations from Symbolic Expressions	14-3
Placing Simscape Blocks in Nonvirtual Subsystems	14-3
Trimming and Linearization Documentation Enhancements	14-3
Changes to Simscape Demos	14-3

R2009b

Pneumatic Blocks Added to Foundation Library	15-2
---	-------------

New and Enhanced Switches	15-3
Intermediate Terms in Simscape Language Equations	15-3
Local Solver Support in Physical Networks	15-3
Simulink Manifest Tool Support	15-4
SimState Support	15-4
Model Reference Accelerator Mode Support	15-4
Physical Port Rotation for Simscape Blocks	15-4
Changes to Simscape Demos	15-5

R2009a

Simscape Language Source Protection	16-2
Expanded MATLAB Support of Simscape Files	16-2
Viewable and Customizable Source Files for Foundation Library Blocks	16-2
Simscape Language Syntax Changes	16-3
Increased Efficiency of Simscape Language Equations Processing	16-4
New Physical Signal Blocks to Facilitate Rounding	16-4
Model Reference Accelerator Mode Support	16-4
Changes to Simscape Demos	16-4

R2008b

Simscape Language	17-2
Explicit Solvers	17-2
New Ways to Model Variable Chambers	17-3
Model Reference Support	17-3
More Solver Performance and Robustness Enhancements .	17-3

R2008a

Trimming Now Available for Simscape Models	18-2
Thermal Unit Conversions Now Supported	18-2
Enhancement to Specifying Units	18-2
New Blocks	18-3
Enhancements to Simulation Algorithms	18-3
“What’s This?” Context-Sensitive Help Available for Simulink Configuration Parameters Dialog	18-3
New Simscape Demo	18-4

R2007b

Code Generation Now Available for Simscape Models	19-2
--	------

New Thermal Block Libraries	19-2
Additional Physical Signal Blocks	19-2
Improved Simulation Performance	19-2
New Simscape Demos	19-3

R2007a

Product Introduction	20-2
Block Libraries Moved from SimHydraulics to Simscape ..	20-2

R2016a

Version: 4.0

New Features

Bug Fixes

Compatibility Considerations

Simscape Language

Lookup Table Improvements: Define 3D tables, interpolate using Akima splines

The `tablelookup` function has been extended to support three-dimensional table lookup. That is, the function can now compute an output value by interpolating the input value against a set of data points in a three-dimensional table.

The interpolation and extrapolation options have been improved:

- There are two interpolation options, `linear` and `smooth`. The `smooth` option replaces the previously available `cubic` and `spline` options. It uses a modified Akima algorithm, as described in “Smooth Interpolation Algorithm”, to produce a continuous curve or surface with continuous first-order derivatives. It also tries to preserve the slope and avoid undulations where the data suggests a flat region. The default is `linear`, which provides the best performance.
- There are two extrapolation options, `linear` and `nearest`. The names of the options are the same as in previous releases, but the underlying algorithms have changed. The `linear` method now ensures a smooth connection between the interpolation region and the extrapolation region. The `nearest` method produces a curve or surface with continuous value at the boundary between the interpolation region and the extrapolation region that does not go above the highest point in the data or below the lowest point in the data.

Compatibility Considerations

The `smooth` interpolation option replaces the `cubic` and `spline` options, available in previous releases. If your existing models use `cubic` or `spline` interpolation, these options will be automatically mapped to `smooth`. The new method is superior in the way it preserves the shape of the data and produces a continuous curve or surface with continuous first-order derivatives. However, this enhancement can produce a change in results for your legacy models.

The new interpolation algorithm tries to preserve the slope and avoid undulations where the data suggests a flat region. It interprets the data as a flat region whenever there are three or more consecutive colinear points in the table data. It then connects those three or more points with a straight line. The colinear points do not have to be horizontal. To

ensure that the region between two data points is flat, insert an additional data point between those two points.

The `linear` extrapolation method now ensures a smooth connection between the interpolation region and the extrapolation region. The `nearest` method extends as a constant that is now equal to the last interpolated value, rather than the last point in the output table data. These enhancements can also produce a change in results for your legacy models.

Event Variables and edge Operator: Capture values at events

Event is a conceptual notation that denotes a change of state in a system. Event modeling lets you perform discrete changes on continuous variables. The two most common applications of event modeling are:

- Trigger-and-hold mechanism, such as a triggered delay. For example, a component has two inputs: `u` and `x` (triggered signal), and one output `y`. When and only when the triggered signal `x` changes value from false to true, output `y` is reset to the value of `u` at current time. `y` remains unchanged all other times.
- Enabled component, acting on a principle similar to Simulink® enabled subsystem. That is, the component has a control signal as input. If the control signal has a positive value, then the component holds certain states to the most recent value, or resets them. When the control signal is negative, the states change according to component equations.

The following new constructs in Simscape™ language let you perform event modeling: event variables, `edge` operator, `when` clause, and `events` section.

Event variables are piecewise constant, that is, they change values only at event instants, and keep their values constant between events. You can declare internal component variables of type integer or real as event variables by setting the `Event=true` attribute.

The `edge` operator takes a scalar boolean expression as input. It returns true, and triggers an event, when and only when the input argument changes value from false to true. The return type of `edge` is *event type*. Event type is a special category of boolean type, which returns true only instantaneously, and returns false otherwise.

The `when` clause serves to update the values of the event variables. The syntax is
`when EventPredicate`

```
var1 = expr1;  
var2 = expr2;  
...  
end
```

EventPredicate is an expression that defines when an event occurs. It must be an expression of event type. The variables in the body of the `when` clause must be declared as event variables. When the event predicate returns true, all the variables in the body of the `when` clause simultaneously get updated to the new values. The `when` clause can have optional `elsewhen` branches. However, an `else` branch is not allowed.

The new `events` section in a component file manages the event updates. The `events` section can contain only `when` clauses. The order of `when` clauses does not matter.

The following example implements a triggered delay component:

```
component Triggered  
  inputs  
    u = 0; % input signal  
    triggered = 0; % control signal  
  end  
  variables(Event=true)  
    x = 0;  
  end  
  outputs  
    y = 0;  
  end  
  equations  
    y == x;  
  end  
  events  
    when edge(triggered>0)  
      x = u;  
    end  
  end  
end
```

For more information, see “Discrete Event Modeling”.

Integral Operator, `integ`: Specify time integration of an expression

The new `integ` operator lets you perform time integration of an expression in the `equations` section of a Simscape file without declaring and initializing extra variables.

The full syntax is:

```
integ(expr, t_L)
```

where:

- `expr` is a Simscape language expression.
- `t_L` is the lower integration limit, specified as a delay time relative to current time. This operand is optional.

The upper integration limit is the current simulation time. If you omit the lower limit, the integration starts from simulation start time.

Improved attributes for member visibility and access

The new attribute `ExternalAccess` controls visibility of members in the user interface, that is, outside Simscape language. In other words, this attribute controls whether parameters, variables, and other members are visible and modifiable in block dialog boxes, simulation logs, variable viewer, and so on. The attribute is applicable to all member classes and has the following values:

- `modify` — The member is modifiable in the block dialogs and visible in the logs and viewer.
- `observe` — The member is visible in the logs and viewer, but not modifiable.
- `none` — The member is visible nowhere outside the language.

The default value of the `ExternalAccess` attribute for a member depends on the value of the `Access` attribute for that member.

Access	Default ExternalAccess
<code>public</code>	<code>modify</code>
<code>protected</code>	<code>observe</code>
<code>private</code>	<code>observe</code>

Members in the base class with `Access=private` are forced to have `ExternalAccess=none`, to avoid potential collision of names between the base class and the derived class.

When `Access` is explicitly set to `private` or `protected`, it does not make sense to explicitly set `ExternalAccess=modify`. In this situation, the compiler issues a warning and remaps `ExternalAccess` to `observe`.

Compatibility Considerations

The `ExternalAccess` attribute replaces the member attribute `Hidden` (which you could previously set for all member classes, such as parameters, variables, and so on, including member components of a composite component). The model attribute `Hidden` (which you can set for components to control whether they show up in a generated library or report) is not affected.

You do not currently need to update the existing component files. Members with a `Hidden` attribute compile without warnings. In a future release, implicit specification of the `Hidden` attribute on a member will start to produce warnings when you attempt to build the component. Eventually, the `Hidden` attribute for members will be removed. When writing new component files, use the new syntax.

If you want to update your existing component files, replace `Hidden=true` with `ExternalAccess=observe`.

Simscape language improved ease of use

In an effort to lift restrictions, simplify the syntax, and extend the language capabilities, the following enhancements have been implemented in this release:

- Arbitrary order of sections in a Simscape file. There may still be no more than one setup section per component. Otherwise, a component file can now contain multiple instances of declaration blocks or implementation sections of the same type, such as branches, equations, and so on. The sections can appear in any order.
- `der` operator now supports higher order derivatives. For example, `der(der(x))` is the second order time derivative of `x`.
- All parameters are now variable-sized. In previous releases, only parameters passed directly to `tablelookup` function could have variable size. Now, you no longer need to specify the `Size=variable` attribute for table data parameters. You can also do arithmetic on table data parameters:

```
component MyLookup
  parameters
    ud = [ 0 1 2 ];
```

```

    yd = [ 0 1 2 ];
end
inputs
    u = 0;
end
outputs
    y = 0;
end
equations
    y == tablelookup( ud, yd+1, u );
end
end

```

- Parameters may now be N-dimensional. This functionality is primarily intended for the three-dimensional lookup tables, introduced in this release (see “Lookup Table Improvements: Define 3D tables, interpolate using Akima splines” on page 1-2).
- Derived private parameters and variables. You can derive a parameter value by specifying an expression containing other parameters. For example:

```

component HasDerivedParameter
    parameters
        Par1 = 0;
        Par2 = 0;
    end
    parameters(Access=private)
        DerivedParameter = Par1 + Par2;
    end
    equations
        % Use DerivedParameter here.
    end
end

```

If `Par1` or `Par2` is modified in the setup section, then `DerivedParameter` will be the new sum of the two (and that change is applied at the moment of the setup assignment). The deriving relationship is broken only if the derived parameter is modified in the setup section.

The derived parameter must be declared with the `Access=private` attribute. The expression used to derive the parameter cannot be time-dependent, that is, it cannot contain operators such as `time`, `der`, `delay`, or `integ`. You can use the same method to initialize a variable.

- Nested parameter and variable access in composite components. For example, if component `C` contains component `B`, which in turn contains component `A`, both `C` and

B can assign values to parameters and variables of component A. In case of conflicting assignment, the “outermost” one (in this example, component C) takes precedence.

- Use of member component parameters and variables in the composite component equations. Similar to the previous example, you can now use parameters and variables of component A in the equations of components B and C. For example:

```
component B
  parameters
    p1 = 0;
  end
  components
    a = A(v1 = 10); % Sets initial condition of a.v1, as in previous releases.
  end
  equations
    der(a.v1) == p1; % Uses variable from component A here. This is new
  end
end
```

- Simple setup function. In general, you cannot specify a block parameter as run-time if the underlying component uses it in the setup function. However, if the setup is restricted to simple operations like error-checking, you can declare it as:

```
function setup %#simple
    ...
end
```

In this case, many of the parameters used in the setup function can be selected as run-time parameters. For more information, see “Simple Setup”.

Additional MATLAB operators allowed in Simscape language

The following MATLAB[®] operators are now allowed in a Simscape file:

Name	Discontinuous
ones	
zeros	
cat	
horzcat	
vertcat	

Name	Discontinuous
sum	
prod	
mod	Yes
any	Yes
all	Yes
min	Yes
max	Yes
ndims	
length	
size	
numel	
isempty	
isequal	Possibly, if arguments are real and have the same size and commensurate units
isinf	Yes
isfinite	Yes
isnan	Yes
double	
int32	Yes
uint32	Yes

Some of these operators were previously allowed only in the declaration section. They are now allowed in all the other sections as well.

All arguments that specify size or dimension must be unitless constants or unitless compile-time parameters.

Improved strictness and accuracy

As a continuation of improvements made in R2015b (see “Units handling refinements in Simscape language” on page 3-2), several more semantic changes have been made to increase robustness and consistency of simulation results:

- Fully typed parameter modification. In the old compiler, parameters could be updated with a value that either had commensurate unit or was unitless. In case of unitless value, the parameter preserved the original unit. In the new compiler, like in MATLAB, you can update a parameter with a value that has different size and unit. As a result, updating to a unitless value no longer preserves the original unit, which introduces a compatibility issue.
- Assertions on `let` variables. The old compiler effectively inlined `let` variable declarations into the code, and then added implicit assertions to the appropriate branches. The new compiler computes the values of `let` variables once per call, which greatly improves performance. As a result, if your `let` statement contains functions such as `sqrt(u)`, `log(u)`, and so on, the implicit assertion `u>=0` is added up front. Therefore, if the `in` part contains an `if` statement, with `u<0` in one of the branches, it triggers the assertion even if the `let` variable is used in another branch.
- Assertion for `a^b` where `a < 0 && ~isinteger(b)`. For minor time steps, the compiler still evaluates power as:

```
if a < 0
    return - ( (-a) ^ b );
else
    return a^b;
end
```

However, it now issues a warning assertion if the above rule is violated for an accepted time step.

Compatibility Considerations

When you rebuild your custom libraries for use with R2016a, you can get compilation errors or simulation-time assertions if your Simscape files violate the new, stricter, language rules.

For example, consider the following component:

```
component MyComp
    components
        r1 = resistor;
        r2 = resistor(R=2);
    end
    function setup
        r1.R = 1;
    end
```



```
end
```

Updating a parameter to a unitless value no longer preserves the original unit, and therefore you will now get a compilation error about units being not commensurate. To avoid the error, specify fully typed values when modifying parameters. For example:

```
component MyComp
  components
    r1 = resistor;
    r2 = resistor(R={2, 'Ohm'});
  end
  function setup
    r1.R = {1, 'Ohm'};
  end
end
```

Another example concerns the assertions on `let` variables. Consider the following code:

```
let
  v = sqrt(u);
in
  if u < 0
    y == 0;
  else
    y == v;
  end
end
```

Even though variable `v` is used only in the `else` branch, the implicit assertion is added at the beginning:

```
assert(u >= 0);
let
  v = sqrt(u);
in
  if u < 0
    y == 0;
  else
    y == v;
  end
end
```

Any negative value of `u`, instead of successfully executing through the `if` branch, will now trigger the assertion. To avoid the error, rewrite the file as follows:

```
if u < 0
    y == 0;
else
    y == sqrt(u);
end
```

Change to Thermal Liquid domain definition

The energy flow rate replaces the thermal flux as a Through variable in the Foundation Thermal Liquid domain.

The new Through variable is part of a broader change to the energy equations used in Thermal Liquid components. The updated equations are now more readable and extensible, enabling you to more easily check and add energy terms. The equations omit kinetic energy terms in a common approximation that has negligible impact on simulation results.

Compatibility Considerations

If you created custom components for the Thermal Liquid domain, you might need to update your code to reflect the new domain definition.

New base classes for Thermal Liquid components

Components in the Thermal Liquid domain inherit from new base classes. Simscape software continues to support the old base classes but these components no longer depend on them. This change affects the converter, pipe, restriction, reservoir, and source components.

Compatibility Considerations

The new base classes omit certain variables defined in the old base classes. If you created custom components that inherit from these component classes and call any of the missing variables, you might need to update your code. This issue does not affect custom components that inherit directly from the old base classes.

Renamed Thermal Liquid component source files

Component files for Pipe (TL), Translational Mechanical Converter (TL), and Rotational Mechanical Converter (TL) blocks have been renamed. Use the new file names to open

the files and view the source code for the blocks. The table shows the old and new files names.

New File Name	Old File Name
pipe_resistive	resistive_tube
pipe_resistive_compressibility	resistive_tube_with_dynamic_compressibility
pipe_resistive_compressibility_inertia	pipeline_segment
translational_converter	translational_converter_steady_compressibility
translational_converter_compressibility	translational_converter_dynamic_compressibility
rotational_converter	rotational_converter_steady_compressibility
rotational_converter_compressibility	rotational_converter_dynamic_compressibility

Foundation Library and Simulation

Run-Time Parameters: Speed up simulation tasks and modify component parameter values without regenerating C code

You can now modify certain block parameters between simulation runs without regenerating C code or triggering the diagram update.

Unlike Simulink blocks, where all parameters are tunable at run time unless otherwise specified, the default for Simscape block parameters is that they are modifiable at compile time. To make a parameter modifiable at run time, you have to designate it as such.

Tunable parameters (that is, parameters that you can modify during a simulation run) are not supported for Simscape blocks. If a Simscape parameter is designated as run-time, it means that you can modify its value between simulation runs without recompiling the model.

To specify run-time parameters:

- 1 Set the preference to show the run-time parameters drop-down in block dialogs. On the MATLAB Toolstrip, click **Preferences**. In the left pane of the Preferences dialog box, select **Simscape**, and then in the right pane, select the **Show run-time parameter settings** check box. Click **OK**.
- 2 Open a block dialog box in your model. Because of the preference setting, there is now an extra drop-down box next to each parameter. By default, all these drop-down boxes display **Compile-time**.
- 3 For the parameter that you want to modify, switch the drop-down value to **Run-time**. If the parameter currently has a numeric value, create a MATLAB variable and replace the parameter value with this variable.
- 4 In the tool bar at the top of the model window, click **Enable Fast Restart**.
- 5 You can now run the simulation repeatedly and change the variable value between simulations without recompiling the model.

For more information, see “Run-Time Parameters”.

Simulation Speed Improvements: Up to 5x speed improvement for simulations using generated code

In this release, there are significant speed improvements both for desktop simulations and for those using generated code. For example, a benchmark medium-sized electrical network, with approximately 200 nodes, shows the following results:

Simulation type	R2015b Time	R2016a Time	Speed Improvement
Desktop simulation	10.4 s	8.0 s	1.3 times
Simulink Coder™ Rapid Simulation Target (RSim)	12.2 s	6.1 s	2.0 times
Embedded Coder® (GRT) with local solver	15.2 s	3.1 s	4.9 times

Stream Logging Data to Disk: Increase capacity for Simscape simulation results

In previous releases, logged simulation data was saved in a workspace variable of type `simscape.logging.Node`, named as specified by the **Workspace variable name** configuration parameter. Saving data to the workspace can slow down the simulation and consume memory. Data logging options, such as **Decimation** or **Limit data points**, helped alleviate the issue by limiting the number of saved data points. You could also try logging data for a subset of blocks, or using time intervals.

The new method of streaming logged data to disk significantly increases the data logging capacity, because you are no longer limited by the system memory. You can now log simulation data for all blocks in a large model, or for selected blocks for a very long simulation time.

To stream data to disk, open the MATLAB Preferences dialog box, go to the **Simscape** pane, and select the **Stream data to temporary disk directory** check box. When this preference is turned on, the simulation data, in the form of a `simlog` object generated during simulation, is stored in an HDF5 file in a temporary directory. The workspace variable of type `simscape.logging.Node`, named as specified by the **Workspace variable name** configuration parameter, still gets created, but instead of storing all the

simulation data it references the `simlog` object in the temporary file. The temporary file persists as long as there is a logging variable name in the workspace that references it.

You view and analyze logged simulation data exactly as before, with all the interaction between the workspace variable and the stored object happening behind the scenes.

There are two new functions that handle saving logged data to a file (other than the temporary file) and loading it from a file into the workspace:

- `simscape.logging.export(simlog, 'fileName')` stores the simulation data in a specified HDF5 file. `simlog` is the simulation log variable name. `fileName` is the name and path to destination file.
- `var = simscape.logging.import('fileName')` creates a workspace variable of type `simscape.logging.Node`, which references the `simlog` object in the specified HDF5 file. `var` is the variable name. If you do not assign a variable name when calling the function, then the workspace variable name is `ans`.

By default, the **Stream data to temporary disk directory** preference is turned off, and data logging works the same as in previous releases.

For more information, see “Stream Logging Data to Disk”.

Simscape Component Block: Efficiently create custom components by selecting Simscape language file directly from block

In previous releases, generating Simscape blocks from textual component files required building the custom block libraries. You can still use the `ssc_build` command to generate a custom block library from a complete package of Simscape component files. However, the new **Simscape Component** block in the Utilities library lets you generate a Simscape block directly from a textual component file, skipping the library build process.

For more information, see “Selecting Component File Directly from Block”.

Simscape Results Explorer Unit Selection: Select y-axis units directly in plot pane

When you display logged simulation data in Simscape Results Explorer, the data along the *x*-axis is always time, in seconds. However, you can now change the *y*-axis units directly on the plot.

Each of the plots now has a drop-down arrow next to the unit name for the y -axis. When you click this arrow, a context menu appears containing names of all the units in the unit registry that are commensurate with the current plot unit, as well as two other options:

- **Default** — Use the default unit.
- **Specify** — Type the unit name or expression in a pop-up window and click **OK**. The specified unit name or expression must be commensurate with the current plot unit.

Once you select the option you want, the drop-down menu collapses and the plot is redrawn in specified units.

For more information, see “Plot Simulation Data in Different Units”.

Compatibility Considerations

Under **Plot options**, you can switch the **Plot signals** option from **Separate** to **Overlay**, to overlay multiple plots on the same axes. In previous releases, you could overlay plots with commensurate y -axis units. Now, if you want to overlay several plots, select the same y -axis unit for all these plots.

PS Lookup Table (3D) Block: Graphically define implicit equations that require lookup tables with three independent variables

The new **PS Lookup Table (3D)** block computes an approximation to some function $f=f(x_1, x_2, x_3)$ given the x_1, x_2, x_3, f data points. The three inputs and the output are physical signals. See the block reference page for details.

Compatibility Considerations

With the addition of three-dimensional table lookup, the axis naming schema has moved from (x, y, z) to (x_1, x_2, x_3) . Parameter names in the **PS Table Lookup (1D)** and **PS Table Lookup (2D)** blocks have been changed, to make them consistent with the **PS Lookup Table (3D)** block. The parameter names in the block source have also been changed, to make them consistent with the new naming schema.

If you use these blocks directly in your models, there is no compatibility impact. However, if you use these blocks as composite component members in Simscape language, you might need to update your code to use the new parameter names in the block source.

Block	Old Parameter Name	New Parameter Name
PS Table Lookup (1D)	x_t	x
PS Table Lookup (1D)	y_t	f
PS Table Lookup (2D)	x_t	x1
PS Table Lookup (2D)	y_t	x2
PS Table Lookup (2D)	z_t	f

Validation of Signal Units: Ensure consistent units specification on Simulink signals connected to Simscape physical networks

You can now specify physical units on Simulink signals. For details, see [Simulink Units: Specify, visualize, and check consistency of units on interfaces](#).

If you specify physical units on a Simulink signal connected to a **Simulink-PS Converter** or a **PS-Simulink Converter** block, this unit must be consistent with the unit specified inside the block. On a **Simulink-PS Converter** block, you specify the unit using the **Input signal unit** parameter. On a **PS-Simulink Converter** block, you specify the unit using the **Output signal unit** parameter. If the parameter value does not match the physical unit of the Simulink signal connected to the block, you get a warning.

If the block parameter value is 1, it matches the Simulink signal unit designated as 1.

Compatibility Considerations

The default Simscape unit registry has been made consistent with the Simulink unit database. If the Simscape and Simulink databases used different symbols for the same unit, both symbols are now in the Simscape database and you can use either one.

Quantity	Unit	Simscape Symbol	Simulink Symbol
Energy	British thermal unit	Btu	Btu_IT
Power	Horsepower	HP	HP_DIN
Viscosity absolute	Poise	Poise	P
Viscosity kinematic	Newt	Newt	newt

In several cases, there was a conflict between the Simulink and Simscape symbols.

Unit	Simscape Symbol	Simulink Symbol
Coulomb	c	C
Degree Celsius	C	degC
Degree Rankine	R	degR
Roentgen	-	R

This conflict resulted in the following changes to Simscape units, with potential compatibility impact:

- The **C** symbol for degree Celsius has been replaced with **degC**. Both **c** and **C** now indicate Coulomb.
- The **R** symbol for degree Rankine has been replaced with **degR**. Symbol **R** now indicates roentgen.
- The **Fh** symbol was not part of the conflict, but the **degF** symbol has been added for consistency. You can use either one for degree Fahrenheit.

If your legacy models use **Simulink-PS Converter** or **PS-Simulink Converter** blocks with unit specified as **C** or **R**, you get a compile-time error. Consult the Upgrade Advisor to migrate the model to the latest version of Simscape software.

If there is no conflict between the old and new symbols (for example, **Btu** and **Btu_IT**), your legacy models do not need to be updated. Only the new symbol (in this case, **Btu_IT**) will appear in the dialog box drop-downs for commensurate units, but you can type **Btu** in unit expressions. However, MathWorks recommends that you use the new symbol names going forward.

To distinguish between absolute and relative temperatures, Simscape unit manager lets you select whether to apply affine conversion (see “Thermal Unit Conversions”). By contrast, Simulink database contains a separate set of temperature units to indicate relative temperatures: **deltaK**, **deltadegC**, **deltadegF**, **deltadegR**. These units have also been added to the Simscape database. If you use them, affine conversion does not apply. The old method, of specifying regular temperature units and selecting whether to apply the affine conversion, works without change; however, you can get warnings about unit mismatch from the Simulink side of your model.

If you add a new unit to your unit registry, by using the **pm_addunit** function, and your unit definition conflicts with the one in the Simulink database, you will get a warning that the Simscape unit is not compatible with the Simulink unit database. If you add a unit that does not exist in the Simulink database, you will get a warning about undefined

unit. Note that this warning applies only to the Simulink database; the Simscape physical network works as expected. For information on how to turn off these warnings, see “Working with Simulink Units”.

Another issue affects legacy models where multiple **Simulink-PS Converter** or **PS-Simulink Converter** blocks share the same Simulink signal. For example, you can no longer connect multiple **Simulink-PS Converter** blocks with different units to the same Simulink source, such as a Constant block.

Thermal Resistor block in the Electrical Elements library

The **Thermal Resistor** block represents a temperature-dependent resistor. When the temperature at the thermal port is T , the resistance is $R = R_0(1 + \alpha(T - T_0))$, where R_0 is the nominal resistance at the reference temperature T_0 and α is the temperature coefficient.

This block was previously part of the Simscape Electronics™ Passive Devices library. The ability of the Simscape Electronics **Resistor** block to model thermal effects, introduced in R2015b, rendered the **Thermal Resistor** block superfluous. Therefore, this block has now been moved to the Simscape Electrical Elements library.

There is no compatibility impact on existing models.

Mass Flow Rate & Thermal Flux Sensor (TL) block renamed

The energy flow rate replaces the thermal flux as a Through variable in the Foundation Thermal Liquid domain. To reflect this change, the **Mass Flow Rate & Thermal Flux Sensor (TL)** block has been renamed to **Mass & Energy Flow Rate Sensor (TL)**. The updated block senses not the thermal energy flow rate but the total energy flow rate—the sum of thermal energy and pressure-volume work terms.

Compatibility Considerations

The variable measured through port Phi of the **Mass & Energy Flow Rate Sensor (TL)** block has changed. Older models dependent on thermal energy flow rate measurements can give inaccurate simulation results.

Thermal Liquid volumetric flow rate source and sensor blocks

The Thermal Liquid library now provides source and sensor blocks for setting and sensing volumetric flow rates:

- Volumetric Flow Rate Source (TL)
- Controlled Volumetric Flow Rate Source (TL)
- Volumetric Flow Rate Sensor (TL)

The sources are ideal and adiabatic. Friction losses and heat exchange with the source environment are ignored. Use the **Volumetric Flow Rate Source (TL)** block for constant flow rates and the **Controlled Volumetric Flow Rate Source (TL)** for variable flow rates.

Custom environment pressure in Thermal Liquid converter blocks

Thermal Liquid converter blocks provide a new option to specify the external environment pressure. Affected blocks include **Translational Mechanical Converter (TL)** and **Rotational Mechanical Converter (TL)**. You can set the environment pressure to the standard atmospheric value of 0.101325 MPa or to a custom value that you specify. The environment pressure adds to the total pressure the converters must overcome to generate force or torque.

Numerical improvements in Thermal Liquid blocks for greater model fidelity

Various block parameters and equations have changed to provide greater model fidelity. Changes include:

- Revised Nusselt number calculations in the **Pipe (TL)** block — In the laminar regime, the Nusselt number is now a constant. This change reflects the negligible thermal entrance effects assumed in pipe segments. In the turbulent regime, the Nusselt number now follows from the Gnielinski correlation. The new correlation improves numerical accuracy relative to the Colburn equation that it replaces.
- Scaling of minimum thermal conductance in the **Thermal Liquid Settings (TL)** block — The minimum thermal conductance in a thermal liquid network now scales with component geometry. This parameter sets a lower bound on thermal conductance which, during flow reversals, helps to smooth out heat flow transitions. The new geometric scaling enables you to more accurately model slender components with large aspect ratios.
- Updated discharge coefficient definition in **Local Restriction (TL)** and **Variable Local Restriction (TL)** blocks — The new discharge coefficient

parameter relates the actual mass flow rate to its theoretical value according to the expression

$$\dot{m} = C_d S_r \sqrt{\frac{2\rho\Delta p}{1 - (S_r / S)^2}},$$

where:

- \dot{m} is the mass flow rate.
- C_d is the discharge coefficient.
- S_r is the restriction area.
- S is the pipe area.
- ρ is the liquid density.
- Δp is the pressure drop across the restriction.

The previous definition corresponds to the more aptly named flow coefficient, given by

$$\dot{m} = C_v S_r \sqrt{2\rho\Delta p}$$

where C_v is the flow coefficient. The change in coefficient definition has a negligible impact on simulation results when the restriction area is less than one quarter of the adjacent pipe area.

- New cross-sectional area parameter for thermal conduction calculations in the **Rotational Mechanical Converter (TL)** block. The block uses the new parameter to determine the aspect ratio of the converter, L/S , where S is the thermal conduction length and S is the specified cross-sectional area. The thermal conduction length is assumed equal to the cross-sectional diameter.

Sign reversal of thermal expansion coefficient

The sign of the **Isobaric coefficient of thermal expansion** parameter in the **Thermal Liquid Settings (TL)** block has changed. The updated block automatically reverses the coefficient sign in older models so you do not have to. The new sign convention is the same used by the National Institute of Standards and Technology (NIST) in the fluid properties database REFPROP:

$$\alpha = \frac{1}{V} \left(\frac{\partial V}{\partial T} \right)_p = -\frac{1}{\rho} \left(\frac{\partial \rho}{\partial T} \right)_p,$$

where:

- α is the isobaric coefficient of thermal expansion
- V is the volume
- T is the temperature
- p is the pressure
- ρ is the density

Functionality being removed or changed

Simscape Language Keyword Name	What Happens When You Use the Keyword?	Use This Instead	Compatibility Considerations
Hidden attribute for members	Still runs	ExternalAccess	See Compatibility Considerations under “Improved attributes for member visibility and access” on page 1-5.

R2015aSP1

Version: 3.13.1

Bug Fixes

R2015b

Version: 3.14

New Features

Bug Fixes

Compatibility Considerations

Simscape Language

Units handling refinements in Simscape language

To increase robustness and consistency of simulation results, several semantic changes have been made with respect to how Simscape language handles math operations on physical units:

- When you declare a member as a value with unit, or use `{value, 'unit'}` in equations, `value` must be a unitless expression. In previous releases, the `value` expression could already have a unit, which sometimes resulted in double conversions by the unit manager.
- Arguments passed to transcendental functions must be dimensionless. Examples of transcendental functions include the exponential function, the logarithm, and the trigonometric functions. Previously, these functions accepted a value with unit as an argument, but did not handle units. For example, if p is pressure, `log(p)` would take a logarithm of the value but ignore the units, and therefore have different results depending on whether pressure was defined in Pa or MPa.

Now you have to pass a dimensionless argument to these functions, by converting to units of 1 before applying the function. In the example above, instead of `log(p)`, you now have to use `log(value(p, 'Pa'))`, or equivalently `log(p/{1, 'Pa'})`.

Compatibility Considerations

When you rebuild your custom libraries for use with R2015b, you can get `ssc_build` errors if your Simscape files violate the new, stricter, rules of units handling. Modify the source file to convert to units of 1, as described above.

Foundation Library and Simulation

Two-Phase Fluid domain and block library

The Foundation library now contains a two-phase fluid domain and Two-Phase Fluid block library. This library contains chambers, reservoirs, local restrictions, and energy converters, sources and sensors, as well as a Two-Phase Fluid Properties (2P) block, which controls fluid properties for the attached circuit.

Use these blocks for modeling two-phase fluid systems, where the working agent is part liquid and part vapor.

For more information, see the block reference pages. See also Two-Phase Fluid Domain for information on the two-phase fluid domain definition.

Variable priority option None for initialization

Variable initialization priority **None** is now available in block dialogs, in addition to **High** and **Low**. In previous releases, all three values were available only in the Simscape language, when writing component files. When using a block, you could only specify **High** or **Low** priority, or leave the variable as **Unused**. With this change, the options available in the **Variables** tab of the block dialog box align exactly with the options available in the Simscape file.

Each variable has a beginning value before the initialization algorithm finds consistent values for the system of equations. When the variable priority is **None**, the algorithm starts at this beginning value but does not remember the beginning value as the algorithm finds a solution. When the variable priority is **Low**, the beginning value becomes a target for the algorithm and the algorithm tries to stay close to the target. When the variable priority is **High**, the beginning value becomes a target for the algorithm and the algorithm tries to meet the target exactly.

The component author specifies default initialization priority, beginning value, and unit for each variable in the Simscape file. When using the block, you can override these default settings for any of the public variables (that is, variables that appear on the **Variables** tab of the block dialog box) by selecting the **Override** check box for that variable and specifying your own values.

If you clear the **Override** check box next to a variable name, then its **Priority**, **Beginning Value**, and **Unit** fields switch back to defaults specified in the component

file. However, if you select the check box again, these fields will retain their last specified value for when they were overridden.

Periodic Operators library

The Physical Signals library now has a Periodic Operators sublibrary, which contains the following blocks:

- PS Constant Offset Estimator
- PS Harmonic Estimator (Amplitude, Phase)
- PS Harmonic Estimator (Real, Imaginary)
- PS RMS Estimator

Use these blocks for various aspects of periodic signal measurements.

Speed and efficiency improvements for simulation of switched linear systems

Performance optimizations implemented in this release allow switched linear models to run faster in desktop simulation, in both Normal and Accelerator modes. The average simulation time for switched linear systems is around 1.3x faster than in previous release. The actual speedup varies depending on the particular model.

To take advantage of the new optimizations, switched linear models:

- Can not contain blocks that register sample times, such as **Counter** or **Random Number** blocks.
- Can not log simulation statistics. Open the Configuration Parameters dialog and, on the **Simscape** pane, clear the **Log simulation statistics** check box.

Domain-specific line style propagation through block icons

Domain-specific line styling, introduced in R2014b, now applies to the block icons as well. Relevant parts of each block icon assume domain-specific line styles and colors, with the connection lines continuing into the icon, so that the block diagram closely resembles a circuit schematic. The connection lines now have rounded corners. These enhancements improve the readability and clarity of block diagrams.

Rotational Inerter and Translational Inerter blocks

The new Rotational Inerter and Translational Inerter blocks represent two-port inertia in rotational and translational mechanical systems, respectively.

Use these blocks in high performance suspension systems, to decouple weave and roll modes, or in applications where you need to model a passively tuned mass-spring-damper response.

Real-Time Performance Advisor checks for physical models

If you have a Simulink Real-Time™ license, the new `Execute real-time application` activity mode in Performance Advisor lets you optimize your model for real-time execution. This mode includes several checks specific to physical models. These checks are organized in the **Simscape checks** folder, with subfolders for the add-on products, such as **SimDriveline checks** or **SimElectronics checks**. The top-level **Simscape checks** are applicable to all physical models. Each of the subfolders contains checks that target specific blocks from that add-on product.

To access the checks, in the Performance Advisor window, under **Activity**, select `Execute real-time application`. In the left pane, expand the **Real-Time** folder, and then the **Simscape checks** folder. Run the top-level Simscape checks. If your model contains blocks from an add-on product, also run the checks in the subfolder corresponding to that product.

For more information on using the Performance Advisor, see Automated Performance Optimization.

Real-Time Simulation Documentation Enhancements

The documentation on real-time and hardware-in-the-loop simulation of Simscape models has been revised and expanded. See Real-Time Simulation.

Variable Viewer link to block diagram renamed

The Variable Viewer option that highlights the relevant block in the block diagram has been renamed from **Highlight block** to **Go to block**, to avoid ambiguity in SimMechanics™ models.

When you right-click in the **Name** column of any row in the Variable Viewer table, a context menu opens with the following options:

- **Go to block** — Highlights the corresponding block in the block diagram, opening the appropriate subsystem if needed. If the row represents a variable, highlights the parent block for this variable.
- **Open block dialog** — Opens the corresponding block dialog box (for a variable, opens the parent block dialog box). This enhancement makes it easy to go from the Variable Viewer to the **Variables** tab in the block dialog box, to modify the variable priorities and targets.

For more information, see Variable Viewer.

Naming schema change within the data logging variable structure

When you enable simulation data logging, the workspace variable containing the logged data mimics the structure of the model. Nodes within the variable have names that are derived from the blocks and variables in the model. However, each node has to be a valid and unique MATLAB identifier, and therefore the block names undergo changes, such as replacing spaces with underscores.

In previous releases, this name processing also involved a built-in string-to-integer function that stripped leading zeros from numbers in block names. For example, for a **Resistor** block named **R0001** in the block diagram, the corresponding logging variable node was named **R1**. If your block diagram contained two resistors, **R1** and **R0001**, then one node was named **R1** and the other **R2**, to satisfy the unique identifier rule. As a result, the logging variable nodes could have names that did not match the block diagram.

The current naming schema preserves the leading zeros in numbers included in block names. In the example above, the node corresponding to resistor **R0001** is now named **R0001**.

Compatibility Considerations

If your models have blocks with names that contain numbers with leading zeros (such as **R0001**), then the names of the corresponding nodes in the data logging structure have changed compared to the previous release. If you have scripts that rely on these node names being the same, you need to update them.

R2015a

Version: 3.13

New Features

Bug Fixes

Foundation Library and Simulation

Variable Viewer link to block diagram

The Variable Viewer tool now provides a direct link to the relevant block in the block diagram.

When you right-click in the **Name** column of any row in the Variable Viewer table, a context menu opens with the following options:

- **Highlight block** — Highlights the corresponding block in the block diagram, opening the appropriate subsystem if needed. If the row represents a variable, highlights the parent block for this variable.
- **Open block dialog** — Opens the corresponding block dialog box (for a variable, opens the parent block dialog box). This enhancement makes it easy to go from the Variable Viewer to the **Variables** tab in the block dialog box, to modify the variable priorities and targets.

For more information, see Variable Viewer.

Improved solver efficiency for model initialization

The initial conditions solver speed increased significantly in this release without sacrificing the robustness.

The initial conditions solver works much faster in general. If the initial condition solve fails, the reaction time is much faster as well. The solver also reacts almost instantaneously when you press CTRL-C.

Sparkline plots for logged data

You can view sparkline mini-plots of logged simulation data for selected blocks and variables directly on the model canvas.

Before using this functionality, you must enable data logging, for all or some of the blocks, and run the simulation. For more information, see Data Logging.

To view the sparkline plots, in the model window, from the top menu bar, select **Display > Simscape > Toggle Sparklines When Clicked**. This action adds the check mark

next to the **Toggle Sparklines When Clicked** menu option, and you can start selecting blocks to display sparkline plots of logged data for their variables.

When you select a block you can see sparkline plots for its variables. Hover over the variable name to see the plot. You can select which variables to plot (the first three are shown by default).

If you select a block for which simulation data is not being logged, it displays **No variables** instead of the sparkline plots. Right-click the block, select **Simscape > Log simulation data**, and rerun the simulation.

Repeatedly selecting a block toggles the display of its sparkline plots on and off.

To clear all plots and start again with a clean canvas, select **Display > Simscape > Remove All Sparklines**. Then you can select more blocks and variables to display their sparkline plots.

Repeatedly selecting the **Toggle Sparklines When Clicked** menu option toggles the ability to view the sparkline plots for the model on or off, as indicated by the check mark.

For more information, see [View Sparkline Plots of Simulation Data](#).

Data logging for models created using `ssc_new`

When you create a new model using the `ssc_new` function, simulation data logging for this model is now turned on by default. Data logging configuration parameters are automatically set to the following values:

- **Log simulation data** — All.
- **Log simulation statistics** — Off.
- **Open viewer after simulation** — Off.
- **Workspace variable name** — `simlog`.
- **Decimation** — 1.
- **Limit data points** — On.
- **Data history (last N steps)** — 10000.

Using data logging is a best practice for Simscape models because it provides access to important simulation and analysis tools. For more information, see [Data Logging](#).

Pressure setting and name change for Thermal Liquid reservoir blocks

The two reservoir blocks in the Thermal Liquid/Elements library serve as reference points in a pipe network, where you can specify an arbitrary reference temperature. In previous versions, the pressure in these blocks was always set to atmospheric pressure. Now you have an option to specify an arbitrary value for the reference pressure, as well as temperature.

Both blocks now have a new parameter, **Reservoir pressure specification**, which lets you select between two values:

- Atmospheric pressure (default)
- Specified pressure — If you select this value, an additional parameter, **Reservoir pressure**, appears in the dialog box to let you enter the reference pressure value.

The block names have been changed:

Old Name	New Name
Temperature Reservoir (TL)	Reservoir (TL)
Controlled Temperature Reservoir (TL)	Controlled Reservoir (TL)

There is no compatibility impact. When you open an existing model containing these blocks, it is updated automatically to use the new version of the blocks.

New examples

Examples introduced in this version are:

- Lithium-Ion Battery Pack With Fault
- Transmission Line

R2014b

Version: 3.12

New Features

Bug Fixes

Compatibility Considerations

Simscape Language

Refrigeration example modeling two-phase fluid flow

A new example shows how you can use Simscape language to model two-phase fluid flow. The example includes a custom two-flow domain and a library of custom components based on this domain.

To open the custom library, type `two_phase_lib` at the MATLAB Command prompt. Double-click any block in the library to open its dialog box, and then click the **Source code** link in the block dialog box to open the Simscape source file for this block in the MATLAB Editor. The custom domain file is located in the same folder as the component files.

To open the Two-Phase Flow Refrigeration Model built of these custom components, type `ssc_refrigeration` at the MATLAB Command prompt.

Foundation Library and Simulation

Domain-specific line styles for representing physical connections

Each Simscape domain now uses a distinct default color and line style for the connection lines. Physical signal lines now also have a distinct style and color. This enhancement improves the readability of block diagrams.

Improved interface for connecting Simscape blocks

The following usability enhancements have been made in this release:

- Visual hint for an invalid connection attempt — You cannot connect ports that belong to different domains, but in previous releases it was not always obvious why the connection was not being made. Now, as you are trying to connect a Simscape port, the cursor appearance changes just as it approaches a port or line that belongs to a different domain, to indicate that the connection here is invalid.
- If you place a Simscape block with two ports (one on each side) over a connection line that belongs to the appropriate domain, the block gets inserted automatically.
- When you connect Simscape blocks, the ports on either side of the connection line disappear. Physical signal output ports also disappear. Physical signal input ports remain, to show the signal direction.

Option to configure logging of Simscape simulation results per block

In previous releases, once you enabled simulation data logging, it included data for the whole model. The new `Use local settings` option lets you log data for selected blocks only. The basic workflow is as follows:


- 1 Enable simulation data logging on a block-by-block basis. In the model window, from the top menu bar, select **Simulation > Model Configuration Parameters**. In the Configuration Parameters dialog box, in the left pane, select **Simscape**, then set the **Log simulation data** parameter to `Use local settings`. Click **OK**.
- 2 Select the blocks in your model by right-clicking on each block and then, from the context menu, selecting **Simscape > Log simulation data**. A check mark appears in front of the **Log simulation data** option.
- 3 Simulate the model. When the simulation is done, the simulation data log contains only the data from the selected blocks.

To stop logging data for a previously selected block, right-click on it and select **Simscape > Log simulation data** again to remove the check mark.


If you set the **Log simulation data** parameter to **All**, the simulation log will contain data from the whole model, regardless of the block selections. Setting the **Log simulation data** parameter to **None** disables data logging for the whole model.

Flat view option and improved diagnostics for Variable Viewer

You can now control the number of rows in the Variable Viewer by switching between the tree view (the default) and the flat view. By default, the Variable Viewer opens in tree view, with variable nodes grouped under the parent port, block, and subsystem nodes.

To switch to the flat view, click  in the Variable Viewer toolbar. The table now contains one row per variable, with the **Name** column showing the full path to the variable, in the following format:

```
SubsystemName --> BlockName --> PortName --> VariableName
```

If the Variable Viewer is in flat view, the buttons that expand and collapse nodes are disabled. To switch back to the tree view, click  in the Variable Viewer toolbar.

Another improvement is better diagnostics of the overall status. A message in the bottom-right corner of the Variable Viewer window tells you whether the viewer displays the variables at start, imported variables (when you load an initial state or SimState), or after update diagram (when the previously computed actual values become unavailable).

Simscape Results Explorer


Simscape Results Explorer is an interactive tool that lets you navigate and plot the simulation data logging results. This tool is an enhanced version of the Simscape Simulation Results Explorer (`ssc_explore.m`) previously posted on MATLAB Central. It is now part of the product. To use the tool:


- 1 In the model window, from the top menu bar, select **Simulation > Model Configuration Parameters**. In the Configuration Parameters dialog box, in the left pane, select **Simscape**.
- 2 Set the **Log simulation data** parameter to **All** or **Use local settings**, to enable data logging.
- 3 Select the **Open viewer after simulation** check box. Click **OK**.


- 4 If using local settings, select the blocks in your model as described in “Option to configure logging of Simscape simulation results per block” on page 5-3.
- 5 Simulate the model. When the simulation is done, the Simscape Results Explorer window opens. It contains the simulation log tree hierarchy in the left pane. When you click on a node in the left pane, the corresponding plot appears in the right pane.

You can control whether the Simscape Results Explorer window is reused when you rerun the simulation, or a new window is opened after the next simulation run, by linking and unlinking the window.

When you first open the Simscape Results Explorer window, it is linked to the current MATLAB session. This means that when you run a new simulation, the results in the window will be overwritten. To retain the current results and open a new window after

the next simulation, click the  button located in the toolbar above the left pane. The

button appearance changes to  and, when the new window opens after simulation, that window will be linked to the session. Only one window can be linked to the session,

so if you have multiple windows open, linking one of them (by clicking on its  button) unlinks the previous one.

Simscape logging into Simulink single output

If you select the **Save simulation output as single object** check box on the **Data Import/Export** pane of the Configuration Parameters dialog box, Simscape log data will now be part of the single output object. In this case, it is not stored as a separate workspace variable.

If the **Save simulation output as single object** check box is not selected, simulation data is stored in the current workspace, with the name of the workspace variable specified by the **Workspace variable name** configuration parameter. The default variable name is `simlog`.

This enhancement makes Simscape data logging compatible with the `parfor` command. For more information, see [Save simulation output as single object](#).

Compatibility Considerations

In previous releases, simulation data was stored as a variable in the base workspace. Now, if the single output is not enabled (which is the default), this variable is stored in

the current workspace, instead of the base workspace. If the single output is enabled, simulation data becomes part of the single output object and is not stored as a separate workspace variable. Therefore, if you have created custom scripts or functions that rely on accessing the `simlog` variable in the base workspace, you will have to modify them accordingly.

Simscape logging integration with Simulation Data Inspector

You can now configure your model to automatically record Simscape logging data, along with the rest of simulation data obtained from a model run, using the Simulation Data Inspector. The basic workflow is as follows:

- 1 Set up your model to log simulation data. In the model window, from the top menu bar, select **Simulation > Model Configuration Parameters**. In the Configuration Parameters dialog box, in the left pane, select **Simscape**, then set the **Log simulation data** parameter to **All** or **Use local settings**. If using local settings, select the blocks to log data from, as described in “Option to configure logging of Simscape simulation results per block” on page 5-3.
- 2 Enable data recording. In the Configuration Parameters dialog box, in the left pane, select **Data Import/Export**, then select the **Record logged workspace data in Simulation Data Inspector** check box.
- 3 Simulate the model. When the simulation is done, a notification bar appears in the Simulink Editor.
- 4 In the notification bar, click the link to open the Simulation Data Inspector and view the results.

For detailed information on how to configure and use the Simulation Data Inspector, see [Inspect Signal Data with Simulation Data Inspector](#).

Infinite resistance and port terminator blocks

Infinite resistance blocks let you specify initial difference for the appropriate Across variable (such as voltage, pressure, temperature) between two nodes without affecting model equations. The following new blocks in Foundation library implement infinite resistance for their respective domains:

- Infinite Flow Resistance (TL)
- Infinite Hydraulic Resistance

- Infinite Pneumatic Resistance
- Infinite Resistance
- Infinite Thermal Resistance

These blocks have no parameters, but their **Variables** tab lets you set the priority and initial target value for the appropriate Across variable.

Similarly, the conserving port terminator blocks now display the **Variables** tab, which lets you set the priority and initial target value for the appropriate Across variable at a node. The affected blocks are:

- Cap (TL)
- Hydraulic Cap
- Open Circuit
- Perfect Insulator
- Rotational Free End
- Translational Free End

The PS Terminator block has been added to the Sinks sublibrary of the Physical Signals library. It lets you cap physical signal output ports that do not connect to other blocks. Unlike conserving ports in physical modeling, or Simulink output ports, unconnected physical signal output ports do not generate warnings. However, you can use a PS Terminator block for clarity, to indicate that the signal was not inadvertently left unconnected.

Performance improvements for linear systems in Normal and Accelerator modes

Linear optimizations have been extended to include affine systems (for example, models containing blocks like **DC Voltage Source**), which results in faster simulation. Also, increased efficiency of generated code for linear and affine systems provides improved performance in Accelerator mode and for code generation.

R2014a

Version: 3.11

New Features

Bug Fixes

Compatibility Considerations

Simscape Language

ssc_build library build process that no longer requires C compiler

In previous releases, the library build process, whether performed using the `ssc_build` command or using the `ssc_mirror` command with the `buildmirror` flag set to `true`, required a C compiler. Before running either of these commands for the first time, you had to set up your compiler by running `mex -setup`.

In-memory execution of the library build process, implemented in this release, increases the process speed and eliminates the need for a C compiler. Therefore, you no longer need to have a C compiler available on your machine in order to build a custom block library.

priority attribute for setting relative priority of variable target values during initialization

The new initialization process, implemented in this release, involves block-level variable initialization. That is, a Simscape block dialog box now has an additional **Variables** tab, which lists all the public variables specified in the underlying component file, along with the initialization priority, target initial value, and unit of each variable. The block user can change the variable priority and target, prior to simulation, to affect the model initialization. The default values for variable priority, target value, and unit come from the variable declaration in the component file.

To enable the block author to specify default priority for a variable, a new attribute is now available. The `priority` attribute can have one of the three values: `priority.high`, `priority.low`, and `priority.none`. The default is `priority.none`, which is equivalent to leaving out the `priority` attribute entirely.

To specify a high or low default priority for a component variable, declare the variable as a field array. For example, the following declaration initializes variable `t` (spring deformation) as 0 mm, with high priority:

```
variables
    t = { value = { 0 , 'mm' }, priority = priority.high }; % Spring deformation
end
```

The old way of declaring variables still works. In fact, leaving out the `priority` attribute (that is, using `priority.none`) is suitable in most cases. The block user can

control the variable initialization priority, as needed, by using the **Variables** tab of the block dialog box.

For example, you can declare the same variable `t` as follows:

```
variables
    t = { 0 , 'mm' }; % Spring deformation
end
```

In this case, the **Variables** tab in the block dialog box will have the **Spring deformation** variable listed initially as **Unused** (which means its priority and target are not used in the initialization process). The block user can modify the variable priority, as well as the target value and unit, in the **Variables** tab of the block dialog box prior to simulation.

If there are no top-level public variables declared in the component file (for example, if the top-level variables are declared as hidden), the **Variables** tab does not appear in the block dialog box. The same is true for composite components, because they also have no top-level public variables.

For more information on block-level variable initialization, see “Variables tab for specifying target value and priority for new initialization process” on page 6-5.

Script for replacing through and across statements with branches and equations

The `ssc_update` script is now available to help you update the legacy component files that contain **across** and **through** statements. The script replaces the **through** statements with the corresponding **branches** section, and adds the equations equivalent to the **across** statements to the **equations** section of the file.

For more information on the old and new syntax, see Compatibility Considerations under “**branches** section for defining the relationship between component Through variables and nodes” on page 7-2.

To run the script, at the MATLAB command prompt, type:

```
ssc_update package
```

where *package* is the name of a top-level package directory, without the leading + character. If you run the `ssc_update` command from inside the package directory

structure, you can omit the argument. The script updates all the legacy component files located in the package. For more information, see `ssc_update`.

Foundation Library and Simulation

Variables tab for specifying target value and priority for new initialization process

New initialization process, implemented in this release, gives you more control over model initialization. Most of the Foundation library blocks now have a new **Variables** tab, which lists all the public variables specified in the underlying component file, along with priority, initial value, and unit. In most cases, the default value for each of these is **Unused**. Once you select the check box next to a variable name, you can specify its priority (**High** or **Low**), target initial value, and unit.

If the underlying component has no top-level public variables (such as a composite component, or one with the top-level variables declared as hidden), then the block dialog box does not have the **Variables** tab. For that reason, the Utilities library blocks, most of the Foundation library sensors and sources, and the absolute majority of blocks in the add-on products are not affected by this change.

The values you specify during block-level variable initialization are not the actual values of the respective variables, but rather their target values at the beginning of simulation ($t = 0$). Depending on the results of the initial conditions solve, some of these targets may or may not be satisfied. The solver tries to satisfy the high-priority targets first, then the low-priority ones:

- At first, the solver tries to find a solution where all the high-priority variable targets are met exactly, and the low-priority targets are approximated as closely as possible. If the solution is found during this stage, it satisfies all the high-priority targets. Some of the low-priority targets might also be met exactly, the others are approximated.
- If the solver cannot find a solution during the first stage, it issues a warning and enters the second stage, where **High** priority is relaxed to **Low**. That is, the solver tries to find a solution by approximating both the high-priority and the low-priority targets as closely as possible.

After you initialize the block variables and prior to simulating the model, you can open the Variable Viewer to see which of the variable targets have been satisfied. For more information on block-level variable initialization and Variable Viewer, see Variable Initialization.

Compatibility Considerations

In previous releases, several Foundation library blocks contained parameters that let you specify an initial value for an internal block variable at the start of simulation. These parameters have now been removed. The following table lists the initialization parameters that have been removed from block dialogs and the names of the corresponding block variables:

Block Name	Parameter Name	Variable Name
Capacitor	Initial voltage	Capacitor voltage
Constant Volume Chamber (TL)	Fluid initial pressure	Pressure
	Fluid initial temperature	Temperature
Constant Volume Hydraulic Chamber	Initial pressure	Pressure (gauge)
Constant Volume Pneumatic Chamber	Initial pressure	Pressure
	Initial temperature	Temperature
Fluid Inertia	Initial flow rate	Flow rate
Hydraulic Piston Chamber	Initial pressure	Pressure (gauge)
Inductor	Initial current	Inductor current
Inertia	Initial velocity	Rotational velocity
Mass	Initial velocity	Velocity
Mutual Inductor	Winding 1 initial current	Primary current
	Winding 2 initial current	Secondary current
Pneumatic Piston Chamber	Initial pressure	Pressure
	Initial temperature	Temperature
Rotational Hard Stop	Initial angular position	Angular position
Rotational Spring	Initial deformation	Deformation
Rotary Pneumatic Piston Chamber	Initial pressure	Pressure
	Initial temperature	Temperature
Thermal Mass	Initial temperature	Temperature

Block Name	Parameter Name	Variable Name
Translational Hard Stop	Initial position	Position
Translational Spring	Initial deformation	Deformation
Variable Hydraulic Chamber	Initial pressure	Pressure (gauge)

Legacy models using these blocks are not affected by this change. If a block had an initialization parameter, then, once you open the model in the current release, this parameter value is automatically mapped to the corresponding block variable, which assumes it as the target value with **High** priority. The simulation results stay the same.

However, if you have a custom composite component (written in Simscape language) that uses one of these Foundation library blocks and references an initialization parameter, trying to build the library or simulate a model containing the custom block produces an error, because the referenced parameter is no longer available.

For example, if you have a custom composite component that contains an Inertia block:

```
component DC_Motor
[...]
parameters
    motor_inertia = { 0.01, 'g*cm^2' };      % Inertia
    init_velocity = { 0, 'rad/s' };          % Initial velocity
    [...]
end
components(Hidden=true)
    motorInertia = foundation.mechanical.rotational.inertia(inertia = motor_inertia,
        initial_velocity = init_velocity);
    [...]
end
[...]
end
```

`ssc_build` will produce an error similar to the following:

```
Error using ne_updateLibraryItem>lBuild (line 35)   File: C:\Work\libraries\MyElecLibrary\DC_Motor.ssc
Line: 32 Reference to parameter 'initial_velocity' is invalid.
```

Update the custom component by removing all references to the initialization parameter:

```
component DC_Motor
[...]
parameters
    motor_inertia = { 0.01, 'g*cm^2' };      % Inertia
    [...]
end
components(Hidden=true)
    motorInertia = foundation.mechanical.rotational.inertia(inertia = motor_inertia);
    [...]
end
```

```
[...]  
end
```

Variable Viewer for analyzing results of new initialization process

A new analysis tool, available for models containing Simscape blocks and blocks from add-on products, provides the back end for the block-level variable initialization by letting you view the variable targets, priority, and actual initial values prior to simulation. To open the tool, in the top menu bar of the model window, select **Analysis > Simscape > Variable Viewer**. For more information, see Variable Viewer.

Statistics Viewer that displays variable source and number of eliminated variables

The Statistics Viewer analysis tool has the following enhancements:

- New top-level statistic, **1-D/3-D Interface**, lists the connections between the two types of physical networks. It appears only for models that connect blocks from SimMechanics Second Generation library to Simscape blocks, or blocks from other add-on products.
- The new **Sources** section lists variable sources for the selected statistic. If you select a connection under the **1-D/3-D Interface** statistic category, the **Sources** section lists the source and destination for this connection. If you select a statistic with a nonzero value under the **1-D Physical System** category, the **Sources** section lists all the variables that fall under this statistic.

For each variable, the **Source** column contains the full path to the variable, starting from the top-level model, with a link to the relevant block. If you click the link in the **Source** column, the corresponding block is highlighted in the block diagram. The **Value** column contains the name of the variable, as it would appear in the **Variables** tab of the block dialog box.

- Additional statistics under the **1-D Physical System** category: **Number of eliminated variables** (further categorized as algebraic and differential variables) and **Number of dynamic variable constraints**. Eliminated variables are continuous variables that are eliminated during optimization and are not seen by the solver. Dynamic variable constraints are constraints involving only dynamic variables and inputs. Such constraints result in Index-2 differential algebraic equations and therefore can cause numerical difficulties or slow down your simulation.

For more information, see Model Statistics.

Fundamental Reluctance block

The new Fundamental Reluctance block in the Magnetic Elements library provides a simplified model of a magnetic reluctance, that is, a component that resists flux flow. Unlike the Reluctance block, which computes reluctance based on the geometry of the section being modeled, the **Fundamental Reluctance** block lets you specify a value of reluctance directly as a block parameter.

Hydro-mechanical converter blocks with fluid compressibility option

The Rotational Hydro-Mechanical Converter and Translational Hydro-Mechanical Converter blocks now contain a drop-down **Compressibility** parameter, with the default value **Off**. If you select **On**, additional parameters appear in the block dialog to let you account for fluid compressibility within the converter block itself. In previous releases, you had to connect a converter to a **Hydraulic Piston Chamber** block to account for fluid compressibility.

Compatibility Considerations

Existing models are not affected by this change, because compressibility in converters is off by default. However, this change makes the **Hydraulic Piston Chamber** block obsolete. MathWorks recommends that you specify fluid compressibility directly in the converter blocks, because the new method provides more accurate results and also because the **Hydraulic Piston Chamber** block may be removed in a future release.

Handling of pressure or temperature below absolute zero during simulation

You can now set the models that use the hydraulic domain to either warn or stop simulating with an error when absolute pressures fall below absolute zero. The default behavior is to stop simulating with an error, which is the same as in previous releases. You can change this by using the **Custom Hydraulic Fluid** block, or the **Hydraulic Fluid** block (available with SimHydraulics[®] block libraries), to have the simulation continue with a warning. See the block reference pages for details.

You can also set the models that use the pneumatic domain to either warn or stop simulating with an error when pressures or temperatures fall below absolute zero. The default behavior is to stop simulating with an error. This check was not performed during simulation in previous releases. You can change the default behavior by using the **Gas**

Properties block, to have the simulation continue with a warning. See the block reference page for details.

Input filtering options for 1-D/3-D connections

The Solver Configuration block now lets you control whether input filtering is applied automatically for models that connect blocks from SimMechanics Second Generation library to Simscape blocks, or blocks from other add-on products. It also lets you specify a global filtering time constant value for all the 1-D/3-D connections within the network. See the block reference page for details.

Software-in-the-loop simulation for physical models

In previous releases, software-in-the-loop (SIL) simulation was not supported for models containing Simscape blocks or blocks from the add-on products. This limitation is now removed.

Change in default settings for `ssc_new`

The `ssc_new` function, which creates a new Simscape model populated by required and commonly used blocks, now uses a different default solver and absolute tolerance. Here is the summary of changes:

	Old setting	New setting
Solver	ode15s (stiff/NDF)	ode23t (mod. stiff/Trapezoidal)
Absolute tolerance	auto	1e-3

Existing models are not affected. When you create new models with `ssc_new`, they will use the new settings. For more information, see the `ssc_new` reference page.

Functionality being removed or changed

Simscape Language Keyword Name	What Happens When You Use the Keyword?	Use This Instead	Compatibility Considerations
<code>across</code>	Still runs	Create equivalent equations	Run <code>ssc_update</code> to update the

Simscape Language Keyword Name	What Happens When You Use the Keyword?	Use This Instead	Compatibility Considerations
			components in an existing package. See “Script for replacing through and across statements with branches and equations ” on page 6-3.
through	Still runs	branches	Run <code>ssc_update</code> to update the components in an existing package. See “Script for replacing through and across statements with branches and equations ” on page 6-3.

R2013b

Version: 3.10

New Features

Bug Fixes

Compatibility Considerations

Simscape Language

branches section for defining the relationship between component Through variables and nodes

In previous releases, the Through and Across variables in a component file were connected to the domain Through and Across variables in the `setup` section, using the `through` and `across` statements. The syntax of these statements was nonintuitive and somewhat confusing.

Starting in this release, define the relationship between the Through variables by using the new `branches` section, located after the `setup` section in the component file. The `branches` section starts with the `branches` keyword, contains one or more branch statements, and ends with the `end` keyword.

Each branch statement has the syntax:

```
a : node1.a -> node2.a;
```

which clearly indicates direction, from `node1` to `node2`. Therefore, `a` is subtracted from the conserving equation identified by `node1.a`, and `a` is added to the conserving equation identified by `node2.a`. If the component has multiple nodes, indicate branches by writing multiple statements in the `branches` section. For syntax and examples, see the `branches` reference page.

To establish the relationship between the Across variables, use the `equations` section of the component file. Add an equation that connects the component Across variable with the respective variables at the component nodes. If there is more than one Across variable, add multiple equations, connecting each variable with its respective nodes.

For more information, see [Defining Relationship Between Component Variables and Nodes](#).

Compatibility Considerations

You do not currently need to update the existing component files. The `across` and `through` statements compile without warnings. In a future release, they will start to produce warnings when you attempt to build the component. Eventually they will be removed. When writing new component files, use the new syntax.

If you want to update your existing component files, the following table summarizes the old and new syntax.

Old Syntax	New Syntax
<code>through(a, node1.a, node2.a);</code>	<code>a : node1.a -> node2.a ;</code>
<code>across(a, node1.a, node2.a);</code>	<code>a == node1.a - node2.a ;</code>

For example, suppose you have an electrical component that uses the old syntax:

```
component my_resistor
  nodes
    p = foundation.electrical.electrical;
    n = foundation.electrical.electrical;
  end
  variables
    i = { 0, 'A' };
    v = { 0, 'V' };
  end
  parameters
    R = { 1, 'Ohm' }; % Resistance
  end
  function setup
    across( v, p.v, n.v ); % voltage across
    through( i, p.i, n.i ); % current through
  end
  equations
    v == i*R;
  end
end
```

Here is the same component rewritten using the new syntax:

```
component my_resistor
  nodes
    p = foundation.electrical.electrical;
    n = foundation.electrical.electrical;
  end
  variables
    i = { 0, 'A' };
    v = { 0, 'V' };
  end
  parameters
    R = { 1, 'Ohm' }; % Resistance
```

```
end
branches
    i : p.i -> n.i ;    % current through
end
equations
    v == p.v - n.v;    % voltage across
    v == i*R;
end
end
```

import statement enabling simplified access to other component classes

With the introduction of composite components in R2012b, class member declarations now include user-defined types, that is, component classes. An `import` mechanism provides a convenient means to accessing classes defined in different scopes, with the following benefits:

- Allows access to model class names defined in other scopes without a fully qualified reference
- Provides a simple and explicit view of dependencies on other packages

The `import` statement can have the following syntax:

```
import package_or_class;
```

or

```
import package.*
```

The first syntax is a qualified import, which imports a specific package or class. The second one is an unqualified import, which imports all subpackages and classes under the specified package.

For more information, see [Importing Domain and Component Classes](#).

connect statement support for vector and matrix physical signals

Simscape language now supports nonscalar (vector-valued or matrix-valued) physical signals in `connect` statements. For more information, see [Nonscalar Physical Signal Connections](#).

Foundation Library and Simulation

Thermal Liquid domain and block library

The Foundation library now contains a thermal liquid domain and Thermal Liquid block library. This library contains thermohydraulic elements, such as chambers, reservoirs, local restrictions, and hydro-mechanical converters. It also contains thermal liquid sources and sensors, as well as a Thermal Liquid Settings (TL) block, which controls thermal liquid domain properties for the attached circuit.

Use these blocks for modeling applications such as:

- Transportation of heated liquid in pipeline networks
- Actuator warm-up due to viscous stresses
- Heat generation and dissipation in complex systems, such as aircraft hydraulic systems and associated heat exchangers

For more information, see the block reference pages. See also Thermal Liquid Domain for information on the thermal liquid domain definition. The Across variables are pressure and temperature, and the Through variables are mass flow rate and thermal flux. Note that the product of each pair of the Through and Across variables (pressure and mass flow rate, temperature and thermal flux) is not power, and therefore these result in a pseudo-bond graph.

Simscape model statistics viewer

A new analysis tool, available for models containing Simscape blocks and blocks from add-on products, lets you view Simscape statistics, such as the number of continuous and discrete variables, number of zero-crossing signals, and number of joints and constraints. To open the tool, in the top menu bar of the model window, select **Analysis > Simscape > Statistics Viewer**.

For more information, see Simscape Model Statistics.

Removal of laminar-turbulent zero-crossings in hydraulic blocks

Hydraulic blocks in the Foundation library no longer produce zero-crossings upon transition between the laminar and turbulent regimes during simulation. This enhancement results in increased simulation efficiency for hydraulic models.

New examples

Examples introduced in this version are:

- Hydraulic Fluid Warming due to Losses
- Optimal Pipeline Geometry for Heated Oil Transportation
- Water Hammer Effect

Functionality being removed or changed

Simscape Language Keyword Name	What Happens When You Use the Keyword?	Use This Instead	Compatibility Considerations
across	Still runs	Create equivalent equations	See Compatibility Considerations under “branches section for defining the relationship between component Through variables and nodes” on page 7-2.
through	Still runs	branches	See Compatibility Considerations under “branches section for defining the relationship between component Through variables and nodes” on page 7-2.

R2013a

Version: 3.9

New Features

Bug Fixes

Compatibility Considerations

Simscape Language

Variable-size domain parameters

Variable-size component parameters have been implemented in Release R2012a, to support the `tablelookup` function. You can now declare variable-size domain parameters and propagate them to components. Variable-size parameters are still not allowed in the component `equations` section outside of the `tablelookup` function. For more information, see [Using Lookup Tables in Equations and Propagation of Domain Parameters](#).

Vector and matrix physical signals

Simscape language now supports nonscalar (vector-valued or matrix-valued) physical signals in its `inputs` and `outputs` declarations. All signals in such vector or matrix should have the same units. For example, the following declaration

```
inputs
  I = {zeros(3), 'm/s'}; % :left
end
```

initializes a component input as a 3-by-3 matrix of linear velocities.

Simulink-PS Converter and PS-Simulink Converter blocks have also been enhanced to handle vector and matrix physical signals.

Foundation Library and Simulation

Random Number and Uniform Random Number blocks

Two new Physical Signal blocks have been added to the Sources library:

- The Random Number block generates normally (Gaussian) distributed random numbers.
- The Uniform Random Number block generates uniformly distributed random numbers.

The block behavior is the same as that of the respective Simulink blocks, except that they generate a physical signal rather than a unitless Simulink signal.

Perfect Insulator block for thermal domain

The new Perfect Insulator block in the Thermal Elements library models a thermal element with no thermal mass and perfect insulation. Use this block as an insulation for thermal ports to prevent heat exchange with the environment and to model an adiabatic process.

Initialization and diagnostic improvements

When you use local solver in a model, an alternative initialization technique is automatically employed if the standard initialization techniques fail. This alternative technique attempts to find consistent states, within numerical tolerance, by taking a small finite step from the user-specified initial states. Therefore, if the alternative technique succeeds, a warning is issued to the command line that user-specified initial conditions may not be satisfied. Employing this alternative technique increases the initialization robustness, especially when there are nonlinear constraints among dynamic states of a model.

The missing reference node diagnostics have been improved to include information about the particular block and variable that needs a reference node. This is especially helpful when multiple domains are involved in the model.

Model Advisor checks for outdated blocks and for physical unit consistency

Model Advisor user interface now contains two checks specific to Simscape models:

- **Check consistency of block parameter units** notifies you about differences between the declared and the actual settings of block parameter units. The check is triggered by default when you run Model Advisor on your model. You can also run this check individually by selecting **By Product | Simscape** or **By Task | Modeling Physical Systems**. The check detects block parameters in which the specified unit is not directly convertible into the default unit expected by the block. For example, it alerts you if a block parameter is declared with the unit of rad/s but the value specified in your pre-R2013a model is in Hz. This situation can be problematic because of the new unit definition for Hz in R2013a (see “Unit definition of Hz now consistent with SI” on page 8-5).

After you run the check, a table of results appears in the right pane of the Model Advisor window. Each cell in the first column of the table contains a link to the problematic block, and the corresponding cell in the second column contains the name of parameter in question, the expected unit, and the specified unit.

Clicking on a link highlights the corresponding block in the model. Double-click the highlighted block, verify the parameter unit setting and correct it, if necessary. Then save and reload the model.

- **Check for outdated Simscape blocks** detects a pre-R2013a version of AC Voltage Source and AC Current Source blocks in your model. The check is triggered by default when you run Model Advisor. You can also run this check individually by selecting **By Product | Simscape**.

After you run the check, a list of links to the outdated blocks appears in the right pane of the Model Advisor window. Clicking on a link highlights the corresponding block in the model.

To update the blocks, scroll down the right pane of the Model Advisor window and click the **Update** button.

- If the automatic update is successful, the Results box displays a message that all blocks have been updated to the current Simscape version.

- If the message says that some of the blocks could not be updated automatically, rerun the check and manually replace the outdated blocks with the latest version from the block library.

For more information on using Model Advisor, see Consulting Model Advisor.

Unit definition of Hz now consistent with SI

The unit definition for Hz is now $1/s$, in compliance with the SI unit system. In previous releases, the unit definition for Hz was rev/s , consistent with the definition of frequency as cycles per second in an electrical context, or revolutions per second in a mechanical context. The old unit definition allowed you to specify frequency in angular units (such as rad/s or rpm) and write frequency-dependent equations without requiring the 2π conversion factor. The main reason for changing the Hz unit definition is to give the block author responsibility for frequency units and their correct interpretation for that block. While a sinusoidal source might reasonably be given a frequency in units of Hz or rpm , angular frequency has no relevance when frequency refers to a nonrotational periodic signal such as the frequency of a PWM source.

As a result of the new unit definition for Hz, frequency units and angular velocity units are no longer directly convertible, and using one instead of the other may result in unexpected conversion factors applied to the numerical values by the block equations.

Drop-down lists of suggested units in block dialogs have been updated to reflect this change. For example, if your block has a **Frequency** parameter with the default unit of HZ, the drop-down list for this parameter now contains only units directly convertible to Hz (such as kHz, MHz and GHz) and does not contain the angular velocity units (such as rpm , deg/s and rad/s). You can still type a unit expression representing angular frequency into the units combo box, and the block will accept it as commensurate with the expected parameter unit, but it is your responsibility to make sure that the specified unit works correctly with the block equations.

For more information, see Units for Angular Velocity and Frequency.

Compatibility Considerations

Two Foundation library blocks, AC Current Source and AC Voltage Source, have been affected by this change. In previous releases, you could specify the **Frequency** parameter for these blocks either in units of HZ or in angular units, such as rad/s or rpm . Starting with Release R2013a, you must specify the **Frequency** parameter in units

of Hz or directly convertible to Hz (such as 1/s, kHz, MHz and GHz) because the internal equation of the block now uses the 2π conversion factor to account for the 1/s unit definition.

If you have a pre-R2013a model that contains these blocks, update it by running the Model Advisor check, **Check for outdated Simscape blocks**, or by using the `slupdate` utility, and then save the model. A related Model Advisor check, **Check consistency of block parameter units**, notifies you about differences between the declared and the actual settings of block parameter units. For more information, see “Model Advisor checks for outdated blocks and for physical unit consistency” on page 8-4.

If you have custom Simscape libraries written in R2012b or earlier, and you have used Hz, kHz, MHz, and GHz as parameter units, then you will need to update your Simscape code to take account of the Hz unit change. Previously the Simscape unit manager automatically converted any value entered in units of Hz into units of rad/s before computation. Therefore, now you need to introduce a factor of 2π into block equations to convert to rad/s and retain the old functionality.

New format for saving simulation data log objects

Data logging functionality lets you log simulation data to the workspace, in the form of a workspace variable. As with any workspace variable, you can save the data log to a MAT-file, and then load the file into workspace at a later date to query and analyze the data.

Starting with Release R2013a, a new format for saving the data log objects in a MAT-file has been introduced. The new format reduces the disk space usage and memory consumption, and makes it faster to save and load simulation data logs.

Compatibility Considerations

There is no backward incompatibility. That is, you can load previously saved MAT-files containing simulation data with no restrictions.

However, there is a forward incompatibility. MAT-files containing simulation data log objects saved in the new format (starting with Release R2013a) cannot be opened in older versions of MATLAB software (Release R2012b or earlier).

New examples

The following example has been introduced in this version:

- Engine Cooling System

Functionality being removed or changed

Simscape Language Keyword Name	What Happens When you use the Keyword?	Use This Instead	Compatibility Considerations
throughs	The change was introduced in Release R2009a. The keyword has now been removed, and using the old syntax produces an error, instead of a warning, when you attempt to build the component.	variables (Balanci	See “Simscape Language Syntax Changes” on page 16-3 in Release R2009a.

R2012b

Version: 3.8

New Features

Bug Fixes

Compatibility Considerations

Simscape Language

Connection of components within a Simscape file

In physical modeling, there are two types of models:

- Behavioral — A model that is implemented based on its physical behavior, described by a system of mathematical equations. An example of a behavioral block implementation is the Variable Orifice block.
- Composite — A model that is constructed out of other blocks, connected in a certain way. An example of a composite, or structural, block implementation is the 4-Way Directional Valve block (available with SimHydraulics block libraries), which is constructed based on four Variable Orifice blocks.

In previous versions, Simscape language supported only behavioral models, that is, models defined by equations. To create a model containing multiple interconnected components, you had to define each component in a separate file, deploy it as a custom block, and then use masked subsystems in block diagrams to connect these blocks into a single composite model.

Now, additional language constructs let you create composite models directly in a Simscape file. A component file may now contain two additional blocks:

- A **components** declaration block, which begins with a **components** keyword and is terminated by an **end** keyword. This block contains declarations for all the constituent components. Each component is defined with its full path to the top-level package directory. Specify the required component parameters by declaring a corresponding parameter in the top-level **parameters** declaration block, and then passing this value on to the constituent component.

For example, the following code includes a Foundation library Resistor block in your custom component file, with a default resistance of 10 Ohm:

```
component MyCompositeModel1
...
parameters
    p1 = {10, 'Ohm'};
    ...
end
components(Hidden=true)
    r1 = foundation.electrical.elements.resistor(R=p1);
```

```

    ...
end
...
end

```

- A `connections` block, located after the `setup` section, which begins with a `connections` keyword and is terminated by an `end` keyword. This block contains information on how the constituent components' ports are connected to one another, and to the external inputs, outputs, and nodes of the top-level component.

For example, the following code includes the Foundation library Voltage Sensor and Electrical Reference blocks in your custom component file, connects the negative port of the voltage sensor to ground, and connects the physical signal output port of the voltage sensor to the external output of the composite component, located on the right side of the resulting block icon:

```

component MyCompositeModel2
...
  outputs
    Out = { 0.0, 'V' }; % V:right
    ...
  end
  components(Hidden=true)
    VoltSensor = foundation.electrical.sensors.voltage;
    Grnd = foundation.electrical.elements.reference;
    ...
  end
  connections
    connect(Grnd.V, VoltSensor.n);
    connect(VoltSensor.V, Out);
    ...
  end
end

```

For more information, see [Creating Composite Components](#).

floor, ceil, fix, and round functions

You can now use the following MATLAB functions in the `equations` section of the Simscape file:

- `floor` performs rounding toward negative infinity.
- `ceil` performs rounding toward positive infinity.

- `fix` performs rounding toward zero.
- `round` performs rounding toward the nearest integer.

The PS Floor, PS Ceil, and PS Fix blocks in the Physical Signals/Nonlinear Operators sublibrary of the Foundation library have been reimplemented using the Simscape language and the corresponding function. The PS Round block has been added to the Nonlinear Operators sublibrary. It performs rounding toward the nearest integer.

Foundation Library and Simulation

Speed and efficiency improvements for simulation of switched linear systems

Switched linear systems are systems that have multiple configurations during simulation, but each configuration is linear. The changes in configuration during simulation may be due to deployment of switches or other elements. A new specialized simulator, implemented for switched linear systems, reduces the number of states and accelerates simulation. The specialized simulator is automatically employed based on the system structure; you do not have to select or enable it explicitly.

Zero-crossing statistics for Simscape logging

If you log simulation data for a Simscape model, you now have an option to log simulation statistics, including zero-crossing data. By default, the zero-crossing data is not logged. If you select the **Log simulation statistics** check box on the **Simscape** pane of the Configuration Parameters dialog box, the simulation log variable contains an additional `SimulationStatistics` node for each block that can produce zero crossings. You can then plot and analyze this data similar to other data logged to the workspace during simulation.

Counter and Repeating Sequence blocks that facilitate discrete sampling

Two new blocks in the Physical Signals/Sources library facilitate discrete sampling in physical modeling:

- The Counter block repeatedly increments the output signal by 1 with every time step, in the range between the minimum (reset) value and the maximum value. You can optionally specify an initial signal value, different from the reset value, and an initial time offset. Use this block, in conjunction with other physical signal blocks, to model discrete behaviors.
- The Repeating Sequence block outputs a periodic piecewise-linear signal. You can optionally specify an initial signal value and an initial time offset. The repeating sequence consists of a number of linear segments, connected to each other. Use this block to generate various types of physical signals, such as pulse, sawtooth, stair, and so on.

Open-circuit terminator blocks for electrical, hydraulic, and mechanical domains in Foundation library

Physical Network block diagrams do not allow unconnected Conserving ports. Previously, if you wanted to leave a port unconnected (open-circuit) you had to add an extra sensor, which cluttered the block diagram.

Now the following blocks represent domain-specific open-circuit terminators:

- The Open Circuit block represents an electrical terminal that draws no current. Use this block to terminate electrical ports that you want to leave open-circuit.
- The Hydraulic Cap block represents a hydraulic plug, that is, a hydraulic port with zero flow through it. Use this block to terminate hydraulic ports that you want to cap.
- The Rotational Free End block represents a mechanical rotational port that rotates freely, without torque. Use this block to terminate mechanical rotational ports that you want to leave unconnected.
- The Translational Free End block represents a mechanical translational port that moves freely, without force. Use this block to terminate mechanical translational ports that you want to leave unconnected.

Viewable and customizable source files for additional Foundation library blocks

In R2009a, many blocks in the Foundation library were implemented using the Simscape language. In 2012b, most of the remaining blocks have been converted.

You can now view the source files for most of the Foundation library blocks. When you open the block dialog box, it contains a link:

View source for *BlockName*

Click this link to open the Simscape source file for the block in the MATLAB Editor. To customize the block for your application, edit the source file and save it in a package directory. Some of the features, such as drop-down lists in block dialog boxes, are not yet available for custom blocks. For more information on packaging Simscape source files, see Simscape File Deployment.

Compatibility Considerations

The PS Math Function block now issues a simulation-time error when the input falls out of the expected domain for the particular function used. For example, if set to `sqrt`, the PS Math Function block now issues an error if it receives negative input during simulation.

New examples

Examples introduced in this version are:

- Lithium Battery Cell - One RC-Branch Equivalent Circuit
- Lithium Battery Cell - Two RC-Branch Equivalent Circuit
- Asynchronous PWM Voltage Source
- Discrete-Time PWM Voltage Source
- Turbojet Engine

Also, the existing Pneumatic Motor example has been modified to show how a pneumatic motor can be modeled using the Simscape language.

R2012a

Version: 3.7

New Features

Bug Fixes

Compatibility Considerations

Modeling Delays in Simscape Language

The new `delay` construct in Simscape language lets you refer to past values of expressions in the `equations` section of the Simscape file. For more information, see the delay reference page, and the new demo, Variable Transport Delay.

New Blocks for Modeling Delays

The new Delays sublibrary of the Physical Signals library contains two blocks:

- PS Constant Delay block returns the input signal delayed by a specified time, which is constant throughout the simulation. You specify the delay time as a block parameter.
- PS Variable Delay block delays the input signal by a variable time. You specify the delay time and the input history as additional inputs.

Use these blocks to model idealized transport phenomena at system level.

1D and 2D Interpolation Available in Simscape Language

The new `tablelookup` function in Simscape language lets you interpolate expressions in the `equations` section of the Simscape file. It computes an output value by interpolating the input value against a set of data points in a one-dimensional or two-dimensional table, and supports three interpolation and two extrapolation options. This functionality is similar to that of the Simulink and Simscape Lookup Table blocks. It allows you to incorporate table-driven modeling directly in your custom block, without the need of connecting an external Lookup Table block to your model. For more information, see the `tablelookup` reference page.

The PS Lookup Table (1D) and PS Lookup Table (2D) blocks in the Foundation library now use the `tablelookup` function in the Simscape language. The block functionality and user interface remain the same.

Input Filtering Usability Enhancements

In previous releases, input filtering was automatically turned on whenever you used an explicit solver in a Simscape model. To turn off input filtering when using an explicit solver, you had to supply first derivative of the input signal as an additional input signal to the Simulink-PS Converter block. For models using other types of solvers, input filtering was not available.

Now input filtering is independent of the solver used in the model. You can control whether you filter input or provide time derivatives for each input signal individually, by configuring the Simulink-PS Converter block connected to that input signal. You can:

- Set the **Filtering and derivatives** parameter to **Filter input**, and select whether you want to use the first-order or second-order filter. Input filtering makes the input signal smoother and generally improves model performance. The additional benefit is that the Simscape engine computes the time derivatives of the filtered input. The first-order filter provides one derivative, while the second-order filter provides the first and second derivatives. If you use input filtering, it is very important to select the appropriate value for the filter time constant.
- Set the **Filtering and derivatives** parameter to **Provide input derivative(s)**, and provide either just the first time derivative, or the first and the second time derivatives, through additional input ports on the Simulink-PS Converter block.

By default, input signals are used as is, without performing input filtering or otherwise providing time derivatives of the input signal. If you use an explicit solver, MathWorks recommends that you provide input derivatives by selecting one of the options listed above. If you do not provide input derivatives, and the solver you use requires them, you get an error message indicating how many input derivatives you need to provide. For more information, see *Harmonizing Simulink and Simscape Solvers and the Simulink-PS Converter block reference page*.

Compatibility Considerations

Input filtering is no longer automatically turned on for models using explicit solvers. Therefore if you have an existing model that uses an explicit solver, its performance and simulation results may be different from the previous version.

Because Simscape solver no longer automatically provides the required input derivatives, you may get an error message indicating how many input derivatives you need to provide.

To preserve the old behavior, open each Simulink-PS Converter block and set the **Filtering and derivatives** parameter to **Filter input**, while keeping the value of the **Input filtering time constant** parameter unchanged. If any of the Simulink-PS Converter blocks in your model had input derivatives provided as additional signals, set the **Filtering and derivatives** parameter for such block to **Provide input derivative(s)**.

Zero Damping Allowed for Resistive Elements

In previous releases, resistive blocks in the Foundation library required positive damping coefficients. Negative damping values are nonphysical. However, zero damping values are useful for model checking and testing. For example, if you use a Rotational Damper block as part of a customized gear model, it is convenient to be able to set the **Damping coefficient** parameter to 0 temporarily, to compute undamped responses.

The following blocks now allow zero damping values, to support model testing:

- Resistor (Electrical Elements library)
- Linear Hydraulic Resistance (Hydraulic Elements library)
- Rotational Damper (Rotational Elements library)
- Translational Damper (Translational Elements library)

Changes to Simscape Demos

The following demo has been added in Version 3.7:

Demo Name	Description
Variable Transport Delay (ssc_transport_delay)	Provides an example of modeling a variable transport delay using Simscape language. The Transport Delay subsystem models signal propagation through media moving between the Input and the Output terminals. The media velocity may vary, thus it is specified through the block port. The distance between the terminals is constant and it is specified as a block parameter. To see the implementation details, look under mask of the Transport Delay subsystem, then right-click the Variable Transport Delay block and select View Simscape source .

R2011b

Version: 3.6

New Features

Bug Fixes

Compatibility Considerations

Simscape Language Enables User-Defined Diagnostics During Simulation

The new `assert` construct in Simscape language lets you implement run-time error messages when the custom block is used in a model. In the component file, you specify the condition to be evaluated, as well as the error message to be output if this condition is violated. When the custom block based on this file is used in a model, it will output this error if the condition is violated during simulation. For more information, see [Programming Run-Time Errors and Warnings](#).

Compatibility Considerations

Run-time error checking has now been implemented in Foundation library, and may affect the behavior of your existing models. For example, in previous releases, `sqrt(x)` for negative numbers was calculated as $\sqrt{|x|} \cdot \text{sign}(x)$ (e.g., `sqrt(-1) = -1`). Now it will give a run-time error if the input becomes negative.

Run-time error checks have been implemented for the following functions:

Function	Condition
<code>sqrt(x)</code>	$x < 0$
<code>log(x)</code>	$x \leq 0$
<code>log10(x)</code>	$x \leq 0$
<code>1 / x</code>	$x == 0$, if x is an expression involving only constants and parameters

The following Foundation library blocks are affected:

- **PS Math Function** block — when the inputs of `log`, `log10`, `sqrt`, and `1/u` violate the conditions listed above, the block will now generate a run-time error.
- **PS Divide** block — when the denominator is zero, the block will now generate a run-time error.

Existing models using these blocks, which ran successfully in previous releases due to lack of error checking, will now generate a run-time error if one of the above conditions is violated.

New Block for Modeling Discrete Delays

The Asynchronous Sample & Hold block, in the new Discrete sublibrary of the Physical Signals library, sets output Y equal to input U when the rising edge of the trigger input becomes greater than zero. Use this block, in conjunction with other physical signal blocks, to model discrete and event-based behaviors.

Specialized Simulator for Linear Systems

The new specialized simulator, implemented for linear systems, reduces number of states and accelerates simulation. The specialized linear simulator is automatically employed based on the system structure, you do not have to select or enable it explicitly.

Rebuilding of Custom Block Libraries Now Required

Due to further scalability improvements, as well as other internal enhancement to facilitate add-on product functionality, you have to rebuild your custom block libraries once you upgrade to Version 3.6 (R2011b). It is required that you rebuild your custom block libraries for use with each new version of Simscape software. For more information, see [When to Rebuild the Custom Library](#).

To rebuild the libraries, run `ssc_build` on the component Simscape files. If you try to use the custom blocks without rebuilding the libraries, you will get an error message.

Running `ssc_clean` before `ssc_build` is strongly recommended but not required. If you run into errors after running `ssc_build`, run `ssc_clean` and then try running `ssc_build` again.

R2011a

Version: 3.5

New Features

Bug Fixes

Compatibility Considerations

Additional Hydraulic Sources

Two new blocks have been added to the Hydraulic Sources library:

- Hydraulic Constant Flow Rate Source block represents an ideal source of hydraulic energy that is powerful enough to maintain specified flow rate at its outlet regardless of the pressure differential across the source. The **Source flow rate** parameter specifies the flow rate through the source.
- Hydraulic Constant Pressure Source block represents an ideal source of hydraulic energy that is powerful enough to maintain the specified pressure differential between its inlet and outlet regardless of the flow rate through the source. The **Pressure** parameter specifies the pressure differential across the source.

Use these blocks for models where flow rate or pressure remain constant throughout simulation.

Improved Simulation Diagnostics

The following improvements in simulation diagnostics have been implemented, to aid debugging:

- Equation dependency diagnostics now point to the specific equations (with line number and file location info) within the component Simscape files.
- Equation dependency diagnostics have been extended to include switched-linear and nonlinear equations in the analysis.
- Equation dependency diagnostics are now triggered on nonlinear solver failures that occur after the start of simulation.
- Zero-crossing related warnings and error messages now point to the specific equations (with line number and file location info) within the component Simscape files.

Improved Scalability

Various scalability improvements have been implemented in this release, accelerating simulation of larger systems.

Compatibility Considerations

Due to this change, you have to rebuild your custom block libraries once you upgrade to Version 3.5 (R2011a). To rebuild the libraries, run `ssc_build` on the component

Simscape files. If you try to use the custom blocks without rebuilding the libraries, you will get an error message.

Running `ssc_clean` before `ssc_build` is strongly recommended but not required. If you run into errors after running `ssc_build`, run `ssc_clean` and then try running `ssc_build` again.

Improved Algorithms for Algebraic Loop Detection and Zero-Crossing Robustness

The following simulation algorithm improvements have been implemented in this release:

- False algebraic loop detection and prevention—The improved algorithm can now recognize false algebraic loops and prevent them from affecting simulation results.
- Performance and zero-crossing robustness improvements—Zero-crossing detection algorithm has been optimized to ignore zero-crossings that do not result in model behavior changes during simulation.

Change in Evaluating Unit Expressions

In the Unit Manager parser, multiplication (*) used to mistakenly have higher precedence than division (/). This issue is now fixed. When evaluating unit expressions, * and / now have the same precedence and are evaluated based on left associativity.

Compatibility Considerations

Due to this change, custom units specified in the unit registry may now evaluate differently. For example, if you have added a unit $m/s*s$, in previous releases it was evaluated as $m/(s*s) = m/s^2$. It will now evaluate to $(m/s)*s = m$. You can use parentheses to preserve the old behavior.

R2010b

Version: 3.4

New Features

Bug Fixes

Compatibility Considerations

Simulation Data Logging Enhancements

The following data logging enhancements have been implemented in this version:

- `plot` and `plotxy` commands have been added. The `plot` command lets you plot logged data against time, while the `plotxy` command plots two sets of data against each other. Also, `plot` and `plotxy` methods are now available for `simscape.logging.Node` and `simscape.logging.Series` objects. For more information, see the respective reference pages.
- New configuration parameter, **Decimation**, lets you downsample logged data by skipping time steps. For more information, see Data Logging Options in the *Simscape User's Guide*.
- Simulation data is now logged according to the value specified for the **Output options** parameter in the **Data Import/Export** pane of the Configuration Parameters dialog box.

Compatibility Considerations

For models that do not use the default value of the **Output options** parameter in the **Data Import/Export** pane of the Configuration Parameters dialog box, logged simulation data may change compared to the previous release.

Zero-Crossing Handling Enhancements

The creation and detection of zero-crossing conditions in Simscape models have been improved.

New Implementation of Relational Functions Without Creating Simulink Zero-Crossing Conditions

The Simscape language now implements relational functions in a different way from how relational operators are implemented. Unlike relational operators, these relational functions now do not create Simulink zero-crossing conditions.

For more information about zero crossings in Simscape models, see *Creating and Detecting Zero Crossings in Simscape Models* in the *Simscape User's Guide*.

For more information about creating models in the Simscape language, see the *Simscape Language Guide* and the equations syntax.

Improved Zero-Crossing Detection Diagnostics

If your model generates a Simulink warning or error about zero crossings involving Simscape blocks, the warning or error message now specifies which blocks are generating this zero-crossing diagnostic message.

If you globally disable zero-crossing detection in a Simulink model containing Simscape blocks, and if you are using a variable-step solver without a local solver, you now receive a diagnostic warning or error. You can control which message that you receive in the **Simscape** pane of the model Configuration Parameters dialog box, through the **Zero-crossing control is globally disabled in Simulink** drop-down list. This option supports context-sensitive or “What's This?” help, by default accessed through right-clicking the item.

See Harmonizing Simulink and Simscape Solvers in the Simscape User's Guide.

C++ Code Generation Support

You can now generate C++ code from Simscape models. Encapsulated C++ code generation is not supported.

For more information, see Code Generation and Limitations in the Simscape User's Guide.

Sparse Solver Enhancement

The implementation and control of matrix linear algebra in the Solver Configuration block have been improved and simplified. Whether you choose the sparse solver or the full solver, your linear algebra choice is now implemented in both model simulation and code generated from the model.

For more information, see the Solver Configuration block reference page.

Component Descriptor Is No Longer Inherited from the Base Class

In Simscape Language, the name of the block built from a component file generally corresponds to the component file name. You can provide a more descriptive name for the block by adding a comment line immediately following the component declaration. This comment line is called the descriptor. For more information, see How to Customize the Block Name.

If you use subclassing, the subclass inherits all of the members (parameters, variables, nodes, inputs and outputs) from the base class (for more information, see Subclassing and Inheritance). In previous releases, descriptor was one of the inherited properties. That is, if you had a subclass component without a descriptor, the resulting block name was derived from the descriptor of the base class, instead of the component file name. This could create issues with library building if both the subclass component and the base class component were in the same sublibrary.

Starting with this release, descriptor is no longer inherited from the base class. In other words, if a component file does not contain a comment line immediately following the component declaration, the component name is always used for the block name.

Compatibility Considerations

For existing subclass components, if the component does not have a descriptor line, upon rebuilding the library the block name will change compared to the previous release. To preserve the old name, add a descriptor line to the subclass component file.

Documentation Enhancements

The Model Simulation chapter has been expanded and improved with revised and new sections, including:

- How Simscape Models Represent Physical Systems
- How Simscape Simulation Works
- Setting Up Solvers for Physical Models
- Customizing Solvers for Physical Models
- Real-Time Simulation
- Finding an Operating Point
- Linearizing at an Operating Point
- Finding and Using Operating Points in an Electronic Circuit

Changes to Simscape Demos

The following demo has been added in Version 3.4:

Demo Name	Description
-----------	-------------

Electrical Transformer (ssc_transformer)	Presents a view inside a transformer core using the electromagnetic blocks from the Magnetic block library.
---	---

R2010a

Version: 3.3

New Features

Bug Fixes

Compatibility Considerations

Magnetic Blocks Added to Foundation Library

Foundation library now contains magnetic domain and Magnetic block library. This library contains electromagnetic elements, such as reluctances, actuators, and electromagnetic converters, as well as magnetic sensors and sources.

Use these blocks to model magnetic circuits that can be represented by a one-dimensional flux flow, for example, solenoids and transformers.

Magnetic block models are based on the following assumptions:

- The magnetic system is assumed lossless. You can model losses in interconnecting systems instead – in electric systems using resistors and in mechanical systems using friction.
- Modeling of superconductors (with zero relative permeability) is not supported.

For more information see the block reference pages. See also Magnetic Domain for information on the magnetic domain definition. The Across variable is magnetomotive force (mmf), and the Through variable is flux. Note that these result in a pseudo-bond graph, because the product of mmf and flux is energy, not power.

Simulation Data Logging Now Available

You can now log simulation data to workspace for debugging and verification purposes. Data logging lets you analyze how internal block variables change with time during simulation. For example, you may want to see that the pressure in a hydraulic cylinder is above some minimum value, or compare it against the pump pressure. If you log simulation data to workspace, you can later query, plot, and analyze it without rerunning the simulation. For more information, see Data Logging in the *Simscape User's Guide*.

Simscape Diagnostics Improvements

Simscape error messages triggered by initial dynamic state inconsistencies and by nonlinear solver convergence failures are now more detailed. These messages report specific components of your models that may have caused the error.

Sparse Jacobian Support

You can now use a Jacobian method with an implicit Simulink solver in your Simscape models. You can choose a method yourself or allow Simulink to determine an appropriate

Jacobian method for you. Depending on the sparsity pattern and number of states of your model, your simulation may be more efficient.

See [Choosing a Jacobian Method for an Implicit Solver](#) in the Simulink documentation.

Ability to Generate Simscape Language Equations from Symbolic Expressions

If you have Symbolic Math Toolbox™ software, you can use the `simscapeEquation` function to generate Simscape language equations from symbolic expressions. For more information, see [Generating Simscape Equations in the Symbolic Math Toolbox documentation](#).

Placing Simscape Blocks in Nonvirtual Subsystems

Nonvirtual subsystems that support continuous states include Enabled subsystems and Atomic subsystems. These subsystems can contain Simscape blocks. However, physical connections and physical signals must not cross nonvirtual boundaries. For more information, see [Restricted Simulink Tools](#) in the *Simscape User's Guide*.

Compatibility Considerations

Simscape solver no longer permits physical connections and physical signals to cross nonvirtual subsystem boundaries, because the semantics of these types of connections are unclear. If either a physical signal or a physical connection crosses a nonvirtual boundary, the solver issues an error upon simulation. To resolve the issue, place all blocks belonging to a given Physical Network in the same nonvirtual subsystem.

Trimming and Linearization Documentation Enhancements

The documentation on Simscape model trimming and linearization has been revised and expanded. See [Finding an Operating Point and Linearizing at an Operating Point](#).

Changes to Simscape Demos

The following demos have been added in Version 3.3:

Demo Name	Description
-----------	-------------

Circuit Breaker (ssc_circuitbreaker)	Implements a simple circuit breaker model.
Solenoid with Magnetic Blocks (ssc_solenoid_magnetic)	Shows how to model a solenoid using the electromagnetic blocks from the new Magnetic block library.

R2009b

Version: 3.2

New Features

Bug Fixes

Compatibility Considerations

Pneumatic Blocks Added to Foundation Library

Foundation library now contains pneumatic domain and Pneumatic block library. This library contains pneumatic elements, such as orifices, chambers, and pneumatic-mechanical converters, as well as pneumatic sensors and sources.

Use these blocks to model pneumatic systems, for applications such as:

- Factory automation — basic pneumatic linear/rotational actuators, valves (variable orifices), and air supply
- Robotics — robotic arms and haptic interfaces
- Gaseous transportation systems and pipelines

You can also use these blocks to model dry air and low pressure flows, for example, for HVAC applications.

Pneumatic block models are based on the following assumptions:

- Working fluid is an ideal gas satisfying the ideal gas law.
- Specific heats at constant pressure and constant volume, c_p and c_v , are constant.
- Processes are adiabatic, that is, there is no heat transfer between components and the environment (except for components with a separate thermal port).
- Gravitational effects are neglected.

For more information see the block reference pages, as well as Modeling Pneumatic Systems in the *Simscape User's Guide*.

Compatibility Considerations

To avoid duplicate block names in different Simscape domains and increase naming consistency across domains, the following hydraulic blocks have been renamed:

Old Block Name	New Block Name
Constant Area Orifice	Constant Area Hydraulic Orifice
Constant Volume Chamber	Constant Volume Hydraulic Chamber
Piston Chamber	Hydraulic Piston Chamber
Resistive Tube	Hydraulic Resistive Tube
Variable Area Orifice	Variable Area Hydraulic Orifice

Old Block Name	New Block Name
Variable Chamber	Variable Hydraulic Chamber
Ideal Hydraulic Flow Rate Sensor	Hydraulic Flow Rate Sensor
Ideal Hydraulic Pressure Sensor	Hydraulic Pressure Sensor
Ideal Hydraulic Flow Rate Source	Hydraulic Flow Rate Source
Ideal Hydraulic Pressure Source	Hydraulic Pressure Source

Old models containing any of these blocks will be updated automatically once you open and save them.

New and Enhanced Switches

The following switching capability enhancements have been implemented in Foundation libraries:

- New Physical Signal PS Switch block has been added to the Nonlinear Operators library. It contains three physical signal input ports, a physical signal output port, and one parameter, **Threshold**. If the second input is greater than or equal to the threshold, then the output is connected to the first input. Otherwise, the output is connected to the third input. The second input never connects to the output.
- Electrical Switch block has been enhanced to use a value specified in the **Threshold** parameter (rather than zero) for opening and closing the switch.

Intermediate Terms in Simscape Language Equations

You can now introduce intermediate terms in Simscape Language equations by using the `let` and `in` keywords. This functionality helps increase the equation readability, as well as avoid duplicating information by defining an intermediate term once and then using it in multiple equations. For more information, see the Simscape Language Guide.

Local Solver Support in Physical Networks

The Solver Configuration block now lets you use sample-based local solver with a specific sample time. In sample-based simulation, all the Physical Network states, otherwise represented as continuous, become discrete states. The solver updates the states once per time step. This option is especially useful for code generation, or hardware-in-the-loop (HIL) simulations. For more information, see the Solver Configuration reference page.

Simulink Manifest Tool Support

Dependency analysis tools for Simscape files have been added in this release. They consist of the following command-line options:

- `simscape.dependency.file` — Perform dependency analysis for a single Simscape file.
- `simscape.dependency.lib` — Perform dependency analysis for a Simscape custom library.
- `simscape.dependency.model` — Perform dependency analysis on a model containing Simscape and Simulink blocks.

Manifest reports generated using Simulink Manifest Tools now also include model dependencies for the Simscape blocks. For more information, see *Checking File and Model Dependencies* in the *Simscape Language Guide*.

SimState Support

Simscape software now supports Simulink SimState feature, introduced in R2009a. This feature allows you to save all runtime data necessary for restoring the simulation state of a model. For more information, see *Saving and Restoring the Simulation State* as the SimState in the *Simulink User's Guide*.

Note When using SimState to save and restore simulations of models involving Simscape blocks, please ensure both 'DstWorkspace' and 'SrcWorkspace' to be 'base' by using:

```
simset('DstWorkspace', 'base', 'SrcWorkspace', 'base')
```

Model Reference Accelerator Mode Support

Simscape and its add-on products now fully support Model Reference Accelerator Mode, both for model simulation and for code generation.

Physical Port Rotation for Simscape Blocks

When you rotate a regular Simulink block, its ports are by default reordered after rotation, to maintain the left-right and top-down block diagram orientation convention

used in control system modeling applications. This convention is not applicable to physical modeling and is potentially confusing, because it results in effectively rotating and flipping the block at the same time.

Therefore, starting with Version 3.2 (R2009b), when you rotate a Simscape block (including blocks from add-on products), its ports are not reordered. This behavior is similar to that of the masked blocks with **Port Rotation** set to **Physical**. For illustration of differences between the *default port rotation* type and the *physical port rotation* type, see *Changing a Block's Orientation* in the *Simulink User's Guide*.

Compatibility Considerations

This change in the behavior of the ports after block rotation may result in visually crossed connection lines in some of your existing block diagrams with rotated blocks. The effect is purely cosmetic and has no impact on actual model connections or simulation.

Changes to Simscape Demos

The following demos have been added in Version 3.2:

Demo Name	Description
Pneumatic Actuation Circuit (ssc_pneumatic_actuator)	This demo shows how the Foundation Library pneumatic components can be used to model a controlled pneumatic actuator. The Directional 5-way valve, Double-acting pneumatic actuator and Pipe blocks are masked subsystems created from Foundation Library blocks.
Pneumatic Motor (ssc_pneumatic_motor)	This demo shows how the Rotational Pneumatic-Mechanical Converter block can be used to approximate the behavior of a pneumatic vane motor.
Pneumatic Motor and Directional Control Valve (ssc_pneumatic_motor_and_valve)	This model shows the pneumatic vane motor, as defined in the Pneumatic Motor demo, deployed in a typical pneumatic circuit.

R2009a

Version: 3.1

New Features

Bug Fixes

Compatibility Considerations

Simscape Language Source Protection

Simscape language files can be protected to enable model sharing without disclosing the component or domain source. You can then share the protected (executable) files without disclosing the file content (similar to P-code vs M-code). While Simscape source files have the extension `.ssc`, Simscape protected files have the extension `.sscp`.

Use the `ssc_protect` command to protect individual files and directories.

Use the `ssc_mirror` command to create a protected copy of a whole package, along with a custom block library built from it.

For more information, see [Using Source Protection for Simscape Files](#).

Expanded MATLAB Support of Simscape Files

MATLAB support of Simscape files has been expanded:

- If you issue the `open` command on a Simscape file, the file will open in the MATLAB Editor. The Simscape file must be on the MATLAB path, or in a package residing in a directory on the MATLAB path. For more information on packaging Simscape files, see [Organizing Your Simscape Files](#).

If you issue the `open` command on a Simscape protected file (`*.sscp`), the corresponding Simscape source file (`*.ssc`) will open, provided it exists in the same directory as the Simscape protected file.

- Issuing the `help` command on a Simscape file displays the domain or component description, that is, all the comments immediately following the domain or component declaration, in the MATLAB Command Window.
- MATLAB Editor now supports syntax highlighting of Simscape files, similar to M-files. For more information, see [Adjust Editor Appearance](#).

Viewable and Customizable Source Files for Foundation Library Blocks

You can now view the source files for many Foundation library blocks. When you open the block dialog box, it contains a link:

View source for *BlockName*

Click this link to open the Simscape source file for this block in the MATLAB Editor. To customize the block for your application, edit the source file and save it in a package directory. For more information, see Simscape File Deployment.

Compatibility Considerations

The block source has been optimized, with some previously defined but unused variables eliminated. Therefore, when you load an old model containing Foundation blocks, you might get warnings, for example:

```
Warning: In instantiating linked block 'model/R1' : Resistor block (mask) does not
have a parameter named 'current_Log'.
```

You can safely ignore these warnings. Once you save the model, the warnings will disappear.

Simscape Language Syntax Changes

The following changes have been implemented in Simscape language:

- The `throughs` keyword has been obsoleted. Use `variables(Balancing=true)` to declare Through variables in a domain.
- The `equation` keyword has been changed to `equations`.
- The name of a Simscape file must match the name of the component or domain it defines. If this is not the case, you will get an error when trying to build a library or use the block in a model.

For more information, see the Simscape Language Guide.

Compatibility Considerations

The changes are relatively minor, but may require modifying your existing Simscape files. The following table summarizes the old and new syntax.

Old Syntax	New Syntax
<code>throughs</code>	<code>variables(Balancing=true)</code>
<code>equation</code>	<code>equations</code>

Increased Efficiency of Simscape Language Equations Processing

Simscape language equations are now processed more efficiently, reducing the time required to process equations with multiple `if` statements.

New Physical Signal Blocks to Facilitate Rounding

Three new Physical Signal blocks have been added to the Nonlinear Operators library:

- PS Ceil block performs rounding of the signal toward positive infinity
- PS Floor block performs rounding of the signal toward negative infinity
- PS Fix block performs rounding of the signal toward zero

Model Reference Accelerator Mode Support

Simscape and its add-on products now support Model Reference Accelerator Mode for model simulation, but not for code generation. Model Reference Accelerator Mode for code generation is supported only by SimMechanics and SimDriveline™ software.

Changes to Simscape Demos

The following demo has been added in Version 3.1:

Demo Name	Description
Creating A New Circuit (ssc_new_elec)	Use this demo as a template for creating a new electrical model. Open the demo and use File > Save As to save it under the desired model name. Then delete the unwanted components and add new ones. This demo also opens an Electrical Starter Palette, which contains links to the most often used electrical components.

R2008b

Version: 3.0

New Features

Bug Fixes

Compatibility Considerations

Simscape Language

New Simscape language extends the Simscape modeling environment by enabling you to create new components that do not exist in the Foundation library or in any of the add-on products. It is a dedicated textual physical systems modeling language with the following characteristics:

- Derives from MATLAB and familiar to those who use MATLAB
- Contains additional constructs specific to physical modeling and excludes constructs that have nothing to do with physical modeling
- Incorporated into the Simscape modeling interface
- Not focused on algorithm development

The Simscape language is intended to make modeling physical systems easy and intuitive. It lets you create new physical domains and components as textual files and then use them in Simscape block diagrams to model the desired physical effects. For more information, see the Simscape Language Guide.

Explicit Solvers

It is now possible to choose any variable-step or fixed-step solver for models containing Simscape blocks. Note, however, that implicit solvers, such as ode14x, ode23t, and ode15s, are still a better choice for a typical model. In particular, for stiff systems, implicit solvers typically take many fewer timesteps than explicit solvers, such as ode45, ode113, and ode1.

By default, you will get a warning when using an explicit solver for a model containing Simscape blocks. For models that are not stiff, however, explicit solvers can be effective, often taking fewer timesteps than implicit solvers. Depending on the type of your model, you can configure your preferences to either turn off this warning (if your model is not stiff) or even change it into an error (to avoid inadvertent use of explicit solvers), by using the **Simscape** pane of the Configuration Parameters dialog box.

If you use an explicit solver, it requires time derivatives of the input signals. By default, needed input derivatives are provided by filtering the input through a low-pass filter. The derivative of the filtered input can then be computed by the Physical Networks simulation engine. The new **Derivatives** tab in the Simulink-PS Converter dialog box lets you turn off input filtering and instead provide the first derivative of input as an additional input signal to the Simulink-PS Converter block. For more information, see the Simulink-PS Converter block reference page.

Because input filtering can appreciably change the input signal and drastically affect simulation results if the time constant is too large, a warning is issued when input filtering is used. The warning indicates which Simulink-PS Converter blocks have their input signals filtered. This warning can also be turned off (or changed to an error) by changing the preferences on the **Simscape** pane of the Configuration Parameters dialog box.

New Ways to Model Variable Chambers

There are now two blocks that let you model fluid compressibility in variable chambers:

- Piston Chamber block lets you model fluid compressibility in a chamber created by the piston in a cylinder. It replaces the Variable Volume Chamber block, available in previous releases.
- Variable Chamber block lets you model fluid compressibility in variable volume chambers of any shape. The instantaneous value of the chamber volume is provided by using a physical signal port.

Compatibility Considerations

The Variable Volume Chamber block, available in previous releases, has been deprecated. It has been replaced by the Piston Chamber block in other (structural) blocks and in demos shipped with the product. If you have used the Variable Volume Chamber block in your models, it will continue to work. Going forward, however, use the Piston Chamber block to model fluid compressibility in cylinder chambers.

Model Reference Support

Simscape software now supports the Simulink model referencing functionality in Normal mode. Other Simulink models can now reference Simscape models in normal (non-code-generation) execution. Simscape models continue to be able to reference Simulink models (that do not contain Simscape blocks) in normal execution. See Limitations for more details.

More Solver Performance and Robustness Enhancements

Version 3.0 contains multiple further enhancements to simulation algorithms, resulting in improved robustness and reliability.

R2008a

Version: 2.1

New Features

Bug Fixes

Trimming Now Available for Simscape Models

Finding and managing operating points by trimming has been implemented for models that include Simscape and SimHydraulics blocks. Simulink Control Design™ product is required for using this functionality. For more information, see Finding an Operating Point in the Simscape documentation.

Thermal Unit Conversions Now Supported

You can now specify temperature for your thermal models in a variety of units, including degrees Celsius, Fahrenheit, and Rankine. The unit manager automatically handles conversions between thermal units.

Thermal units sometimes require an affine conversion, that is, a conversion that performs both multiplication and addition. In situations when you deal with a relative, rather than absolute, temperature, you need to convert using just the linear term. Thermodynamic variables in block dialogs are automatically tagged as appropriate and handled by the unit manager. However, when an input or output signal is related to thermodynamic variables and contains units of temperature, you must decide whether affine conversion needs to be applied. The Simulink-PS Converter and PS-Simulink Converter block dialogs now contain the **Apply affine conversion** checkbox. If you select it, the unit manager uses the affine conversion, otherwise it applies the default linear conversion.

For more information, see Thermal Unit Conversions in the Simscape documentation, as well as the Simulink-PS Converter and PS-Simulink Converter block reference pages.

The `pm_addunit` command has also been modified to support affine conversions. Its second argument, `conversion`, may now be either a positive real scalar or a 1x2 array. If this argument has two elements, then it is specifying an affine conversion, with the first element (a positive real number) being the linear conversion coefficient, and the second being the offset.

Enhancement to Specifying Units

Simscape block dialogs have drop-down combo boxes for units next to a parameter value. You can either select a unit from the drop-down list, or type a commensurate unit name (or a mathematical expression with unit names) directly into the units combo box of the block dialog. For more information, see How to Specify Units in Block Dialog Boxes in the Simscape documentation.

Similarly, the Simulink-PS Converter and PS-Simulink Converter block dialogs now contain a drop-down list, which is prepopulated with some common input or output units. You can either select a unit from the list or type a unit name, or a mathematical expression with unit names. Note that you must still match the unit type:

- For a PS-Simulink Converter block, these units must be commensurate with the units of the input physical signal coming into the block.
- Signal units that you specify in a Simulink-PS Converter block must match the input type expected by the Simscape block connected to it.

New Blocks

Version 2.1 contains two new blocks:

- Gyration block in the Electrical Elements library simulates an ideal gyrator, which can be used to implement an inductor with a capacitor.
- PS Abs block in the Physical Signals library returns absolute value of input signal.

Enhancements to Simulation Algorithms

Version 2.1 contains multiple enhancements to simulation algorithms, resulting in improved robustness and reliability.

“What’s This?” Context-Sensitive Help Available for Simulink Configuration Parameters Dialog

R2008a introduces “What's This?” context-sensitive help for parameters that appear in the Simulink Configuration Parameters dialog, including those on the **Simscape** pane. This feature provides quick access to a detailed description of the parameters, saving you the time it would take to find the information in the Help Browser.

To use the “What's This?” help, do the following:

- 1 Place your cursor over the label of a parameter.
- 2 Right-click. A **What's This?** context menu appears.

For example, the following figure shows the **What's This?** context menu appearing after a right-click on the **Start time** parameter in the **Solver** pane.



- 3 Click **What's This?**. A context-sensitive help window appears showing a description of the parameter.

New Simscape Demo

The following demo has been added in Version 2.1:

Demo Name	Description
House Heating System (ssc_house_heating_system)	The demo represents a simple house heating system consisting of a heater, thermostat, and a house structure with four thermally distinguishable parts: inside air, house walls, windows, and roof. You can investigate system behavior with the heating system turned on or off, and plot the heat cost and indoor versus outdoor temperatures.

R2007b

Version: 2.0

New Features

Bug Fixes

Code Generation Now Available for Simscape Models

Code generation has been implemented for models that include Simscape and SimHydraulics blocks. For more information, see [Code Generation](#) in the Simscape documentation.

New Thermal Block Libraries

Version 2.0 contains new block libraries of fundamental thermal elements, sensors, and sources:

- Conductive Heat Transfer
- Convective Heat Transfer
- Radiative Heat Transfer
- Thermal Mass
- Thermal Reference
- Ideal Heat Flow Source
- Ideal Heat Flow Sensor
- Ideal Temperature Source
- Ideal Temperature Sensor

Additional Physical Signal Blocks

The new Physical Signal blocks introduced in Version 2.0 are listed below:

- PS Constant
- PS Math Function
- PS Max
- PS Min
- PS Sign

Improved Simulation Performance

In Version 2.0, various solver improvements have led to improved simulation performance:

- Enhanced handling of dependent dynamic states (higher-index DAEs)

Simscape can now handle dependencies among the dynamic states as long as they are linear in the states and independent of time and inputs. This allows you, for example, to connect capacitors in parallel (even with their parasitic series resistances set to 0), inductors in series, and so on.

- Significant reduction of the number of equations, which substantially increased simulation speed

The typical speedup of your models is between 5 and 10 times. There are some models that are below and above this range. Also, the number of states and equations changed between releases. This means that you will have to reset any calculations that relied on the states (such as initial state setting).

The changes to the simulation technology are significant. You may find that some of your models may require different or tighter tolerances to converge, while others will require no change. Refer to the troubleshooting section in the User's Guide for help in finding the cause of a problem if simulation failed.

New Simscape Demos

The following demos have been added in Version 2.0:

Demo Name	Description
DC Motor Thermal Circuit (ssc_dc_motor_thermal_circuit)	The demo illustrates how the thermal behavior of a motor can be simulated in lumped parameters.
Round Rod Heat Conduction (ssc_round_rod_heat_conduction)	The demo illustrates the usage of thermal blocks for developing a model of a long iron rod that is heated with a heat source through its left face. The right face and the outer cylindrical surface are open to atmosphere, with a force heat convection.

R2007a

Version: 1.0

New Features

Compatibility Considerations

Product Introduction

Simscape software extends the Simulink product line with tools for modeling and simulating multidomain physical systems. It enables you to describe multidomain physical systems containing mechanical, hydraulic, and electrical components as physical networks.

Simscape key features are:

- Single modeling environment for modeling and simulating physical systems, such as mechanical, electrical, and hydraulic systems
- Foundation library of physical modeling building blocks and fundamental mathematical elements
- Connection blocks to bridge modeling domains
- Full simulation and limited editing capabilities for models built with SimMechanics, SimDriveline, or SimHydraulics blocks (no license for these products required as long as the products are installed)
- Ability to specify units of parameters and variables, with all unit conversion handled automatically

Simscape software can be used for a variety of automotive, aerospace, defense, and industrial equipment applications. Together with SimMechanics, SimDriveline, SimHydraulics, and SimPowerSystems™ (all available separately), Simscape lets you model complex interactions in electromechanical and hydromechanical systems.

Block Libraries Moved from SimHydraulics to Simscape

The Foundation and Utilities block libraries that used to be included in SimHydraulics (V1.0 and V1.1) are now part of Simscape product.

Compatibility Considerations

Several blocks that used to be in SimHydraulics V1.1 and are now part of Simscape software have undergone changes that have compatibility impact. These blocks are:

- Fluid Inertia
- Inertia
- Mass

- PS Integrator
- Rotational Spring
- Translational Spring

Each of these blocks has a parameter that specifies the initial condition for use in computing the block's initial state at the beginning of a simulation run. In this version, there is a difference in the way these initial conditions are computed, and as a result, the blocks work differently than they used to in the previous version. For details, see the block reference pages.

