

COMSOL Multiphysics

Release Notes

COMSOL Multiphysics Release Notes

© 1998–2018 COMSOL

Protected by patents listed on www.comsol.com/patents, and U.S. Patents 7,519,518; 7,596,474; 7,623,991; 8,219,373; 8,457,932; 8,954,302; 9,098,106; 9,146,652; 9,323,503; 9,372,673; and 9,454,625. Patents pending.

This Documentation and the Programs described herein are furnished under the COMSOL Software License Agreement (www.comsol.com/comsol-license-agreement) and may be used or copied only under the terms of the license agreement.

Support for implementation of the ODB++[®] Format was provided by Mentor Graphics Corporation pursuant to the ODB++ Solutions Development Partnership General Terms and Conditions. ODB++ is a trademark of Mentor Graphics Corporation.

COMSOL, the COMSOL logo, COMSOL Multiphysics, COMSOL Desktop, COMSOL Server, and LiveLink are either registered trademarks or trademarks of COMSOL AB. All other trademarks are the property of their respective owners, and COMSOL AB and its subsidiaries and products are not affiliated with, endorsed by, sponsored by, or supported by those trademark owners. For a list of such trademark owners, see www.comsol.com/trademarks.

Version: COMSOL 5.4

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries, contact Technical Support, or search for an address and phone number. You can also visit the Worldwide Sales Offices page at www.comsol.com/contact/offices for address and contact information.

If you need to contact Support, an online request form is located at the COMSOL Access page at www.comsol.com/support/case. Other useful links include:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/community
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM010001

C o n t e n t s

Chapter I: Release Notes

New Products	12
COMSOL Compiler™	12
The Composite Materials Module	12
COMSOL Multiphysics	13
General New Functionality	13
New Functionality in the Application Builder	13
New Functionality in COMSOL Server™	15
New and Improved General Functionality in COMSOL Multiphysics	15
New Functionality in the Physics Interfaces	17
New Geometry and Mesh Functionality	18
New and Updated Operators, Functions, and Definitions	19
New Functionality in Studies and Solvers	19
New and Improved Results and Visualization Functionality	22
New Java® API Methods	24
General Backward Compatibility Considerations	24
Backward Compatibility with Version 5.3a	25
Backward Compatibility with Version 5.3.	26
Backward Compatibility with Version 5.2a	27
Backward Compatibility with Version 5.2.	28
Backward Compatibility with Version 5.1.	29
Backward Compatibility with Version 5.0.	30
Backward Compatibility with Version 4.4.	31
Backward Compatibility with Version 4.3b	32
Backward Compatibility with Version 4.3a	33
Backward Compatibility with Version 4.3.	35
AC/DC Module	36
New Functionality in Version 5.4.	36
Backward Compatibility with Version 5.2 and older.	37
Backward Compatibility with Version 5.0 and older.	38

Acoustics Module	40
New Functionality in Version 5.4.	40
New and Updated Model Examples in 5.4	41
Backward Compatibility with Version 5.3a	42
Backward Compatibility with Version 5.3.	42
Backward Compatibility with Version 5.2.	42
Backward Compatibility with Version 5.1.	42
Backward Compatibility with Version 4.3a and Older	43
Backward Compatibility with Version 4.3.	43
Batteries & Fuel Cells Module	44
New Functionality in Version 5.4.	44
New and Updated Applications and Models in Version 5.4	46
Backward Compatibility with Version 5.3a	47
CAD Import Module, Design Module, and LiveLink™ Products for CAD	48
New Functionality in Version 5.4.	48
New Functionality in LiveLink™ for AutoCAD® in Version 5.4	48
New Functionality in LiveLink™ for Inventor® in Version 5.4	48
New Functionality in LiveLink™ for PTC® Creo® Parametric™ in Version 5.4.	49
New Functionality in LiveLink™ for PTC® Pro/ENGINEER® in Version 5.4.	50
New Functionality in LiveLink™ for Revit® in Version 5.4	50
LiveLink™ for Solid Edge®	50
LiveLink™ for SOLIDWORKS®	51
Backward Compatibility with Previous Versions	51
CFD Module	53
New Functionality in Version 5.4.	53
New Model in Version 5.4.	54
Backward Compatibility with Version 5.3a	54
Backward Compatibility with Version 5.0.	55
Backward Compatibility with Version 4.4.	55
Backward Compatibility with Version 4.3b	56
Backward Compatibility with Version 4.3a	57
Backward Compatibility with Version 4.3.	57

Chemical Reaction Engineering Module	62
New Functionality in Version 5.4.	62
New Application and Updated Models in Version 5.4	63
Backward Compatibility with Version 5.3a and Earlier	64
Backward Compatibility with Version 5.0 and Earlier	64
Corrosion Module	68
New Functionality in Version 5.4.	68
New and Updated Models in Version 5.4.	69
Backward Compatibility with Earlier Versions	70
ECAD Import Module	71
New Functionality in Version 5.4.	71
Backward Compatibility with Version 5.3a	71
Backward Compatibility with Version 5.2a	71
Backward Compatibility with Version 5.0.	71
Electrochemistry Module	72
New Functionality in Version 5.4.	72
New and Updated Models in Version 5.4.	74
Backward Compatibility with Version 5.3a	74
Electrodeposition Module	75
New Functionality in Version 5.4.	75
New and Updated Models in Version 5.4.	77
Backward Compatibility with Version 5.3a	77
Fatigue Module	78
New Functionality in Version 5.4.	78
Backward Compatibility with Version 5.3.	78
Geomechanics Module	79
New Functionality in Version 5.4.	79
New Models in Version 5.4.	80
Heat Transfer Module	81
New and Improved Functionality in Version 5.4	81
New Applications in Version 5.4	87

Updated Applications in Version 5.4	88
Backward Compatibility with Version 5.3a	89
Backward Compatibility with Version 5.3.	91
Backward Compatibility with Version 5.2a	93
Backward Compatibility with Version 5.2.	93
Backward Compatibility with Version 5.1.	94
Backward Compatibility with Version 5.0.	95
Backward Compatibility with Version 4.4.	97
Backward Compatibility with Version 4.3b	99
Backward Compatibility with Version 4.3a	99
Backward Compatibility with Version 4.3.	100
Obsolete Features	102
MEMS Module	103
New Functionality in Version 5.4.	103
New Models in Version 5.4	107
Backward Compatibility with Version 5.2.	108
Backward Compatibility with Version 4.3b	108
Microfluidics Module	109
New Functionality in Version 5.4.	109
Backward Compatibility with Version 5.3a	109
Backward Compatibility with Version 4.3a	110
Backward Compatibility with Version 4.3.	110
Mixer Module	111
New Functionality in Version 5.4.	111
Molecular Flow Module	112
Backward Compatibility with Version 5.0 and Earlier	112
Multibody Dynamics Module	113
New Functionality in Version 5.4.	113
Nonlinear Structural Materials Module	115
New Functionality in Version 5.4.	115
New Models in Version 5.4	116

Optimization Module	118
New Functionality in Version 5.4.	118
New and Updated Models in Version 5.4.	119
Backward Compatibility with Version 4.3a	119
Particle Tracing Module	120
New Functionality in Version 5.4.	120
New Applications in Version 5.4	120
Backward Compatibility with Version 5.3 and Earlier	120
Backward Compatibility with Version 5.2a and Earlier	121
Backward Compatibility with Version 5.2.	122
Backward Compatibility with Version 5.1.	122
Backward Compatibility with Version 5.0 and Earlier	123
Pipe Flow Module	124
Backward Compatibility with Version 5.1 and older.	124
Plasma Module	125
New Functionality in Version 5.4.	125
New and Updated Applications and Models in Version 5.4	125
Backward Compatibility with Version 5.3a and Earlier	126
Backward Compatibility with Version 5.3 and Earlier	126
Backward Compatibility with Version 5.2 and Earlier	127
Backward Compatibility with Version 5.1.	127
Backward Compatibility with Version 4.4.	127
Ray Optics Module	129
New Functionality in Version 5.4.	129
New Applications in Version 5.4	131
Backward Compatibility with Version 5.3 and Earlier	131
Backward Compatibility with Version 5.2a and Earlier	132
Backward Compatibility with Version 5.2.	133
Backward Compatibility with Version 5.0.	135
RF Module	136
New Functionality in Version 5.4.	136
New and Updated Models in Version 5.4.	138

Rotordynamics Module	141
New Functionality in Version 5.4	141
New Models in Version 5.4	142
Semiconductor Module	144
New Functionality in Version 5.4	144
New Models in Version 5.4	147
Structural Mechanics Module	149
New Functionality in Version 5.4	149
New Models in Version 5.4	154
Backward Compatibility with Version 5.2	155
Backward Compatibility with Version 4.3b	156
Backward Compatibility with Version 4.3a	156
Subsurface Flow Module	157
New Functionality in Version 5.4	157
New Models in Version 5.4	158
Backward Compatibility with Version 5.3a and Earlier	158
Wave Optics Module	159
New Functionality in Version 5.4	159
New and Updated Models in Version 5.4	162
Material Library	163
New and Updated Material Data in Version 5.4	163
LiveLink™ for MATLAB®	164
New Functionality in Version 5.4	164
Backward Compatibility with Version 5.2	165
Backward Compatibility with Version 5.0	165
The COMSOL API for Use with Java®	166
COMSOL 5.4 API Changes	166
COMSOL 5.3a API Changes	167
COMSOL 5.3 API Changes	167
COMSOL 5.2a API Changes	167
COMSOL 5.1 API Changes	168

COMSOL 5.0 API Changes	171
COMSOL 4.4 API Changes	173
COMSOL 4.3b API Changes	174
COMSOL 4.3a API Changes	174
COMSOL 4.3 API Changes	175
COMSOL 4.2a API Changes	176
COMSOL 4.1 API Changes	177
COMSOL 4.0a Java API Changes	178



Release Notes

COMSOL Multiphysics[®] version 5.4 includes the all-new COMSOL Compiler[™], extended and improved versions of the Application Builder, COMSOL Server[™], and COMSOL Multiphysics[®], a new Composite Materials Module, as well as updates and improvements for all COMSOL Multiphysics[®] add-on products. These *Release Notes* provide information regarding new functionality in version 5.4 for all COMSOL[®] software products.

New Products

COMSOL Compiler™

Using the new COMSOL Compiler™, you can compile apps developed using the Application Builder into standalone, executable apps that can be deployed and run on any supported platform (Windows®, Linux®, and macOS) without a license file. Such executable apps can be deployed within an organization or distributed to a wide range of users. A license for COMSOL Compiler™ is needed for compiling and creating executable apps.

The Composite Materials Module

The Composite Materials Module is a new add-on to the Structural Mechanics Module for simulations of composite materials. This module provides tools for structural (and, together with other add-on products, multiphysics) modeling in laminate and layered materials.

There are two different formulations for layered shells, one that is based on a classical laminate theory and one that is based on a more complete 3D theory. The former is implemented in a material model in the existing Shell interface, whereas the latter forms the basis for the new Layered Shell interface.

The user interface includes tools for specifying isotropic and anisotropic material properties in layered materials together with dedicated visualization tools used to display the layers graphically. By combining the Composite Materials Module with the Heat Transfer Module, you get more sophisticated heat transfer and thermal expansion functionality. Similarly, you can combine the Composite Materials Module with the AC/DC Module for layered materials functionality within electrostatics and electric currents. Applications include structural analysis of composite laminates, possibly coupled to the Heat Transfer Module and the AC/DC Module for multiphysics analysis of composite materials.

COMSOL Multiphysics

General New Functionality

IMPORT AND EXPORT OF PREFERENCES

You can now import and export your preference settings to make it easier to migrate preference settings from earlier versions of a model or from one COMSOL Multiphysics installation to another. The **Import** and **Export** buttons are available in the **Preferences** dialog.

GRAPHICS SERVER IMPROVEMENTS

- File export of images and animations when done from an API client (for example, MATLAB[®]) no longer requires a graphics server. This is the case both for the export of result features and when using the new `image().export()` method. This relaxed requirement is especially beneficial when running the server on macOS, which does not support a graphics server.
- COMSOL Multiphysics can now connect to a graphics server. If connecting both COMSOL Multiphysics and an API client to the same server, they will handle the plotting individually. COMSOL Multiphysics will continue plotting in its window, and the API client will plot in windows on the server.

New Functionality in the Application Builder

IMPROVEMENTS TO THE FORM EDITOR

- All **Form** objects can now act as **Settings Form** nodes in the **Model Builder**, when the **Show in Model Builder** check box in the **Settings** window for **Form** nodes is selected.
- You can now add local declarations and methods in **Form** objects. Declarations added from the main **Declarations** node are considered global and have their values synchronized between **Form** instances. Local declarations are useful when you want declarations set in one **Form** instance to not affect the values of the declarations in other **Form** instances. Local methods provide a way to share local methods between form objects without adding them to the global namespace.
- A **Center on screen** setting is now available in the **Size** section of the **Main Window** feature. When selected, the app is centered in the middle of the desktop upon launch.

- The **Size** setting for **Button** and **Toggle Button** form objects has changed its name to **Style**. In addition to the existing **Small** and **Large** options, there is a new **Flat** option. The **Flat** style is the same size as **Large** but with a transparent button background and no border. This look appears on Windows and in the web client. The **Flat** and **Large** styles look the same on macOS and Linux.
- You can now specify the background color for **Button** objects. When **Default** is selected, the platform controls the background color.
- The **Graphics** form object toolbar now includes the new **Orthotropic Projection** toggle button.
- The **GUI Commands** part of the **Editor Tools** window now includes graphics commands for the new **Orthotropic Projection**, **Show Selection Colors**, and **Show Material and Texture** functionality.
- The **Results Table** form object now supports evaluation groups.

IMPROVEMENTS TO THE METHOD EDITOR AND APPLICATION LANGUAGE

- All methods can now act as model methods or method calls in the **Model Builder**, when the **Show in Model Builder** check box in the **Settings** window for **Method** nodes is selected.
- You can now add command-line arguments to apps and method calls.
- Method inputs of the double types now have an optional unit definition.
- These methods have been added: `model.methodCall().methods()`, to get all model methods in a model, and `model.methodCall(String).inputNames()`, to get the names of input parameters of method reference by given method call. Both methods return string arrays. These methods simplify dynamic creation of **Method Call** features through the external API.

See also the *Application Programming Guide* for a complete overview of the application language for programming methods in the Application Builder.

New Functionality in COMSOL Server™

- There is now a live search box on the **Application Library** page for filtering the applications shown.
- There is an explicit **Remember me** check box on the web login form that lets users decide if their session should be persistent or not.
- The installer now verifies that the Windows service details are valid and that the service account is able to access the installation directory and the shared working directory.
- There is now an option to send notifications as emails, in addition to the built-in notifications panel.
- COMSOL Client can now be launched through **Run in COMSOL Client** links from a web browser, even for anonymous and automatically logged in users.
- The Application Library now supports sorting in descending order.

New and Improved General Functionality in COMSOL Multiphysics

- For improved navigation in models with many nodes, you can create custom groupings of nodes by adding **Group** nodes in the model tree. Group nodes can add organization to the model tree by grouping features that belong together, such as features that belong to a particular part of the geometry or are related to a particular aspect of the physics.
- It is now possible to add multiple **Parameters** nodes for grouping parameters for different purposes. You can also exclude parameters in a **Parameters** node from parameter selections (in parametric sweeps, for example). Parameter cases can be added as **Case** subnodes to use different parameter values in different cases, which is useful in connection with the new **Parameter switch** option for parametric sweeps.
- You can now add selection colors using color themes and color settings for selections under **Definitions** and for nodes in geometry sequences (in the latter case, when the **Resulting objects selection** check box has been selected) in 2D and 3D. It is also possible to control if selection colors as well as material colors and texture should be shown from the Graphics window toolbar and from the **View** nodes' settings.
- The **Development** toolbar now includes a **Compare** button. Using the compare functionality, you can compare the current model or app with the contents of an MPH-file on the file system, such as an earlier version of the same model or app.

The comparison then appears in the **Comparison Results** window, where you can filter the differences, inspect the details, and export the comparison to an XML-file.

- It is now possible to create **Form** objects in the Application Builder and use them as **Settings Form** nodes under **Global Definitions** in the **Model Builder** or as modal dialog boxes for performing specific tasks. You can add **Settings Form** nodes and show them as dialog boxes from the **Forms** section in the **Development** toolbar. Also, all applicable methods can now act as model methods in the **Model Builder**.
- Improved memory allocation for multicore computers, primarily those using Windows[®]. It is activated automatically for a multicore architecture but can also be activated using the **Memory allocator** settings on the **Multicore and Cluster Computer** page in the **Preferences** dialog box.
- For global parameters, you can now add multiple **Parameters** nodes for better organization of parameters in your models. A **Show in parameter selections** check box can be cleared to exclude parameters from a **Parameters** node to appear in parameter selections. You can also add **Case** subnodes for parameter cases and switch between parameter cases to use different sets of parameter values.
- The **Select Study** page in the Model Wizard has been reorganized to provide a better structure and overview for selecting a study type.
- The support for model inputs has been extended so that most physical properties can now act as model inputs and are represented as predefined variables. There is a new **Common Model Inputs** node under **Global Definitions**, where you can view and change default scalar values for all model inputs. By default, any material using a model input will now use the common model input. The **Common Model Inputs** node is added automatically when a model input is requested. You can also add **Model Input** nodes under a model component to define model inputs locally in all or part of the component's geometry.
- The **Find** tool's dialog box includes a new **Advanced** tab that provides more specific search methods, including filtering to search specifically for node tags, node labels, node names, node types, descriptions, and settings.
- In the **Graphics** window toolbar for 3D geometries, a new button is available for switching between orthotropic and perspective projection.
- Timestamps now appear for messages in the **Messages** window. They can be turned off using the **Add timestamps to messages** check box on the **General** page in the **Preferences** dialog box.
- For closable windows in the COMSOL Desktop[®], you can now close them using Ctrl+F4 (Command+W on macOS).


- You can now select multiple rows in tables (in, for example, the **Parameters** and **Variables** nodes).
- In the **Selection List** window, you can now filter the list of objects or entities by choosing **Only List Selected**, **Only List Visible**, or **Only List Applicable** from the **Filter List** menu.
- Selections for Equation View contributions now appear in the Graphics window when you have selected a single variable in the list under **Variables** in an **Equation View** node's **Settings** window.

IMPROVED HANDLING OF BATCH LICENSES

In version 5.4, the way that batch licenses are checked out has changed. In previous versions, if you submitted a batch job from the operating system command line, a full COMSOL Multiphysics license was checked out. One such checkout consists, roughly speaking, of one COMSOL Desktop user interface license and one batch license for cluster and remote computing. In version 5.4, you can submit a batch job from the operating system command line with only a batch license being checked out. This means that one concurrent user can submit a batch job and still keep working on the same or a different model in the COMSOL Desktop environment. If a second batch job is run simultaneously as the first one by the same user, then a second license is checked out, unless the second batch job is just a parametric variation of the first one, which still check out only one license key.


New Functionality in the Physics Interfaces

- The **Discretization** section in the **Settings** windows for the main physics interfaces nodes is now always visible and includes the fundamental settings for controlling the element (shape function) order and, in some cases, the element type. To access more advanced discretization settings, select **Advanced Physics Options** from the **Show** menu in the **Model Builder** toolbar. The **Discretization** option has been removed.
- New Reacting Flow, Diluted Species multiphysics coupling feature.
- Updated formulation for the **Flux** and the **No Flux** nodes included in the Transport of Diluted Species interface.
 - The flux relative to the convective flux is prescribed instead of the total flux.
 - The new formulation implies that the nodes can be used on a moving mesh geometry without compensating for the boundary velocity.

- The inverse time step measure variable $\text{dt}2\text{Inv}$, solved for in transport interfaces, has been turned into a state variable and is no longer visible in the user interface.
- **Go to Source** buttons () are now available for the coupled physics interfaces in multiphysics coupling features.

New Geometry and Mesh Functionality

GEOMETRY FUNCTIONALITY

- It is now possible to highlight the results from building a geometry feature by clicking the **Highlight Result** button () in the toolbar at the top of **Settings** windows for geometry features.
- There is a new virtual geometry operation, Collapse Face Region, in 3D for automatic detection and removal of narrow regions of faces. You can select it from the **Virtual Operations** menu when right-clicking a 3D Geometry node. By default, the tolerance is set automatically, but you can also provide a manual tolerance (in terms of a maximum width for a narrow region).
- For the Remove Details operation, you can now control whether it is carried out on the entire object or on a selection of domains, faces, or edges, using only the selection settings in **Entities to Process** section in the **Settings** window for **Remove Details**.
- For 1D geometries, you can now specify intervals using interval lengths in addition to specifying coordinates in the **Settings** window for **Interval** nodes.

MESHING FUNCTIONALITY

- Faster meshing of boundary layers.
- In the **Adapt** node, you can now choose to perform mesh adaptation on boundaries and edges, and it is also possible to modify a mesh by adapting it to a specified isotropic or anisotropic metric.
- You can now refine meshes on edges, faces, and domains interactively for all mesh element types.
- For physics-controlled meshing, you can now control which physics that will contribute to the meshing in a multiphysics simulation.
- It is now possible to import planar 2D meshes into a 3D component.
- You can now import second-order elements as linear elements for mesh import in 2D and 3D using COMSOL Multiphysics files.

- The Adapt, Convert, and Refine operations are now available for imported mesh sequences.
- Improved quad meshing through the use of templates for planar equilateral triangles, quarter circles, and semicircles.
- You can now also create swept meshes in geometries with isolated geometric entities.

NEW MESHING TUTORIAL

The new tutorial *Swept Meshing of a Bracket Geometry* demonstrates how to partition a 3D geometry to create a swept mesh. This tutorial model explores different strategies for partitioning the geometry and demonstrates how to combine hexahedral and tetrahedral meshes. Application Library path:

COMSOL_Multiphysics/Meshing_Tutorials/bracket_swept_mesh

New and Updated Operators, Functions, and Definitions

- For **Interpolation** functions, the primitive functions and inverse functions are now available in the **Related Functions** section (called **Random Sampling** in earlier functions). It is now possible to define the primitive function and random sampling of any interpolation function such that there is also only one actual function and only one argument for values coming from an input file or from a result table.
- For **Boundary System** nodes that define boundary coordinate systems, you can now add **Reverse Normal** and **Domain Normal** subnodes to reverse or define the normal direction for some selected boundaries and domains, respectively.
- Bulk viscosity and parameter of nonlinearity are now available as material properties for air in the Built-In and Liquids and Gases material libraries.

New Functionality in Studies and Solvers

STUDIES AND STUDY STEP FUNCTIONALITY

- Parameters can now be separated into groups using multiple **Parameters** nodes, and for each group, it is possible to add parameter cases of values as **Case** subnodes. For **Parametric Sweep** nodes, there is a new **Parameter switch** sweep type by which you can sweep over those parameter cases.
- Improved mesh adaptation:
 - The solutions are now stored in a way that is similar to how solutions are stored for parametric sweeps. A mesh refinement level parameter is automatically added,

which makes result comparisons easier because all the solutions can be obtained from one data set.

- All adaptation except **Rebuild mesh** can now also be used with imported meshes.
- A new **General modification** refinement method is available in mesh adaptation studies.
- Improved functionality in the **Adapt** node in meshing sequences. See [Meshing Functionality](#) above.
- The **Stop** buttons in the **Progress** window now appear only for levels where stopping the solution is applicable.

SOLVER FUNCTIONALITY

- It is now possible to reuse the sparsity pattern of previously assembled matrices in order to improve the performance of the solution process. To do so, and if not selected by default, select the **Reuse sparsity pattern** check box in the **Advanced** solver node's **Assembly Settings** section.
- The model reduction functionality has been improved and reorganized. You can add **Global Reduced Model Input** nodes under **Global Definitions** to declare global variables for use as controlling inputs to **Reduced Model** nodes. It is now possible to define the expressions for these variables in one place. These expressions will be used both for the reduced and unreduced version of the model. You can also import reduced-order model files by choosing **Reduced Model Import** from the **Reduced Models** menu on the **Global Definition** node's context menu.
- The default values for the **Factor in error estimate** for the **Direct** linear solvers is now 1 (it was 400 in earlier versions). This change applies only to direct linear solvers when used as the main solver. This change works in the direction of easier error tests (it makes the linear solver error estimate 400 times smaller).
- The FFT solver and study steps now support initial expressions. Such initial expressions make it possible to couple a field to another field. The mapping occurs before the FFT.
- A transpose-free QMR iterative solver is now available as a **TFQMR** solver choice in the **Solver** list for the **Iterative** solver nodes. It can be an alternative to GMRES. TFQMR uses less memory than GMRES, especially for problems where the GMRES restart number needs to be increased to reach convergence. Problems in this category are problems where multigrid or other high-quality preconditioners cannot be used.

- Anderson acceleration is now available with pseudo time stepping in the **Fully Coupled** and **Segregated** solver steps.
- For the boundary element method, a fast and memory-efficient Hierarchical LU solver or preconditioner is now available.
- You can now add stop conditions to the **Time Explicit** solver.
- An option to add a scale factor for the residual scaling has been added to the **Dependent Variables** nodes in the solver configurations. It can be useful in cases where convergence is hampered by the residual error.
- For the eigenvalue solver and study steps, you can now select a **Real symmetric solver consistency check** check box, which is available when the **Use real symmetric eigenvalue solver** list is set to **Automatic**. It can be used to avoid that the eigenvalue solver assumes a positive definite mass matrix in cases where that condition is not strictly fulfilled.
- For time-dependent simulations, the maximum time step can now be an expression. The expression is evaluated while solving and can, for instance, depend on the time parameter itself.
- Maximum time step control is now available for the time-dependent modal solver.
- The default behavior for the automatic damped Newton method with automatic scaling has been improved. The old behavior meant updating the weights used for termination in every iteration. This strategy is sometimes problematic for the damping logic and can lead to divergence of the nonlinear iterations. The new algorithm updates the weights only if they change two orders of magnitude, and the automatic Newton solver is then restarted from the current solution. A new **Update weights for automatic scales** check box is available and selected by default for all the Newton solvers — **Automatic (Newton)**, **Automatic highly nonlinear (Newton)**, and **Constant (Newton)** — for the stationary and parametric cases. It makes it possible to turn off the update of weights for automatic scales.
- For the detailed solver log, the time-dependent solver now lists any algebraic variables at the start of the solution process.

CLUSTER AND BATCH FUNCTIONALITY

- You can now choose to store the solutions from a cluster simulation on a single node instead of on all nodes, which can be beneficial for memory management, because

memory usage for the solution can otherwise grow linearly with the number of nodes in the cluster.

- You can now make use of batch licenses from the command line using the new `-usebatchlic` argument.

New and Improved Results and Visualization Functionality

NEW AND IMPROVED DATA SET AND EVALUATION FUNCTIONALITY

- You can now add **Evaluation Group** nodes to create evaluation groups, which provide a more structured approach to setting up and maintaining numerical evaluation nodes in a model. You can add numerical evaluation nodes to an **Evaluation Group** node, and the numerical data from those evaluations appear in a special table window for that evaluation group. The evaluation groups can be updated automatically after the solver has finished using the new **Reevaluate all evaluation groups after solving** check box in the **Results** node's **Settings** window. There is also support for computing the sum of multiple numerical evaluations, for example.
- New Extrusion 1D and Extrusion 2D data sets for extruding a solution from 1D to 2D or 2D to 3D for use in postprocessing. The data set to extrude can be any data set with an output dimension equal to the **Extrusion** data set's dimension.
- You can now use solution-dependent settings in the parameter ranges in the Parametric Curve and Parametric Surface data sets.
- A **Zoom extents** check box is now available in the **Settings** window for the **Export>Animation** feature.
- The **Eva1** feature in the COMSOL Multiphysics API can now retrieve data for Shell data sets.
- Variables for the plane parameters in Surface data sets are now available.
- It is now possible to use solution-dependent expressions in the Contour and Isosurface data sets.
- Particle Bin and Ray Bin data sets are now available for evaluations of particles and rays in the Particle Tracing Module and the Ray Optics Module and Acoustics Module, respectively.
- Response Spectrum 2D and Response Spectrum 3D data sets are now available for response spectrum analysis in the Structural Mechanics Module.

NEW AND IMPROVED GRAPHICS AND PLOT FUNCTIONALITY

- 3D images can now be exported using the glTF file format (GL transmission format). The images are saved as binary GLB files.
- A new Surface Slit plot type is now available for 3D surface plots where you see the result of evaluating one expression on one side and another expression on the other side.
- Improved annotation plots:
 - In the **Settings** window for **Annotation** plot nodes, you can now choose an anchor point and specify if the label should be horizontal or vertical.
 - The point markers are better looking, and the background color and show frame settings can be inherited from the parent node.
 - In the **Settings** window for **ID Plot Group** nodes, you can choose to plot annotations using the secondary y -axis.
- It is now possible to plot arrows on streamlines, and a new arrowhead arrow type has been added.
- For isosurface plots, you can now add isosurface-level labels.
- Improved control of the polar plot axis: You can now specify a zero angle (right, up, left, or down); angular unit (radians or degrees); rotation direction (clockwise or counterclockwise); and symmetric angle range.
- A parameter indicator now appears to display the solutions time, parameter value, or phase for the currently selected plot in the upper-left corner of the **Graphics** window.
- A **Selection** subnode is now available for the **Mesh** plot nodes.
- It is now possible to plot points in Particle Trajectory plots so that the sphere radii have a constant radius when zooming.
- The plot nodes that were previously called **Far Field** are now called **Radiation Pattern** to better reflect their applicability.
- You can now add a variable name for the azimuthal angle to evaluate it in 2D axisymmetric radiation pattern plots.
- The **Plot First**, **Plot Previous**, **Plot Next**, and **Plot Last** buttons are now also available for 1D plots.
- The rendering of graph plots has been improved.
- In the **Graphics** window for 3D plots, new **Rotate Right 90°** and **Rotate Left 90°** buttons are now available, and the **Go to ZX View** button has been replaced by the **Go to XZ View** button.

- It is now possible to control the radius scale for points in plots such as Scatter Volume plots, where a scale factor relative to the automatic radius is available.
- The Far Field plot is now called Radiation Pattern.
- Using the API and methods, you can create Point Data, Line Data, and Surface Data plots of raw data in the form of point data, element data, and color data. These plots can appear in the COMSOL Desktop but cannot be directly created there. You can also create 2D and 3D Tube Data, Annotation Data, and Arrow Data plots using methods.

IMPROVED REPORT GENERATION

- You can now create your own custom templates for report generation. For example, you can define a report template that generates reports containing simulation results only with large plot images and saves it in a dedicated report templates folder for easy access in future modeling sessions.
- Report image generation no longer uses the **Graphics** window of the COMSOL Desktop.
- The **Progress** window in the COMSOL Desktop now tracks the progress of the report writing stage.
- There is now complete COMSOL API for Java[®] support for reports.
- In the **Comments** field in the **Properties** window of the nodes in the Model Builder tree, you can now use syntax to add formatting, Greek letters, exponents, and some other special characters to the node comments. The comments will then include that formatting in model reports.

New Java[®] API Methods

- The `ModelUtil.modelsUsedByOtherClients()` method returns the tags of the models used by other clients.
- The `view.geom()` and `view.getSDim()` methods return the geometry and the space dimension for the view, respectively.

General Backward Compatibility Considerations

COMSOL Multiphysics version 5.3a can open MPH-files saved from COMSOL Multiphysics versions 4.0–5.3.

COMSOL Multiphysics version 5.3a can run Java[®] files saved from COMSOL Multiphysics versions 4.0–5.3. However, the Java[®] files may need to be modified in

accordance with information following in this document and may need to be recompiled with the `comsol compile` command in version 5.3a.

Backward Compatibility with Version 5.3a

ADAPTATION AND ERROR ESTIMATES

The property `rmethod` has been renamed `meshadaptmethod`, with additional possible values. The old property (`rmethod`) and its values can still be used.

MASS PROPERTIES AND MASS CONTRIBUTIONS

In the **Density** section, the **Integration frame** list in previous versions is now called **Density input frame**.

In the API, the corresponding property name has been changed from `integrationFrame` to `densityFrame`.

NO FLUX FORMULATION

The formulation of the No Flux feature has been changed in the Transport of Diluted Species interface in version 5.4 so that the flux relative to the convective flux is set to zero. The new formulation is used when opening a model from a previous session. In the previous formulation, the total flux was set to zero.

LEGACY FLAG `-DCS.LEFTRIGHTSELECTION` HAS BEEN REMOVED

The legacy flag `-Dcs.leftrightselection` for selections of geometric entities using left-right click has been removed in version 5.4.

AMD ACML LIBRARIES

The AMD ACML library is no longer supported by AMD. It has been replaced by standard BLAS and LAPACK as an alternative to Intel MKL. On Linux, the BLIS library is also available. Intel MKL is still the default BLAS library.

STRICTER SOLUTIONINFO VALIDATION

The `SolutionInfo` validation now ensures that `SolutionInfo.isValid()` returns true before using other `SolutionInfo` methods, unless stated otherwise in the documentation of the methods.

PDE, BOUNDARY ELEMENTS INTERFACE

The PDE, Boundary Elements interface has been reimplemented and improved. Backward compatibility is provided for MPH files but not for the Java API.

STABILIZATION IN SOME PHYSICS INTERFACES

Model files created in version 5.3 and earlier retain the old stabilization formulation. The new formulation is obtained by replacing the physics interface with a new one. The improved stabilization breaks Model Java-file backward compatibility for time-dependent models that contain one of the affected physics interfaces. Please contact COMSOL Support to learn how to retain backward compatibility for your specific Model Java-file.

STORE SOLUTION ON DISK

The **Store solution on disk** option has been removed.

FUNCTIONS IN GEOMETRY FEATURES

The button **Rebuild with Updated Functions** has been removed from the following geometry features: **Parametric Curve**, **Parametric Surface**, and **Sweep**. Instead, use the **Refresh** or **Clear Functions** button in the function features.

In the API, the method `importData()` is obsolete for the geometry features `ParametricCurve`, `ParametricSurface`, and `Sweep`. Instead, use the `refresh()` method in the function features.

PARTICIPATION FACTOR

Support for participation factors in the eigenvalue solver has been replaced by a new and improved **Participation Factors** feature, which is available under **Definitions>Variable Utilities** in a model component for computing participation factors in structural simulations.

INTERPOLATION CURVES

For interpolation curves, in the case when relative tolerance is zero (which is the default), the algorithm in 5.3a has changed so that the shape of the curve becomes somewhat different (while still interpolating the given points).

ITERATIVE SOLVERS

For the iterative solvers, the error handling mechanism has been improved to contribute to a general robustness of the computation. The improved mechanism is enabled per default and can give the following effects for nonlinear iterations:

- Increased total number of linear iterations
- Failure with the error “Divergence of the linear iterations”, if the preconditioner is extremely ill-conditioned.

UNITS IN THE SIZE EXPRESSION NODE

The size expression is now interpreted in the model component’s unit system rather than in the geometry’s unit system. Also, the spatial coordinates (x, y, z , and so on) are now always in the component’s unit system. In version 5.3, when evaluating on a grid, the spatial coordinates were in the geometry’s unit system.

STRESS/STRAIN PLOTS

Principal stress/strain plots now evaluate the stresses and strains in the centers of mesh elements instead of in the mesh nodes.

Backward Compatibility with Version 5.2a

FRAME DEFINITIONS

In version 5.3a, all frames are always defined. If you open models from version 5.2a or earlier, there will be a **Permanently Define All Frames** button under **Frames** in the General section of the **Component** nodes. When you click the button, all frames are defined, and the button disappears.

FRAME CONTROL

In previous versions of COMSOL Multiphysics, it was possible to apply multiple frame-controlling physics on the same selection. For example, it was possible to add two Solid Mechanics interfaces (with the **Include geometric nonlinearity** option enabled) on the same selection and solve. In this case, an “override rule” was applied between the controlling physics, effectively meaning that the last physics in the tree had control of the frame deformation for the overlapping domains. This situation could cause unexpected results.

In version 5.3, multiple physics interfaces controlling the same frame are not allowed on the same selection and will cause an error when trying to solve. However, it is still possible to use multiple frame-controlling physics with overlapping selection, but you

now have to explicitly disable frame control on all but one of the physics. This gives you better control on what is going on. The functionality to disable frame control for physics has been incorporated in the **Settings** window for study steps to allow choosing different physics to control the frame in different study steps.

Frame-Scoped Variables

Certain variables pertaining to frame deformation are now defined with “frame prefix”. For example, the relative element volume is now called `spatial.relVol`. Previously, these variables were added by the physics interface controlling the frame deformation so they used a “physics prefix” (for example, `ale.relVol`). For backward compatibility of old models, the physics interfaces still define the old variables, but they are aliases of the new frame variables.

API SYNTAX FOR ERROR HANDLING

The old syntax for an error

```
feature("foo").feature("prob1").feature("error1")
```

is no longer supported. Instead, use the following documented syntax:

```
feature("foo").problem("error1")
```

FREE QUAD MESHING

Models saved in version 5.2a with the free quad tessellation method set to **Automatic** (in a **Free Quad** feature) or face meshing method set to **Quadrilateral** (in a **Swept** feature), will, when opened in 5.3, have the tessellation method or face meshing method set to **Legacy version 5.2a** or **Quadrilateral (legacy version 5.2a)**, respectively.

Backward Compatibility with Version 5.2

FREE QUAD MESH

The new free quad meshing algorithm is used in new models, but for models created in earlier versions, the legacy algorithm is used instead. The default for the new `method` property is therefore `auto` in new models, and `legacy52` in migrated models.

GEOMINFO CHECK() METHOD

The return value of the `check()` method in `GeomInfo` has been removed. Now, `check()` throws an error if the geometry is invalid.

THE CONST PROPERTY IN SOLVERS

The `const` property available for many solvers has been removed in version 5.2a and replaced with the `cname` and `c1ist` properties, which are string arrays for the constant names and corresponding constant values, respectively. For the **Dependent Variables** node, the new property `initparametersmethod`, which can be set to `auto` (the default) or `manual`, has been added to control automatic synchronization of values of parameters to use for initial expressions. The **Dependent Variables** node now also includes the `cname` and `c1ist` properties.

Backward Compatibility with Version 5.1

CREATING SELECTIONS

In version 5.2, the `selresult` property replaced the `createselection` property. `createselection` is still supported for backward compatibility.

SELECTIONS IN PART INSTANCES

For backward compatibility for selections in part instances, there is a **Keep noncontributing selections** check box, cleared by default, in the **Selection Settings** section of the Settings window for **Part Instance** nodes. If you select the **Keep noncontributing selections** check box, the **Keep column** is disabled, and the selection is kept if the **Contribute to** value is **None**. In the COMSOL API, the default is an active **Keep noncontributing selections** setting.

MESH PARTS

For backward compatibility regarding STL/VRML import, and for COMSOL API compatibility, the old user interface for mesh import (with a specified file name and import properties) and the corresponding properties are available in the COMSOL API but not in the COMSOL Desktop, unless it is the active option. This means that the old user interface can only be reached if the filename is set in combination with the STL import type (the API type `st1vrml`), which cannot be done from the new user interface.

DORMAND-PRINCE 5 TIME-STEPPING SOLVER

The Dormand-Prince 5 Runge-Kutta solver in 5.1 does not use field norm scaling in version 5.1 when estimating errors. This means that the errors of, for instance, an ODE might become very small when solved together with a field with many DOFs, and the time steps taken might then be too large. In version 5.2a, the Runge-Kutta solver uses field norm scaling, which means that old models using the Dormand-Prince 5 might

need tighter tolerances to produce results in earlier versions that are similar to those in 5.2a.

Backward Compatibility with Version 5.0

MESH IMPORT

The method used to automatically partition the boundary of imported meshes in 3D has been improved. If you have an existing model, you can work with it without being affected. However, if you click the **Import** button of the **Import** feature in the meshing sequence, the file is read again and the new partitioning method is used. The exception is if the **Import** feature had **Boundary partitioning** set to **Manual**. In this case, the modified parameter values are preserved under the **Feature detection** setting, which uses the same algorithm as in version 5.0.

COMSOL tries to map the old selection on boundaries to the new boundaries, but it is not always possible to do accurately when new faces have appeared or old faces have disappeared. You may have to manually review and update boundary, edge, and point selections after reimporting the mesh.

If you have a Java[®] or MATLAB[®] program that imports meshes, the number of geometric entities may have changed compared to older versions.

SECURITY SETTINGS

In version 5.2a, the **Allow external process and libraries** check box on the **Security** page in the **Preferences** dialog box is cleared by default to not allow applications to start external processes on the computer. The default setting in 5.0 is set to allow such external processes.

DISPLAY OF MAX/MIN MARKERS IN PLOTS

The display of max/min markers in plots is now off by default to make plots fit better when using a small graphics canvas. In previous versions, the display of max/min markers in plots was on by default.

MERGED MPH-FILE FORMAT

The MPHAPP file and MPH file formats have been merged since version 5.1, and all application files use the .mph file suffix. You can still open MPHAPP files created in version 5.0.

COMSOL SERVER

COMSOL Server from version 4.4 is now called the COMSOL Multiphysics Server.

MESH IMPORT

An edge in an imported mesh that has a common start and end vertex, or that lacks start and end vertices, is now split into two edges with distinct start and end vertices. For meshes where this happens, the numbering of all geometric entities may change when the mesh is rebuilt.

For MPH files created in earlier versions, selections are automatically updated with the new entity numbers.

LOCAL COORDINATE VALUES

For model components created in version 5.0 and onward, the variable names `xi1`, `xi2`, and `xi3` are reserved for the predefined local coordinate variables. To access the previous behavior (that local coordinate variables do not exist), you can use the API method `model.modelNode(<tag>).defineLocalCoord(false)`.

TRANSPORT OF DILUTED SPECIES

Running Java[®]-files from previous versions may fail due to the new default name for the Transport of Diluted Species interface. This can be avoided by adding a command that specifies the identifier in accordance with the name of the interface. For example, when creating an interface using the old name (`chds`):

```
model.physics().create("chds", "DilutedSpecies", "geom1",  
    new String[][]{{"c"}});
```

Add the following line to specify the identifier accordingly:

```
model.physics("chds").identifier("chds");
```

The Java[®] API syntax for creating and accessing vectors and tensors in the Transport of Diluted Species interface has changed as well as the syntax for setting physics properties. This does not affect MPH files. See the backward compatibility notes for the Chemical Reaction Engineering Module for additional information that also applies to the Transport of Diluted Species interface.

DOCUMENTATION

The *COMSOL Multiphysics Programming Reference Manual* replaces the *COMSOL API for Use with Java[®] Reference Manual*.

MODEL NODES ARE NOW COMPONENT NODES

The **Model** nodes (as they were called in previous versions), which contained separate model components in a model file, are called **Component** nodes in version 5.2a.

CHANGES TO PARAMETRIC SWEEPS

Old models that use stationary parametric sweeps are loaded with the **Reuse solution for previous step** list set to **Yes**. The **Run continuation for** list is set to the parameter used, unless the continuation algorithm would not have been used for this model in previous versions (for example, if multiple parameters are used or if the parameter list is not monotonous).

CHANGES TO THE PHYSICS SELECTION IN STUDY SETTINGS

The names of the states of a physics interface in the physics tree, which you can modify under **Physics and Variables Selection** in the study steps' settings windows, have changed:

- **Provide Degrees of Freedom** is now called **Disable in Solvers**.
- **Disable** is now called **Disable in Model**.

REVISED FORMULATION FOR LAMINAR INFLOW AND LAMINAR OUTFLOW

The formulations of the laminar inflow and laminar outflow conditions have been corrected. The modified formulation gives a more accurate mass flux. Some models may now produce a slightly different flow field.

Laminar inflow and laminar outflow are available in the fluid flow physics interfaces in the following modules:

- Batteries & Fuel Cells Module
- CFD Module
- Corrosion Module
- Electrochemistry Module
- Electrodeposition Module
- Heat Transfer Module
- Microfluidics Module
- Plasma Module
- Subsurface Flow Module

NEW DEFAULT FOR GRAPHICS OPTIMIZATION

Under **Graphics and Plot Windows** in the **Preferences** dialog box, the default settings in the **Optimize for** list is now **Quality** instead of **Performance**.

VELOCITY/ACCELERATION INTEGRATION VARIABLE

For the Solid Mechanics interface (and all related multiphysics interfaces) and the Truss interface, the help variable `u0` (velocity integration variable) is used in **Prescribed Velocity** and **Prescribed Acceleration** features for Time Dependent study types. This variable computes the displacement for each point where the condition is prescribed. It is changed to use the full feature scope in order to avoid collisions in cases where several such features exist within the same model component. As a result, when opening and running an old model that uses such features together with a segregated solver, an error message appears, stating that not all dependent variables occur in at least one of the segregated solver steps. The relevant action is to manually add the velocity integration variable to the segregated step containing the corresponding displacement field. Alternatively, you can regenerate any affected solver sequence.

MATH LIBRARIES ON AMD PROCESSORS

MKL is now the default math library on AMD processors. Switching to the ACML math library for AMD processors might improve performance in some cases.

Backward Compatibility with Version 4.3a

ERROR ESTIMATION CHANGES IN THE SOLVERS

The **Automatic** method for **Check error estimate** for direct linear solvers and for **Validate error estimate** has changed. For nonlinear and time-dependent problems, the underlying nonlinear solver does not accept termination for a linear solution step that does not fulfill the error estimate (unless the step size is very small). The motivation for this change is that the old method can lead to premature termination of the nonlinear solution process, which in some cases introduces a large error for the computed solution.

This change can cause the solvers to take more nonlinear steps for stationary problems and more time steps for time-dependent problems, and it may also lead to convergence problems. For such cases, use the **No** method to obtain the old behavior. However, doing so can hide numerical problems and potentially lead to large numerical errors.

TERMINATION CRITERION FOR STATIONARY SOLVERS

The default termination criterion for stationary solvers has changed. In 4.3a, the settings corresponded to **Solution**; now, the default is **Solution or residual**. This change in default termination criterion might affect models created in earlier versions of COMSOL Multiphysics if you regenerate the solver sequence in 5.2a.

BACKWARD EULER INITIALIZATION TIME STEP

A new setting in the **Advanced** section of the settings window for the **Time-Dependent Solver**, called **Fraction of initial step for Backward Euler**, provides an option for entering a dimensionless quantity that determines the size of the time step for the backward Euler method (in terms of the initial step). This value can improve the accuracy of the initialization step but can also affect the start-up of some models. The default value is 0.001 (this differs from earlier versions, which used a value of 1). When opening models created in version 4.3a or earlier, the value for this fraction is set to 1 to maintain compatibility with those versions.

LOADING EXTERNAL PHYSICS BUILDER JAR FILES

External physics builder JAR archives compiled with earlier versions of COMSOL Multiphysics include a manifest file that contains a reference to the CDO library, which has been removed. To load such JAR files in version 5.2a, you must first do one of the following:

- Delete the META-INF/MANIFEST.MF file in the archive source on the file system, and then recompile the JAR archive.
- Manually remove the line with `org.eclipse.emf.cdo` in the META-INF/MANIFEST.MF file in the JAR archive. You can do this directly in a file archive manager such as 7-Zip or similar.

HIGHLIGHTING GEOMETRY OBJECTS WHEN DRAWING IN 2D

When you have drawn one geometry object on top of another object, toggling of the highlighting of these objects occurs when you click several times. It is important that you click without moving the cursor in a position where the objects overlap in order to toggle. For example, if you draw a circle (C1) and then draw another smaller circle (C2) inside of C1, then first clicking on C2 may highlight C1. If you click for a second time without moving the cursor, it will highlight C2. In previous versions, you would move the cursor after the first click in order to highlight C2.

NEW TERMINOLOGY FOR CONSTRAINT TYPES

The following constraint types have new names in version 5.2a:

- *Bidirectional, symmetric* is now *Apply reaction terms on: All physics (symmetric)*.
- *Unidirectional* is now *Apply reaction terms on: Individual dependent variables*.

WEAK CONSTRAINTS UPDATE FOR THE LAMINAR FLOW INTERFACE

The weak constraint formulations for the following boundary conditions have been updated:

- Symmetry
- The Slip option in the Wall feature

These boundary conditions are now formulated using the same set of Lagrange multipliers as all of the other boundary conditions in the Laminar Flow interface. The Lagrange multiplier `un_1m` has been removed.

Models saved in version 4.3 will include `un_1m` until the model is re-solved. In some cases, occurrences of `un_1m` in the solver sequence must be replaced manually. This is the case if `un_1m` was the only Lagrange multiplier component in a segregated group or the only Lagrange multiplier component of a Vanka smoother. Alternatively, you can generate a new automatic solver sequence. Models saved in versions earlier than 4.3 must either be re-solved in version 5.2a for postprocessing, or opened and re-saved in version 4.3 before being opened in version 5.2a.

AUTOMATICALLY CREATED IDENTITY/CONTACT PAIRS

The pairs created by the Form Assembly feature in the geometry can differ from their forms in 4.3 for certain 3D and 2D geometries. The pairs should now be the same for the CAD and COMSOL representations.

AC/DC Module

New Functionality in Version 5.4

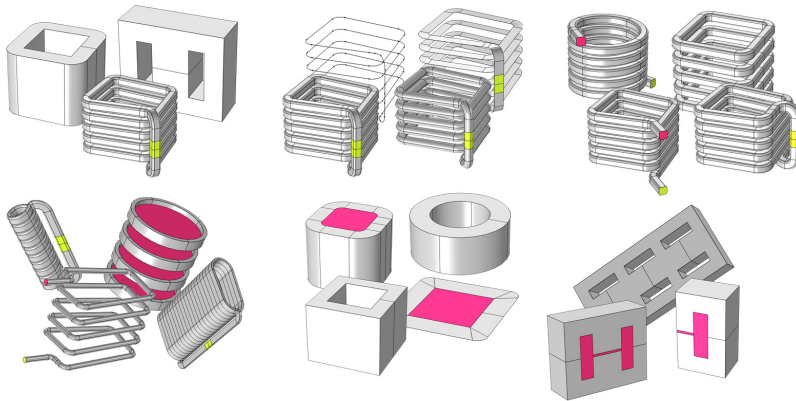
ELECTRIC CURRENTS, LAYERED SHELL INTERFACE

The new Electric Currents, Layered Shell interface is used to compute electric fields, currents, and potential distributions in thin conducting layered shells under conditions where inductive effects are negligible; that is, when the skin depth is much larger than the studied device. It supports stationary modeling on faces in 3D.

This interface replaces the old Electric Currents, Shell interface, except for models from version 5.3a or earlier and for piezoresistive modeling. The old interface has been renamed Electric Currents, Single Layer Shell and is available in the Model Wizard and Add Physics dialog box from Piezoresistivity under Structural Mechanics.

ADDITIONAL NEW FUNCTIONALITY

- A new geometry part library for a large variation of coil and core shapes is now available. Important selections, for example, for feed boundaries, are automatically generated. Some examples of generic shapes are shown below.



- The material model for magnetic saturation has been revised so that there is no longer a need to input both B-H and H-B curves. Only the B-H curve needs to be input and the (inverse) H-B curve is computed internally when needed. The

nonlinear magnetic energy and coenergy densities are now also supplied as output variables.

NEW APPLICATIONS

A new Layered Shell folder has been added to the AC/DC Module Application Library. The folder contains three examples that use the Electric Currents, Layered Shell interface. Two of these models are new and showcase a verification model and a planar PCB coil. The third model showcases the thermal deformation of a circuit, containing a thin coil. This is an update of a model previously available only within the Structural Mechanics Module and Heat Transfer Module.

- *Solid Multilayer Shell Comparison* studies the current conduction in a thin multilayer structure and compares results from the new shell formulation with results from a volumetric simulation using the Electric Currents interface where the shell layers are treated and meshed as solid (resolving the thickness).
- *Planar PCB Coil* simulates a spiral coil on a printed circuit board (PCB) involving several layers and vias between the layers.
- *Heating Circuit* models a device consisting of an electrically resistive layer deposited on a glass plate. The layer causes Joule heating when a voltage is applied to the circuit. The layer's properties determine the amount of heat produced.

Backward Compatibility with Version 5.2 and older

COIL FEATURES

Multi-Turn Coil features in old models will be mapped to the new **Coil** feature using the Homogenized Multi-Turn Conductor model. **Single-Turn Coil** features in old models will import as is with a warning that the feature is obsolete and will be removed in future versions.

The **User Defined Coil Geometry** subnode, which is available for 3D **Coil** nodes with the User-Defined Coil type, is now a domain feature. You can now select coil input and output boundaries using the new **Input** and **Output** subnodes. The selection is set up correctly when opening old models.

REMOVED THE OLD INFINITE ELEMENTS FEATURES

The old **Infinite Elements** feature, which are obsolete since version 4.2, have now been discontinued and will be automatically removed when opening old models.

COIL CURRENT CALCULATION

When models saved in previous versions are opened in version 5.1, the Coil Current Calculation study steps are migrated to Coil Geometry Analysis. Old solutions stored in the model can still be postprocessed, and the solver sequence will be regenerated automatically (with a Stationary solver) the first time the study is solved.

The Coil Geometry Analysis study step (formerly Coil Current Calculation) will now solve for all of the coils in the active interfaces. To solve only for specific coils (specified using the `CoilName` property), set the `SpecifyCoil` property to 1.

The default solver sequence generated by the Coil Geometry Analysis is different in version 5.1. Code that accesses specific solver features in the generated solver sequence may need to be reviewed.

OTHER COIL IMPROVEMENTS

Harmonic Perturbation subnodes under coil features (**Single-Turn Coil**, **Multi-Turn Coil**) are now global features, so the call to the create method should use the appropriate space dimension (-1):

```
model.physics("mf").feature("std1").create("hp1",  
    "CoilHarmonicPerturbation", -1);
```

Global features have no selections, so code that accesses the selection of the Harmonic Perturbation features may need to be reviewed.

Some of the improvements in the 3D **Multi-Turn Coil** features may require a review of existing code that uses the COMSOL API.

Subnodes required to set up the coil features are now added automatically. Existing code that uses the coil features may need to be updated.

The parameters `eCoil` and `length` have been moved from the **Multi-Turn Coil** features (boundary and domain) to the new subfeature `UserDefinedCoilGeometry`.

OTHER IMPROVEMENTS

New functionality introduced in version 5.1 is disabled by default when opening models created in previous versions:

- New boundary conditions for **Gauge Fixing** features
- Accurate coil voltage calculation

This functionality can be enabled using the appropriate inputs in the Settings window. Refer to the documentation for the individual features for more details.

Acoustics Module

New Functionality in Version 5.4

The following new functionality is available:

- The new Port boundary condition is used to excite and absorb acoustic waves that enter or leave waveguide structures, like ducts and channels. The condition is available in the Pressure Acoustics, Frequency Domain interface.
- A new Nonlinear Acoustics Westervelt feature is available with the Pressure Acoustics, Time Domain interface. This feature can be used to model high-amplitude acoustics in the time domain as experienced in some transducers, acoustic horns, and ultrasound applications. The new functionality also includes shock-capturing stabilization and specialized solver handling.
- The Atmosphere attenuation and the Ocean attenuation fluid models are used to model the attenuation for atmospheric air and for sea water. They are included in the frequency-domain Pressure Acoustics interfaces and the Ray Acoustics interface.
- The new Exterior Field Calculation feature represents an update of the existing Far-Field Calculation feature.
- The Exterior Field Calculation feature can now be used in all transient Pressure Acoustics interfaces when combined with a Time to Frequency FFT study step.
- The Pressure Acoustics, Boundary Element interface now has multiphysics couplings to Thermoviscous Acoustics and Poroelastic Waves as well as Interior Velocity and Interior Displacement boundary conditions. In addition, all the impedance models available for the FEM interface are also available for the BEM interface.
- The linearized Navier-Stokes and Thermoviscous Acoustics physics interfaces now have an option to use an adiabatic formulation for the governing equations.
- Gradient-term suppression (GTS) stabilization is now available for the Linearized Navier-Stokes Equation physics interfaces.
- Several updates to the default plots generated by the physics interface of the Acoustics Module: New color schemes, eigenfrequency list for eigenfrequency studies, new and updated Exterior Field default plots. The Far-Field plots have been renamed to Radiation Pattern plots.
- A new Modulated Gaussian Pulse option is available for background and incident fields in the transient Pressure Acoustics interfaces.

- The Air and Water, Liquid materials have been updated with acoustics-specific quantities like the value for the bulk viscosity and the coefficient of nonlinearity.
- The Ray Acoustics interface now includes improved absorption/attenuation calculation method in domains, improved behavior for Rayleigh roughness, new input for Exterior and Unmeshed Domains material data, and two new options for the Compute Intensity feature.

New and Updated Model Examples in 5.4

- *Duct with Right Angled Bend* model studies the acoustic behavior of a duct or waveguide with a right-angled bend. The model uses Port boundary conditions at the inlet and outlet. The ports can capture and treat nonplane propagating modes in waveguides, extending the analysis above the first cutoff frequency. The transmission loss and the scattering coefficients of the system are determined.
- *Small Concert Hall Acoustics* analyzes the acoustics of a small concert hall using the Ray Acoustics physics interface. The model setup includes an omnidirectional sound source, Wall boundary conditions for specular and diffuse scattering, sound pressure evaluation on a boundary, a Receiver data set, an Impulse Response plot, and an energy decay curve. The results are compared to a simple reverberation time estimate.
- *Lumped Receiver with Full Vibroacoustic Coupling* shows the analysis of the interaction between a vibration isolation mounting and a miniature hearing aid transducer (a Knowles TEC30033 balanced armature receiver) using a lumped representation of the transducer. The lumped model is simplified as an equivalent electroacoustic circuit. The vibration and acoustic characteristics of the lumped model are then coupled to a multiphysics model of the vibration isolation system to achieve a full system analysis.
- *Loudspeaker Driver — Transient Analysis* performs a full transient analysis of a loudspeaker driver, which allows the modeling of nonlinear effects. It extends the linear frequency domain analysis done in the *Loudspeaker Driver* tutorial model. The output includes the total harmonic distortion (THD) and the dynamic BL curve. The model also describes how to calculate the intermodulation distortion (IMD).
- *Nonlinear Acoustics — Modeling of the 1D Westervelt Equation* shows how to use the new Nonlinear Acoustics (Westervelt) model in the Pressure Acoustics, Time Domain physics interface. The model results are compared to an analytical solution.

- The *Absorptive Muffler and Perforated Muffler* models have been updated to use the new Port boundary condition.
- *Vibrating Particle Water* has been updated to use the new Adiabatic formulation option in the Thermoviscous Acoustics interface.
- In the *Helmholtz Resonator with Flow* and *Ultrasound Flow Meter with Generic Time-of-Flight Configuration* models, the CFD part of the model has been updated to use the Fully Developed Turbulent Flow Inlet option. This update significantly simplifies setting up the model.

Backward Compatibility with Version 5.3a

For the Linearized Navier-Stokes interfaces, the override rules for boundary conditions have changed. To obtain the old behavior of a model, disable the boundary conditions that should not contribute to the solution.

Backward Compatibility with Version 5.3

For the Linearized Euler, Transient interface, initial conditions are only used for the dependent variables. The initial conditions for the time derivatives have therefore been removed. For a model with fields rho, u, and p, remove the Java[®] lines setting drho/dt, du/dt, and dp/dt (these parameters change names when the field names change).

Backward Compatibility with Version 5.2

The default settings have changed for the Incident Pressure Field in the Pressure Acoustics interfaces. Add the following line to obtain the old behavior of this feature when ipf1 is a subfeature to pwr1:

```
model.physics("acpr").feature("pwr1").feature("ipf1").
set("c", "acpr.c_c");
```

Backward Compatibility with Version 5.1

- Plane Wave Radiation has been removed from the Linearized Potential Flow, Frequency Domain and Linearized Potential Flow, Transient interfaces in 1D axisymmetry.
- A new Lagrange multiplier variable (`ta.lm_slip`) has been added to the slip conditions in the Thermoacoustics, Frequency Domain interface. If you run a

model with a segregated solver, you will need to add this variable to the group containing the velocity degree of freedom. You can also regenerate the default solver. In the API, you need to add `comp1_ta_lm_slip` to the declaration of the segregated step; for example, adding:

```
model.sol("sol1").feature("s1").feature("se1").feature("ss1").  
set("segvar", new String[]{"comp1_p", "comp1_u", "comp1_T",  
"comp1_ta_lm_slip"});
```

- The Waveguide end impedance option has been removed from 2D, 1D axisymmetric, and 1D as it had no physical meaning in these space dimensions.

Backward Compatibility with Version 4.3a and Older

- The old **Perfectly Matched Layer** (PML) node that is located under the physics node has been discontinued. From the next version on, only the **Perfectly Matched Layer** nodes defined under the **Definitions** node are supported.
- In order for the old **Perfectly Matched Layer** (PML) node to be backward compatible, the PML feature must be placed after any domain **Monopole Source**, domain **Dipole Source**, or **Background Pressure Field** nodes.

Backward Compatibility with Version 4.3

- The symbols for volume sources for the acoustics monopole (Q_m ; was Q) and dipole (\mathbf{q}_d ; was \mathbf{q}) volume sources for pressure acoustics have changed.
- The default value for the **Typical wave speed** property in the Acoustic-Piezoelectric Interaction, Frequency Domain interface has changed to 343 m/s.

Batteries & Fuel Cells Module

New Functionality in Version 5.4

- A tortuosity option has been added for effective transport parameter correction for domain nodes modeling transport in porous media. This option is available in the Primary Current Distribution and Secondary Current Distribution; Tertiary Current Distribution, Nernst-Planck; Lithium-Ion Battery; Battery with Binary Electrode; Single Particle Battery; and Electrophoretic Transport interfaces.
- An Equilibrium Potential Handling (Primary Condition) setting has been added to the Electrode Surface and Porous Electrode Surface nodes. When there are multiple Electrode Reaction/Porous Electrode Reaction subnodes present, this setting controls which equilibrium potential will be used in Current Distribution Initialization studies.
- A new Ion Exchange Membrane domain feature has been added to the Tertiary Current Distribution, Nernst-Planck interface.
- An improved Ion Exchange Membrane Boundary feature is now available in the Tertiary Current Distribution, Nernst-Planck interface that supports transport of multiple ions across the membrane.
- PARDISO is now the default for the Direct solver in the Current Distribution and Battery interfaces.
- Accurate boundary fluxes have been added to the Electrophoretic Transport interface.
- New Equilibrium Reaction and Surface Equilibrium Reaction features are available in the Transport of Concentrated Species interface.
- A new Fast Irreversible Surface Reaction node has been added to the Tertiary Current Distribution, Nernst-Planck and Transport of Diluted Species interfaces.
- A new Fast Irreversible Electrode Reaction electrode kinetics expression type has been added to the Electrode Reaction subnode of the Electrode Surface node in the Electroanalysis and Transport of Diluted Species interfaces. The same feature has also been added to the Internal Electrode Surface and Thin Electrode Surface features in the Tertiary Current Distribution, Nernst-Planck interface.
- The Electrostatics interface now supports the Frequency Domain, Perturbation study step.

- The Electric potential dependent variable now uses quadratic elements in the Tertiary Current Distribution, Nernst-Planck; Lithium-Ion Battery; Battery with Binary Electrode; and Lead-Acid Battery interfaces.
- A new Heat of Reaction section has been added to the Electrode Reaction and Porous Electrode Reaction nodes in the electrochemistry interfaces. The new section supports setting the thermoneutral voltage.
- An improved Periodic Condition feature in the Primary Current Distribution; Secondary Current Distribution; and Tertiary Current Distribution, Nernst-Planck interfaces provides an option to set periodicity separately for the electrolyte and electrode phase, and to set the potential offset.
- A new Lumped Battery interface is available. It provides a simplified (compared to, for instance, the Lithium-Ion Battery or Single Particle Battery interfaces) approach to battery modeling.
- A new Battery Equivalent Circuit entry is available in the Model Wizard.
- The Electrical Circuit interface has been added to the Batteries & Fuel Cells Module.
- The Battery Open Circuit Voltage node in the Electrical Circuit interface is now available in the Batteries & Fuel Cells Module.
- dE_{eqdT} data for NMC, LTO, and NCA electrodes has been added to the Batteries & Fuel Cells Material Library.
- Updated E_{eq} and cE_{eqref} data for NCA electrodes in the Batteries & Fuel Cells Material Library.
- The Circuit Terminal node and Circuit Terminal boundary condition in the Electrode Surface node have been added to the Batteries & Fuel Cells Module.
- The Circuit Terminal operation node in the Single Particle Battery interface has been added to the Batteries & Fuel Cells Module.
- There is a new **Stress and Strain** section in the **Particle Intercalation** node in the Lithium-Ion Battery and Battery with Binary Electrode interfaces. The section settings are used to calculate stress and strain variables.
- The default solver for the battery interfaces now has the **Reuse sparsity pattern** check box enabled in the **Advanced** solver node's **Assembly Settings** section. This setting typically reduces computation time by around 10% for time-dependent models.
- For both Porous Electrode Reaction and Electrode Reaction nodes in the battery interfaces (Lithium-Ion Battery, Battery with Binary Electrode, Lead-Acid Battery,

and Single Particle Battery), the Equilibrium Potential E_{eq} and $dE_{eq}dT$ parameter inputs are now taken from Materials by default.

- Improved default plots. Vector plots for models not using Porous Electrode domains now make use of streamline plots with the new arrow feature.
- Updated formulation for the Flux and No Flux nodes included in the mass transport interfaces.
 - The flux relative to the convective flux is prescribed instead of the total flux.
 - The new formulation implies that the nodes can be used on a moving mesh geometry without compensating for the boundary velocity.
 - Affected interfaces: Transport of Diluted Species; Transport of Diluted Species in Porous Media; Transport of Concentrated Species; Tertiary Current Distribution, Nernst-Planck; Electroanalysis; and Electrophoretic Transport.
- Updated equation display for governing equations and boundary conditions in the mass transport interfaces. Equations updated to be formulated in terms of the flux relative to the convective flux.
- The new setting **Account for Stefan velocity** is available in the Flux, Mass Fraction, and Electrode Surface Coupling nodes in the Transport of Concentrated Species interface.

New and Updated Applications and Models in Version 5.4

- *Discharge and Self-Discharge of a Lead-Acid Battery* model updated to use the new Equilibrium Potential Handling (Primary Condition) setting.
- *Parameter Estimation of a Time-Dependent Lumped Battery Model* updated to use the new Lumped Battery interface.
- *Vanadium Redox Flow Battery* updated to use the new Ion Exchange Membrane domain node, including automatic Donnan potentials at the internal boundaries. Also, the model now uses the new Fast Irreversible Electrode Reaction kinetics expression.
- *Ohmic Losses and Temperature Distribution in a Passive PEM Fuel Cell* model updated to use the new Thermoneutral Voltage option in the Heat of Reaction section.
- New *Electrode Utilization in a Large Format Lithium-Ion Battery Pouch Cell* tutorial added.
- *Thermal Modeling of a Cylindrical Lithium-Ion Battery in 2D* updated to use the new Lumped Battery interface.

- New *Diffusion-Induced Stress in a Lithium-Ion Battery* model added. The model makes use of the new Stress and Strain section in the Particle Intercalation node.
- New *Equivalent Circuit Model for a Ni-MH Battery* model added. The model makes use of the new Battery Equivalent Circuit entry in the Model Wizard.

Backward Compatibility with Version 5.3a

For both Porous Electrode Reaction and Electrode Reaction nodes, in the battery interfaces (Lithium-Ion Battery, Battery with Binary Electrode, Lead-Acid Battery, and Single Particle Battery), the Equilibrium Potential E_{eq} and dE_{eqdT} parameter inputs are now taken from Materials by default. This may have an impact on the Java[®] API back compatibility if user-defined settings for these two parameters are used.

NO FLUX FORMULATION

The formulation of the No Flux feature has been changed in the mass transport interfaces in version 5.4 so that the flux relative to the convective flux is set to zero. The new formulation is used when opening a new model from a previous version. In the previous formulation, the total flux was set to zero.

Affected interfaces: Transport of Diluted Species, Transport of Diluted Species in Porous Media, Transport of Concentrated Species, Tertiary Current Distribution, Electroanalysis, and Electrophoretic Transport.

CAD Import Module, Design Module, and LiveLink™ Products for CAD

New Functionality in Version 5.4

The CAD file import functionality included with these products has been extended to support new versions for some of the supported file formats (see under Read from File, CAD on www.comsol.com/products/specifications/cad).

CAD IMPORT MODULE GEOMETRY KERNEL UPGRADE

The CAD Import Module, the Design Module, and the LiveLink™ products for CAD utilize the Parasolid® geometry kernel from Siemens PLM for solid modeling operations, geometry repair, and defeaturing. (Without these products, a COMSOL-native geometry modeling kernel is used.) The CAD Import Module released with COMSOL 5.4 includes an upgraded version of the Parasolid kernel. As a result, a number of stability issues have been fixed, which makes the import of CAD models and solid operations more robust.

New Functionality in LiveLink™ for AutoCAD® in Version 5.4

- The name and value of dimensional parameters that are controlled by a formula in the CAD file (read-only parameters) can now be transferred to the COMSOL Multiphysics model during synchronization and become available as parameters in the simulation setup.
- Material selections, which are generated based on the material assignments in the CAD file, are now also available as object selections in the appropriate input selection lists of geometry features following the LiveLink™ feature in a geometry sequence.
- The LiveLink™ interface now supports AutoCAD® 2019.

New Functionality in LiveLink™ for Inventor® in Version 5.4

- The name and value of dimensional parameters that are controlled by a formula in the CAD file (read-only parameters) can now be transferred to the COMSOL Multiphysics model during synchronization and become available as parameters in the simulation setup.

- Material selections, which are generated based on the material assignments in the CAD file, are now also available as object selections in the appropriate input selection lists of geometry features following the LiveLink™ feature in a geometry sequence.
- The LiveLink™ interface now supports Inventor® 2019.
- The updated tutorial *Electrical Heating in a Busbar Assembly* now analyzes a larger busbar assembly designed to conduct a direct current from a current source to the anode in an electrolysis process. Selections defined in the CAD assembly are used to assign materials and boundary settings in the simulation.
 - Application Library path:
LiveLink_for_Inventor/Tutorial_Examples/busbar_llinventor

New Functionality in LiveLink™ for PTC® Creo® Parametric™ in Version 5.4

- Support for PTC® Creo® Parametric™ 2.0 by the LiveLink™ interface will be removed in the next major release of COMSOL Multiphysics®. File import of PTC® Creo® Parametric™ 2.0 will continue to be supported.
- The name and value of dimensional parameters that are controlled by a formula in the CAD file (read-only parameters) can now be transferred to the COMSOL Multiphysics model during synchronization and become available as parameters in the simulation setup.
- Material selections, which are generated based on the material assignments in the CAD file, are now also available as object selections in the appropriate input selection lists of geometry features following the LiveLink™ feature in a geometry sequence.
- Expanding on the functionality that synchronizes selections based on material assignments to the CAD design in PTC® Creo® Parametric™, the LiveLink™ interface now adds support for user-defined selections. In the added COMSOL Selections interface in PTC® Creo® Parametric™, you can define selections that are synchronized to the COMSOL Multiphysics model. You can choose to synchronize selections for assembly components, features, bodies, faces, edges, or points, which become selections in the model when the design is synchronized with the COMSOL Multiphysics model.
- The LiveLink™ interface now supports PTC® Creo® Parametric™ 5.0.

- The new tutorial *Two-Pole Three-Phase Induction Motor* illustrates how to compute the torque of a three-phase squirrel-cage induction motor at various speeds. The torque is computed by averaging the results from simulations performed on 2D cross sections at several locations along the induction motor. The 3D geometry and selections, which are used for the physics settings, are transferred from PTC® Creo® Parametric™ by the LiveLink™ interface. By defining a cutting plane in the 3D geometry, the 2D geometry for the simulation is generated in the COMSOL Multiphysics model by the Cross Section operation, which also retains the selections defined in 3D.
- Application Library path:
LiveLink_for_PTC_Creo_Parametric/Tutorial_Examples/induction_motor_llcreop

New Functionality in LiveLink™ for PTC® Pro/ENGINEER® in Version 5.4

- The name and value of dimensional parameters that are controlled by a formula in the CAD file (read-only parameters) can now be transferred to the COMSOL Multiphysics model during synchronization and become available as parameters in the simulation setup.

New Functionality in LiveLink™ for Revit® in Version 5.4

- The name and value of mass parameters that are controlled by a formula in the CAD file (read-only parameters) can now be transferred to the COMSOL Multiphysics model during synchronization and become available as parameters in the simulation setup.
- The LiveLink™ interface now supports Revit® 2019.

LiveLink™ for Solid Edge®

NEW FUNCTIONALITY IN VERSION 5.4

- The name and value of dimensional parameters that are controlled by a formula in the CAD file (read-only parameters) can now be transferred to the COMSOL Multiphysics model during synchronization and become available as parameters in the simulation setup.

- Material selections, which are generated based on the material assignments in the CAD file, are now also available as object selections in the appropriate input selection lists of geometry features following the LiveLink™ feature in a geometry sequence.
- The LiveLink™ interface now supports Solid Edge® 2019.

BACKWARD COMPATIBILITY WITH VERSION 5.0

The default value for the `keepfree` property of the `LiveLinkSolidEdge` function is now set to `on`. Previously, the default was set to `off`.

LiveLink™ for SOLIDWORKS®

NEW FUNCTIONALITY IN VERSION 5.4

- User-defined selections now support assembly-level pattern features present in the SOLIDWORKS® assembly files.
- Material selections, which are generated based on the material assignments in the CAD file, are now also available as object selections in the appropriate input selection lists of geometry features following the LiveLink™ feature in a geometry sequence.
- The name and value of dimensional parameters that are controlled by a formula in the CAD file (read-only parameters) can now be transferred to the COMSOL Multiphysics model during synchronization and become available as parameters in the simulation setup.

Backward Compatibility with Previous Versions

BACKWARD COMPATIBILITY WITH VERSION 5.3A

The new `fillholes` property has been added for the `Import` function. The following applies:

- The combination of the `fillholes` property set to `on` and the `knit` property set to `solid` is equivalent to the `knit` property set to `solid` in previous versions.
- The combination of the `fillholes` property set to `off` and the `knit` property set to `surface` is equivalent to the `knit` property set to `surface` in version 5.3a.
- The combination of the `fillholes` property set to `on` and the `knit` property set to `surface` is equivalent to the `knit` property set to `surface` in version 5.3 and previous versions.

The new `fillholes` property has been added for the `Knit` function. The `Knit` function with the `fillholes` property set to `on` corresponds to the `Knit` function in previous versions.

BACKWARD COMPATIBILITY WITH VERSION 5.2

Geometry Repair Tolerances

The geometry repair tolerance can be automatic, relative, or absolute in version 5.2a. In previous versions, the repair tolerance was always a relative tolerance.

- If you have not set the relative repair tolerance `repairtol`, you will get the new default repair tolerance type: `auto`. This will result in a valid geometry in almost all cases. In extreme cases, the result can have a different topology than in 5.2.
- In rare cases, if you have set the `repairtol` tolerance, version 5.2a can fail when building the geometry. If so, set `repairtoltype` to `auto` (select **Automatic** from the **Repair tolerance** list) to get the 5.2 behavior.

BACKWARD COMPATIBILITY WITH VERSION 5.0

The default value for the `keepfree` property of the `LiveLinksSOLIDWORKS` function is now set to `on`. Previously, the default was set to `off`.

CFD Module

New Functionality in Version 5.4

- Large Eddy Simulation (LES) is now selectable from the Model Wizard. The three new interfaces LES RBVM (residual-based variational multiscale), LES RBVMWV (residual-based variational multiscale with viscosity), and LES Smagorinsky are all based on variational multiscale methods.
- Two new transport interfaces, Phase Transport and Phase Transport in Porous Media, can be coupled to free flow and porous media flow interfaces, respectively, to simulate n-phase flow.
- New predefined multiphysics coupling between Phase Transport in Porous Media and Darcy's Law interfaces: Multiphase Flow in Porous Media.
- Three new non-Newtonian models for yield-stress fluids: Bingham-Papanastasiou, Herschel-Bulkley-Papanastasiou, and Casson-Papanastasiou.
- The **Fully developed flow** option for Inlet and Outlet features is now available for all turbulence models.
- All turbulence models are available in the Two-Phase Flow, Level Set and Phase Field interfaces.
- New features in Two-Phase Flow include Interior Wetted Wall for both Level Set and Phase Field, and Thin Barrier for Level Set. The Wetted Wall and Interior Wetted Wall features in Two-Phase Flow, Level Set are now compatible with fluid-structure interaction (FSI).
- Other changes in Two-Phase Flow include conservation corrections and linear shape functions as the new default discretization.
- New formulations for the Mixture Model and Euler-Euler Model interfaces.
- New boundary conditions in the Mixture Model interfaces: Mixture Continuity and Periodic Flow Condition.
- An Interior Wall boundary condition is now available in the High Mach Number Flow interfaces.
- Updated formulation for the Flux and No Flux nodes included in the Transport of Diluted Species and Transport of Concentrated Species interfaces. The flux relative to the convective flux is prescribed instead of the total flux.

- A new setting, **Account for Stefan velocity**, is available in the Flux and Mass Fraction nodes in the Transport of Concentrated Species interface.
- The Transient with Initialization study, applicable to some of the Turbulent Flow interfaces, has been renamed Time Dependent with Initialization.
- The new option, **Automatic from physics**, for **Pseudo time stepping**, toggles between **On** and **Off**, depending on whether **Turbulence model type** is set to **RANS** or **None**.
- The default solver for the fluid flow interfaces now has the **Reuse sparsity pattern** check box enabled in the **Advanced** solver node's **Assembly Settings** section. This setting typically reduces computation time by around 10% for time-dependent models.

New Model in Version 5.4.

The new *Two-Phase Flow over a Low Permeable Lens* model, under Multiphase Tutorials in the Application Library, uses the new predefined multiphysics coupling, Multiphase Flow in Porous Media, to simulate two-phase flow in a porous medium containing a low permeable lens.

Backward Compatibility with Version 5.3a

MIXTURE MODEL EQUATION FORMULATION

The Mixture Model equation formulation has been revised to increase the accuracy and facilitate the solution of a number of problem setups. As a consequence of the revision, old models can return slightly different results in the new version compared to previous versions.

EULER-EULER MODEL

The results from models using the Euler-Euler Model interfaces may differ in version 5.4 due to the more general formulation for the viscous stress in the momentum equation for the dispersed phase. The behavior in the old formulation may be reproduced by dividing the dispersed phase viscosity, μ_D , by the variable `ee.phidPos`.

NO FLUX FORMULATION

The formulation of the No Flux feature has been changed in the Transport of Diluted Species interface in version 5.4 so that the flux relative to the convective flux is set to zero. The new formulation is used when opening a model from a previous session. In the previous formulation, the total flux was set to zero.

FULLY DEVELOPED FLOW REPLACES LAMINAR INFLOW AND LAMINAR OUTFLOW

The **Laminar inflow** and **Laminar outflow** options on Inlets and Outlets, respectively, have been replaced with the **Fully developed flow** option. Models built in versions prior to 5.4 will contain warnings about obsolete functionality.

Backward Compatibility with Version 5.0

TRANSPORT OF CONCENTRATED SPECIES

See [Chemical Reaction Engineering Module](#).

Backward Compatibility with Version 4.4

INLET FEATURE

The old inlet features still exist, but have been excluded from the physics context menu. Old models retain old **Inlet** features, but adding a new **Inlet** feature comes with the new functionality. Old Model Java[®] files create **Inlet** features of the old types. The Normal stress condition can still be prescribed on inlets using an **Open Boundary** or a **Boundary Stress** feature.

NEW NAMES FOR MASS TRANSPORT INTERFACES

Running Model Java[®] files from previous versions may fail due to the new default names in the Transport of Diluted Species and Transport of Concentrated Species interfaces. This can be avoided by adding a command that specifies the identifier in accordance with the Name of the interface. For example, when creating a Transport of Concentrated Species interface using the old name (`chcs`):

```
model.physics().create("chcs", "ConcentratedSpecies", "geom1",  
    new String[][]{{"wO2", "wH2O", "wN2"}});
```

Add the following line to specify the identifier accordingly:

```
model.physics("chcs").identifier("chcs");
```

WALL DISTANCE EQUATION

The old Continuity node is still included in the interface but is excluded from the feature list. Hence, old models retain their old Continuity node when opened in version 5.4, and old Model Java[®] files are fully backward compatible.

EULER-EULER MODEL

The results from models using the Euler-Euler Model interfaces may differ in version 5.4 due to the more general formulation for the viscous stress in the momentum equation for the dispersed phase. The behavior in the old formulation may be reproduced by dividing the dispersed phase viscosity, μ_D , by the variable `ee.phidPos`.

FULLY DEVELOPED FLOW REPLACES LAMINAR INFLOW AND LAMINAR OUTFLOW

The **Laminar inflow** and **Laminar outflow** options on Inlets and Outlets respectively have been replaced with the **Fully developed flow** option. Models built in versions prior to 5.4 will contain warnings about obsolete functionality.

Backward Compatibility with Version 4.3b

BUBBLE-INDUCED TURBULENCE IN BUBBLY FLOW

Equation terms accounting for bubble-induced turbulence in the Turbulent Bubbly Flow interface are now added correctly. As a result, models with Turbulent Bubbly Flow now show higher levels of turbulence.

MIXTURE MODEL EQUATION FORMULATION

The Mixture Model equation formulation has been revised to increase the accuracy and facilitate the solution of a number of problem setups. As a consequence of the revision, old models can return slightly different results in the new version than they did in previous versions.

EULER-EULER MODEL

The results from models using the Euler-Euler Model interfaces may differ in version 5.4 due to the more general formulation for the viscous stress in the momentum equation for the dispersed phase. The behavior in the old formulation may be reproduced by dividing the dispersed phase viscosity, μ_D , by the variable `ee.phidPos`.

FULLY DEVELOPED FLOW REPLACES LAMINAR INFLOW AND LAMINAR OUTFLOW

The **Laminar inflow** and **Laminar outflow** options on Inlets and Outlets respectively have been replaced with the **Fully developed flow** option. Models built in versions prior to 5.4 will contain warnings about obsolete functionality.

MIXTURE MODEL EQUATION FORMULATION

The Mixture Model equation formulation has been revised to increase the accuracy and facilitate the solution of a number of problem setups. As a consequence of the revision, old models can return slightly different results in the new version than they did in previous versions. As part of the revision, a penalty diffusion on the dispersed phase has been deactivated, because after the other revisions, it made the equation system too nonlinear and therefore more difficult to converge. However, you can still activate the penalty diffusion in the **Advanced** section of the Mixture Model physics interface settings.

EULER-EULER MODEL

The results from models using the Euler-Euler Model interfaces may differ in version 5.4 due to the more general formulation for the viscous stress in the momentum equation for the dispersed phase. The behavior in the old formulation may be reproduced by dividing the dispersed phase viscosity, μ_D , by the variable `ee.phidPos`.

FULLY DEVELOPED FLOW REPLACES LAMINAR INFLOW AND LAMINAR OUTFLOW

The **Laminar inflow** and **Laminar outflow** options on Inlets and Outlets respectively have been replaced with the **Fully developed flow** option. Models built in versions prior to 5.4 will contain warnings about obsolete functionality.

FLUID-STRUCTURE INTERACTION

The Fluid-Structure Interaction (FSI) multiphysics interface has been updated. The separate `vWall` field is no longer required and has been removed. FSI models from 4.3 and earlier versions that include Time Dependent study steps will be affected in the following ways:

- Model files for Java[®] will fail. Any reference to the `vWall` field must be removed.

- Models older than 4.3 (that is, 4.2a and older) must clear their time-dependent solutions when opened in 5.4. An alternative procedure is to:
 - Open the model in version 4.3.
 - Right-click any study with a time-dependent study step and select *Update Solution*.
 - Save the model.
 - Open the model in version 5.4.

FSI models with only stationary study steps will not be affected. Note that `vWall` will still be available as a variable. Hence, references to `fsi.vWall` made in, for example, another physics interface still work.

BRINKMAN EQUATIONS AND FREE AND POROUS MEDIA FLOW

The momentum equations in the Brinkman Equations interface and the Free and Porous Media Flow interface have been corrected. The term $-Q_{br} \cdot \mathbf{u} / \varepsilon_p^2$ in the right-hand side previously lacked the factor $1/\varepsilon_p^2$, where ε_p is the porosity.

REACTING FLOW INTERFACE NAME CHANGE

The Reacting Flow interfaces have been renamed Reacting Flow in Porous Media. If you open a model using either the Reacting Flow, Diluted Species or the Reacting Flow, Concentrated Species interface, the interface is renamed Reacting Flow in Porous Media.

WEAK CONSTRAINTS UPDATE FOR FLUID FLOW INTERFACES

The weak constraint formulations for the following boundary conditions in the following interfaces have been updated:

Laminar Flow and Creeping Flow

- Symmetry
- The Slip boundary condition in the **Wall** feature

Turbulent Flow, k - ε and Turbulent Flow, k - ω

- Symmetry
- The Wall functions boundary condition in the **Wall** feature
- The Moving Wall (wall functions) boundary condition in the **Wall** feature
- The Slip boundary condition in the **Wall** feature

Turbulent Flow, low-Re k - ε and Turbulent Flow, Spalart-Allmaras

- Symmetry
- The Slip option in the **Wall** feature

Rotating Machinery

- Symmetry
- The Slip boundary condition in the **Wall** feature
- The Wall Functions boundary condition in the **Wall** feature (turbulent flow only)
- The Moving Wall (wall functions) boundary condition in the **Wall** feature (turbulent flow only)
- Rotating Wall (turbulent flow only)

Bubbly Flow

- Symmetry
- The Slip boundary condition for the liquid phase in the **Wall** feature
- The Wall Functions boundary condition for the liquid phase in the **Wall** feature (turbulent flow only)

Mixture Model

- Symmetry
- The Slip boundary condition for the mixture in the **Wall** feature
- The Wall Functions boundary condition for the mixture in the **Wall** feature (turbulent flow only)

Euler-Euler Model, Laminar Flow

- Symmetry
- The Slip boundary condition for either phase in the **Wall** feature
- The Slip boundary condition for either phase in the **Inlet** feature

Brinkman Equations and Free and Porous Media Flow

- Symmetry
- The Slip boundary condition in the **Wall** feature

Two-Phase Flow, Level Set

- Symmetry
- The Slip boundary condition in the **Wall** feature
- The Wetted Wall boundary condition in the **Wall** feature
- The Moving Wetted Wall boundary condition in the **Wall** feature

- The Wall Functions boundary condition in the **Wall** feature (turbulent flow only)
- The Moving Wall (wall functions) boundary condition in the **Wall** feature (turbulent flow only)

Two-Phase Flow, Phase Field

- Symmetry
- The Slip boundary condition in the **Wall** feature
- The Wall function boundary condition in the **Wall** feature (turbulent flow only)
- The Moving Wall (wall functions) boundary condition in the **Wall** feature (turbulent flow only)

Nonisothermal Flow and Conjugate Heat Transfer

- Symmetry
- The Slip boundary condition in the **Wall** feature
- The Wall Functions boundary condition in the **Wall** feature (turbulent flow k - ϵ and turbulent flow k - ω only)
- The Moving Wall (wall functions) boundary condition in the **Wall** feature (turbulent flow k - ϵ and turbulent flow k - ω only)

High Mach Number Flow

- Symmetry
- The Slip boundary condition in the **Wall** feature
- The Wall Functions boundary condition in the **Wall** feature (turbulent flow k - ϵ only)
- The Moving Wall (wall functions) boundary condition in the **Wall** feature (turbulent flow k - ϵ only)

These boundary conditions are now formulated using the same set of Lagrange multipliers as all of the other boundary conditions for the dependent velocity variables. The previously used Lagrange multiplier `un_1m` has been removed.

When you open models saved in version 4.3, they include `un_1m` until the model is resolved. In some cases, occurrences of `un_1m` in the solver sequence must be replaced manually. This is the case if `un_1m` was the only Lagrange multiplier component in a segregated group or the only Lagrange multiplier component of a Vanka smoother. Alternatively, you can generate a new automatic solver sequence. Models saved in versions prior to version 4.3 must either be re-solved in version 5.4 for postprocessing, or opened and re-saved in version 4.3 before being opened in version 5.4.

Weak constraints for the **Interior Wall** feature are no longer available.

REVISION OF THE TURBULENCE MODELS

The formulations of some variables in the turbulence models have been revised in order to improve accuracy. Models using a turbulence model can display a different convergence behavior in version 5.4 than in version 4.3 and the results can differ slightly between the versions.

EULER-EULER MODEL

The results from models using the Euler-Euler Model interfaces may differ in between versions 4.3 and 5.4 due to the more general formulation for the viscous stress in the momentum equation for the dispersed phase. The behavior in the old formulation may be reproduced by dividing the dispersed phase viscosity, μ_D , by the variable `ee.phidPos`.

FULLY DEVELOPED FLOW REPLACES LAMINAR INFLOW AND LAMINAR OUTFLOW

The **Laminar inflow** and **Laminar outflow** options on Inlets and Outlets respectively have been replaced with the **Fully developed flow** option. Models built in versions prior to 5.4 will contain warnings about obsolete functionality.

Chemical Reaction Engineering

Module

New Functionality in Version 5.4

- New features in the Transport of Diluted Species interface: Partition Condition, Surface Reactions, and Fast Irreversible Surface Reaction.
- New features in the Transport of Concentrated Species interface; Equilibrium Reaction and Surface Equilibrium Reaction.
- A new Reacting Flow, Diluted Species coupling feature.
- A new User Defined Species feature is available in the Thermodynamics interface.
- The Thermodynamic System feature replaces the Property Package feature. The feature used for thermodynamics calculation is now called Thermodynamic System.
 - The system phases are reflected in the default label. For example, a Vapor-Liquid System node is created for a system including a vapor and a liquid phase.
- The Thermodynamics node is added automatically when creating a thermodynamics feature (Thermodynamic System or User-Defined Species).
- Parameters and functions are sorted under the respective Thermodynamic System feature.
 - Species properties are collected under a node with the species name.
 - Mixture properties and Equilibrium Calculation nodes are collected under a Mixture node.
 - Properties for a specific phase are collected in nodes with the name of the phase.
- Improved user interface for controlling the thermodynamic models and property models in a Thermodynamic System.
 - Two settings, **Liquid phase model** and **Gas phase model**, are now used to control the thermodynamic models for two-phase and three-phase systems.
 - All property models can be inspected and edited in a table.
- Improved user interface for binary interaction parameters in a Thermodynamic System.
 - All binary interaction parameters currently in use can be inspected and edited in a table.

- Improved user interface for the Maxwell-Stefan diffusivities in the Transport of Concentrated Species interface. Using the table input, each species pair is clearly shown.
- Support for creating the new Equilibrium Reaction and Surface Equilibrium Reaction features (in the Transport of Concentrated Species interface), from the Generate Space-Dependent Model feature in the Reaction Engineering interface.
- Support for automatic definition of the equilibrium constant is now available in the Reaction Engineering interface and the Chemistry interface.
- Support for automatic evaluation of the heat of reactions for isothermal condition is now available in the Reaction Engineering interface and the Chemistry interface when fully coupled to a Thermodynamic System.
- Improved sorting of postprocessing variables in mass transport interfaces.
- Updated formulation for the Flux and No Flux nodes included in the mass transport interfaces.
 - The flux relative to the convective flux is prescribed instead of the total flux.
 - The new formulation implies that the nodes can be used on a moving mesh geometry without compensating for the boundary velocity.
 - Affected interfaces: Transport of Diluted Species, Transport of Diluted Species in Porous Media, Transport of Concentrated Species, Nernst-Planck Equations, and Electrophoretic Transport.
- Updated equation display for governing equations and boundary conditions in the mass transport interfaces. Equations updated to be formulated in terms of the flux relative to the convective flux.
- A new setting, **Account for Stefan velocity**, is available in the Flux and Mass Fraction nodes in the Transport of Concentrated Species interface.

New Application and Updated Models in Version 5.4

New application: *Equilibrium Reaction – Gas Phase Conversion of Ethylene to Ethanol*. This application can be used to investigate how equilibrium compositions are affected by initial conditions and operating conditions.

The *Separation Through Dialysis* model has been updated to use the new Partition Condition feature. The model now includes a single Transport of Diluted Species interface.

The *Dissociation in a Tubular Reactor* model has been updated to use the Thermodynamics functionality. The model uses the new User Defined Species feature, as well as the new automatic definition of the equilibrium constant. The model is now available in the Thermodynamics folder of the Application Library included with the Chemical Reaction Engineering Module.

The *HI Batch Reactor* model has been updated to use the Thermodynamics functionality. The model is now available in the Thermodynamics folder of the Application Library included with the Chemical Reaction Engineering Module.

Backward Compatibility with Version 5.3a and Earlier

NO FLUX FORMULATION

The formulation of the No Flux feature has been changed in the Transport of Diluted Species interface in version 5.4 so that the flux relative to the convective flux is set to zero. The new formulation is used when opening a model from a previous session. In the previous formulation, the total flux was set to zero.

Affected interfaces: Transport of Diluted Species, Transport of Diluted Species in Porous Media, Transport of Concentrated Species, Nernst-Planck Equations, and Electrophoretic Transport.

Backward Compatibility with Version 5.0 and Earlier

REACTION ENGINEERING AND CHEMISTRY INTERFACES

The API syntax for creating surface species has changed. Running Java[®] files from previous versions may fail due to the new default name indexes, including **ads** for surface species and **s** for solids in the Reaction Engineering and Chemistry interfaces. The surface species names and solid species names need to be changed accordingly in any Java[®] code:

- Change any species index (**s**) to (**ads**)
- Change any species index (**S**) to (**s**)

The following shows an example of the needed code change:

Version 5.0 and earlier:

```
model.physics("re").feature("rch1").set("formula", "A+B(S)=>C+D(s)");
model.physics("re").feature("spec1").set("specName", "As(s)");
model.physics("chem").feature("rch1").set("formula", "A+B(S)=>C+D(s)");
```



```
model.physics("chem").feature("spec1").set("specName", "As(s)");
```

Change this to

Version 5.2 and later:

```
model.physics("re").feature("rch1").set("formula", "A+B(s)=>C+D(ads)");  
model.physics("re").feature("spec1").set("specName", "As(ads)");  
model.physics("chem").feature("rch1").set("formula", "A+B(s)=>C+D(ads)");  
model.physics("chem").feature("spec1").set("specName", "As(ads)");
```

TRANSPORT OF CONCENTRATED SPECIES INTERFACE

Updated Internal Variable Names

The internal variable names of vector and tensor components have changed in the *Transport of Concentrated Species* interface. In most cases, users will not be affected by this change. When opening a model file created in a previous version, the previous variable definitions remain. However, when updating or recomputing the solution, new variable names will be generated. This means that if any of these variable names have been used in a user-defined expression or plot, they need to be updated manually as described below.

In version 5.1, the spatial direction indicator string (x, xy, and so on) has been moved to the end of the variable name. This applies to all vector and tensor variables generated by the interface.

For example, the diffusive flux vector components have changed (in 5.0 and earlier) from

```
tcs.dfluxx_w1, tcs.dfluxy_w1, tcs.dfluxz_w1
```

to (in 5.2 and later)

```
tcs.dflux_w1x, tcs.dflux_w1y, tcs.dflux_w1z
```

and the components of the diffusion tensor, defined by the Fick's Law diffusion model, have been changed (in 5.0 and earlier) from

```
tcs.Dfxx_w1, tcs.Dfxy_w1, tcs.Dfxz_w1, tcs.Dfyx_w1, ...
```

to (in 5.2 and later)

```
tcs.Df_w1xx, tcs.Df_w1xy, tcs.Df_w1xz, tcs.Df_w1yx, ...
```

Java® API Syntax Changes

The API syntax for setting physics properties has been changed in the Transport of Concentrated Species interface. Now the property name corresponds to the name of

the GUI section containing the property. In order to run Java® files from previous versions, apply the syntax changes as given in the examples below:

Version 5.0 and earlier:

```
model.physics("tcs").prop("DiffusionModel").setIndex("DiffusionModel",
"MaxwellStefan", 0);
model.physics("tcs").prop("DiffusionModel").setIndex("DiffusionModel",
"MixtureAveraged", 0);
model.physics("tcs").prop("DiffusionModel").setIndex("DiffusionModel",
"FicksLaw", 0);
model.physics("tcs").prop("Convection").setIndex("Convection", "0", 0);
model.physics("tcs").prop("Migration").setIndex("Migration", "0", 0);
model.physics("tcs").prop("Regularization").setIndex("Regularization", "1",
0);
```

For version 5.2 and later, change this to:

```
model.physics("tcs").prop("TransportMechanism").set("DiffusionModel",
"MaxwellStefan");
model.physics("tcs").prop("TransportMechanism").set("DiffusionModel",
"MixtureAveraged");
model.physics("tcs").prop("TransportMechanism").set("DiffusionModel",
"FicksLaw");
model.physics("tcs").prop("TransportMechanism").set("Convection", false);
model.physics("tcs").prop("TransportMechanism").set("Migration", false);
model.physics("tcs").prop("AdvancedSettings").set("Regularization", "0");
```

SURFACE REACTIONS INTERFACE

Updated Internal Variable Names

The same pattern for vector and tensor variable name changes, as described in the [Transport of Concentrated Species Interface](#) section above, holds true for the Surface Reactions interface in version 5.4.

Java® API Syntax Changes

The API syntax for setting the diffusion tensor of surface species, the molar mass and density of the bulk species, and initial values have been changed in the Surface Reactions interface.

Setting Diffusion Tensor

The surface species name needs to be included in the first argument of the set method. In order to run Model Java® files from previous versions, apply the syntax changes as shown in the example below:

Version 5.0 and earlier:

```
model.physics("sr").feature("sp1").set("D_0", new String[]{"Dcs1", "0", "0",
"0", "Dcs1", "0", "0", "0", "Dcs1"});
```

```
model.physics("sr").feature("sp1").set("D_1", new String[]{"Dcs2", "0", "0",  
"0", "Dcs2", "0", "0", "0", "Dcs2"});
```

For version 5.2 and later, change this to:

```
model.physics("sr").feature("sp1").set("D_cs1", new String[]{"Dcs1", "0",  
"0", "0", "Dcs1", "0", "0", "0", "Dcs1"});  
model.physics("sr").feature("sp1").set("D_cs2", new String[]{"Dcs2", "0",  
"0", "0", "Dcs2", "0", "0", "0", "Dcs2"});
```

Setting Molar Mass and Density

The bulk species name needs to be included in the first argument of the `setIndex` method. Apply the Java® API syntax changes as shown in the example below:

Version 5.0 and earlier:

```
model.physics("sr").feature("sp1").set("M_0", "Mcb1");  
model.physics("sr").feature("sp1").set("M_1", "Mcb2");  
  
model.physics("sr").feature("sp1").set("rho_0", "rhocb1");  
model.physics("sr").feature("sp1").set("rho_1", "rhocb2");
```

For version 5.2 and later, change this to:

```
model.physics("sr").feature("sp1").setIndex("M_cb1", "Mcb1", 0);  
model.physics("sr").feature("sp1").setIndex("M_cb2", "Mcb2", 0);  
  
model.physics("sr").feature("sp1").setIndex("rho_cb1", "rhocb1", 0);  
model.physics("sr").feature("sp1").setIndex("rho_cb2", "rhocb2", 0);
```

Setting Initial Values

Apply the Java® API syntax changes as shown in the example below:

Version 5.0 and earlier:

```
model.physics("sr").feature("init1").set("cs1", "cs10");  
model.physics("sr").feature("init1").set("cs2", "cs20");  
model.physics("sr").feature("init1").set("cb1", "cb10");  
model.physics("sr").feature("init1").set("cb2", "cb20");
```

For version 5.2 and later, change this to:

```
model.physics("sr").feature("init1").setIndex("initcs", "cs10", 0);  
model.physics("sr").feature("init1").setIndex("initcs", "cs20", 1);  
model.physics("sr").feature("init1").setIndex("initcb", "cb10", 0);  
model.physics("sr").feature("init1").setIndex("initcb", "cb20", 1);
```

Corrosion Module

New Functionality in Version 5.4

- A tortuosity option has been added for effective transport parameter correction for domain nodes modeling transport in porous media. This option is available in the Primary Current Distribution and Secondary Current Distribution; Tertiary Current Distribution, Nernst-Planck; Lithium-Ion Battery; Battery with Binary Electrode; Single Particle Battery; and Electrophoretic Transport interfaces.
- An Equilibrium Potential Handling (Primary Condition) setting has been added to the Electrode Surface and Porous Electrode Surface nodes. When there are multiple Electrode Reaction/Porous Electrode Reaction subnodes present, this setting controls which equilibrium potential will be used in Current Distribution Initialization studies.
- A new Ion Exchange Membrane domain feature has been added to the Tertiary Current Distribution, Nernst-Planck interface.
- An improved Ion Exchange Membrane Boundary feature is now available in the Tertiary Current Distribution, Nernst-Planck interface that supports transport of multiple ions across the membrane.
- PARDISO is now the default for the Direct solver in the Current Distribution and Battery interfaces.
- Accurate boundary fluxes have been added to the Electrophoretic Transport interface.
- A new Fast Irreversible Surface Reaction node has been added to the Tertiary Current Distribution, Nernst-Planck and Transport of Diluted Species interfaces.
- A new Fast Irreversible Electrode Reaction electrode kinetics expression type has been added to the Electrode Reaction subnode of the Electrode Surface node in the Electroanalysis and Transport of Diluted Species interfaces. The same feature has also been added to the Internal Electrode Surface and Thin Electrode Surface features in the Tertiary Current Distribution, Nernst-Planck interface.
- The Electrostatics interface now supports the Frequency Domain, Perturbation study step.
- The Electric Potential dependent variable now uses quadratic elements in the Tertiary Current Distribution, Nernst-Planck interface.

- A new Heat of Reaction section has been added to the Electrode Reaction and Porous Electrode Reaction nodes in the electrochemistry interfaces. The new section supports setting the thermoneutral voltage.
- An improved Periodic Condition feature in the Primary Current Distribution; Secondary Current Distribution; and Tertiary Current Distribution, Nernst-Planck interfaces provides an option to set periodicity separately for the electrolyte and electrode phase, and to set the potential offset.
- Improved point/edge (2D/3D) conditions at the intersection between a dissolving (corroding) Deforming Electrode Surface multiphysics node and a Nondeforming Boundary multiphysics node.
- The Level Set interface is now available in the Corrosion Module.
- Improved default plots. Vector plots for models not using Porous Electrode domains now make use of streamline plots with the new arrow feature.
- Updated formulation for the Flux and No Flux nodes included in the mass transport interfaces.
 - The flux relative to the convective flux is prescribed instead of the total flux.
 - The new formulation implies that the nodes can be used on a moving mesh geometry without compensating for the boundary velocity.
 - Affected interfaces: Transport of Diluted Species: Transport of Diluted Species in Porous Media, Tertiary Current Distribution, Nernst-Planck; Electroanalysis; and Electrophoretic Transport.
- Updated equation display for governing equations and boundary conditions in the mass transport interfaces. Updated equations are formulated in terms of the flux relative to the convective flux.

New and Updated Models in Version 5.4

A new *Under-Deposit Corrosion* model, using the Level Set interface, is now available with the Corrosion Module Application Library.

A new *Localized Corrosion* model, using the Level Set interface, is now available with the Corrosion Module Application Library.

NO FLUX FORMULATION

The formulation of the No Flux feature has been changed in the Transport of Diluted Species interface in version 5.4 so that the flux relative to the convective flux is set to zero. The new formulation is used when opening a model from a previous session. In the previous formulation, the total flux was set to zero.

Affected interfaces: Transport of Diluted Species; Transport of Diluted Species in Porous Media; Tertiary Current Distribution, Nernst-Planck; Electroanalysis; and Electrophoretic Transport.

ECAD Import Module

New Functionality in Version 5.4

- Support for the ODB++(X) and NETEX-G formats will be removed in the next major release of COMSOL Multiphysics.
- For PCB files (ODB++® and IPC-2581) the import generates domain and boundary selections based on the electrical nets defined in the file.
- The buttons **Select All Metal And Dielectric Layers for Import** and **Clear All Imports** have been added to the **Layers to import** table to enable a more efficient workflow when importing ECAD files.

Backward Compatibility with Version 5.3a

The `elevation` property of the `Import` function is now also initialized when setting the `manualelevation` property to `off`. Previously, it was only initialized when setting the `filename` property, the `manualelevation` property is `off`, and the value of either the `height` or `importtype` properties is changed.

Backward Compatibility with Version 5.2a

- The default value for the `sellayer` property of the `Import` function is now `on`. Previously, the default was `off`.
- The default value for the `sellayershow` property of the `Import` function is now `all`. Previously, the default was `dom`.

Backward Compatibility with Version 5.0

- The default value for the `grouping` property of the `Import` function is now `layer`. Previously, the default was `all`.
- For ODB++® and ODB++(X) files (the `type` property is `odb`), new rules apply for the initialization of the `importlayer` property of the `Import` function. Now, only layers of the types *Metal* and *Dielectric* are initialized with the string `on` in the `importlayer` string array. Previously, this also included layers of the type *Drill*.

Electrochemistry Module

New Functionality in Version 5.4

- A tortuosity option has been added for effective transport parameter correction for domain nodes modeling transport in porous media. This option is available in the Primary Current Distribution and Secondary Current Distribution; Tertiary Current Distribution, Nernst-Planck; Lithium-Ion Battery; Battery with Binary Electrode; Single Particle Battery; and Electrophoretic Transport interfaces.
- An Equilibrium Potential Handling (Primary Condition) setting has been added to the Electrode Surface and Porous Electrode Surface nodes. When there are multiple Electrode Reaction/Porous Electrode Reaction subnodes present, this setting controls which equilibrium potential will be used in Current Distribution Initialization studies.
- A new Ion Exchange Membrane domain feature has been added to the Tertiary Current Distribution, Nernst-Planck interface.
- An improved Ion Exchange Membrane Boundary feature is now available in the Tertiary Current Distribution, Nernst-Planck interface that supports transport of multiple ions across the membrane.
- PARDISO is now the default for the Direct solver in the Current Distribution and Battery interfaces.
- Accurate boundary fluxes have been added to the Electrophoretic Transport interface.
- A new Fast Irreversible Surface Reaction node has been added to the Tertiary Current Distribution, Nernst-Planck and Transport of Diluted Species interfaces.
- A new Fast Irreversible Electrode Reaction electrode kinetics expression type has been added to the Electrode Reaction subnode of the Electrode Surface node in the Electroanalysis and Transport of Diluted Species interfaces. The same feature has also been added to the Internal Electrode Surface and Thin Electrode Surface features in the Tertiary Current Distribution, Nernst-Planck interface.
- The Electrostatics interface now supports the Frequency Domain, Perturbation study step.
- The Electric Potential dependent variable now uses quadratic elements in the Tertiary Current Distribution, Nernst-Planck interface.

- A new Heat of Reaction section has been added to the Electrode Reaction and Porous Electrode Reaction nodes in the electrochemistry interfaces. The new section supports setting the thermoneutral voltage.
- An improved Periodic Condition feature in the Primary Current Distribution; Secondary Current Distribution; and Tertiary Current Distribution, Nernst-Planck interfaces provides an option to set periodicity separately for the electrolyte and electrode phase, and to set the potential offset.
- Improved default plots. Vector plots for models not using Porous Electrode domains now make use of streamline plots with the new arrow feature.
- Updated formulation for the Flux and No Flux nodes included in the mass transport interfaces.
 - The flux relative to the convective flux is prescribed instead of the total flux.
 - The new formulation implies that the nodes can be used on a moving mesh geometry without compensating for the boundary velocity.
 - Affected interfaces: Transport of Diluted Species: Transport of Diluted Species in Porous Media, Tertiary Current Distribution, Nernst-Planck; Electroanalysis; and Electrophoretic Transport.
- Updated equation display for governing equations and boundary conditions in the mass transport interfaces. Updated equations are formulated in terms of the flux relative to the convective flux.
- Updated formulation for the Flux and the No Flux nodes included in the mass transport interfaces.
 - The flux relative to the convective flux is prescribed instead of the total flux.
 - The new formulation implies that the nodes can be used on a moving mesh geometry without compensating for the boundary velocity.
 - Affected interfaces: Transport of Diluted Species, Transport of Diluted Species in Porous Media, Tertiary Current Distribution, Nernst-Planck; Electroanalysis, and Electrophoretic Transport.
- Updated equation display for governing equations and boundary conditions in the mass transport interfaces. Equations are updated to be formulated in terms of the flux relative to the convective flux.

New and Updated Models in Version 5.4

The *Electrodialysis* model has been updated to make use of the new Ion Exchange Membrane domain feature and the improved Periodic Condition feature in the Tertiary Current Distribution, Nernst-Planck interface.

Backward Compatibility with Version 5.3a

NO FLUX FORMULATION

The formulation of the No Flux feature has been changed in the Transport of Diluted Species interface in version 5.4 so that the flux relative to the convective flux is set to zero. The new formulation is used when opening a model from a previous session. In the previous formulation, the total flux was set to zero.

Affected interfaces: Transport of Diluted Species; Transport of Diluted Species in Porous Media; Tertiary Current Distribution, Nernst-Planck; Electroanalysis; and Electrophoretic Transport.

Electrodeposition Module

New Functionality in Version 5.4

- A tortuosity option has been added for effective transport parameter correction for domain nodes modeling transport in porous media. This option is available in the Primary Current Distribution and Secondary Current Distribution; Tertiary Current Distribution, Nernst-Planck; Lithium-Ion Battery; Battery with Binary Electrode; Single Particle Battery; and Electrophoretic Transport interfaces.
- An Equilibrium Potential Handling (Primary Condition) setting has been added to the Electrode Surface and Porous Electrode Surface nodes. When there are multiple Electrode Reaction/Porous Electrode Reaction subnodes present, this setting controls which equilibrium potential will be used in Current Distribution Initialization studies.
- A new Ion Exchange Membrane domain feature has been added to the Tertiary Current Distribution, Nernst-Planck interface.
- An improved Ion Exchange Membrane Boundary feature is now available in the Tertiary Current Distribution, Nernst-Planck interface that supports transport of multiple ions across the membrane.
- PARDISO is now the default for the Direct solver in the Current Distribution and Battery interfaces.
- Accurate boundary fluxes have been added to the Electrophoretic Transport interface.
- A new Fast Irreversible Surface Reaction node has been added to the Tertiary Current Distribution, Nernst-Planck and Transport of Diluted Species interfaces.
- A new Fast Irreversible Electrode Reaction electrode kinetics expression type has been added to the Electrode Reaction subnode of the Electrode Surface node in the Electroanalysis and Transport of Diluted Species interfaces. The same feature has also been added to the Internal Electrode Surface and Thin Electrode Surface features in the Tertiary Current Distribution, Nernst-Planck interface.
- The Electrostatics interface now supports the Frequency Domain, Perturbation study step.
- The Electric Potential dependent variable now uses quadratic elements in the Tertiary Current Distribution, Nernst-Planck interface.

- A new Heat of Reaction section has been added to the Electrode Reaction and Porous Electrode Reaction nodes in the electrochemistry interfaces. The new section supports setting the thermoneutral voltage.
- An improved Periodic Condition feature in the Primary Current Distribution; Secondary Current Distribution; and Tertiary Current Distribution, Nernst-Planck interfaces provides an option to set periodicity separately for the electrolyte and electrode phase, and to set the potential offset.
- Improved point/edge (2D/3D) conditions at the intersection between a dissolving (corroding) Deforming Electrode Surface multiphysics node and a Nondeforming Boundary multiphysics node.
- Improved default plots. Vector plots for models not using Porous Electrode domains now make use of streamline plots with the new arrow feature.
- Updated formulation for the Flux and No Flux nodes included in the mass transport interfaces.
 - The flux relative to the convective flux is prescribed instead of the total flux.
 - The new formulation implies that the nodes can be used on a moving mesh geometry without compensating for the boundary velocity.
 - Affected interfaces: Transport of Diluted Species; Transport of Diluted Species in Porous Media; Tertiary Current Distribution, Nernst-Planck; Electroanalysis; and Electrophoretic Transport.
- Updated equation display for governing equations and boundary conditions in the mass transport interfaces. Updated equations are formulated in terms of the flux relative to the convective flux.
- Updated formulation for the Flux and the No Flux nodes included in the mass transport interfaces.
 - The flux relative to the convective flux is prescribed instead of the total flux.
 - The new formulation implies that the nodes can be used on a moving mesh geometry without compensating for the boundary velocity.
 - Affected interfaces: Transport of Diluted Species, Transport of Diluted Species in Porous Media, Tertiary Current Distribution, Nernst-Planck; Electroanalysis, and Electrophoretic Transport.
- Updated equation display for governing equations and boundary conditions in the mass transport interfaces. Equations are updated to be formulated in terms of the flux relative to the convective flux.

New and Updated Models in Version 5.4

There is a new *Pulse Reverse Plating* tutorial in the Electrodeposition Application Library.

Backward Compatibility with Version 5.3a

NO FLUX FORMULATION

The formulation of the No Flux feature has been changed in the Transport of Diluted Species interface in version 5.4 so that the flux relative to the convective flux is set to zero. The new formulation is used when opening a model from a previous session. In the previous formulation, the total flux was set to zero.

Affected interfaces: Transport of Diluted Species; Transport of Diluted Species in Porous Media; Tertiary Current Distribution, Nernst-Planck; Electroanalysis; and Electrophoretic Transport.

Fatigue Module

New Functionality in Version 5.4

FATIGUE EVALUATION FOR MEMBRANES

Fatigue evaluation is now available also for stresses and strains computed in the Membrane interface. This can also be used for obtaining more accurate results on the boundaries of solid bodies. In this case a very thin membrane, only used for stress evaluation, is added on top of the solid.

MEAN STRESS CORRECTION

In the Stress-Life method, it is now possible to take mean stress effects into account. Three methods are available: Goodman, Gerber, and Soderberg.

IMPROVED FATIGUE EVALUATION FOR SHELLS

When performing fatigue evaluation on shells, it is now possible to directly specify that the stresses or strains be taken from either the top or bottom side. The third option is the same as in previous versions — the through-thickness position is taken from the default result location specified in the Shell interface.

Backward Compatibility with Version 5.3

Several of the fatigue laws for low-cycle fatigue are customarily expressed in terms of number of load reversals.

In previous versions, the distinction between *number of load reversals* to failure and *number of load cycles* to failure has not been clear, neither in the theory sections nor in result output. In version 5.3a, the result is always in terms of cycles, and the documentation has been clarified at a number of places.

The affected fatigue models are **Basquin**, **Coffin-Manson**, **Combined Basquin and Coffin-Manson**, **Morrow**, **Smith-Watson-Topper (SWT)**, **Wang-Brown**, and **Fatemi-Socie**.

Geomechanics Module

New Functionality in Version 5.4

DAMAGE MODELING

For brittle materials, damage caused by cracking is an important failure mechanism. The new **Damage** subnode under **Linear Elastic Material** makes it possible to incorporate continuum-based damage models, where the material softens as an effect of the severity of the loading. Various effective scalar damage models based on equivalent strain measures (**Rankine**, **Smooth Rankine**, **Norm of elastic strain tensor**, and **User defined**) are available. You can also select different rules for the damage evolution: **Linear strain softening**, **Exponential strain softening**, and **User defined**.

Softening material models are numerically problematic due to strong localization of the damage. For this reason, you often need to use regularization methods which smooth the damage over a finite region. Two such methods are available: **Crack band** and **Implicit gradient**.

The damage functionality is available with both the Geomechanics Module and the Nonlinear Structural Materials Module. With the Geomechanics Module, you can also use the **Mazars damage for concrete** model, which considers not only tensile cracking but also compressive failure.

SUCTION IN EXTENDED BARCELONA BASIC MODEL

The suction degree of freedom is no longer available in the **Extended Barcelona Basic Model** and has been replaced by a user-defined parameter. The old **Initial suction** parameter has been removed.

MULTIPLICATIVE DECOMPOSITION IN NONLINEAR ELASTIC MATERIAL

Even though the **Nonlinear Elastic Material** is mainly intended for small strains, it can also be used under geometric nonlinearity. The removal of inelastic strains from the total strains is now done using multiplicative decomposition, just as for other materials. By using the check box **Additive strain decomposition**, you can retrieve the formulation previously used.

BURGERS VISCOELASTIC MODEL

A built-in viscoelastic model has been added: the Burgers model. As with the other viscoelastic models, it is accessed by adding a **Viscoelasticity** subnode under the **Nonlinear Elastic Material**.

New Models in Version 5.4

SLOPE STABILITY IN A DAM EMBANKMENT

This example shows how to run a slope stability analysis in a dam embankment, including the pore pressure effect. The fluid transport in the porous soil is described by Darcy's law, and an elastoplastic analysis is carried out with the Mohr-Coulomb model. The material properties are parameterized with respect to the factor of safety using the shear reduction technique.

BRITTLE DAMAGE IN UNIAXIAL TENSION

In this tutorial model, different methods for modeling brittle damage are compared in a uniaxially loaded bar. The new **Damage** feature is used. The localization of the deformation during fracture is studied using different discretization orders and regularization methods.

CRACKING OF A NOTCHED BEAM

In this example, two models for brittle damage are used to evaluate the fracture of a notched concrete beam subjected to three-point bending. The new **Damage** feature is used. The results are compared with experimental data.

WETTING AND DRYING PATHS FOR PARTIALLY SATURATED SOIL WITH THE EXTENDED BARCELONA BASIC SOIL MODEL

Experiments on dry and wet soil samples are performed to understand their behavior under different loading conditions. In this example, the Extended Barcelona Basic (BBM_x) soil model is used to simulate the wetting and drying paths in partially saturated soil samples under cyclic loading.

Heat Transfer Module

New and Improved Functionality in Version 5.4

EXTENDED CAPABILITIES FOR HEAT TRANSFER IN THIN STRUCTURES

The features and interfaces for the modeling of heat transfer in thin structures have been completely redesigned to take advantage of the new **Layered Material** functionality. Layered materials are defined on boundaries but correspond to 3D materials with one or multiple layers. They contain information regarding their composition and the thickness of their layers. It is possible to preview the layered material structure to define properties and heat loads or constraints layer by layer. Additionally, the functionality offers a postprocessing tool to analyze and visualize the solution as a 3D structure.

In the Heat Transfer interface, the **Thin Layer**, **Thin Film**, and **Fracture** features now support layered materials. Once a **Layered material** has been selected in the **Layer Selection** section, the feature is applicable on the boundaries corresponding to the selected layered material. Subfeatures are available for the definition of heat fluxes, heat sources, temperature constraints, and surface-to-ambient radiation.

The Heat Transfer in Shells, Heat Transfer in Films, and Heat Transfer in Fractures interfaces, found under the **Heat Transfer>Thin Structures** branch, contain features for the modeling of heat transfer by conduction and convection in layered materials. The **Solid**, **Fluid**, and **Porous Medium** boundary features solve the energy equation in each layer, either with a general formulation or a lumped model for thermally thin or thermally thick structures. The **Heat Source** and **Heat Source, Interface** features account for heat sources inside the layers and between them. Other edge and boundary interface features are also available to define heat fluxes, temperature constraints, and surface-to-ambient radiation at layer edges and interfaces between layers.

Multiphysics couplings are also supported on layered structures: the **Electromagnetic Heating**, **Layered Shell** and **Thermoelectric Effect**, **Layered Shell** couplings can be added on boundaries where either a Heat Transfer interface with the **Thin Layer**, **Thin Film**, or **Fracture** features or a Heat Transfer in Shells interface is applicable together with an Electric Currents, Layered Shells interface.

SURFACE-TO-SURFACE RADIATION WITH SPECULAR REFLECTION AND TRANSMISSION

A new algorithm, the ray shooting method, is available in the Surface-to-Surface Radiation interface for view factor computation. It handles mixed diffuse-specular reflection on surfaces as well as transmission in semitransparent surfaces. Although rough surfaces reflect incident rays in all direction regardless of the incident direction and are well-modeled by diffuse surfaces, very smooth surfaces reflect the incident ray according to the law of reflection and should be modeled as specular.

By selecting the **Ray shooting** method in the **Surface-to-surface radiation method** list, it makes the new **Opaque Surface** feature available to model mixed diffuse-specular reflection accurately and realistically for a wide range of surfaces. When some layers are not fully opaque and transmit a fraction of the incident irradiation, the new **Semi-Transparent Surface** feature can be used. This is useful, for example, to model windows made of glass that reflect and absorb a fraction of the sun radiation and transmit the rest. The feature provides settings for the surface emissivity, reflectivity, and transmissivity.

EXTENDED CAPABILITIES FOR SURFACE-TO-SURFACE RADIATION WITH WAVELENGTH-DEPENDENT MATERIAL PROPERTIES

The Surface-to-Surface Radiation physics interface now supports an arbitrary number of subdivisions of the spectral domain. The graphical user interfaces of all radiation features; namely, **Diffuse Surface**, **Diffuse Mirror**, **Prescribed Radiosity**, **Opacity**, and **External Radiation Source**, have been reorganized to specify the material properties and user inputs in a more compact way. The surface emissivity, for example, can be provided either by giving its value for each spectral band, or by entering a wavelength-dependent expression, with `rad.lambd` giving access to the wavelength variable. The same improvements also apply to the new wavelength-dependent material properties and user inputs required by the new specular radiation functionalities; namely, surface reflectivity, surface transmissivity, and critical angle.

EXTENDED CAPABILITIES FOR RADIATION IN PARTICIPATING MEDIA

Several improvements have been implemented for the modeling of radiation in participating media. A new option is available to define the radiation scattering, the **Heney-Greenstein** phase function. It offers an interesting compromise between complexity and accuracy to represent scattering due to particles in a participating medium. In order to simplify the phase definition, it is now possible to normalize the phase function for **Linear anisotropic** and **Polynomial anisotropic** scattering types.

In addition, several new quadrature sets have been added for the discrete ordinates method (DOM) in the Radiation in Participating Media and Radiation in Absorbing-Scattering Media interfaces. In addition to the **Level Symmetric Even** set, two additional S_N approximations are available: **Level Symmetric Hybrid** and **Equal Weight Odd**. These sets differ in the moment conditions they satisfy, and introduce 8 to 168 discrete ordinates, depending on the selected order. A T_N approximation, the **Quasi-uniform** weight approximation, is also available. It discretizes the angular space by using a reference octahedron with 8 triangular faces, further discretized in function of the order of the method, to produce from 8 to 512 discrete ordinates. The choice of the quadrature set and order depends on the accuracy needs and the available computational resources.

RADIATION IN ABSORBING-SCATTERING MEDIA INTERFACE

The new Radiation in Absorbing-Scattering Media physics interface can be found under the **Radiation** group of the **Heat Transfer** branch. It provides features to model the propagation, absorption, and scattering of radiation in a semitransparent medium. In particular, it is well-suited for the modeling of light diffusion in a nonemitting medium.

Like the Radiation in Participating Media interface, it solves the Radiative Transfer Equation, but with no emission term. Therefore, the two interfaces share the same discretization methods (**PI approximation** and **Discrete Ordinates Method**) and options for the definition of scattering.

The **Absorbing-Scattering Medium** domain node defines the absorption and scattering properties and the radiative heat source due to absorption. On boundaries, the **Incident Intensity** node allows you to specify the radiative intensity along incident directions, and the **Opaque Surface** node specifies a wall that absorbs all the radiative intensity, producing heat.

REFACTORIZING OF HEAT TRANSFER WITH RADIATION FUNCTIONALITIES INTO MULTIPHYSICS COUPLINGS

The **Surface-to-surface radiation** and **Radiation in participating media** check boxes, previously found under the **Physical Model** section of the Heat Transfer interface, have been removed. This means that the Heat Transfer interface does not support the surface-to-surface radiation features and radiation in participating media features anymore, with the exception of the **Surface-to-Ambient Radiation** feature. In particular, the **Diffuse Surface**, **Diffuse Mirror**, **Prescribed Radiosity**, **Opacity**, and **External Radiation Source** features are now available exclusively in the Surface-to-Surface Radiation interface. The **Radiation in Participating Media**, **Opaque Surface**, **Incident Intensity**, and

Continuity on Interior Boundary features are now available exclusively in the Radiation in Participating Media interface.

To achieve the modeling of heat transfer with radiation, the following multiphysics predefined interfaces and couplings have been added:

- The new Heat Transfer with Surface-to-Surface Radiation predefined multiphysics interface, available under the **Heat Transfer>Radiation** branch, automatically adds a Heat Transfer interface and a Surface-to-Surface Radiation interface, coupled through the **Heat Transfer with Surface-to-Surface Radiation** multiphysics coupling.
- The new Heat Transfer with Radiation in Participating Media predefined multiphysics interface, available under the **Heat Transfer>Radiation** branch, automatically adds a Heat Transfer interface and a Radiation in Participating Media interface, coupled through the **Heat Transfer with Radiation in Participating Media** multiphysics coupling.
- The new Heat Transfer with Radiation in Absorbing-Scattering Media multiphysics interface, available under the **Heat Transfer>Radiation** branch, automatically adds a Heat Transfer interface and a Radiation in Absorbing-Scattering Media interface, coupled through the **Heat Transfer with Radiation in Absorbing-Scattering Media** multiphysics coupling.
- The new Heat Transfer with Radiative Beam in Absorbing Media multiphysics interface, available under the **Heat Transfer>Radiation** branch, automatically adds a Heat Transfer interface and a Radiative Beam in Absorbing Media interface, coupled through the **Heat Transfer with Radiative Beam in Absorbing Media** multiphysics coupling.

REFACTORIZING OF FUNCTIONALITIES INTO SUBFEATURES

In order to achieve a better clarity and usability of the user interface, the following subfeatures have been added. They provide the settings corresponding to a particular configuration of the parent feature:

- The new **Convectively Enhanced Conductivity** subfeature is available under the **Fluid** and **Moist Air** nodes. It provides the settings previously available under the **Equivalent Conductivity for Convection** section of the **Fluid** and **Moist Air** nodes, to account for convective heat flux by enhancing the fluid thermal conductivity according to the Nusselt number.
- The new **Thermal Damage** subfeature is available under the **Biological Tissue** node. It provides the settings previously available under the **Damaged Tissue** section of the **Biological Tissue** node, for the definition of the **Transformation model**.

In addition, the Rosseland approximation for radiative heat transfer in media with high optical thickness is now handled by the new **Optically Thick Participating Medium** subnode, available under most of the domain features of the Heat Transfer interface. It provides the settings to account for the radiative heat source by enhancing the thermal conductivity. Although it is used to model radiation, it is conceptually different from the **Participating Medium** feature that solves for additional variables (radiative intensity or incident radiation) to compute the radiative heating. It replaces the **Rosseland approximation** option of the **Radiation discretization method** list previously available under the **Participating Media Settings** section.

Finally, the new **Phase Change Material** subfeature replaces the **Phase Change Material** feature, and extends its capabilities to solids and porous media. It is available under the **Solid**, **Fluid**, and **Porous Medium** nodes, and provides the same settings as the obsolete feature, to specify the properties of a phase change material according to the apparent heat capacity formulation. When added under the **Solid** node, a single density should be defined for all phases to ensure mass conservation on the material frame.

VOLUME REFERENCE TEMPERATURE MODEL INPUT

The features defined on the material frame, with an input field for the density, have a new model input: the **Volume reference temperature**. This is the temperature for the evaluation of the density in the reference geometry, which should match the geometry in the material frame. By default, this model input is set to the **Common model input** variable, `minput.Tempref`. This ensures mass conservation during the simulation. This concerns the following features: **Solid**, **Porous Medium**, **Biological Tissue**, **Building Material**, **Shape Memory Alloy**, **Thin Layer**, and **Fracture** in the Heat Transfer interface, and **Solid**, **Porous Medium** in the Heat Transfer in Shells interface.

MISCELLANEOUS

- The new **Heat and Moisture Flow** multiphysics interfaces are available under the **Heat Transfer>Heat and Moisture Transport** branch to model coupled heat transfer and moisture transport in air by laminar and turbulent flows. The different interfaces couple the laminar and turbulent versions of the Single-Phase Flow interfaces with the Heat Transfer in Moist Air and Moisture Transport in Air interfaces. The **Heat and Moisture**, **Moisture Flow**, and **Nonisothermal Flow** multiphysics coupling nodes handle turbulent mixing and heat and moisture wall functions for turbulent flows as well as account for the temperature and moisture content dependency of material properties in the fluid flow equations in air.
- The new Heat Transfer in Solids and Fluids interface, available in the **Model Wizard** under the **Heat Transfer** branch, replaces the Heat Transfer interface that was

previously available uniquely through the addition of the Conjugate Heat Transfer multiphysics interface. A **Solid** feature is active by default on all domains, and a **Fluid** feature has an empty selection by default. Similarly to the Heat Transfer in Fluids interface, its default discretization order is 1.

- The **Thermal Insulation** feature is now available on interior boundaries. It ensures that no heat transfer occurs at the boundaries this feature is applied on, hence interrupting the default temperature continuity condition. Note that the default **Thermal Insulation** feature is still not applicable on interior boundaries.
- A new **Ambient Thermal Properties** node is available under the **Definitions** node. It contains the **Ambient Settings** section that was previously available in the Heat Transfer interface. Several **Ambient Thermal Properties** nodes may be added in a single model. The ambient variables defined by this node remain available from the physics features.
- In the **External Radiation Source** global feature, it is now possible to define the **Solar position** from a **Weather station** setting made in an **Ambient Thermal Properties** node under **Definitions**. When an **Ambient Thermal Properties** node is added with **Ambient data** set to **Meteorological data (ASHRAE 2013)** or **Meteorological data (ASHRAE 2017)**, this node can be selected from the **Ambient data** list in the **External Radiation Source** feature.
- The user input for the **Reference temperature**, used in the incompressible flow formulation for nonisothermal flows, and for the computation of the reference enthalpy in the heat transfer equation, has been moved from the **Physical Model** section of the Single-Phase Flow interface to the **Physical Model** section of the Heat Transfer interface. In addition, its default value has changed from 298.15 K to 293.15 K.
- Additional surface plots are generated by default in the **Temperature** plot group of 3D models when the **Thermal Contact** feature is active on some boundaries, to display the temperatures on the upside and downside of the slit defined by the feature on these boundaries.
- In the **Symmetry for Surface-to-Surface Radiation** global feature, it is now possible to define the symmetry planes by selecting points manually from the **Graphics** window, instead of providing their coordinates. The **Point selection** option can be selected from the **Selection method** list.
- The default presmoothing and postsmoothing in the GMRES iterative solver have changed from SOR line to SCGS when the **Nonisothermal Flow** coupling is used. In addition, in 3D models with a **Local Thermal Non-Equilibrium** coupling, the default

Segregated node has been replaced by a **Fully Coupled** node, in order to solve for the two (solid and fluid) temperatures in a single step.

- In 1D models, the default value of the **Cross sectional perimeter** has been changed to $2 * (\text{sqrt}(\text{pi} * \text{ht} . \text{Ac}))$ in the **Physical Model** section, for consistency with the default **Cross sectional area**.
- The definitions of the sensible enthalpy ($\text{ht} . \text{DeltaH}$) and total energy flux ($\text{ht} . \text{Teflux}$) postprocessing variables have been improved in porous media and thin structures.

New Applications in Version 5.4

ISOTHERMAL BOX

This new tutorial model shows how to model an isothermal box used to transport refrigerated medical materials. The goal is to keep the material between a restricted temperature range from 2°C to 8°C for 24 hours. Ice-to-water phase change plays a crucial role. Eutectic plates containing ice act as energy storage by absorbing the exterior warmth during the melting process that occurs at 0°C, hence maintaining the content at low temperature. The simulation is performed for 72 hours so that the temperature evolution can also be observed after the cooling ice packages have completely melted.

ACTION ON STRUCTURES EXPOSED TO FIRE

Five new tutorial models contain verification examples from the European Standard EN 1991-1-2:2010-12, Eurocode 1: Actions on structures — Part 1–2: General actions: Actions on structures exposed to fire.

- The *Action on Structures Exposed to Fire — Cooling Process* tutorial addresses the first verification example of the norm. A transient cooling process due to convection is modeled. The tutorial verifies that the numerical results obtained with COMSOL Multiphysics are within the validity ranges specified in the norm.
- The *Action on Structures Exposed to Fire — Heating Process* tutorial addresses the second verification example of the norm. It describes a heating process due to convection and radiation using a temperature-dependent thermal conductivity. The tutorial verifies that the numerical results obtained with COMSOL Multiphysics are within the validity ranges specified in the norm.
- The *Action on Structures Exposed to Fire — Heat Transfer in Multiple Layers* tutorial addresses the third verification example of the norm. It describes a heating

process through layers using a temperature-dependent thermal conductivity. The tutorial computes heat transfer through layers with different material properties.

- The *Action on Structures Exposed to Fire — Thermal Elongation* tutorial addresses the fourth verification example of the norm. The tutorial verifies that the calculated elongation matches the expected values.
- The *Action on Structures Exposed to Fire — Thermal Stress in a Beam* tutorial addresses the seventh verification example of the norm. It describes the nonlinear mechanical behavior of a beam that is exposed to a temperature gradient. The numerical results obtained with COMSOL Multiphysics are compared with the reference values specified in the norm.

Updated Applications in Version 5.4

USE OF THE NEW INTERFACES AND FEATURES FOR LAYERED MATERIALS

The following tutorials have been updated to use the new **Thin Layer** feature, applicable on layered materials: *Fluid-Structure Interaction in Aluminum Extrusion*, *Double-Pipe Heat Exchanger*, *Finned Pipe*, *Forced Convection Cooling of an Enclosure with Fan and Grille*, *Heating Circuit*, *Sun's Radiation Effect on Two Coolers Placed Under a Parasol*, *Shell-and-Tube Heat Exchanger*, *Heat Transfer in a Surface-Mount Package for a Silicon Chip*, *Natural Convection Cooling of a Vacuum Flask*, *Disk-Stack Heat Sink*, *Composite Thermal Barrier*, and *Silica Glass Block Coated with a Copper Layer*.

The following tutorials have been updated to use the new **Heat Transfer in Shells** interface, applicable to layered materials: *Disk-Stack Heat Sink*, *Shell Conduction*.

USE OF THE NEW HEAT TRANSFER WITH SURFACE-TO-SURFACE RADIATION INTERFACE

Several tutorials have been updated to use the new Heat Transfer with Surface-to-Surface Radiation multiphysics interface: *Electronic Chip Cooling*, *Free Convection in a Light Bulb*, *Flash Method*, *Radiation in a Cavity*, *Sun's Radiation Effect on Two Coolers Placed Under a Parasol*, *Rapid Thermal Annealing*, and *Thermo-Photo-Voltaic Cell*.

USE OF THE NEW HEAT TRANSFER WITH RADIATION IN PARTICIPATING MEDIA INTERFACE

The *Radiative Cooling of a Glass Plate* tutorial has been updated to use the new Heat Transfer with Radiation in Participating Media multiphysics interface.

USE OF THE NEW HEAT TRANSFER IN SOLIDS AND FLUIDS INTERFACE

Several tutorials have been updated to use the new Heat Transfer in Solids and Fluids interface, including *Electronic Chip Cooling*, *Free Convection in a Light Bulb*, *Tin Melting Front*, *Finned Pipe*, *Thermal Performances of Windows*, and *Forced Convection Cooling of an Enclosure with Fan and Grille*. Note that when the Heat Transfer in Solids interface was used in these tutorials, the discretization order was changed from quadratic to linear by using the new interface. Therefore, some small changes in the results can be observed. This is the case for the *Thermal Performances of Windows* tutorial in particular.

USE OF THE NEW MOVING MESH NODE UNDER DEFINITIONS

The *Fluid Damper* tutorial has been updated to use the **Moving Mesh** node under **Definitions** instead of the Moving Mesh interface. In addition, it has been updated by using the **Incompressible flow** option in the Laminar Flow interface, as the density defined in the material is not temperature dependent, by setting the initial time to 0 for the time-dependent study, and by improving the settings for postprocessing.

The *Dynamic Wall Heat Exchanger* tutorial has been updated to use the **Moving Mesh** node under **Definitions** instead of the Moving Mesh interface.

Backward Compatibility with Version 5.3a

- The Heat Transfer in Thin Shells (htsh), Heat Transfer in Thin Films (htsh), and Heat Transfer in Fractures (htsh) interfaces that do not support the **Layered Material** functionality are now obsolete. When opening a model created in version 5.3a or earlier in the GUI, these interfaces are not migrated. They should be manually replaced by the corresponding Heat Transfer in Shells (htlsh), Heat Transfer in Films (htlsh), and Heat Transfer in Fractures (htlsh) interfaces that support the **Layered Material** functionality. The **Thin Layer**, **Thin Film**, and **Fracture** features are replaced by the **Solid**, **Fluid**, and **Porous Medium** features in the new interfaces.
- The **Thin Layer**, **Thin Film**, and **Fracture** features (and their subfeatures) that do not support the **Layered Material** functionality are now obsolete. When opening a model created in version 5.3a or earlier in the COMSOL Desktop, these features are not migrated. They should be manually replaced by the corresponding features that support the **Layered Material** functionality. Note that the pair features for thin structures are no longer available for layered materials. The obsolete **Thin Layer**, **Thin Film**, and **Fracture** features remain available through the COMSOL API so that existing scripts remain compatible. However, it is recommended to update the

scripts to use the new features as the obsolete features will be discontinued in future versions.

- The **Surface-to-surface radiation** check box has been removed from the **Physical Model** section of the Heat Transfer interface. As of Version 5.4, surface-to-surface radiation models should use the Surface-to-Surface Radiation physics interface coupled with a Heat Transfer domain physics interface via the Heat Transfer with Surface-to-Surface Radiation multiphysics coupling feature. The Heat Transfer interface does not support the **Diffuse Surface**, **Diffuse Mirror**, **Prescribed Radiosity**, **Opacity**, and **External Radiation Source** features anymore. Models built prior to v5.3a using surface-to-surface radiation features formerly available in the Heat Transfer interfaces will mark these features as obsolete. Support for features marked as obsolete is not ensured in future versions, thus it is strongly recommended to transfer all these features to a Surface-to-Surface Radiation physics interface and use a Heat Transfer with Surface-to-Surface Radiation multiphysics coupling feature.
- The **Radiation in participating media** check box has been removed from the **Physical Model** section of the Heat Transfer interface. From Version 5.4, radiation in participating media models should use the Radiation in Participating Media physics interface coupled with a Heat Transfer physics interface via the **Heat Transfer with Radiation in Participating Media** multiphysics coupling feature. The Heat Transfer interface does not support the **Radiation in Participating Media**, **Opaque Surface**, **Incident Intensity**, and **Continuity on Interior Boundary** features anymore. Models built prior to v5.3a using radiation features formerly available in the Heat Transfer interfaces will mark these features as obsolete. Support for features marked as obsolete is not ensured in future versions, thus it is strongly recommended to transfer all these features to a Radiation in Participating Media physics interface and use a **Heat Transfer with Radiation in Participating Media** multiphysics coupling feature.
- The **Ambient Settings** section has been removed from the Heat Transfer interface. When opening a model created in version 5.3a or earlier in the COMSOL Desktop, in which some of the input fields of this section have been edited, an **Ambient Thermal Properties** node is automatically created under **Definitions** with the same settings. If some ambient variables have been selected in features, the feature inputs are updated as well.
- The **Phase Change Material** feature has been replaced by the **Phase Change Material** subfeature. When opening a model created in version 5.3a or earlier in the COMSOL Desktop, a **Fluid** node with the **Phase Change Material** subfeature is

automatically created in place of the obsolete feature. This node is labeled **Phase Change Material** in the model.

- The **Damaged Tissue** section has been removed from the **Biological Tissue** feature. When opening a model created in version 5.3a or earlier in the COMSOL Desktop, in which the **Include damage integral analysis** check box of this section has been selected, a **Thermal Damage** node is automatically added under **Biological Tissue** with the same settings. Note that for the **Temperature threshold** option, it is no longer possible to perform simultaneous hyperthermia and cryogenic analysis. The **Hyperthermia analysis** option is therefore selected by default when opening the model.
- The **Rosseland approximation** option, which was available in the **Radiation discretization method** list of the **Radiation in Participating Media** feature, has been removed. When opening a model created in version 5.3a or earlier in the COMSOL Desktop, in which this option has been selected, an **Optically Thick Participating Medium** subnode is automatically created under the active domain node.
- The **Medium Properties** feature has been renamed **Absorbing Medium** in the Radiative Beam in Absorbing Medium interface, with identical settings and modeling. When opening a model created in version 5.3a or earlier in the COMSOL Desktop, an **Absorbing Medium** feature is automatically created in place of the obsolete feature. This node is still labeled **Medium Properties** in the model.
- The **Radiation in Participating Media** feature has been renamed **Participating Medium** in the Radiative Beam in Absorbing Medium interface. When opening a model created in version 5.3a or earlier in the COMSOL Desktop, a **Participating Medium** feature is automatically created in place of the obsolete feature. This node is still labeled **Radiation in Participating Media** in the model.
- The **Change Thickness** and **Change Cross Section** features have been renamed **Thickness** and **Cross Section**, respectively. When opening a model created in version 5.3a or earlier in the COMSOL Desktop, these features are not migrated. They should be manually replaced by the new features.

Backward Compatibility with Version 5.3

- The **Electromagnetic Heat Source** and **Boundary Electromagnetic Heat Source** coupling features are now obsolete. They have been merged into a single **Electromagnetic Heating** coupling feature. When opening a model created in version 5.3 or earlier in the COMSOL Desktop, the **Electromagnetic Heat Source** and **Boundary Electromagnetic Heat Source** features are automatically migrated to **Electromagnetic**

Heating features with selections (respectively a domain and a boundary selection) that match the selections of the migrated features. The obsolete **Electromagnetic Heat Source** and **Boundary Electromagnetic Heat Source** features remain available through the COMSOL API so that existing scripts remain compatible. However, it is recommended to update the scripts to use the new **Electromagnetic Heating** feature as the obsolete feature will be discontinued in future versions. Note that temperature coupling is enforced in the 5.3a version of the **Electromagnetic Heating** coupling feature.

- The **Thermoelectric Effect** and **Boundary Thermoelectric Effect** coupling features have been merged into a new **Thermoelectric Effect** coupling feature. When opening a model created in version 5.3 or earlier in the COMSOL Desktop, the **Thermoelectric Effect** and **Boundary Thermoelectric Effect** features are automatically migrated to the new **Thermoelectric Effect** feature with selections (respectively a domain and a boundary selection) that match the selections of the migrated features. The obsolete **Thermoelectric Effect** and **Boundary Thermoelectric Effect** features remain available through the COMSOL API so that existing scripts remain compatible. However, it is recommended to update the scripts to use the new **Thermoelectric Effect** feature as the obsolete feature will be discontinued in the future. Note that temperature coupling is enforced in the 5.3a version of the **Thermoelectric Effect** coupling feature.
- The **Inflow Heat Flux** boundary condition is now obsolete. It can be replaced by the new **Inflow** boundary condition and the **Boundary Heat Source** condition.
- The **Enable conversions between material and spatial frame** check box, displayed in the **Advanced Settings** section of the Heat Transfer interfaces when the model component supports moving mesh, has been removed. This means that conversion between material and spatial frames is always effective when these frames differ.
- The **Energy absorption** option has been renamed **Arrhenius kinetics** in the **Transformation model** list of the **Irreversible Transformation** and **Biological Tissue** nodes.
- The **Convective moisture flux** option has been renamed **Convective moisture flux, pressures difference** in the **Moisture Flux** node.
- The ρ_{Int} , $C_{p\text{Int}}$, and γ_{Int} variables, defined by the **Fluid** domain feature, have been removed.
- The q_{evap} variable, defined by the **Moist Surface** and **Wet Surface** boundary nodes, has been replaced by the q_{evapTot} variable.

The **General** option for **Thin film model** has been removed from the **Thin Film** fallback node.

Backward Compatibility with Version 5.2a

The obsolete features **Convective Heat Flux** and **Out-of-plane Convective Heat Flux** are removed in Version 5.3. These were marked as obsolete with a warning message since Version 5.0. These features will be removed from your model when opening in Version 5.3. Use instead the corresponding **Heat flux** and **Out-of-Plane Heat Flux** features, with the option **Convective heat flux**. The variables `ht.chflux`, `ht.chflux_u`, `ht.chflux_d`, and `ht.chflux_z` defined by these features have been also removed from equation view and postprocessing variables menu.

The new default option in **Thin Moisture Barrier** feature is **Vapor barrier material properties**, and the **Moisture barrier material properties** option has been removed. When loading a MPH file that was created before version 5.3 this latter option will be cleared, and the corresponding user inputs used to define the **Moisture transfer coefficient** directly. However, for the API, it will be necessary to update the code with an existing option name.

In the **Equivalent Conductivity for Convection** section of the **Fluid** node, the new **Automatic** option for **Temperature difference** has become the default option. When loading a MPH file that was created before version 5.3, the **User defined** option will be set automatically with the previous value. However, for the API, it will be necessary to update the code to set the **User Defined** option.

Backward Compatibility with Version 5.2

Several enhancements or bug fixes may require an update of the existing applications or induce a change in the results. Here is the list of the changes:

- When **Solar position** is selected in **External Radiation Source**, the new option **Update time from solver** is selected by default. When loading an MPH file that was created before version 5.2a this option will be cleared. However, for the API, for time-dependent studies, it will be necessary to update the code to get the same behavior as in previous versions. In most cases, removing the t term in the Second field is the best option. Alternatively, clearing the **Update time from solver** option will ensure that the same behavior as before is obtained.
- The crosswind stabilization of the heat transfer and fluid flow interfaces has been updated. In general, the update reduces the smoothing induced by the stabilization compared to previous version. In addition, when you use the **Nonisothermal Flow** multiphysics coupling and the fluid is defined as incompressible, the uncoupled versions of the streamlines and crosswind stabilization are used.

- Since **Pair Thin Film** and **Pair Fracture** are now pair features, their selection needs to be redefined.
- The definition of the `ht.feature.fc(RH,T,pA)` function when **Moist air** fluid type is used in **Fluid** feature has been corrected. This change can affect the results in simulations where it is used.
- The frame support has been enhanced to support moving mesh combined with deformed geometry in the same component.
- The **Prescribed Radiosity** feature has been updated so that it no longer ignores the **Temperature** model input.
- In surface-to-surface boundary features, the ambient temperature definition has been fixed when the **Opacity Controlled** and **Defined ambient temperature on each side** options are used simultaneously.

Backward Compatibility with Version 5.1

CHANGES IN VARIABLE NAMES

The names of the heat capacity variables have changed in several features for consistency:

- In the **Bioheat** feature, `ht.cb` has become `ht.cp_b`.
- In the **Thin Layer** feature, `ht.cs` has become `ht.cp_s`.
- In the **Thin Rod** feature, `ht.c1` has become `ht.cp_1`.
- In the **Thin Layered Shell** feature, `htsh.cs1` has become `htsh.cp_s1`, and so on.
- In the **Heat Transfer in Porous Media** feature, `ht.c_pp` has become `ht.cp_p`, `ht.c_pp1` has become `ht.cp_p1`, and so on.
- In the **Immobile Fluids** feature, `ht.c_pg1` has become `ht.cp_g1`, and so on.
- In the **Fracture** feature, `ht.c_pfr` has become `ht.cp_fr`.

The names of the thermal resistance and absolute thermal resistance variables have changed in the **Isothermal Domain Interface** feature:

- `ht.idi1.Rs` has become `ht.idi1.R_t`
- `ht.idi1.Rsth` has become `ht.idi1.R_tth`

The name of the fluid viscosity variable has changed in the **Local Thermal Non-Equilibrium** multiphysics coupling: `1tne1.mu_f` has become `1tne1.mu`.

OBSOLETE FEATURES

A number of features are now obsolete because they have been merged or replaced by a more general features. However, when models from COMSOL 5.0 or before contain these features, some of the obsolete features are visible in version 5.2a with a warning sign notifying that the feature is now obsolete (see [Table 1-1](#)). Some features have been completely removed (see [Table 1-2](#)). A model containing any of them can be opened in 5.2a, but the feature will be missing.

Updating the models is recommended. Use the new feature and remove the obsolete feature to make sure that the model will remain compatible with future versions of COMSOL Multiphysics.

The table below summarizes the changes:

TABLE 1-1: OBSOLETE FEATURES THAT ARE OBSOLETE BUT THAN CAN BE LOADED IN 5.1

OBSOLETE FEATURE	REPLACE IN 5.2A BY
Convective Heat Flux	Heat Flux with the Convective heat flux option selected
Surface-to-Ambient Radiation	Diffuse Surface with Include surface-to-surface radiation deselected
Surface-to-Surface Radiation	Diffuse Surface with Include surface-to-surface radiation selected
Highly Conductive Layer	Thin Layer with Layer type set to Conductive
Thin Thermally Resistive Layer	Thin Layer with Layer type set to Resistive

TABLE 1-2: OBSOLETE FEATURES THAT ARE NO LONGER AVAILABLE

OBSOLETE FEATURE	REPLACE IN 5.2A BY
Infinite Element	Infinite Element is no longer defined in the physics interface. Infinite elements are defined in the Definitions section instead.
Electrochemical Heat flux	Boundary Heat Source
Reaction Heat Flux	Boundary Heat Source
Heat Transfer in Participating Media	Heat Transfer in Solids contributing to Radiation in Participating Media
Porous Matrix	Heat Transfer in Porous Media

Note that it is not possible to add any of the obsolete features from the COMSOL Desktop in version 5.2a. Only the new features are available.

OBsolete Postprocessing Variables Deleted in 5.1

List of deleted postprocessing variables: `ndflux_acc`, `ndflux_acc_u`, `ndflux_acc_d`, `ntflux_acc`, `ntflux_acc_u`, `ntflux_acc_d`, `nteflux_acc`, `nteflux_acc_u`, `nteflux_acc_d`, `qout`, `qout_u`, `qout_d`, `qin`, `qin_u`, `qin_d`, `qw`, `qw_u`, and `qw_d`.

Changes in Variable Names

The variable `ht.Tvar`, used for nonisothermal flow wall functions, becomes a fully scoped variable in the following features:

- **Heat Flux** (`ht.hf1.Tvar`), **Line Heat flux**, **Point Heat flux**, and **Out of plane heat flux**.
- **Heat Source** (`ht.hs1.Tvar`), **Boundary heat source**, **Line heat source**, and **Point heat source**.
- **Diffuse Surface** (`ht.ds1.Tvar`).

The fully scoped variables `Ptot`, `Pbtot`, `Pltot`, and `Pstot` become `P0`, `Pb`, `P1`, and `Ps`, respectively, in the following features:

- **Heat Flux**
- **Inflow Heat Flux**
- **Line Heat Flux**
- **Heat Source**
- **Boundary Heat Source**
- **Line Heat Source**
- **Line Heat Source** (**Thin Rod** subfeature)
- **Line Heat Source on Axis**
- **Layer Heat Source** (**Thin Layer** subfeature)

In the **Thermal Contact** feature:

- `Qfric`, `QfricPow`, and `Qrate` become `Qb`, `Pb`, and `Qbpart`, respectively
- `Qfric0`, `Qrate0`, `Econtact0`, `hrad0`, `hgap0`, and `hconstr0` are removed and migrated to existing variables `Qb`, `Qbpart`, `Econtact`, `hrad`, `hgap`, and `hconstr`, respectively, in old models.

The **Change Effective Thickness** feature from the Heat Transfer in Thin Shells interface merges `htsh.de` into the existing variable `htsh.ds`.

The Heat Transfer in Thin Shells interface also merges the property `de` and variable `htsh.de` into existing property `ds` and variable `htsh.ds`. An API command such as

`model.physics("htsh").prop("de").set("de", [...])` now does not have any effect.

Backward Compatibility with Version 4.4

OUT-OF-PLANE HEAT TRANSFER UPDATE

The out-of-plane heat transfer property, which was available in 2D and 1D geometries, has been removed as of COMSOL Multiphysics 5.0. Now, all geometries are considered 3D geometries and thickness or area parameters are always available.

API Users

With the **Overall heat transfer rate** option (previously named **Total power** or **Total heat flux**), the thickness parameters (`dz ENTR` and `Ac INPUT`) have been removed for the following features:

- **Heat Source**
- **Heat Flux**
- **Boundary Heat Source**
- **Layer Heat Source**
- **Inflow Heat Flux**

Compatibility with previous code is maintained for most of the cases. However, three configurations require manual updates of the code:

- If you specified the thickness parameter twice (there were two instances of specifying the parameter).
- If you first specified the thickness and in a second command the total power value.
- If you first specified the thickness and then activated the out-of-plane heat transfer option.

For these 3 cases, you need to rewrite this part of the Java[®] code by doing the following. For a heat source example (2D and 1D axisymmetric geometries):

```
Ptot_50 = Ptot_44_value*ht.d/dz ENTR_44_value,
```

where `Ptot_44_value` is the total power value used in previous versions, `Ptot_50` is the total power value used since COMSOL Multiphysics 5.0, and `dz ENTR_44_value` is the value used in previous versions.

In 1D geometries, the following update should be performed: `Ptot_50 = Ptot_44_value*ht.d/Ac INPUT_44_value`,

where `Ac INPUT_44_value` is the value used in previous versions.

Flux Variables

The flux variables are always given in W/m^2 (also for 2D or 1D geometries). To obtain the corresponding flux variables in W/m on a boundary in 2D, for example, the flux variables have to be multiplied by the thickness `ht.d`.

The following variables have been updated following this convention:

- Conductive heat flux, `ht.dflux`
- Convective heat flux, `ht.cflux`
- Translational heat flux, `ht.trlflux`
- Total energy flux, `ht.teflux`
- Total heat flux, `ht.tflux`
- Boundary convective heat flux, `ht.chflux`
- Radiative heat flux, `ht.rflux`

All flux variables now have the following sign convention: positive heat flux corresponds to heating, whereas negative heat flux corresponds to cooling. The variables `chflux` and `rflux` may have opposite signs compared to previous versions, depending on the context where they are defined.

INFLOW HEAT FLUX IS ONLY APPLICABLE ON NON-SOLID DOMAINS

The **Inflow Heat Flux** feature is no longer applicable on boundaries adjacent to the following domain features: **Heat Transfer in Solids**, **Biological Tissue**, and **Isothermal Domain**. This restriction avoids ill-posed configurations due to the null velocity field in these features. This may change model results if this boundary condition was previously applied to a boundary adjacent to a nonfluid domain.

HEAT TRANSFER WITH PHASE CHANGE NOW USES A VOLUMETRIC FORMULATION

The variables `ht.thetai` now represent volume fractions (they were mass fractions before). This implies changes in the definition of density and heat capacity at constant pressure close to the phase change interface. Far from the phase change interface, the density and heat capacity are unchanged.

Backward Compatibility with Version 4.3b

INTERIOR FLUXES UPDATED

Because the interior flux variable definitions have been revised, models that use interior fluxes in definitions, physics interfaces, or postprocessing show different results in versions 5.0 and later than in earlier versions.

IMPROVED ENERGY BALANCE

Changes introduced in the equation formulation to enable better energy balance may slightly modify the results.

Among the changes, the reference enthalpy, H_{Ref} , the value of which is defined up to a constant, is now set to 0 J/kg at the reference temperature and pressure. This changes the absolute values of the enthalpy and several variables based on it. In particular, the value of the convective heat flux is changed. Even the sign may be changed in some cases. However, the quantity of interest, the net convective heat flux, is identical because the shift induced by the change of H_{Ref} applies for the enthalpy definition at the inlet as well as at the outlet.

Backward Compatibility with Version 4.3a

HEAT TRANSFER IN POROUS MEDIA

Old models using the **Porous** feature will be supported in the COMSOL Multiphysics interface. However, it is recommended to replace this feature with Heat Transfer in Porous Media (see [Obsolete Features](#)). For a model using the API, it will be necessary to update the model to use Heat Transfer in Porous Media.

SURFACE-TO-SURFACE RADIATION

The surface radiosity, previously named J , is now named $ht.J$ by default (assuming that ht is the physics interface tag).

Models created using versions 4.3 and 4.3a that contain surface-to-surface radiation support the old syntax in multiphysics interfaces. It is, however, recommended that you update the models to use the new syntax. Models created in a version before 4.2a need to be opened in version 4.3 or 4.3a and resaved before being opened in 5.0 or a later version.

For a model using the API, you must update the model to use the new syntax.

RADIATION IN PARTICIPATING MEDIA

The radiative intensity variable names, previously named `I1` and so on, became `ht.I1` (when there are fewer than 10 discrete directions) or `ht.I01` (when there are 10 or more discrete directions) by default.

Models created in versions 4.3 and 4.3a that include radiation in participating media support the old syntax in multiphysics interfaces. It is, however, recommended that you update the models to use the new syntax. Models created in a version before 4.2a need to be opened in version 4.3 or 4.3a and resaved before being reopened in 5.0 or a later version. In addition, for all versions, it may be necessary to regenerate the default solver to get the correct solver settings.

The `S2` quadrature in 2D has been replaced by the LSE symmetric quadrature. It has been updated so that it coincides with the other quadratures. LSE symmetric quadratures were already used for `S4`, `S6`, and `S8` in 2D and for all 3D quadratures. This quadrature update can result in significant changes in 2D models using the `S2` quadrature. However, the theoretical error is not different and the 2D and 3D implementations are now consistent.

Backward Compatibility with Version 4.3

THE DEFAULT MODEL LIST HAS BEEN REMOVED

Heat transfer physics interfaces no longer have the default model list. This list was used to change the default feature for heat transfer models. You can obtain a similar result by adding the corresponding domain feature immediately below the default feature and setting its domain selection to **All domains**.

Any model files for Java[®] that modify the default model value require a manual update.

IMPROVED STABILIZATION OF HEAT TRANSFER IN SOLIDS

The streamline diffusion stabilization for **Heat Transfer in Solids** and **Biological Tissue** features has been improved. It now accounts for contributions from linear source terms from the **Heat Source**, **Out-of-Plane Convective Cooling**, **Out-of-Plane Radiation**, and **Out-of-Plane Heat Flux** features. This improves the robustness of the convergence when these contributions are large. This change may modify the convergence behavior of existing models.

FRAME SELECTION IN DISCRETIZATION SECTION

The **Frame type** list, which was previously available when a moving mesh was detected, has been removed. The frame type is now controlled by the features.

UPDATE OF FEATURE VARIABLE NAMES

In order to avoid name conflicts between feature variable names, some of them have been renamed. This change concerns contributive features. Because they are contributing, it is possible to have similar features active on the same boundary. To make it possible to distinguish the variables from each feature, the feature scope has been introduced in the variable name so that variable names now use a prefix. These changes affect the following feature variables:

- Heat flux
- Out-of-plane heat flux
- Convective cooling
- Out-of-plane convective cooling
- Boundary heat source
- Heat source
- Line heat source
- Point heat source
- Edge heat flux/Point heat flux
- Electrochemical reaction heat flux
- Reaction heat flux

For example, in a **Boundary Heat Source** feature, `ht.surf` has been renamed into `ht.bhs1.surf` (assuming that `ht` is the physics interface tag). In the Heat Flux feature, the variable that was previously named `ht.q0_hf1` is now `ht.hf1.q0`.

Any model files for Java[®] that use the old variable names in expressions (such as expressions used for plotting or data evaluation that include such old variable names) need to be updated manually.

NEW DEFAULT FLUID FEATURES AND OPAQUE SUBFEATURE

All MPH files created in version 4.3 are converted to appear and behave the same way as in 5.0 or a later version.

Due to these new default features, model files for Java can be simplified. In addition, model files for Java that add the **Fluid** feature or the **Opaque** subfeature with the default tag require a manual update to avoid duplicate tag conflicts.

WEAK CONSTRAINTS UPDATE FOR FLUID FLOW USER INTERFACES

The weak constraint formulations for some boundary conditions in the following interfaces have been updated:

- Laminar Flow
- Turbulent Flow, k - ϵ
- Turbulent Flow, Low Re k - ϵ
- Nonisothermal Flow
- Conjugate Heat Transfer

See the [CFD Module](#) release notes for details.

Weak constraints for the **Interior Wall** feature are no longer available.

REVISION TO THE TURBULENCE MODELS

The formulations of some variables in the turbulence models have been revised in order to improve accuracy. Simulations using a turbulence model can display a different convergence behavior in versions 5.0 and later compared to version 4.3, and the numerical result can differ slightly between the versions.

Obsolete Features

The following features have been removed:

- **Inflow Heat Flux** (replaced by **Inflow** and **Boundary Heat Source**)
- **Electromagnetic Heat Source** and **Electromagnetic Heat Source** (replaced by **Electromagnetic Heating**)
- **Boundary Thermoelectric Effect** (merged with **Thermoelectric Effect**)
- **Convective Heat Flux** (replaced by **Heat Flux** with **Convective heat flux option**)
- **Out-of-plane Convective Heat Flux** (replaced by **Out-of-plane Heat Flux** with **Convective heat flux option**)
- **Porous Matrix** (replaced by **Porous Medium**)
- **Infinite Element** (replaced by **Infinite Element Domain** declared in the **Definitions** node)
- **Electrochemical Heat Flux** (replaced by **Boundary Heat Source**)
- **Reaction Heat Flux** (replaced by **Boundary Heat Source**)
- **Heat Transfer in Participating Media** (replaced by **Heat Transfer with Radiation in Participating Media**)

MEMS Module

New Functionality in Version 5.4

NEW PHYSICS INTERFACE: ELECTROMECHANICS, BOUNDARY ELEMENTS

The **Electromechanics, Boundary Elements** interface solves the same class of problems as the **Electromechanics** interface, except it uses a boundary element discretization for the electrostatics part of the model. Model setup can be easier, as infinite element domains and moving mesh are not necessary.

CHANGES IN SYMMETRY AND ANTISYMMETRY FOR SOLID MECHANICS IN 2D AXISYMMETRY

In the 2D axisymmetric version of the Solid Mechanics interface, the nodes **Symmetry** and **Antisymmetry** are now obsolete. They have been replaced by the **Symmetry Plane** node. The reason is that the only possible true symmetry in an axisymmetric model is with respect to a cut perpendicular to the Z -axis. The old features allowed unphysical constraints.

NEW MODAL SUPERPOSITION STUDY TYPES

Two new predefined study sequences for modal superposition have been added: **Time Dependent, Prestressed, Modal** and **Frequency Domain, Prestressed, Modal**. In both cases, three study steps are generated, where the first one is a stationary step in which the prestress state used for the subsequent eigenfrequency computation is determined.

RENAMED STUDY TYPES

Some study types have been renamed in order to get a more consistent naming:

- **Frequency Domain Modal** is now **Frequency Domain, Modal**
- **Time Dependent Modal** is now **Time Dependent, Modal**
- **Prestressed Analysis, Eigenfrequency** is now **Eigenfrequency, Prestressed**
- **Prestressed Analysis, Frequency Domain** is now **Frequency Domain, Prestressed**

PARTICIPATION FACTORS FOR EIGENFREQUENCY STUDIES

When you add an **Eigenfrequency** study from the **Add Study** window, a **Participation Factors** node is now automatically added under **Definitions**, as long as there is at least one structural mechanics physics interface in the model. You will thus always have access to the participation factor variables after running such a study.

MATERIAL ACTIVATION

It is now possible to activate a material in a stress free state using the new **Activation** subnode under **Linear Elastic Material** in the Solid Mechanics and Membrane interfaces. A user-defined logical expression is used for the activation. The same feature can also be used to remove material. Typical applications are various types of manufacturing processes where material is added.

HOMOGENIZATION OF PERIODIC MATERIALS

Inhomogeneous materials can often be represented by a unit cell that is assumed to be part of a periodic structure. This is sometimes called a *representative volume element* (RVE). The new **Cell Periodicity** feature in the Solid Mechanics interface provides you with several options for analyzing such unit cells:

- Automatic computation of equivalent homogenized anisotropic material properties (stiffness, compliance, thermal expansion), which can be accessed as a Global Material.
- Computation of the local stresses and strains in the unit cell, given external average stress and strain fields.
- Computation of free thermal or hygroscopic expansion of the unit cell, given average temperature or moisture concentration.

BURGERS VISCOELASTIC MODEL

One more built-in viscoelastic model has been added: the Burgers model. As with the other viscoelastic models, it is accessed by adding a **Viscoelasticity** subnode under the **Linear Elastic Material**.

VOLUME REFERENCE TEMPERATURE

A new concept, *volume reference temperature*, has been introduced. It serves two purposes: it replaces the earlier *strain reference temperature* for thermal expansion, and it is the temperature at which the mass density is computed. These two phenomena are physically related to each other, and the default is to use the same value, defined as a **Common model input**. For the mass density, the concept is new and is used to avoid violations of mass conservation. For thermal expansion, the default location where you specify the strain free temperature has moved from the **Thermal Expansion** node or **Thermal Expansion** multiphysics coupling to **Common model input**.

When you open an old model, the value of the strain reference temperature is converted to a **User defined** volume reference temperature.

UTILITY FUNCTIONS FOR EXTERNAL MATERIALS

When programming your own constitutive relations using the external materials functionality, there are several operations for which there is a recurring need. Examples of this are various tensor operations, computation of principal values and orientations, and matrix inversion. There is now a library of utility functions that covers many common operations employed when programming material models for solid mechanics.

TIME STEP PASSED TO EXTERNAL MATERIALS

The time or parameter step is now passed as an argument to the material function when using the **General Stress-Deformation Relation** socket.

NEW SETTINGS FOR MIXED FORMULATION

In earlier versions, a mixed (displacement-pressure) formulation could be activated using the **Nearly incompressible material** check box in a material model. It has been replaced by the **Use mixed formulation** list with the following three choices: **None**, **Pressure formulation**, and **Strain formulation**. The pressure formulation is the same mixed formulation as before, while the strain formulation is new. Instead of a pressure, the volumetric strain is used as an extra degree of freedom.

CHANGES IN FLUID-STRUCTURE INTERACTION INTERFACES

The **Fluid-Structure Interaction, Fixed Geometry** multiphysics coupling present in previous releases is now obsolete. It has been merged with an updated **Fluid-Structure Interaction** multiphysics coupling. This makes it easier to switch between the two formulations. In the current version, the only difference lies in whether a **Deforming Domain** is active on a fluid domain or not.

Old models where you have used **Fluid-Structure Interaction, Fixed Geometry** are not changed. You are, however, advised to update to the new type of modeling.

RIGID DOMAIN IN 2D AXISYMMETRY

The Rigid Domain material model is now also available in 2D axisymmetry. It has only a single degree of freedom: the axial translation.

RIGID CONNECTOR FOR EDGES AND POINTS

The selection in a **Rigid Connector** can now be a combination of boundaries, edges, and points. It is now also possible to add a rigid connector at the edge and point levels, as long as there are no higher-order geometrical objects in the selection.

FLEXIBLE FORMULATION OF RIGID CONNECTOR

There are now two formulations of the **Rigid Connector** feature, **Rigid** and **Flexible**. In the default formulation, all the selected boundaries, edges, and points behave as if they were connected by a common rigid body. In some cases, this gives an unwanted stiffening or unrealistic local stresses. You can then switch to a flexible formulation where the constraint is applied only in an average sense. The flexible formulation is available only for a rigid connector with a pure boundary selection and not when the selection also contains edges or points.

TABULAR OUTPUT OF EIGENFREQUENCIES

When running an Eigenfrequency study, a table containing the natural frequencies, modal damping, and quality factors is now generated by default.

SPRING AND REACTION FORCES

A new reaction force variable containing the sum of reaction forces from constraints and reaction forces from spring foundation has been added to all structural mechanics physics interfaces. The variables, which are available in the result menus, are called **Total reaction force** and have component names like `solid.RFtotalx`. If there are spring foundations in your model, you will find the forces under the category **Spring and Damping forces** in the result menus.

REACTION-FREE SYMMETRY CONDITION

The symmetry conditions in the structural mechanics interfaces have been extended to account for cases where the symmetry plane is allowed to translate in the direction of its normal rather than being a fixed plane. The main application is for use as a boundary condition in truncated structures having no net reaction force. In the new section **Normal Direction Condition** in the settings for the **Symmetry** node, you can adjust the behavior of this boundary condition. There are four different options: **No displacement** (the default), **Free displacement**, **Prescribed force**, and **Prescribed displacement**.

ROLLER CONDITION WITH ANALYTICAL NORMAL ORIENTATION

The default roller constraint acts along a normal to the selected boundaries. The normal is computed numerically. For a certain mesh node, this direction is kept constant during the analysis, so it cannot represent finite sliding on a curved boundary.

You can now, however, prescribe that the boundary slides on an analytical surface. There are two situations where you may need to do this:

- The geometry or mesh representation of the actual boundary is imperfect. This can lead to unexpected locking of the deformation, since the computed normals do not have the intended directions.
- In a geometrically nonlinear analysis, the use of an analytical surface makes it possible to also model finite sliding along curved boundaries.

In the settings in the **Roller** node, select a **Normal orientation**, which can be either **Automatic**, **Plane**, **Cylinder**, or **Sphere**.

New Models in Version 5.4

SQUEEZE-FILM DAMPING OF PERFORATED PLATES

This benchmark model compares the damping coefficients of perforated plates from computation results versus experimental data. The simulation includes 18 different geometric configurations. It uses the Bao's perforation model, which is built into the Thin Film Flow physics interface. Two limiting cases are also simulated: no perforation and zero relative pressure at the perforations.

S-PARAMETER OF A THIN-FILM BAW RESONATOR

This model shows how to compute the S-parameter for a piezoelectric MEMS device by extending the tutorial Thin-Film BAW Composite Resonator. The measurement of the S-parameter is commonly used to characterize such devices for RF applications. The terminal feature in the Electrostatics physics interface provides straightforward access to the computed S-parameters for frequency domain study types.

DISPERSION DIAGRAM OF A THIN-FILM BAW STRUCTURE

This model shows how to create dispersion diagrams from simulation results by extending the tutorial Thin-Film BAW Composite Resonator. The dispersion curve can be plotted against both real and imaginary values of the wave number, corresponding to the propagating modes and evanescent modes, respectively. Each mode can also be visualized by plotting its displacement field. The consideration about such propagating and evanescent modes is an important part of the process in selecting appropriate design parameters for the layer structure of a BAW resonator.

Backward Compatibility with Version 5.2

The friction slip velocity is no longer defined as a dependent variable. When running a model using the COMSOL API, you will need to remove the reference to this variable in the solver settings. The friction slip velocity is usually defined as `<comp>_<solid>_vslip_<pairname>`, where `<comp>` is the tag of the component, `<solid>` is the tag of the Solid Mechanics physics interface where the **Friction** node is defined, and `<pairname>` is the name of the contact pair.

Backward Compatibility with Version 4.3b

For models using contact, plot expressions are not updated when you open an old model where the pair names have been edited manually. Also, opening an old model where a pair name has been manually edited might affect the solver configuration. It is recommended that you regenerate a new default solver configuration.

Microfluidics Module

New Functionality in Version 5.4

UPDATED PHYSICS INTERFACE: LAMINAR TWO-PHASE FLOW, MOVING MESH

The **Laminar Two-Phase Flow, Moving Mesh** interface now uses the **Deforming Domain** feature under **Definitions**. This facilitates model setup, as the fluid domain settings and boundary conditions are now specified in the **Laminar Flow** interface, and the deforming domains and boundary conditions are set up under the **Definitions** node. This also allows other physics to be included in a straightforward way, as the moving mesh functionality is common to all physics in the model. It is now also possible to specify a mass flux in the External Fluid Interface boundary condition.

UPDATED FLUX FORMULATION

Updated formulation for the Flux and No Flux nodes included in the Transport of Diluted Species and the Transport of Diluted Species in Porous Media interfaces.

- The flux relative to the convective flux is prescribed instead of the total flux.
- The new formulation implies that the nodes can be used on a moving mesh geometry without compensating for the boundary velocity.
- Updated equation display for governing equations and boundary conditions. Equations updated to be formulated in terms of the flux relative to the convective flux.

Backward Compatibility with Version 5.3a

NO FLUX FORMULATION

The formulation of the No Flux feature has been changed in the Transport of Diluted Species and Transport of Diluted Species in Porous Media interfaces in version 5.4 so that the flux relative to the convective flux is set to zero. The new formulation is used when opening a model from a previous session. In the previous formulation, the total flux was set to zero.

Backward Compatibility with Version 4.3a

The Free Molecular Flow and Transitional Flow user interfaces now form the Molecular Flow Module. A Molecular Flow Module license is required to use these interfaces.

Backward Compatibility with Version 4.3

BRINKMAN EQUATIONS AND FREE AND POROUS MEDIA FLOW

The momentum equations in the Brinkman Equations interface and the Free and Porous Media Flow interface have been corrected. The term $-\mathbf{Q}_{br} \cdot \mathbf{u} / \epsilon_p^2$ on the right-hand side previously lacked the factor $1/\epsilon_p^2$, where ϵ_p is the porosity.

MOBILITY

In physics interfaces modeling migration (for example, Transport of Diluted Species), anisotropic mobility is now supported, and the mobility can be set to follow the Nernst-Einstein relation. The default setting when making new models has been changed to the Nernst-Einstein relation. Model files for Java[®] generated prior to 4.3b using migration will have to be modified manually to account for these changes.

Mixer Module

New Functionality in Version 5.4

- All Mixer Module interfaces now use the Moving Mesh functionality under the **Definitions** node in the **Model Builder**.
- The v2-f and Realizable k- ϵ turbulence models are now available in the Rotating Machinery, Nonisothermal Flow interface.
- New studies for the Rotating Machinery, Nonisothermal Flow interfaces: Frozen Rotor, One-Way Coupled, NITF; Frozen Rotor, One-Way Coupled with Initialization, NITF; Time Dependent, One-Way Coupled, NITF; and Time Dependent, One-Way Coupled with Initialization, NITF.
- All turbulence models are now available in the Rotating Machinery, Multiphase Flow interface.
- Free Surface and Rotating Shaft boundary conditions are now available in the Rotating Machinery, Multiphase Flow interface.
- A new **Navier slip** option is available for Interior Wall features in all Laminar Flow interfaces.
- The default solver for the fluid flow interfaces now has the **Reuse sparsity pattern** check box enabled in the **Advanced** solver node's **Assembly Settings** section. This setting typically reduces computation time by around 10% for time-dependent models.

Molecular Flow Module

Backward Compatibility with Version 5.0 and Earlier

Due to the fact that multiple species are now supported, some variable names have changed. In general, variables will now have `_G` appended to their names, where `G` is the name specified for the **Incident molecular fluxes** in the **Dependent Variables** section. So, if a model is updated and resolved, the expressions used in results should be changed as follows:

- The number density should be changed from `fmf.N` to `fmf.N_G`.
- The pressure should be changed from `p` to `fmf.p_G`.
- The reconstructed number density should be changed from `fmf.N` to `fmf.ndr1.Nr_G`, where `ndr1` is the tag of the **Number Density Reconstruction** feature.
- Any other variables can be updated using the **Replace Expression** button available during results processing.

In addition, new variables exist for the total pressure, number density, and so on:

- The total number density, `fmf.ntot`.
- The total pressure, `fmf.ptot`.
- The total incident molecular flow, `fmf.Gtot`.

Multibody Dynamics Module

New Functionality in Version 5.4

FLEXIBLE ATTACHMENT

There are now two formulations of the **Attachment** feature: **Rigid** and **Flexible**. In the default formulation, all the selected boundaries behave as if they were connected by a common rigid body. In some cases, this behavior results in an unwanted stiffening or unrealistic local stresses. To avoid that, you can then switch to a flexible formulation where the constraint is applied only in an average sense.

FLUID-STRUCTURE INTERACTION

Two predefined multiphysics interfaces for coupling the Multibody Dynamics interface with fluid flow have been added: **Fluid-Multibody Interaction** and **Fluid-Multibody Interaction, Assembly**. The former uses a standard **Fluid-Structure Interaction** multiphysics coupling, and it is used when the geometry is in the Union state.

For multibody dynamics models, it is, however, more common that the Assembly state is used when building the geometry. In this case, the **Fluid-Multibody Interaction, Assembly** interface must be used. When invoked, it generates the new **Fluid-Structure Interaction, Pair** multiphysics coupling.

ROLLING ELEMENT BEARING

Models for six types of rolling element bearings have been added to the Multibody Dynamics interface:

- Deep groove ball bearing
- Angular contact ball bearing
- Self-aligning ball bearing
- Spherical roller bearing
- Cylindrical roller bearing
- Tapered roller bearing

Each of the bearings can have either a single or double row of rolling elements.

The bearing model includes a nonlinear representation of the contact stiffness between the rolling elements and the inner and outer races. To include such a bearing in your model, add the **Radial Roller Bearing** node, which requires the Rotordynamics Module.

LOCAL FRAME FOR RESULT PRESENTATION

You can now evaluate and plot displacements relative to any rigid domain, attachment, or gear. In the **Results** section of the Multibody Dynamics interface settings, select any such object from the **Body defining reference frame** list. You will then get access to variables for the displacements and velocities relative to that object.

ROLLER CONDITION WITH ANALYTICAL NORMAL ORIENTATION

The default roller constraint acts along a normal to the selected boundaries. The normal is computed numerically. For a certain mesh node, this direction is kept constant during the analysis, so it cannot represent finite sliding on a curved boundary. You can now, however, prescribe that the boundary slides on an analytical surface. There are two situations where you may need to do this:

- The geometry or mesh representation of the actual boundary is imperfect. This can lead to unexpected locking of the deformation, since the computed normals do not have the intended directions.
- In a geometrically nonlinear analysis, the use of an analytical surface makes it possible to also model finite sliding along curved boundaries.

In the settings in the **Roller** node, select a **Normal orientation**, which can be either **Automatic**, **Plane**, **Cylinder**, or **Sphere**.

Nonlinear Structural Materials Module

New Functionality in Version 5.4

MULLINS EFFECT

The Mullins effect is a damage-like phenomenon in rubber, where the stress-strain curve depends on maximum stress that the material has been subjected to. You can now model this effect by adding a **Mullins Effect** subnode under a **Hyperelastic Material**. There are two different models available for the Mullins effect: **Ogden-Roxburgh** and **Miehe**.

DAMAGE MODELING

For brittle materials, damage caused by cracking is an important failure mechanism. The new **Damage** subnode under **Linear Elastic Material** makes it possible to incorporate continuum-based damage models, where the material softens as an effect of the severity of the loading. Various effective scalar damage models based on equivalent strain measures (**Rankine**, **Smooth Rankine**, **Norm of elastic strain tensor**, and **User defined**) are available. You can also select different rules for the damage evolution: **Linear strain softening**, **Exponential strain softening**, and **User defined**.

Softening material models are numerically problematic due to strong localization of the damage. For this reason, you often need to use regularization methods that smooth the damage over a finite region. Two such methods are available: **Crack band** and **Implicit gradient**.

The damage functionality is available with both the Nonlinear Structural Materials Module and the Geomechanics Module. With the latter, there are also models specific to concrete available.

TRANSFORMATION DIRECTION CONTROL FOR SHAPE MEMORY ALLOYS

You can now prescribe the transformation direction for the **Lagoudas** model in the **Shape Memory Alloy** material. Use the new **Transformation direction** setting. This setting can significantly improve convergence for the common cases when the transformation direction is known a priori.

CHANGES FOR HYPERELASTIC MATERIALS WITH LOW COMPRESSIBILITY

Many hyperelastic materials, like rubber, have very low compressibility. To solve such problems, special numerical methods, like mixed formulations, must be used. The treatment of incompressibility or near incompressibility has been changed for most hyperelastic materials. A **Compressibility** list has been added. From that, you can choose one of the following material types:

- **Compressible material.** This option is only available for those material models that are intended to incorporate a significant flexibility.
- **Nearly incompressible material, quadratic volumetric strain energy.** This is the type of mixed formulation that was available in previous versions, when the now obsolete **Nearly incompressible material** check box was selected.
- **Nearly incompressible material, Hartmann-Neff volumetric strain energy.** This new method is similar to the previous formulation but is more accurate in maintaining incompressibility.
- **Incompressible material.** In this new formulation, a constraint is used in order to enforce incompressibility.

MULTIPLICATIVE DECOMPOSITION IN NONLINEAR ELASTIC MATERIAL

Even though the **Nonlinear Elastic Material** is mainly intended for small strains, it can also be used under geometric nonlinearity. The removal of inelastic strains from the total strains is now done using multiplicative decomposition, just as for other materials. By using the **Additive strain decomposition** check box, you can retrieve the formulation previously used.

BURGERS VISCOELASTIC MODEL

One more built-in viscoelastic model has been added: the Burgers model. As with the other viscoelastic models, it is accessed by adding a **Viscoelasticity** subnode under the **Nonlinear Elastic Material** node.

New Models in Version 5.4

SHAPE MEMORY ALLOY SELF-EXPANDING STENT

This model studies the crimping at low temperature of a shape memory alloy stent, followed by heating up to body temperature. The results show the stress increase during heating and the pressure applied to the inner wall of the artery.

BRITTLE DAMAGE IN UNIAXIAL TENSION

In this tutorial model, different methods for modeling brittle damage are compared in a uniaxially loaded bar. The new **Damage** feature is used. The localization of the deformation during fracture is studied using different discretization orders and regularization methods.

CRACKING OF A NOTCHED BEAM

In this example, two models for brittle damage are used to evaluate the fracture of a notched concrete beam subjected to three-point bending. The new **Damage** feature is used. The results are compared with experimental data.

Optimization Module

New Functionality in Version 5.4

TOPOLOGY OPTIMIZATION USING HELMHOLTZ FILTERING

The capability for topology optimization has been improved with support for Helmholtz filtering, projection, and interpolation. A new **Density Model** feature available under **Definitions>Topology Optimization** in model components provides settings for adding filtering, projection, and interpolation.

PARAMETRIC SWEEPS AND DERIVATIVE-FREE OPTIMIZATION

It is now possible to add a **Parametric Sweep** study step over an **Optimization** study step and vice versa if the **Optimization** study uses a derivative-free method (Nelder-Mead, Monte Carlo, BOBYQA, COBYLA, or Coordinate search). There can only be one **Optimization** study step, while more **Parametric Sweep** study steps are allowed.

OTHER IMPROVEMENTS

- The **Keep objective values in table** check box is now available for Levenberg-Marquardt and MMA optimization methods with least-squares objectives. It is enabled by default. This means that an objective table will be made while running. Consequently, the API backward compatibility can be lost in some cases. API users need to update the code that tries to access or make some specific table after the optimization has been run. The table could have already been used by the optimization solver.

When the data source is **Results table** or **Local table**, you can also select items in the **Data column** by clearing or selecting the corresponding check box in the **Use** column. You can then, for example, use the same results table to contain both experimental input and output data when performing a time-dependent optimization using a time-dependent load.

- Equation display has been added to the **Optimization** interface.
- Improved stability on Windows® 10.

New and Updated Models in Version 5.4

A new model, *Exporting and Importing a Topology Optimized Hook* (`hook_optimization_st1`), is available and includes a study of the optimal material distribution in a hook.

In addition, the following models have been updated to make use of the enhanced capabilities for topology optimization: `mbb_beam_optimization`, `loaded_knee`, `tesla_microvalve_optimization`, and `reversed_flow`. In earlier versions, these models used a Tikhonov regularization. They now make use of a filter where the mesh edge length generally works as a default for the filter length, which makes it possible to eliminate a parameter in the models — a parameter that otherwise required substantial trial and error to get right.

Backward Compatibility with Version 4.3a

The **Optimization** check box in the Stationary and Time Dependent study steps have been removed. Instead, use the Optimization study step. The **Optimization** check box does appear in models created in earlier versions of COMSOL Multiphysics where you have selected that check box.

Particle Tracing Module

New Functionality in Version 5.4

ACCUMULATORS FOR VELOCITY REINITIALIZATION

The **Accumulator** is now available as a subnode of the **Velocity Reinitialization** node. When a particle is redirected or stopped by the **Velocity Reinitialization**, the accumulated variables can be incremented in the mesh element occupied by the particle at that instant.

OFFSET VELOCITY DISTRIBUTIONS OF RELEASED PARTICLES

When particles are released with a velocity distribution, it is possible to offset all initial velocities in the distribution by an arbitrary expression. In previous versions, it was only possible to offset the velocity distribution by the velocity of a rotating frame of reference with the **Rotating Frame** feature.

The main use for this functionality is to trace particles in a frame of reference that is moving at some velocity relative to the laboratory frame.

New Applications in Version 5.4

QUASI-2D TURBOMOLECULAR PUMP

The Monte Carlo simulation of flow in a turbomolecular pump can be greatly simplified if the mean radius of the blades is much greater than the spacing between them. Under these conditions, the rotating blades of the pump can be approximated as an infinite row of blades having only translational velocity.

This benchmark model shows the effect of blade velocity, angle, and spacing on the transmission probability of molecules across a single-stage turbomolecular pump in the free molecular flow regime. The results agree well with earlier publications using the same simplifying assumptions.

Backward Compatibility with Version 5.3 and Earlier

NOMENCLATURE CHANGES FOR MULTIPLE PARTICLE SPECIES

The **Override Properties** feature has been renamed to **Particle Properties** to emphasize that it can be used to release multiple species of particles in the same model. The list

Inherit Properties in the settings for most particle release features, such as **Inlet** and **Release from Grid**, has been renamed to **Released particle properties**.

TURBULENT DISPERSION CHANGES

The turbulent dispersion models used by the **Drag Force** in the Particle Tracing for Fluid Flow interface have been modified and a deprecated turbulent dispersion model has been removed.

The option **Discrete random walk, variable time step** has been removed from the **Turbulent dispersion model** list. When an old model using this obsolete option is loaded in version 5.3a, the **Discrete random walk** option is now selected. However, **Discrete random walk** actually behaves more like the old **Discrete random walk, variable time step** model because it only samples the turbulent velocity perturbation at discrete time intervals based on the eddy lifetime or eddy crossing time in the flow.

CHANGES TO RELEASE FROM EDGE AND POINT

In the Particle Tracing for Fluid Flow interface, the **Release from Edge** and **Release from Point** features now allow the initial particle velocity to be selected from a list containing previously computed velocity fields, similar to the behavior of the **Release** and **Inlet** features. This makes the **Release from Edge** and **Release from Point** features easier to couple with another physics interface that models the fluid flow.

CHANGES TO PERIODIC ELECTRIC AND MAGNETIC FORCES

In models built in version 5.3 or earlier, if an **Electric Force** or **Magnetic Force** was present and the **Multiply force by phase angle** check box was selected, then in version 5.3a, **Time harmonic** will be selected from the new **Time dependence of field list**, which replaces this check box.

Backward Compatibility with Version 5.2a and Earlier

REFERENCE EDGE FEATURES HAVE BEEN REMOVED

The **Reference Edge** subnode, which was previously used to specify the orientation of the transverse beam directions for the **Particle Beam** feature in 3D, has been removed. Instead, the orientation of the transverse beam directions is specified using an extra selection field in the settings window for the **Particle Beam** feature itself. Similarly, the **Reference Point** subnode for specifying the beam center has been replaced by an extra selection in the settings window for the **Particle Beam** feature.

Backward Compatibility with Version 5.2

TURBULENT DISPERSION CHANGES

In the settings window for the **Drag Force** for the Particle Tracing for Fluid Flow interface, the **Turbulent dispersion** check box has been replaced by the **Turbulent dispersion model** list. If the check box is selected in a model created in version 5.2 or earlier, then after opening the model in version 5.2a, the option **Discrete random walk** will be selected from the list. If the check box is cleared, then **None** will be selected from the list.

PARTICLE DENSITY SPECIFICATIONS MERGED

In version 5.2, when specifying particle properties using the **Particle Properties** node or **Override Properties** node in the Particle Tracing for Fluid Flow interface, it is possible to specify two different values of the particle density, depending on whether the **Compute particle mass** check box is selected. In version 5.2a, these two values have been merged into a single parameter value; that is, the value or expression for the **Particle density** will be retained when selecting or clearing the **Compute particle mass** check box in the physics interface **Advanced Settings** section.

PARTICLE BEAM CHANGES

The names of several settings have been changed in the **Particle Beam** feature for the Charged Particle Tracing interface.

In the **Sampling from phase space distribution** list, **Uniform** has been renamed **KV**.

In the **Longitudinal velocity distribution** list, **Uniform** has been renamed **None** and **Gaussian** has been renamed **Normal**.

The parameters **Position refinement factor** and **Release distribution accuracy order** have been removed. Due to some changes in the algorithm for initializing particle positions, in version 5.2a, the distribution of particle positions no longer depends on the finite element mesh and may differ from the distribution in version 5.2.

Backward Compatibility with Version 5.1

RENAMED PHYSICS INTERFACE SETTINGS

The **Release type** list in the settings windows for the Charged Particle Tracing and Particle Tracing for Fluid Flow interfaces in earlier versions has been renamed **Particle release specification**. For the Charged Particle Tracing interface, the options **Static** and

Transient have been renamed to **Specify current** and **Specify release times** (the default), respectively. For the Particle Tracing for Fluid Flow interface, the options **Static** and **Transient** have been renamed to **Specify mass flow rate** and **Specify release times** (the default), respectively.

Backward Compatibility with Version 5.0 and Earlier

OLD NODES FOR PARTICLE-FIELD AND FLUID-PARTICLE INTERACTIONS ARE OBSOLETE

The old **Particle-Field Interaction** and **Fluid-Particle Interaction** nodes, which could be added directly to the Charged Particle Tracing and Particle Tracing for Fluid Flow interfaces, respectively, are now obsolete. They will be removed in a future version. Simulations of particle-field and fluid-particle interactions should instead use the dedicated multiphysics nodes.

ELASTIC COLLISION FORCE NODE HAS BEEN REPLACED

If a model created in version 5.0 or earlier containing the **Elastic Collision Force** node is opened in version 5.1 or later, this node will be replaced. If the **Collision model** in the old **Elastic Collision Force** node was set to **Monte Carlo**, a **Collisions** node with the **Elastic** subnode will be created. If instead the **Collision model** was set to **Friction**, the **Friction Force** subnode will be added.

Pipe Flow Module

NEW MULTIPHYSICS COUPLINGS BETWEEN THE PIPE FLOW INTERFACE AND SINGLE-PHASE FLOW INTERFACES

Connecting a pipe segment modeled with the Pipe Flow interface with a 3D single-phase flow body is significantly easier through the introduction of the new multiphysics coupling: **Pipe Connection**. The old **Pipe Connection** nodes inside the Pipe Flow interface and Laminar Flow Interface are now obsolete, since the same functionality is available in a more convenient manner using this new multiphysics coupling.

Backward Compatibility with Version 5.1 and older

REVISED FORMULATION OF THE PIPE FLOW INTERFACE

The Pipe Flow interface has been amended in version 5.2 with an inertial term. The stabilization and element shape functions have also been changed. Models created in previous versions will be opened using the old interface with a notification. For new models, only the new interface is available in the physics interface list.

Plasma Module

New Functionality in Version 5.4

MODIFIED PHYSICS INTERFACE: BOLTZMANN EQUATION, TWO TERM APPROXIMATION

The **Boltzmann Equation, Two-term Approximation** interface has been redesigned and is now available in 0D instead of 1D.

GLOBAL MODELING OF NON-MAXWELLIAN DISCHARGES

It is now possible to compute the EEDF using a two-term approximation of the Boltzmann equation for global plasma models.

OTHER NEW FEATURES

Some other key new features include:

- The transport properties for electrons can now be computed from the EEDF directly.
- A pulsed option has been added to the **Metal Contact** feature in the **Plasma, Time Periodic** interface. This allows the discharge to be driven by a pulse rather than a continuous application of the power.
- A **Species Group** feature has been added to the **Plasma, Time Periodic** interface, which can be used to improve mass conservation in models where there is a chemical composition of multiple different elements (for example, a mixture of Ar/N₂).
- A **Species Constraint Group** feature has been added to the **Plasma** and **Plasma, Time Periodic** interfaces that can be used to constrain the mole fraction of a given species. This is useful for complex chemistries, as you no longer have to deactivate and reactivate species and reactions during model development.
- The default solver settings for the **Plasma, Time Periodic** interface have been improved. The new settings may decrease the number of iterations required for convergence and are more robust.

New and Updated Applications and Models in Version 5.4

NEW MODELS

- Pulsed CCP Discharge.

- Global Model of an Argon Discharge with non-Maxwellian EEDF.

Backward Compatibility with Version 5.3a and Earlier

Due to the way the detailed balance is computed for excitation collisions, models set up from scratch could provide different results from the same model in version 5.3a. Models are most likely to be affected if:

- The mole fraction of the metastable species is high.
- The electron density is very high (above around $1 \times 10^{20} \text{ 1/m}^3$).

The new method for computing the detailed balance is more accurate and physically correct than the method used in version 5.3a and previous.

Backward Compatibility with Version 5.3 and Earlier

There have been several changes that could affect the solution obtained when running a model in version 5.3a versus 5.3 or earlier. This is due to the following changes and bug fixes:

- When electron impact reaction rate constants are computed from cross section data, due to a bug, the result will be different for the Maxwellian or Druyvesteyn electron energy distribution functions. For DC discharges, where Townsend coefficients are recommended, this should not make much of a difference. For inductively coupled or microwave discharges, the average electron density and plasma potential could change by as much as 5%, but typically around 2%. If significant differences in the solution are obtained, it is recommended that you contact support@comsol.com.
- Surface species are no longer available from the context menu. Instead, surface species will appear in the Model Builder when a surface reaction containing surface species is added. If the surface reaction is deleted, then the dependent surface species will also be deleted.
- The rate constant for a surface reaction that uses sticking coefficients has changed, and is now proportional to the molecular weight of the reactant instead of the mean molecular weight. In systems where the molecular weight is the same for all species, this change won't affect the results. In models containing species with very different molecular weights, the surface reaction rates may change according to the square root of the molecular weight of the reactant divided by the mean molecular weight. In such systems, it is recommended to activate the new **Mixture diffusion correction** property, which adds additional correction terms to the mass flux vector for the

heavy species. Overall discharge properties should not be significantly affected by this change.

- The definition of surface site concentrations and surface reaction rate rates has been corrected for models that include a surface chemical mechanism where the surface species have different site occupancy numbers. This should only result in small changes to the solution, if any. The total surface site concentration is now a property on the physics interface level, rather than computed for each species on the feature level.
- The equation to compute binary diffusion coefficients from characteristic length parameters has changed. This may produce very slight changes in the diffusion coefficients for the heavy species in the Plasma and Heavy Species Transport interfaces. Any change in the solution due to this should be a tiny fraction of a percent.

Backward Compatibility with Version 5.2 and Earlier

The old Inductively Coupled Plasma and Microwave Plasma interfaces can still be opened, edited, and solved. When such a model is opened, a warning is given indicating that the interface is obsolete and will eventually be removed.

Backward Compatibility with Version 5.1

Models using the Reduced Electric Fields study will interpret the unit in the edit field for the Reduced electric fields as Townsend. For example, when opening a model created in version 5.1, if the value for the Reduced electric fields was 10, this will now appear as 10[Td] in the study settings.

Backward Compatibility with Version 4.4

Old models will be updated to use the new boundary conditions, but re-solving a model still gives the same results as before. The following changes to the model are made upon opening:

- If an **Electric Potential** feature is applied to boundaries adjacent to a **Plasma Model** feature, it is converted to a **Metal Contact** feature. The settings from the original model are retained. If an **Electric Potential** feature is applied to boundaries adjacent to a **Charge Conservation** feature, it remains the same.
- If an **Electric Potential** feature is applied to a combination of boundaries adjacent to a **Plasma Model** and **Charge Conservation** feature, a **Metal Contact** feature is created for

the boundaries adjacent to the **Plasma Model** feature, and the **Electric Potential** feature remains on boundaries adjacent to the **Charge Conservation** feature.

- Any **External Surface Charge Accumulation** boundary conditions are replaced by a new **Dielectric Contact** feature. The settings from the original model are retained. The **External Surface Charge Accumulation** boundary condition is now obsolete.

Ray Optics Module

New Functionality in Version 5.4

ENHANCEMENTS TO THE PART LIBRARY

The Part Library for the Ray Optics Module has been overhauled in Version 5.4, with several major benefits.

Because the lenses and mirrors have been substantially changed, legacy parts with the old geometry sequences will be available for a limited time.

Fewer Internal Boundaries

The lenses in the new Part Library generally have fewer internal edges compared to their counterparts in version 5.3a. The mesh discretization error for reflection and refraction is usually the worst for rays that reach a boundary element adjacent to an edge in 3D, therefore the newer parts are much less prone to discretization error. The result is significantly better accuracy when tracing rays through complicated optical systems in version 5.4.

NEW PARTS AND PART VARIANTS

The Conic Mirror Off Axis 3D part is now available as either a standard or standalone variant. This affects the shape and orientation of the face opposite the reflecting surface.

The lens parts Spherical Equi-Concave, Equi-Convex, Plano-Concave, and Plano-Convex lenses can each be specified using one of four variants:

- Radius of curvature and center thickness
- Effective focal length and center thickness
- Radius of curvature and edge thickness
- Effective focal length and edge thickness

The following new parts are also available:

- Elliptical Planar Mirror
- Spherical Mirror
- Rectangular Planar Annulus

OPTICAL DISPERSION MODELS

The **Medium Properties** node for the Geometrical Optics interface now supports a wide range of popular optical dispersion models, such as the **Sellmeier** equation and its variants. If an optical material having a certain set of optical dispersion coefficients is loaded into the Geometrical Optics model, then the dispersion relation can be detected automatically. This makes it possible to trace rays through glasses made by different manufacturers, which might use different sets of optical dispersion coefficients.

Some of the built-in optical dispersion models include temperature dependence, making it easier to set up and run high-fidelity structural-thermal-optical-performance (STOP) analyses.

WAVELENGTH DISTRIBUTIONS AT RAY RELEASES

Polychromatic light can now be specified in one of two ways: either specifying a distribution of frequency values or a distribution of vacuum wavelengths.

IMPROVED ACCURACY FOR RAY TRACING IN ABSORBING MEDIA

The algorithm for computing ray intensity power in an absorbing medium, where the refractive index is complex-valued, has been improved. The new algorithm has significantly less numerical error, especially when the time steps or optical path length intervals in the study are very large.

GEOMETRY NORMALS FOR RAY-BOUNDARY INTERACTIONS

The **Use geometry normals for ray-boundary interactions** check box can be selected in the settings for the Geometrical Optics interface. If the geometry is undeformed, this causes the rays to query normal directions from the parameterized geometry rather than the boundary mesh, if the normals can be accessed in this way. Selecting this check box makes it possible to trace rays in an undeformed configuration more accurately, even if the mesh is not very fine.

NEW OPTION FOR INTENSITY COMPUTATION: COMPUTE POWER

It is now possible to solve for just intensity, just power, or both, along the ray paths. With either intensity or power, the polarization along each ray is known.

The algorithm for reinitializing ray intensity and power is now significantly more robust for any intensity computation method. In particular, models of enclosed systems, where the wavefront is not well-defined and rays can undergo a large number of reflections or refractions, are much more robust and accurate in version 5.4 compared to version 5.3a.

NEWTONIAN TELESCOPE

This tutorial model shows how to trace unpolarized light rays through a Newtonian telescope. The incoming light is reflected off a paraboloidal mirror onto a flat, elliptically shaped secondary mirror that folds the optical path by 90 degrees toward a flat focal plane. The telescope geometry is parameterized in terms of the primary mirror's focal length and diameter as well as the distance of the focal plane from the optical axis.

HUBBLE SPACE TELESCOPE

The Hubble Space Telescope (HST) is an example of a standard Cassegrain telescope. The “General Conic Mirror On Axis 3D” part from the COMSOL Multiphysics Part Library is used to construct the HST Ritchey-Chretien geometry. Multiple release features are used to generate on- and off-axis ray and spot diagrams.

PETZVAL LENS STOP ANALYSIS, ISOTHERMAL

This model demonstrates an integrated structural-thermal-optical performance (STOP) analysis of an optical system. The Petzval Lens tutorial is used as the basis for this model, together with a simple barrel geometry. A **Parametric Sweep** is performed over several uniform temperature values and the effect on the displacement fields and image quality as a function of temperature is shown.

PETZVAL LENS STOP ANALYSIS

This model demonstrates an integrated structural-thermal-optical performance (STOP) analysis of an optical system. The *Petzval Lens* tutorial is used as the basis for this model, together with a simple barrel geometry. The lens assembly is placed inside a thermo-vacuum enclosure where the exterior temperature is significantly different from the interior. The lens assembly is exposed to this exterior via a pair of windows. The resulting temperature gradients and displacement fields within the optical system are shown as well as the effect on image quality.

Backward Compatibility with Version 5.3 and Earlier

DIFFRACTION GRATING SETTINGS MOVED TO A SUBNODE

In version 5.3 and earlier, the **Grating** feature for the Geometrical Optics interface automatically released transmitted and reflected rays of diffraction order zero. In

version 5.3a, the rays of diffraction order zero are controlled by a **Diffraction Order** subnode, as are all other diffraction orders that might be released.

NEW OPTIONS FOR SPECIFYING DIFFRACTION GRATING ORIENTATION

In the **Grating** feature for the Geometrical Optics interface, the options to control grating orientation in 3D have been reorganized and expanded. It is now possible to specify either the direction of the grating lines or the direction of periodicity in the grating surface. The input **Grating orientation specification** has been renamed **Direction of grating lines**.

NEW OPTIONS FOR SPECIFYING REFLECTANCE OR TRANSMITTANCE

In the Geometrical Optics interface, the settings to control **Thin dielectric films on boundary** in the settings for the **Material Discontinuity** feature have been renamed and rearranged.

The old options **Single layer, specified reflectance** and **Single layer, specified transmittance** have been renamed to **Specify reflectance** and **Specify transmittance**, respectively. For both of these options and for the **Anti-reflective coating** option, which has not been renamed, the **Treat as single layer dielectric film** check box is selected when opening the model in version 5.3a. Otherwise, this check box is cleared, which is the default behavior.

ADJUSTMENT TO THE RAY TERMINATION FEATURE

In the **Ray Termination** feature for the Geometrical Optics interface, the **Termination criteria** list has been renamed **Additional termination criteria**.

REMOVED CHECK BOXES FOR OBSOLETE FUNCTIONALITY

In the settings for the Geometrical Optics interface, the **Allow propagation outside selected domains** check box has been removed. This check box was flagged as obsolete in version 5.3, so clearing it would have created a Warning node.

In the Settings window for the **Release from Grid** feature, the check box **Suppress interaction with coinciding exterior boundaries** has also been removed.

Backward Compatibility with Version 5.2a and Earlier

REFERENCE EDGE FEATURES HAVE BEEN REMOVED

The **Reference Edge** subnode, which was previously used to specify the orientation of some optical components in 3D, has been removed. Instead, the orientation of these

components is specified directly in the parent feature settings using an extra selection input. The following features are affected:

- **Linear Polarizer**
- **Linear Wave Retarder**
- **Mueller Matrix**
- **Grating**

Backward Compatibility with Version 5.2

ANISOTROPIC REFRACTIVE INDICES CAN NO LONGER BE SPECIFIED

In version 5.2 and earlier, it is possible to specify an anisotropic refractive index in the settings window for the **Medium Properties** node, despite the fact that the anisotropy of the refractive index is not taken into account when modeling ray propagation. In version 5.2a, only isotropic refractive indices can be specified in the settings window for the **Medium Properties** node. If an anisotropic refractive index was specified in an older version, then when opening the model in version 5.2a, all of the diagonal terms will be populated using the first diagonal value and all off-diagonal terms will be set to zero.

RENAMED INTENSITY COMPUTATION METHODS

The options in the **Intensity Computation** list in the settings window for the Geometrical Optics interface have been renamed.

- **Using principal curvatures** has been renamed to **Compute intensity**.
- **Using principal curvatures and ray power** has been renamed to **Compute intensity and power**.
- **Using curvature tensor** has been renamed to **Compute intensity in graded media**.
- **Using curvature tensor and ray power** has been renamed to **Compute intensity and power in graded media**.

RENAMED POLARIZATION SETTING

In the **Initial polarization type** list, available for most ray release features in the Geometrical Optics interface, the option **Partially coherent** has been renamed to **Partially polarized**.

CHANGES TO PERTURBATIONS FOR SOLAR RADIATION

The treatment of perturbations due to sunshape and surface roughness has been changed in version 5.2a. This change affects the **Illuminated Surface** and **Solar Radiation** features for the Geometrical Optics interface.

The **Use corrections for finite source diameter** check box has been replaced by the **Corrections for finite source diameter** list. If the check box was selected, **Create light cones at release points** will be selected from the list after opening the model in version 5.2a. If the check box was cleared, **None** will be selected from the list.

For the **Illuminated Surface**, the **Standard deviation of the angle of incidence** parameter has been renamed **Surface slope error** and is now used to define a Rayleigh distribution for perturbations to the surface normal, rather than the angle of incidence of radiation. To account for the different effect that the surface slope error has on the model, any value or expression entered for the **Standard deviation of the angle of incidence** in version 5.2 or earlier will be divided by 2, before being used to specify the **Surface slope error** in version 5.2a.

The **Number of rays in wave vector** space $N_{w,r}$ has been removed. The effect of surface roughness can no longer multiply the number of degrees of freedom created by a release feature at each release point.

ACTIVATION CONDITIONS ON OPTICAL COMPONENTS

The following boundary conditions require some form of ray intensity computation in order to have any effect on the solution:

- **Linear Polarizer**
- **Linear Wave Retarder**
- **Circular Wave Retarder**
- **Ideal Depolarizer**
- **Mueller Matrix**

If any of these features is present in a model created in version 5.2 or earlier, and **None** is selected from the **Intensity computation** list in the settings window for the Geometrical Optics interface, then the feature will be automatically disabled when the model is opened in version 5.2a. To enable these features, select any option other than **None** from the **Intensity Computation** list.

RENAMED SETTINGS

The Wall condition **Bounce** has been renamed to **Specular reflection**.

CHANGES TO FREQUENCY-DEPENDENT MATERIAL PROPERTIES

The **Frequency-dependent refractive indices** check box has been renamed to **Allow frequency distributions at release features**.

Expressions involving the ray frequency must always be included within the `noenv()` operator. Some expressions for user-defined refractive indices may fail to be evaluated properly in version 5.2a unless this operator is applied.

THIN DIELECTRIC FILM SETTINGS

In version 5.0, if the ray intensity is computed, the option to add **Thin Dielectric Film** nodes to a **Material Discontinuity** node is always available. In version 5.2a, the Thin Dielectric Film subnode can only be added if **Add layers to surface** or **Add layers to surface, repeating** is selected from the **Thin dielectric films on boundary** list in the **Material Discontinuity** settings window. If a model created in version 5.0 is opened in version 5.2a, and any **Thin Dielectric Film** nodes have been added, then **Add layers to surface** is selected from the **Thin dielectric films on boundary** list instead of the default **None**.

CHANGES TO THE CIRCULAR WAVE RETARDER

The **Retardance** of the **Circular Wave Retarder** now indicates the rotation angle of the polarization ellipse of a ray that passes through the device. In version 5.0, the retardance corresponds to half of this rotation angle. The behavior of the **Circular Wave Retarder** is now consistent with the documentation. If a model created in version 5.0 is opened in version 5.2a, the expression for the **Retardance** of any **Circular Wave Retarder** node is multiplied by 2.

OBSOLETE DEPOSITED RAY POWER NODE

The **Deposited Ray Power** node for absorbing domains is obsolete and will be removed in future versions. For simulations in which rays generate heat due to absorption, the **Ray Heat Source** multiphysics node should be used instead.

RF Module

New Functionality in Version 5.4

ADDITIONAL RF MATERIAL LIBRARY FOR SUBSTRATES

The RF Module enhances the RF material library with more than 40 substrate materials from Isola group to assist in modeling printed RF, microwave, and millimeter-wave circuits.

UNIFORM ANTENNA ARRAY FACTOR FUNCTION

It is possible to evaluate the radiation pattern of an antenna array quickly from a single antenna radiation pattern by using an asymptotic approach, such as multiplying the far-field of a single antenna with a uniform array factor.

3D FAR-FIELD AND RCS FUNCTIONS FROM 2D AXISYMMETRIC COMPONENT MODEL

2D axisymmetric modeling becomes more useful for a quick estimation of the far-field response of the equivalent 3D model by utilizing new far-field functions. 3D far-field norm functions in 2D axisymmetric geometry are available in these cases:

- Antenna models using circular port excitation with a positive azimuthal mode number
- Scattered field analysis excited by the predefined circularly polarized plane wave type

MORE FAR-FIELD POSTPROCESSING VARIABLES

Maximum variables for directivity, gain, and realized gain have been added. These are available for global evaluation without plotting a 3D far-field pattern, when the selection for the far-field calculation feature is spherical for 3D and circular for 2D axisymmetric components, and its center is on the origin.

ELECTRICALLY THICK LAYER IN TRANSITION BOUNDARY CONDITION

The electrically thick layer option makes the two domains adjacent to the transition boundary uncoupled. The boundary performs like an interior impedance boundary condition but the geometry no longer has to be a domain.

FAR-FIELD ANALYSIS IN TRANSIENT STUDY

A far-field domain/calculation feature is now available in the Electromagnetic Wave, Transient physics interface. With this feature, a wideband antenna far-field pattern analysis in the frequency domain can be obtained by performing a transient response analysis and a time-to-frequency fast Fourier transform (FFT).

CIRCULARLY POLARIZED BACKGROUND FIELD IN 2D AXISYMMETRIC COMPONENT MODEL

A circularly polarized plane wave is available for the scattered field formulation when modeling with a 2D axisymmetric component. By exciting an axisymmetric scatterer with a circularly polarized background field in a 2D axisymmetric geometry, the far-field and radar cross section (RCS) of the same scatterer in 3D, illuminated by a linearly polarized background field, can be quickly estimated using the `norm3DEfar` function.

USER INTERFACE ENHANCEMENT IN PORT FEATURE

In and Outport Direction

Arrow visualization on the port feature helps to quickly identify the inport (active and excited port) and outport (passive and listener port) through the Graphics window. The arrow points in the direction of power flow. An excited port presents an inward arrow on the port boundary while a listener port has an outward arrow on the port boundary. Lumped ports support this visualization feature.

Numeric TEM with Voltage Drop Direction

When numeric-type ports are analyzed by boundary mode analysis, degenerated modes make it problematic to compute S-parameters accurately. The voltage drop direction represents the red arrow tangential to the port boundary, and it fixes the port mode field polarization. The direction can be changeable by clicking the **Toggle Voltage Drop Direction** button in the Settings window of the **Integration Line for Voltage** node, which is a subnode to the numeric TEM-type port feature.

MORE FEATURES FOR TIME EXPLICIT PHYSICS INTERFACE

Perfect electric conductor (PEC), perfect magnetic conductor (PMC), and surface current density now can be applied on interior boundaries as well. A lumped port is added and it calculates S-parameters. Lumped ports can only be added on exterior boundaries.

NEW STUDY SEQUENCES IN THE MODEL WIZARD FOR ONE-WAY COUPLED MULTIPHYSICS

When adding an electromagnetic heating multiphysics interface like Microwave Heating in the Model Wizard, there are now two new study sequences available. The Sequential Frequency-Stationary study first solves a frequency-domain equation for electromagnetics and then uses the electromagnetic heat source as a source term when solving a subsequent stationary heat transfer equation. The Sequential Frequency-Transient study first solves a frequency-domain equation for electromagnetics and then uses the electromagnetic heat source as a source term when solving a subsequent time-dependent heat transfer equation. For both study sequences, it is assumed that the electromagnetics problem does not depend on the computed temperature distribution.

THE REFRACTIVE INDEX ELECTRIC DISPLACEMENT FIELD MODEL NOW ALLOWS INPUT OF FULLY ANISOTROPIC TENSORS

When the **Refractive index** option is selected for the **Electric displacement field** list in wave equation features, it is now allowed to input a fully anisotropic tensor. A matrix-matrix multiplication is used to transform this refractive index tensor to the relative permittivity tensor.

DEFAULT COLOR TABLE FOR DEFAULT PLOTS CHANGED TO RAINBOWLIGHT

To improve legibility of the default black texts that appear on low-level parts of a plot, the default color table has been changed to RainbowLight for default plots. Now, the low-level parts are rendered with a lighter blue tone, improving the legibility.

NODAL CONSTRAINTS AND EXPLICIT-ORTHONORMAL NULL-SPACE FUNCTION

The default constraint type is now Nodal instead of the previous Elemental. This allows more efficient constraint elimination through the use of the explicit-orthonormal null-space function.

New and Updated Models in Version 5.4

TRANSIENT ANALYSIS OF A PRINTED DUAL-BAND STRIP ANTENNA

A wideband antenna study, such as S-parameter and far-field pattern analyses, can be obtained by performing a transient response analysis and a time-to-frequency fast Fourier transform (FFT). This model runs a time-dependent study first and then

transforms the dependent variable, the magnetic vector potential A , and a voltage signal at a lumped port from the time domain to the frequency domain. S-parameters and far-field radiation data is generated from the frequency domain data. The computed S-parameters shows two resonance in the given frequency range, as expected for this dual-band antenna design

STUDY OF A DEFECTIVE MICROSTRIP LINE VIA FREQUENCY-TO-TIME FFT ANALYSIS

While transient analyses are useful for time domain reflectometry (TDR) to handle signal integrity (SI) problems, many RF and microwave examples are addressed using frequency domain simulations generating S-parameters. However, from the frequency domain data, it is difficult to identify sources for this signal degradation. This example simulates a microstrip line in the frequency domain with a couple of line width discontinuities and performs frequency-to-time fast Fourier transform. The computed results help to identify the physical discontinuities and impedance mismatches on the transmission line by analyzing the signal fluctuation in the time domain.

UPDATED MODELS

Application Library path for the updated examples using parts from the RF Part Library:

- RF_Module/Antennas/double_ridged_horn_antenna
- RF_Module/Couplers_and_Power_Dividers/branch_line_coupler
- RF_Module/Couplers_and_Power_Dividers/wilkinson_power_divider
- RF_Module/Transmission_Lines_and_Waveguides/sma_connectorized_gcpw

Application Library path for the updated examples using the Isosurface plot type:

- RF_Module/Antennas/biconical_frame_antenna
- RF_Module/Antennas/dipole_antenna
- RF_Module/Antennas/helical_antenna
- RF_Module/Antennas/log_periodic_antenna
- RF_Module/Antennas/vivaldi_antenna
- RF_Module/EMI_EMG_Applications/airplane_antenna_crosstalk
- RF_Module/Filters/cascaded_cavity_filter
- RF_Module/Filters/cylindrical_cavity_filter_evanescent

Application Library path for the updated example on calculating an axial ratio:

- RF_Module/Antennas/helical_antenna

Application Library path for the updated example on visualizing material properties:

- RF_Module/Microwave_Heating/sar_in_human_head

Application Library path for the updated example on using an array factor:

- RF_Module/Antennas/microstrip_patch_antenna_inset

Rotordynamics Module

New Functionality in Version 5.4

DISK PROPERTIES FROM GEOMETRICAL DIMENSION

When adding a circular **Disk** in the Beam Rotor interface, you can now also compute the mass properties from geometrical size and mass density.

FLOATING RING BEARING

A new **Floating Ring Bearing** model has been added to the Hydrodynamic Bearing interface. In a floating ring bearing, an extra ring is placed between the bearing and journal surfaces so that two lubrication films are formed. These films are connected by oil channels in the ring. The ring can be either fully floating or semifloating.

ROTOR COUPLING

The new **Rotor Coupling** feature in the Beam Rotor and Solid Rotor interfaces allows for shafts that are not coupled in all directions. This makes it possible to model, for example, spline couplings. You can also model connections between shafts with a certain flexibility.

MISALIGNMENT

You can now model misalignment in bearings by adding the new **Misalignment** subnode. It is available for:

- **Journal Bearing, Thrust Bearing, Radial Roller Bearing,** and **Rotor Coupling** in the Solid Rotor interface.
- **Journal Bearing, Thrust Bearing, Radial Roller Bearing,** and **Rotor Coupling** in the Beam Rotor interface.
- **Hydrodynamic Journal Bearing, Hydrodynamic Thrust Bearing,** and **Floating Ring Bearing** in the Hydrodynamic Bearing interface.

Both parallel and angular misalignment can be modeled.

TILT ANGLE IN TILTED PAD BEARING

The pads in tilted pad bearings are usually free to tilt about a pivot point. In the equilibrium state, the tilt angle is such that the net moment on the pad due to the pressure in the fluid film is zero. It is now possible to compute the equilibrium tilt

angle automatically by performing nonlinear iterations to achieve moment equilibrium.

FOUNDATIONS FOR ROLLER BEARINGS

Foundation for the roller bearings have been added. In the **Foundations** section in the settings for **Radial Roller Bearing**, you can now choose to use **Fixed**, **Moving**, or **Flexible** foundation. Using this functionality, it is possible to model stator-rotor assemblies connected through roller bearings.

STATIC EQUILIBRIUM OF JOURNAL

You can now compute the equilibrium position of the journal in the bearing for a given static load. This is available for **Hydrodynamic Journal Bearing** and **Floating Ring Bearing** in the Hydrodynamic Bearing interface.

ROLLER FORCE DISTRIBUTION

You can now visualize the roller force distribution in roller bearings on the boundaries connected to the inner race for both the Solid Rotor and Multibody Dynamics interfaces. The variables, which are available in the results menus, have names like `rotsld.rrb1.fbx`, `rotsld.rrb1.fby`, and so on.

New Models in Version 5.4

RECIPROCATING ENGINE WITH HYDRODYNAMIC BEARINGS

A single cylinder reciprocating engine supported on hydrodynamic bearings is studied. A starting torque is applied to bring the engine to the required rpm. The loading torque is switched on once the engine picks up speed. After the start-up, the engine operates on its own, driven by the cylinder pressure. The engine assembly is modeled using the Multibody Dynamics interface in the Multibody Dynamics Module, and the bearing is modeled using the Hydrodynamic Bearing interface in the Rotordynamics Module. A **Solid-Bearing Coupling** multiphysics coupling is used. The results include stress in the crankshaft and foundation, pressure in the bearings, engine speed variation, generated power, brake horsepower, and orbit of the shaft in the bearing.

TURBOCHARGER SUPPORTED ON FLOATING RING BEARINGS

In this model, a turbocharger supported on two floating ring bearings is studied. The film forces acting on the ring cause it to rotate within the bearing. In a steady state, the ring rotates at a constant angular speed, and the torque on the ring due to the inner and outer films is balanced. The Beam Rotor interface is used to model the

turbocharger rotor, which is coupled with the floating ring bearing using multiphysics couplings. The inner and outer films in the floating ring bearings are connected by six channels in the ring. The results include stress in the rotor, pressure distribution in the bearing, speed variation of the ring, total moment on the ring, and flow rate through the ring channels.

EFFECT OF BEARING MISALIGNMENT ON ROTOR VIBRATION

In this example, a rotor supported on two hydrodynamic bearings is analyzed. An eccentric disk located between the two bearings causes the rotor to whirl. One of the bearings is misaligned with the axis of the rotor. The Beam Rotor with Hydrodynamic Bearing interface in the Rotordynamics Module is used for the combined simulation of rotors and bearings. The analysis is performed for two cases, first without misalignment and then including the misalignment. Results for both cases are compared to understand the effect of the misalignment. Some of the results are the orbit of the disk, frequency spectrum of the rotor response at the disk, and pressure distribution in the bearings.

STEP THRUST BEARING

In this tutorial model, a six-step thrust bearing is analyzed. A step thrust bearing consists of a stepped bearing surface on which the end of the shaft rotates. The entire assembly is submerged in a lubricant. The shaft collar is assumed to be spinning without any axial motion in the bearing. The results include pressure distribution in the bearing, pressure variation in radial and circumferential directions of the pad, and load-carrying capacity of the bearing.

Semiconductor Module

New Functionality in Version 5.4

SCHRÖDINGER-POISSON EQUATION

A new **Schrödinger-Poisson Equation** multiphysics interface is introduced to create a bidirectional coupling between an **Electrostatics** physics interface and a **Schrödinger Equation** physics interface to model charge carriers in quantum-confined systems. The electric potential from the Electrostatics interface contributes to the potential energy term in the Schrödinger Equation interface. A statistically weighted sum of the probability densities from the eigenstates of the Schrödinger Equation interface contributes to the space charge density in the Electrostatics interface. All spatial dimensions (1D, 1D axial symmetry, 2D, 2D axial symmetry, and 3D) are supported.

A dedicated **Schrödinger-Poisson** study type is available for automated generation of self-consistent iterations in the solver sequence. A new benchmark model, *Self-Consistent Schrödinger-Poisson Results for a GaAs Nanowire*, demonstrates the usage of this new feature.

TRAP-ASSISTED SURFACE RECOMBINATION

A new boundary condition, **Trap-Assisted Surface Recombination**, replaces the **Surface traps** check box in the features **Insulation**, **Thin Insulator Gate**, and **Insulator Interface**. Unlike the old check box, which only allows one option of explicit traps, the new boundary condition has both options of SRH recombination and explicit traps like its domain counterpart (the Trap-Assisted Recombination domain condition). In addition, the new boundary condition is extended to include **Schottky contacts**. A problem in the FEM formulation (surface charge has no effect) has been fixed.

WKB TUNNELING MODEL

A new tunneling feature based on the WKB approximation is now available to account for the additional current densities for the carrier transport across a heterojunction or a Schottky barrier via quantum tunneling. A new benchmark model, *Heterojunction Tunneling*, demonstrates the usage of this new feature.

HETEROJUNCTIONS (THERMIONIC EMISSION OPTION) AND SCHOTTKY CONTACTS

A new section, **Extra Current Contribution**, is now available for adding a user-defined extra contribution or for using the new **WKB Tunneling Model** mentioned above.

HETEROJUNCTIONS

For the **Thermionic Emission** option, a single value of A^* (Richardson's coefficient) is computed by picking the smaller effective mass between the two sides for the thermionic emission current densities on both sides, so that the total current density consistently cancels to zero at equilibrium.

INSULATOR TUNNELING

The variable for the perpendicular electric field in the insulator (`semi.E_ins`) is now available for the user-defined tunneling type, making it more convenient to define the tunneling current density.

STABILITY IMPROVEMENTS

The Fletcher mobility model and the SRH, Auger, and Direct recombination models have been improved by incorporating nonnegative carrier concentration values in the formulation.

TERMINAL NAMING WHEN DUPLICATING

When duplicating or pasting metal contacts, insulated gates, or electrostatic terminals, the duplicated entity now has a new unique terminal name. Before version 5.4, the duplicated entity had the same terminal name as the original entity.

SMALL-SIGNAL TERMINAL CURRENT

The terminal current variable for the small-signal analysis now includes the contribution from the displacement current. This makes it convenient to compute lumped parameters such as the differential capacitance of a biased Schottky contact.

SMALL-SIGNAL CONTINUOUS TRAPPING LEVELS

The small-signal analysis can now be performed on systems with continuous trapping levels.

SEMICONDUCTOR EQUILIBRIUM STUDY STEP

The formulation is improved for current-driven metal contacts.

SI SOLAR CELL MODEL

The updated model *si_solar_cell_1d* uses AM 1.5 solar irradiance and silicon absorption spectra for the photogeneration rate.

HETEROJUNCTION 1D MODEL

The updated model shows 4 different ways to achieve better convergence: Study 1, manual scaling; Study 2, inherit solution from Study 1 (same as before); Study 3, Semiconductor Equilibrium step as initial condition; and Study 4, ramp both doping and thermionic current from $1e-8$ (doping ramp was off before). Outdated solver tweaks (initial damping, iteration number) have been removed; comments, the model description, and the model documentation have been updated; and labels have been renamed.

GAN_DOUBLE_HETEROSTRUCTURE_LED MODEL

The ramping and tweaking of initial values and solver settings have been replaced by the Semiconductor Equilibrium study step. Solver tweaks for the current bias study have been removed. The model description, setup comments, and model documentation have been updated.

EEPROM MODEL

Study 1 has been changed to the default solver and manual scaling for better convergence (the model had been using the highly nonlinear Newton setting and then barely converged).

CORRECTIONS AND BACKWARD COMPATIBILITY

- The nonradiative recombination heat source is corrected with a minor offset of $(3/2)kT$. This is a very small correction, since the majority of the heat source is contributed by the band gap energy, which is much larger than kT .
- The *gan_double_heterostructure_led* model has changed the InGaN thickness from 5 nm to 50 nm to match reference paper.
- A **Vertical height** input is available for 1D axisymmetric geometries, with the default value 1 [um] (had been hard-coded to 1 [m]).
- The default solver suggestion now has sparsity pattern caching turned on for both stationary and time-dependent studies and all formulations.
- For the **Schottky Contact**, the input for the metal work function is hidden for user-defined barrier heights.
- For the Schrödinger Equation interface, periodic boundary conditions are available for 2D axially symmetric geometries.

SELF-CONSISTENT SCHRÖDINGER-POISSON RESULTS FOR A GAAS NANOWIRE

This benchmark model simulates a GaAs nanowire using the self-consistent Schrödinger-Poisson theory to compute the electron density and the confining potential profiles. The predefined Schrödinger-Poisson multiphysics coupling feature is combined with the dedicated Schrödinger-Poisson study type to provide a streamlined procedure for model setup and automated creation of self-consistent iterations with tunable parameters for optimizing the convergence under different conditions. The computed electron density and confining potential profiles compare well with the figures in the reference paper with both profiles reproducing the distinct Friedel-type spatial oscillations at low temperatures.

HETEROJUNCTION TUNNELING

This benchmark model simulates a graded heterojunction using the thermionic emission formulation for the charge transfer over the junction. It shows the additional contribution to the current density from the quantum tunneling effect across the potential barrier, using the WKB approximation. Even though the simulated system is 1D in nature, a 2D model is created to illustrate the procedure for simulating a more general configuration. Also demonstrated is the setup of user-defined ternary material properties. The computed I-V curves, their temperature dependence, and energy band diagrams all agree well with results from the literature.

INTERFACE TRAPPING EFFECTS OF A MOSCAP

This tutorial compares experimental data from the literature with a COMSOL Multiphysics model of a MOSCAP with interface traps (surface states). The Trap-Assisted Surface Recombination feature is used to simulate the effects of the trap charges and the processes of carrier capturing and emitting by the traps. The effect of the fixed charges in the gate oxide is also included. The computed values of the capacitance and conductance as functions of the gate voltage and frequency reproduce the qualitative behavior of the experimental data with comparable magnitudes. The model uses the quasi-Fermi level formulation and shows how to plot quantities such as the trap occupancy as a function of the energy.

FORWARD RECOVERY OF A PIN DIODE

This tutorial simulates the turn-on transient (forward recovery) of a simple PIN diode, based on the book “Fundamentals of Power Semiconductor Devices” by B. J. Baliga

(p. 242, 2008 edition). The diode is current driven with a constant ramp rate of $1e9$, $2e9$, and $1e10$ A/cm²/sec as well as a steady-state current density of 100 A/cm². The resulting time evolution of the device voltage and electron concentration compare well to those shown in the book (Fig. 5.30–5.31). For a more sophisticated example that includes band gap narrowing, carrier-carrier scattering, and external load circuit elements, see the tutorial “Reverse Recovery of a PIN Diode”.

REVERSE RECOVERY OF A PIN DIODE

This tutorial simulates the turn-off transient (reverse recovery) of a simple PIN diode with an inductive load, loosely based on the book “Fundamentals of Power Semiconductor Devices” by B. J. Baliga (p. 256, 2008 edition). Unlike the book, which assumes an initial constant current ramp rate followed by an abrupt bottoming out of the device voltage at the supplied reverse voltage, this model uses the circuit capability of COMSOL Multiphysics to simulate the inductive load with a flyback diode in a more realistic fashion. The resulting time evolution of the current, voltages, and carrier concentrations compare well to those shown in the book (Fig. 5.42–5.45).

Structural Mechanics Module

New Functionality in Version 5.4

NEW SHELL INTERFACE IN 2D AXISYMMETRY

The Shell interface is now available in a 2D axisymmetric geometry. The formulation and functionality are essentially the same as in its 3D counterpart. Also, the **Solid-Shell Connection** multiphysics coupling has been extended to 2D axisymmetry.

CHANGED INPUT OF THICKNESS AND OFFSET IN THE SHELL INTERFACE

The input of thickness and offset properties has been moved to a separate node named **Thickness and Offset**. One such node is always present in the model, and you can add more as required.

This means that the old input of these properties in the interface settings, as well as the **Change Thickness** node, has been removed. Old models are automatically converted to use the new syntax.

CHANGES IN SYMMETRY AND ANTISYMMETRY FOR 2D AXISYMMETRIC INTERFACES

In the 2D axisymmetric version of the Solid Mechanics and Membrane interfaces, the nodes **Symmetry** and **Antisymmetry** are now obsolete. They have been replaced by the **Symmetry Plane** node. The reason is that the only possible true symmetry in an axisymmetric model is with respect to a cut perpendicular to the Z -axis. The old features allowed unphysical constraints.

RESPONSE SPECTRUM ANALYSIS

Response spectrum analysis, which is used for estimating the peak response of short, nondeterministic, dynamic events, has been added. It can be used for any combination of structural mechanics interfaces in 2D and 3D. A new node, **Response Spectrum**, has been added under **Defintions>Variable Utilities**. Here, some variables are added, and you can also automate the setup of a special study sequence. Most of the settings for response spectrum evaluation are done in the new **Response Spectrum 2D** and **Response Spectrum 3D** data sets.

There are several modal combination methods, such as SRSS, CQC (der Kiureghian), absolute sum, ten percent method, grouping method, and double sum (Rosenblueth).

Four spatial combination methods are available: SRSS, 100-40-40, CQC3, and SRSS3.

It is also possible to separate periodic and rigid modes using the Gupta and Lindley-Yow methods and to add a static correction for mass not represented by the eigenmodes.

NEW MODAL SUPERPOSITION STUDY TYPES

Two new predefined study sequences for modal superposition have been added: **Time Dependent, Prestressed, Modal** and **Frequency Domain, Prestressed, Modal**. In both cases, three study steps are generated, where the first one is a stationary step in which the prestress state used for the subsequent eigenfrequency computation is determined.

RENAMED STUDY TYPES

Some study types have been renamed in order to get a more consistent naming:

- **Frequency Domain Modal** is now **Frequency Domain, Modal**
- **Time Dependent Modal** is now **Time Dependent, Modal**
- **Prestressed Analysis, Eigenfrequency** is now **Eigenfrequency, Prestressed**
- **Prestressed Analysis, Frequency Domain** is now **Frequency Domain, Prestressed**

PARTICIPATION FACTORS FOR EIGENFREQUENCY STUDIES

When you add an **Eigenfrequency** study from the **Add Study** window, a **Participation Factors** node is now automatically added under **Definitions**, as long as there is at least one structural mechanics physics interface in the model. You will thus always have access to the participation factor variables after running such a study.

MATERIAL ACTIVATION

It is now possible to activate a material in a stress-free state using the new **Activation** subnode under **Linear Elastic Material** in the Solid Mechanics and Membrane interfaces. A user-defined logical expression is used for the activation. The same feature can also be used to remove material. Typical applications are various types of manufacturing processes where material is added.

HOMOGENIZATION OF PERIODIC MATERIALS

Inhomogeneous materials can often be represented by a unit cell that is assumed to be part of a periodic structure. This is sometimes called a *representative volume element*

(RVE). The new **Cell Periodicity** feature in the Solid Mechanics interface provides several options for analyzing such unit cells:

- Automatic computation of equivalent homogenized anisotropic material properties (stiffness, compliance, thermal expansion), which can be accessed as a Global Material.
- Computation of the local stresses and strains in the unit cell, given external average stress and strain fields.
- Computation of free thermal or hygroscopic expansion of the unit cell, given average temperature or moisture concentration.

BURGERS VISCOELASTIC MODEL

One more built-in viscoelastic model has been added: the Burgers model. As with the other viscoelastic models, it is accessed by adding a **Viscoelasticity** subnode under the **Linear Elastic Material**.

VOLUME REFERENCE TEMPERATURE

A new concept, *volume reference temperature*, has been introduced. It serves two purposes: It replaces the earlier *strain reference temperature* for thermal expansion, and it is the temperature at which the mass density is computed. These two phenomena are physically related to each other, and the default is to use the same value, defined as a **Common model input**. For the mass density, the concept is new, and is used to avoid violations of mass conservation. For thermal expansion, the default location, where you specify the strain-free temperature, has moved from the **Thermal Expansion** node or **Thermal Expansion** multiphysics coupling to **Common model input**.

When you open an old model, the value of the strain reference temperature is converted to a **User Defined** volume reference temperature.

UTILITY FUNCTIONS FOR EXTERNAL MATERIALS

When programming your own constitutive relations using the external materials functionality, there are several operations for which there is a recurring need. Examples of this are various tensor operations, computation of principal values and orientations, and matrix inversion. There is now a library of utility functions that covers many common operations you would employ when programming material models for solid mechanics.

TIME STEP PASSED TO EXTERNAL MATERIALS

The time or parameter step is now passed as an argument to the material function when using the **General Stress-Deformation Relation** socket.

NEW SETTINGS FOR MIXED FORMULATION

In earlier versions, a mixed (displacement-pressure) formulation could be activated using the **Nearly incompressible material** check box in a material model. It has been replaced by the **Use mixed formulation** drop-down list with the following three choices: **None**, **Pressure formulation**, and **Strain formulation**. The pressure formulation is the same mixed formulation as was used before, while the strain formulation is new. Instead of a pressure, the volumetric strain is used as an extra degree of freedom.

NEW FLUID-STRUCTURE INTERACTION INTERFACES

The fluid-structure interaction (FSI) couplings have been extended to allow interaction with shells and membranes. There are now six entry points for FSI under **Fluid Flow** in the **Add Physics** window:

- **Fluid-Solid Interaction**
- **Fluid-Solid Interaction, Fixed Geometry**
- **Fluid-Shell Interaction**
- **Fluid-Shell Interaction, Fixed Geometry**
- **Fluid-Membrane Interaction**
- **Fluid-Membrane Interaction, Fixed Geometry**

If you also have a license for the Multibody Dynamics Module, you will find two more entry points:

- **Fluid-Multibody Interaction**
- **Fluid-Multibody Interaction, Assembly**

The **Fluid-Structure Interaction, Fixed Geometry** multiphysics coupling present in previous releases is now obsolete. It has been merged with an updated **Fluid-Structure Interaction** multiphysics coupling. This makes it easier to switch between the two formulations. In the current version, the only difference lies in whether a **Deforming Domain** is active on a fluid domain or not.

Old models, where you used **Fluid-Structure Interaction, Fixed Geometry**, are not changed. You are, however, advised to update to the new type of modeling.

RIGID DOMAIN IN 2D AXISYMMETRY

The Rigid Domain material model is now available in 2D axisymmetry. It has only a single degree of freedom: the axial translation.

RIGID CONNECTOR FOR EDGES AND POINTS

The selection in a **Rigid Connector** can now be a combination of boundaries, edges, and points. It is now also possible to add a rigid connector at the edge and point levels, as long as there are no higher-order geometrical objects in the selection.

FLEXIBLE FORMULATION OF RIGID CONNECTOR

There are now two formulations of the **Rigid Connector** feature: **Rigid** and **Flexible**. In the default formulation, all the selected boundaries, edges, and points behave as if they were connected by a common rigid body. In some cases, this gives an unwanted stiffening or unrealistic local stresses. You can then switch to a flexible formulation where the constraint is applied only in an average sense. The flexible formulation is available only for a rigid connector with a pure boundary selection and not when the selection also contains edges or points.

DRILL ADDED TO BOLT PARTS

In the Bolts section of the Part Libraries, there are now geometries that, in addition to the bolt itself, contain a domain intended to be used as a “drill” when building the geometry. This is particularly useful when working with the **Bolt Thread Contact** feature.

TABULAR OUTPUT OF EIGENFREQUENCIES

When running an Eigenfrequency study, a table containing the natural frequencies, modal damping, and quality factors is now generated by default.

SPRING AND REACTION FORCES

A new reaction force variable, containing the sum of reaction forces from constraints and reaction forces from spring foundations, has been added to all structural mechanics physics interfaces. The variables, which are available in the result menu, are called **Total reaction force** and have component names like `solid.RFtotalx`. If there are spring foundations in your model, you will find the forces under the category **Spring and Damping forces** in the result menus.

REACTION-FREE SYMMETRY CONDITION

The symmetry conditions in the structural mechanics interfaces have been extended to account for cases where the symmetry plane is allowed to translate in the direction of

its normal rather than being a fixed plane. The main application is for use as a boundary condition in truncated structures having no net reaction force. In the new section **Normal Direction Condition** in the settings for the **Symmetry** node, you can adjust the behavior of this boundary condition. There are four different options: **No displacement** (the default), **Free displacement**, **Prescribed force**, and **Prescribed displacement**.

ROLLER CONDITION WITH ANALYTICAL NORMAL ORIENTATION

The default roller constraint acts along a normal to the selected boundaries. The normal is computed numerically. For a certain mesh node, this direction is kept constant during the analysis, so it cannot represent finite sliding on a curved boundary. You can now, however, prescribe that the boundary slides on an analytical surface. There are two situations where you may need to do this:

- The geometry or mesh representation of the actual boundary is imperfect. This can lead to unexpected locking of the deformation, since the computed normals do not have the intended directions.
- In a geometrically nonlinear analysis, the use of an analytical surface makes it possible to also model finite sliding along curved boundaries.

You can now, in the settings in the **Roller** node, select a **Normal orientation**, which can be either **Automatic**, **Plane**, **Cylinder**, or **Sphere**.

IMPROVED PRESCRIBED ROTATION IN THE BEAM INTERFACE

The settings for prescribed rotations in the **Prescribed Displacement/Rotation** node in the Beam interface have been changed. The new representation is the same as in the corresponding node in the Shell interface, making it easier to prescribe finite rotations in a geometrically nonlinear case. In a model created in an earlier version, the settings are converted to the new style automatically.

New Models in Version 5.4

EARTHQUAKE ANALYSIS OF A BUILDING

This example shows how to verify the integrity of a structure exposed to an earthquake using response spectrum analysis. The building is modeled as a steel frame using beam elements. Displacements and stresses are computed.

SHOCK RESPONSE OF A MOTHERBOARD

Electronic equipment often needs to be certified to function after having been subjected to a specified shock load. In this example, the effect of an 11-ms half sine

shock with 50-g amplitude on a circuit board is investigated using response spectrum analysis. Results are compared with a full time-domain solution.

CONNECTING BEAMS AND SOLIDS

This example demonstrates the use of the **Solid-Beam Connection** multiphysics coupling to create transitions between the Solid Mechanics and Beam interfaces. Two different connection types are discussed, and a comparison of the stress distributions at the transition is made.

PRESTRESS OF MAIN BEARING CAP BOLTS

This example shows how to model prestressed bolts. The bolt geometry is taken from the Part Libraries. For comparison, one of the bolts is modeled using a thread contact formulation, whereas the other bolt is connected to the bolt hole by a pure continuity condition. The reduced stiffness in the threaded part of the bolt is taken into account by a reduction of the stiffness of the material.

MICROMECHANICAL MODEL OF A COMPOSITE

In this example, a simplified micromechanical model of a unit cell with periodic boundary conditions is analyzed. The homogenized elastic and thermal properties of a composite material are computed based on individual properties of the fiber and matrix. A comparison of numerical values is made against values obtained from the Rule of Mixture.

SPHERICAL CAP WITH CENTRAL POINT LOAD

This benchmark studies the nonlinear deformation of a spherical cap subjected to a point load at center using 2D axisymmetric Solid Mechanics and Shell interfaces. The results from the Shell interface are compared with the Solid Mechanics interface as well as with analytical results given in the benchmark for different discretizations.

Backward Compatibility with Version 5.2

The friction slip velocity is no longer defined as a dependent variable. When running a model using the COMSOL API, you will need to remove the reference to this variable in the solver settings. The friction slip velocity is usually defined as `<comp>_<solid>_vslip_<pairname>`, where `<comp>` is the tag of the component, `<solid>` is the tag of the Solid Mechanics physics interface where the **Friction** node is defined, and `<pairname>` is the name of the contact pair.

Backward Compatibility with Version 4.3b

- If an old model using the Beam interface is opened, the Euler-Bernoulli formulation is used. The Beam Formulation selection is still shown but cannot be changed from Euler-Bernoulli. If Advanced Physics Options is enabled, then the new section **Backward Compatibility** is shown. If you clear the **Use pre 4.4 formulation** check box, then the new formulation is used. Doing this enables the use of Timoshenko beams, but you must manually handle solver settings like segregation and scaling. This legacy option does not support beams mixed with solids or shells in cases where the same names for degrees of freedom were used in both physics interfaces.
- For models using mechanical contact, plot expressions are not updated when you open an old model where the pair names have been edited manually. Also, opening an old model where a pair name has been manually edited might affect the solver configuration. It is recommended that you regenerate a new default solver configuration.

Backward Compatibility with Version 4.3a

- When using version 5.2a, in the Solid Mechanics interface, the property `ControlsSpatialFrame` from version 4.3a no longer exists, and COMSOL Multiphysics now always assumes that the displacements control the spatial frame. Remove all lines setting the value of `ControlsSpatialFrame`. See Support Knowledge Base article 1156.
- When opening old models that include contact, the penalty factor control is set to **User defined** and the contact normal penalty factor is set to the expression used in the model.

Subsurface Flow Module

New Functionality in Version 5.4

Updated formulation for the Flux and No Flux nodes included in the Transport of Diluted Species and Transport of Diluted Species in Porous Media interfaces.

- The flux relative to the convective flux is prescribed instead of the total flux.
- The new formulation implies that the nodes can be used on a moving mesh geometry without compensating for the boundary velocity.
- Updated equation display for governing equations and boundary conditions. Equations updated to be formulated in terms of the flux relative to the convective flux.

There is an updated formulation for the Gravity feature included in Darcy's Law, Fracture Flow, and Richards' Equation interfaces.

There is a new formulation for the Symmetry node included in the Two-Phase Darcy's Law interface.

NEW PHASE TRANSPORT INTERFACES

The new **Phase Transport in Porous Media** interface is used to simulate the transport of multiple immiscible phases through a porous medium. The interface solves for the averaged volume fractions (saturation) of the phases without tracking the interface between the different phases. Microscopic interfacial effects are taken into account through capillary pressure functions.

The new **Multiphase Flow in Porous Media** interface combines the functionality of the Darcy's Law and Phase Transport in Porous Media interfaces. This multiphysics interface is intended to model flow and transport of multiple immiscible phases in a porous medium.

The new **Phase Transport** interface is used to simulate the transport of multiple immiscible phases in free flow. This interface solves for the averaged volume fractions of the phases in laminar or creeping flow, without tracking the interface between them.

TWO-PHASE FLOW IN A POROUS MEDIUM: BUCKLEY-LEVERETT MODEL

This benchmark models two-phase flow in a porous medium in 1D, where the flow can be analytically described by the Buckley-Leverett equation.

TWO-PHASE FLOW OVER A LOW PERMEABLE LENS

This example demonstrate two-phase flow in a porous medium that contains a low permeable lens. The heavier phase infiltrates the porous medium, so the low permeable lens is infiltrated only when a critical saturation is reached.

RESERVOIR WITH HORIZONTAL WELLS

This example models a thin oil reservoir with two horizontal wells. The reservoir contains two phases: water and oil. The oil is recovered by injecting water through the bottom well, so the oil production rate and the water-oil production ratio over time are computed.

Backward Compatibility with Version 5.3a and Earlier

NO FLUX FORMULATION

The formulation of the No Flux feature has been changed in the Transport of Diluted Species and Transport of Diluted Species in Porous Media interfaces in version 5.4 so that the flux relative to the convective flux is set to zero. The new formulation is used when opening a model from a previous session. In the previous formulation, the total flux was set to zero.

Wave Optics Module

New Functionality in Version 5.4

TRANSITION BOUNDARY CONDITION FOR THE ELECTROMAGNETIC WAVES, BEAM ENVELOPES INTERFACE

This new feature allows modeling of electrically thin layers using a boundary feature; thereby, the user does not need to add a domain mesh. This will be useful for modeling thin metallic layers in, for instance, mirror applications. However, this special implementation for the Electromagnetic Waves, Beam Envelopes interface also works fine for thin dielectric layers, like antireflection coatings.

IMPEDANCE BOUNDARY CONDITION FOR THE ELECTROMAGNETIC WAVES, BEAM ENVELOPES INTERFACE

This feature allows truncation of the simulation domain at an interface between two different material domains. The exterior material could be a highly conducting material, like a metal, but for this special implementation for the Electromagnetic Waves, Beam Envelopes interface, it could also be a dielectric layer, like a glass substrate.

SLIT PORTS FOR THE ELECTROMAGNETIC WAVES, BEAM ENVELOPES INTERFACE

A slit port is used on an interior boundary for exciting the simulation domain with an incident wave and then absorbing scattered waves matching the port mode field. A domain-backed slit port is normally backed by a perfectly matched layer (PML) that absorbs the remaining part of the scattered radiation that was not absorbed by the port. A PEC-backed slit port has a regular port on one side, and on the other side a perfect electric conductor (PEC) boundary condition is applied.

Domain-backed slit ports can be good to use when exciting the domain with a Gaussian beam, as the reflected Gaussian beam will not be perfectly absorbed by the port. However, together with the backing PML domain, almost all reflected radiation can be absorbed.

NEW STUDY SEQUENCES IN THE MODEL WIZARD FOR ONE-WAY COUPLED MULTIPHYSICS

When adding an electromagnetic heating multiphysics interface, like the Laser Heating interface, in the Model Wizard, there are two new study sequences available. The Sequential Frequency-Stationary study first solves a frequency-domain equation for electromagnetics and then uses the electromagnetic heat source as a source term when solving a subsequent stationary heat transfer equation. The Sequential Frequency-Transient study first solves a frequency-domain equation for electromagnetics and then uses the electromagnetic heat source as a source term when solving a subsequent time-dependent heat transfer equation. For both study sequences, it is assumed that the electromagnetics problem does not depend on the computed temperature distribution.

THE REFRACTIVE INDEX ELECTRIC DISPLACEMENT FIELD MODEL NOW ALLOWS INPUT OF FULLY ANISOTROPIC TENSORS

When the **Refractive index** option is selected for the **Electric displacement field** list in wave equation features, it is now possible to input a fully anisotropic tensor. A matrix-matrix multiplication is used to transform this refractive index tensor to the relative permittivity tensor.

DEFAULT COLOR TABLE FOR DEFAULT PLOTS CHANGED TO RAINBOWLIGHT

To improve legibility of the default black texts appearing on low-level parts of the plot, the default color table has been changed to RainbowLight for default plots. Now, the low-level parts are rendered with a lighter blue tone, improving the legibility.

UNIFORM ANTENNA ARRAY FACTOR FUNCTION

It is possible to evaluate the radiation pattern of an antenna array quickly from a single antenna radiation pattern by using an asymptotic approach such as multiplying the far-field of a single antenna with a uniform array factor.

3D FAR-FIELD AND RCS FUNCTIONS FROM 2D AXISYMMETRIC COMPONENT MODEL

2D axisymmetric modeling becomes more useful for a quick estimation of the far-field response of the equivalent 3D model by utilizing new far-field functions. 3D far-field norm functions in 2D axisymmetric geometry are available in these cases:

- Antenna models using circular port excitation with a positive azimuthal mode number
- Scattered field analysis excited by the predefined circularly polarized plane wave type

MORE FAR-FIELD POSTPROCESSING VARIABLES

Maximum variables for directivity, gain, and realized gain have been added. These are available for global evaluation without plotting a 3D far-field pattern, when the selection for the far-field calculation feature is spherical for 3D and circular for 2D axisymmetric components and its center is on the origin.

ELECTRICALLY THICK LAYER IN TRANSITION BOUNDARY CONDITION

The electrically thick layer option makes the two domains adjacent to the transition boundary uncoupled. The boundary performs like an interior impedance boundary condition, but the geometry no longer has to be a domain.

CIRCULARLY POLARIZED BACKGROUND FIELD IN 2D AXISYMMETRIC COMPONENT MODEL

A circularly polarized plane wave is available for the scattered field formulation when modeling with a 2D axisymmetric component. By exciting an axisymmetric scatterer with a circularly polarized background field in a 2D axisymmetric geometry, the far-field and radar cross section (RCS) of the same scatterer in 3D, illuminated by a linearly polarized background field, can be quickly estimated using the `norm3DEfar` function.

GUI ENHANCEMENT IN PORT FEATURE

Input and Output Direction

Arrow visualization on the port feature helps to quickly identify inport (active and excited port), and output (passive and listener port) through the Graphics window. The arrow points in the direction of power flow. An excited port presents an inward arrow on the port boundary while a listener port has an outward arrow on the port boundary.

MORE FEATURES FOR TIME-EXPLICIT PHYSICS INTERFACE

Perfect electric conductor (PEC), perfect magnetic conductor (PMC), and surface current density now can be applied on interior boundaries.

NODAL CONSTRAINTS AND EXPLICIT-ORTHONORMAL NULL-SPACE FUNCTION

The default constraint type is now Nodal instead of the previous Elemental. This allows more efficient constraint elimination through the use of the explicit-orthonormal null-space function.

New and Updated Models in Version 5.4

OPTICALLY ANISOTROPIC WAVEGUIDE

In this model, a modal analysis is performed while parametrically sweeping the length of a waveguide from 0.5 μm to 4 μm to derive the dispersion curve for the anisotropic core. Both transverse and longitudinal anisotropy are considered in two separate modeling steps.

FOCUSING LENS

Optical lenses of millimeter size cannot easily be analyzed with the Electromagnetic Waves, Frequency Domain interface on standard workstations due to the large number of mesh elements required.

This model explains how the Electromagnetic Waves, Beam Envelopes interface can be used to analyze a plano-convex lens with a 1-mm focal length. The results are compared with the Fresnel diffraction formula. The two results agree very well.

UPDATED MODELS

Application Library path for model using the new Transition boundary condition

- Wave_Optics_Module/Optical_Scattering/beam_splitter

Application Library path for model using the new Impedance boundary condition

- Wave_Optics_Module/Verification_Examples/fresnel_equations

New study in the Directional Coupler model using two copropagating modes

- Wave_Optics_Module/Waveguides_and_Couplers/directional_coupler

Material Library

New and Updated Material Data in Version 5.4

In version 5.4, the following improvements have been made:

- Added data for several types of engineering polymers: ECTFE (polyethylene chlorotrifluoroethylene), polyaryletherketone (PAEK), and polyvinylidene fluoride (PVDF).
- LiPb and Li17Pb83 have been added.
- Data has been added for Haynes 244, Inconel 783, Waspaloy, Alloy 901, Inconel 718, Ti-5.3Al-3Mo-1.5V, Ti-6Al-4V, and TRIP 980.
- Data has been added for Al-Zn, Bi-Sn, Sn-Zn, Cu-Sn, and Ag-Sn eutectic liquids.
- The expansion data for Haynes 242 has been refit with additional data added.
- An error in the stress-rupture data for Super 304H has been fixed.

LiveLink™ for MATLAB®

New Functionality in Version 5.4

LiveLink™ for MATLAB® now supports MATLAB® 2018a and 2018b.

The following sections list new and improved functionality in LiveLink™ for MATLAB®.

USER INTERFACES

The user interfaces for `mphnavigator`, `mphsearch`, and `mphmodellibrary` have been updated. They are installed as part of LiveLink™ for MATLAB® and can be started using their respective command names. Furthermore, it is possible to install **Apps** buttons available in the MATLAB® user interface for these commands using the installer found in `<comsol install dir>\mli\toolbox`.

The user interface for `mphnavigator` shows additional information when a solution or a study is selected. The **Tools>Solutions** menu provides more information about solvers, studies, and datasets as well as values of any swept parameters, time steps, or eigenvalues.

It is now possible to add a report to an existing model using `mphnavigator`. It is also possible to generate reports directly from `mphnavigator`.

Help for nodes can be obtained in two different ways. When clicking the **Help** button, help is opened using the COMSOL Documentation. When the new **Help for COMSOL API** button is clicked, help for the API methods and classes is opened for the selected node.

PLOTTING

The function `mphplot` has been updated to support new plot types added to COMSOL 5.4.

It is now possible to create plots of some nodes in the model tree (for example, physics, geometries, and functions). Such plots are created directly by COMSOL Multiphysics as an image and provide information that was not available previously. For showing where the potential boundary condition is applied in the busbar model, you can use

```
mphplot(model.physics('ec').feature('pot1'))
```

The function `mphviewselection` has improved graphics for displaying overlapping faces in 2D.

JOIN DATA SETS

The function `mphinterp` and other wrapper functions that access numerical data from models have improved support for Join data sets.

MPHLAUNCH

It is now possible to connect to a graphics COMSOL Multiphysics Server. Such a server must be started using the `-graphics` argument and allows plotting by the server.

MPHSOLINFO AND MPHSOLUTIONINFO

Both functions have been updated with improved handling of complex models; for example, models with multiple nested parametric sweeps and time-dependent models.

Backward Compatibility with Version 5.2

The functions `mpheval`, `mphglobal`, and `mphinterp`, which extract numerical data, now support complex values by default; that is, the property `'complexout'` is `'on'` by default. If you want to have the previous behavior, set `'complexout'` to `'off'`.

Backward Compatibility with Version 5.0

- The `mphint2` command: For data sets other than Solution, Particle, Cut Line, Cut Plane, Time Integral, Time Average, Surface, and Line, the integration order now corresponds to the element refinement. The default value is set to 4 instead of 1. This change can affect the solution in cases where the problem is not resolved well enough.

The COMSOL API for Use with Java[®]

COMSOL 5.4 API Changes

CAD IMPORT MODULE, DESIGN MODULE, AND LIVELINK PRODUCTS FOR CAD CHANGES

The new `fillholes` property has been added for the `Import` function. The following applies:

- The combination of the `fillholes` property set to `on` and the `knit` property set to `solid` is equivalent to the `knit` property set to `solid` in previous versions.
- The combination of the `fillholes` property set to `off` and the `knit` property set to `surface` is equivalent to the `knit` property set to `surface` in version 5.3a.
- The combination of the `fillholes` property set to `on` and the `knit` property set to `surface` is equivalent to the `knit` property set to `surface` in version 5.3 and previous versions.

The new `fillholes` property has been added for the `Knit` function. The `Knit` function with the `fillholes` property set to `on` corresponds to the `Knit` function in previous versions.

ECAD IMPORT MODULE CHANGES

- The properties `selnetcontributetobnd`, `selnetcontributetodom`, `selnetkeepbnd`, `selnetkeepdom`, `selnetnamebnd`, `selnetnamedom`, `selnetshowbnd`, `selnetshowdom`, `selnettagbnd`, and `selnettagdom` have been added to the `Import` function. The properties apply to the IPC-2581 and ODB++[®] file types, and enable working with the selections generated for the electrical nets found in the imported files.
- The `elevation` property of the `Import` function is now initialized when setting the `manualelevation` property to `off`. Previously, it was only initialized when setting the `filename` property, and when the `manualelevation` property is `off` and the value of either the `height` or `importtype` properties is changed.

COMSOL 5.3a API Changes

PARTITION OBJECTS PROPERTIES

The `keep` property has been deprecated in version 5.3a. Instead, use the new properties `keepinput` and `keeptool`. If you set `keep` to `on`, both `keepinput` and `keeptool` are set to `on`. If you get the value of `keep`, it is `on` if `keepinput` and `keeptool` are `on`.

COMSOL 5.3 API Changes

NEW COMPONENT SYNTAX

The geometry component is now included in the code for features that appear in a component. For example,

```
model.geom("geom1")
```

in earlier versions is now instead

```
model.component("comp1").geom("geom1")
```

to indicate which component the geometry belongs to. To keep the old syntax in generated Model Java[®] files, clear the **Use component syntax** check box under **Code generation** on the **Methods** page in the **Preferences** window. All old scripts and Java[®] source code files will continue to work using the old syntax.

ECAD IMPORT MODULE CHANGES

- The default value for the `sellayer` property of the `Import` function is now `on`. Previously, the default was `off`.
- The default value for the `sellayershow` property of the `Import` function is now `all`. Previously, the default was `dom`.

COMSOL 5.2a API Changes

AC/DC MODULE CHANGES

New Coil Feature

In the Magnetic Fields, Magnetic and Electric Fields, and Rotating Machinery, Magnetic interfaces, a new unified **Coil** feature has replaced the **Multi-Turn Coil** feature. The functionality of the old **Multi-Turn Coil** feature can be replicated by using the new **Coil** feature with the Conductor model set to Homogenized Multi-Turn, but all of the

other parameters are unchanged. The only change required is to the create statements such as:

```
model.physics("mf").feature().create("mtcd1",  
  "MultiTurnCoilDomain", 3);
```

which should be modified to:

```
model.physics("mf").feature().create("mtcd1", "Coil", 3);  
model.physics("mf").feature("mtcd1").set("ConductorModel",  
  "Multi");
```

User-Defined Coil Geometry Subfeature Selection

In 3D models, when using a user-defined geometry for a **Coil** feature, the **User Defined Coil Geometry** subfeature has been transformed from a boundary feature to a domain feature. New **Input** and **Output** subnodes are used to select the coil input or output boundaries instead. Any code setting the selection of this feature should be modified to set the selection on the **Input** subfeature instead.

Relative Permeability for Magnetic Shielding in Magnetic Fields, No Current

The relative permeability parameter μ_r in the **Magnetic Shielding** feature in the Magnetic Fields, No Currents interface is now a scalar quantity. Setting a tensor quantity will not cause exception, but only the first component of the tensor will be used as the value.

ACOUSTICS MODULE CHANGES

The default settings have changed for the Incident Pressure Field in the Pressure Acoustics interfaces. Add the following line to obtain the old behavior of this feature when `ipf1` is a subfeature to `pwr1`:

```
model.physics("acpr").feature("pwr1").feature("ipf1").  
  set("c", "acpr.c_c");
```

COMSOL 5.1 API Changes

ACOUSTICS MODULE CHANGES

The ratio of specific heats is now a material parameter. If the default value was used in a model, add the following line to the Java[®] file to obtain the old behavior:

```
model.physics("cpf").feature("cpf1").set("gamma_mat", "userdef");
```


ELECTROCHEMISTRY MODULES CHANGES

There are changes for the Tertiary Current Distribution, Nernst-Planck interface. The Java[®] API syntax for creating and accessing vectors and tensors has changed, as well as the syntax for setting physics properties. See the following examples:

- Specifying the diffusion tensor.

Old syntax:

```
model.physics("tcdee").feature("ice1").set("D_0", new
String[]{"D1", "0", "0", "0", "D1", "0", "0", "0", "D1"});
model.physics("tcdee").feature("ice1").set("D_1", new
String[]{"D2", "0", "0", "0", "D2", "0", "0", "0", "D2"});
model.physics("tcdee").feature("ice1").set("D_2", new
String[]{"D3", "0", "0", "0", "D3", "0", "0", "0", "D3"});
```

New syntax (includes the species name in the first argument of the set method):

```
model.physics("tcdee").feature("ice1").set("D_c1", new
String[]{"D1", "0", "0", "0", "D1", "0", "0", "0", "D1"});
model.physics("tcdee").feature("ice1").set("D_c2", new
String[]{"D2", "0", "0", "0", "D2", "0", "0", "0", "D2"});
model.physics("tcdee").feature("ice1").set("D_c3", new
String[]{"D3", "0", "0", "0", "D3", "0", "0", "0", "D3"});
```

- Accessing tensor components (in the definitions of other variables, for example).

Old syntax:

Dxx_c1, Dxy_c1, Dxz_c1...

New syntax (component index after the species name):

D_c1xx, D_c1xy, D_c1xz...

- Specifying the reaction rate vector.

Old syntax:

```
model.physics("tcdee").feature("react1").set("R_0", new
String[]{"R1"});
model.physics("tcdee").feature("react1").set("R_1", new
String[]{"R2"});
model.physics("tcdee").feature("react1").set("R_2", new
String[]{"R3"});
```

New syntax (includes the species name in the first argument of the set method):

```
model.physics("tcdee").feature("react1").set("R_c1", new
String[]{"R1"});
model.physics("tcdee").feature("react1").set("R_c2", new
String[]{"R2"});
model.physics("tcdee").feature("react1").set("R_c3", new
String[]{"R3"});
```

- Accessing vector components.

Old syntax:

```
tfluxx_c1, tfluxy_c1, tfluxz_c1
```

New syntax (component index after the species name):

```
tflux_c1x, tflux_c1y, tflux_c1z
```

- Specifying initial values.

Old syntax:

```
model.physics("tcdee").feature("init1").setIndex("c2", "c20", 0);
model.physics("tcdee").feature("init1").setIndex("c3", "c30", 0);
model.physics("tcdee").feature("init1").setIndex("V", "V0", 0);
```

New syntax:

```
model.physics("tcdee").feature("init1").setIndex("initc", "c20",
1);
model.physics("tcdee").feature("init1").setIndex("initc", "c30",
2);
model.physics("tcdee").feature("init1").setIndex("initphil",
"V0");
```

- Selecting properties.

Old syntax:

```
model.physics("tcdee").prop("Convection").set("Convection", 1,
"0");
model.physics("tcdee").prop("ConvectiveTerm").setIndex("Convectiv
eTerm", "noncons", 0);
```

New syntax (the property name corresponds to the section in the user interface):

```
model.physics("tcdee").prop("TransportMechanism").
set("Convection", 1, "0");
model.physics("tcdee").prop("AdvancedSettings").setIndex("Convect
iveTerm", "noncons", 0);
```

For information about API backward compatibility for the Surface Reactions interface, see the [Chemical Reaction Engineering Module](#) release notes.

ECAD IMPORT MODULE CHANGES

- The default value for the grouping property of the Import function is now set to layer. Previously, the default was all.
- For ODB++® and ODB++(X) files (the type property is odb), new rules apply for the initialization of the importlayer property of the Import function. Now, only

layers of the types *Metal* and *Dielectric* are initialized with the string on in the `importlayer` string array. Previously this also included layers of type *Drill*.

LIVELINK™ FOR SOLID EDGE® CHANGES

The default value for the `keepfree` property of the `LiveLinkSolidEdge` function is now set to `on`. Previously, the default was set to `off`.

LIVELINK™ FOR SOLIDWORKS® CHANGES

The default value for the `keepfree` property of the `LiveLinkSOLIDWORKS` function is now set to `on`. Previously, the default was set to `off`.

COMSOL 5.0 API Changes

DOCUMENTATION

The *COMSOL Multiphysics Programming Reference Manual* replaces the *COMSOL API for Use with Java® Reference Manual*.

DEPRECATED METHODS

The following methods were deprecated in COMSOL 5.0:

TABLE I-3: DEPRECATED METHODS AND THEIR REPLACEMENTS

DEPRECATED METHOD	NEW METHOD
<code>com.comsol.model.OptFeature.field()</code>	<code>OptFeature.comp()</code>
<code>com.comsol.model.OptFeature.field(String)</code>	<code>OptFeature.comp(String)</code>
<code>com.comsol.model.Group.identifier()</code>	<code>Group.paramName()</code>
<code>com.comsol.model.Material.identifier()</code>	<code>ModelEntity.tag()</code>
<code>com.comsol.model.MaterialModel.identifier()</code>	<code>ModelEntity.tag()</code>
<code>com.comsol.model.ModelNode.identifier()</code>	<code>ModelEntity.tag()</code>
<code>com.comsol.model.physics.Physics.identifier()</code>	Use <code>tag()</code> instead.
<code>com.comsol.model.physics.MultiphysicsCoupling.identifier()</code>	<code>ModelEntity.tag()</code>
<code>com.comsol.model.Group.identifier(String)</code>	<code>Group.paramName(String)</code>
<code>com.comsol.model.Material.identifier(String)</code>	<code>ModelEntity.tag(String)</code>
<code>com.comsol.model.MaterialModel.identifier(String)</code>	<code>ModelEntity.tag(String)</code>

TABLE I-3: DEPRECATED METHODS AND THEIR REPLACEMENTS

DEPRECATED METHOD	NEW METHOD
<code>com.comsol.model.ModelNode. identifier(String)</code>	<code>ModelEntity.tag(String)</code>
<code>com.comsol.model.physics.Physics. identifier(String)</code>	Use <code>tag(string)</code> instead.
<code>com.comsol.model.physics. MultiphysicsCoupling.identifier(String)</code>	<code>ModelEntity.tag(String)</code>
<code>com.comsol.model.ModelEntity.name()</code>	<code>ModelEntity.label()</code>
<code>com.comsol.model.ModelEntity.name(String)</code>	<code>ModelEntity. label(String)</code>
<code>com.comsol.model.ParameterEntity. set(String, int, double)</code>	Use the 0-based version (<code>setIndex</code>) instead.
<code>com.comsol.model.ParameterEntity. set(String, int, double)</code>	Use the 0-based version (<code>setIndex</code>) instead.
<code>com.comsol.model.ParameterEntity. set(String, int, double[])</code>	Use the 0-based version (<code>setIndex</code>) instead.
<code>com.comsol.model.ParameterEntity. set(String, int, int)</code>	Use the 0-based version (<code>setIndex</code>) instead.
<code>com.comsol.model.ParameterEntity. set(String, int, int[])</code>	Use the 0-based version (<code>setIndex</code>) instead.
<code>com.comsol.model.ParameterEntity. set(String, int, int, double)</code>	Use the 0-based version (<code>setIndex</code>) instead.
<code>com.comsol.model.ParameterEntity. set(String, int, int, int)</code>	Use the 0-based version (<code>setIndex</code>) instead.
<code>com.comsol.model.ParameterEntity. set(String, int, int, String)</code>	Use the 0-based version (<code>setIndex</code>) instead.
<code>com.comsol.model.ParameterEntity. set(String, int, String)</code>	Use the 0-based version (<code>setIndex</code>) instead.
<code>com.comsol.model.ParameterEntity. set(String, int, String[])</code>	Use the 0-based version (<code>setIndex</code>) instead.

PHYSICS INTERFACES

- The default **Initial Value** features in the following interfaces have been extended:
 - Transport of Concentrated Species
 - Reacting Flow in Porous Media (rfcs)
 - Reacting Flow
 - Rotating Machinery, Reacting Flow

Previously, only the initial mass fractions could be specified. Now, the initial values can be specified in terms of mass fractions, mole fractions, molar concentrations, number densities, or densities. When opening an old Model MPH file, **Mixture specification** is set to **Mass fractions**, and the mass fractions specified are entered in the respective text fields. In old model files for Java[®], commands specifying the initial value of a mass fraction return an error in version 5.0 and later. To specify the initial mass fraction of a species w2, which is the second in the list of species (seen in the **Dependent Variables** section when clicking the interface), change the COMSOL API Java[®] command

```
model.physics("chcs").feature("init1").set("w2", 1, "0.1");
```

to

```
model.physics("chcs").feature("init1").set("w0", 2, "0.1");
```

- The Pressure Acoustics, Frequency Domain interface and the Boundary Mode Acoustics interface have a new default feature. If the default feature has been edited in a Java[®] file, the following line should be added to the Java[®] file to obtain the old behavior for Pressure Acoustics, Frequency Domain:

```
model.physics("acpr").feature().create("pam1",  
"PressureAcousticsModel").selection().all();
```

or

```
model.physics("acbm").feature().create("pam1",  
"PressureAcousticsModel").selection().all();
```

for Boundary Mode Acoustics.

This line should be added directly after the line where the physics interface is created.

- For models using mechanical contact, the variable field name in the solver settings for the old contact variables is different than in earlier versions. For example:
 - In version 5.2 and later: `mod1_solid_contact_old_p1`,
`mod1_solid_cm_old_p1`.
 - In version earlier than 5.0: `mod1_solid_contact_p1_old`,
`mod1_solid_cm_p1_old`.
- The parameter `form` in the property `EquationForm` can no longer be set to `ModeAnalysis` in 3D and 2D axisymmetric calculations for the Pressure Acoustics, Frequency Domain and Pressure Acoustics, Transient interfaces. In these cases, the parameter `modeFreq` has been removed.

COMSOL 4.3b API Changes

PHYSICS INTERFACES

- The *Crosswind diffusion* formulation has been updated for all physics interfaces supporting Crosswind diffusion. Models solved with the new formulation can give different results than models solved in versions earlier than 4.3b. Java[®] files can be modified to retain old crosswind formulations. Please contact COMSOL Support for details.
- The default settings have changed for the Background Pressure Field in the Pressure Acoustics interfaces. Add the following line to obtain the old behavior of this feature:


```
model.physics("acpr").feature("bpf1").set("c", 1, "acpr.c_c");
```
- In the Acoustics Module, the **Far-Field Calculation** feature is no longer available in 1D and 1D axisymmetric models.
- The shape function property `border` has been deprecated and replaced by `order`.

COMSOL 4.3a API Changes

GENERAL API CHANGES

- Class files compiled with COMSOL Multiphysics version 4.3 or earlier need to be recompiled if they contain code that changes the contents of `model.selection()`.
- The `solutionintitle` property governs whether to show solution information in plot titles. But since solution information has never been shown for 1D plots,

regardless of the content of the property, the property has been removed from 1D plot groups.

FLUID FLOW INTERFACES API

- Fluid flow features in the CFD Module and Microfluidics Module that previously created a scalar Lagrange multiplier field with default name `model.un_lm` now create a vectorial Lagrange multiplier field with default name `model.u_lm`. The default component names in 3D are `model.u_lm`, `model.v_lm`, and `model.w_lm`, respectively. Java® files must be updated accordingly.
- Weak constraints for the Interior Wall feature are no longer available. Any references to its weak constraint parameter (`weakConstraints`) or Lagrange multipliers must be removed.
- The **Automatic** setting for the **Pseudo time stepping** property now sets the variable `<phys_tag>.locCFL` to the built-in variable `CFLCMP`, which in turn triggers a PID regulator via the automatic solver suggestions. Here, `<phys_tag>` is the physics interface tag. Java® files where pseudo time stepping is active and the local CFL number is set to **Automatic** must be modified by adding the command

```
model.physics(<tag>).prop("PseudoTimeProperty").set("CFLNumExpr", 1, "Manual");
```

before calling a solver.

COMSOL 4.3 API Changes

- In the Acoustics Module, the far-field variables `pfar` and `Lp_far` have new names with full scope. They are now referred to as `acpr.ffc1.pfar` and `acpr.ffc1.Lp_far`, respectively. Any model files for Java® that use the old variable names in expressions (such as expressions used for plotting or evaluation that include such old variable names) require a manual update.
- The following methods

```
model.physics(<tag>).feature(<ftag>).params();  
model.physics(<tag>).prop(propname).params();
```

are deprecated and replaced by the methods

```
model.physics(<tag>).feature(<ftag>).param();  
model.physics(<tag>).prop(propname).param();
```

- Class files compiled with COMSOL Multiphysics version 4.1 or earlier need to be recompiled.
- The **Far Field** feature in the Electromagnetic Waves interface has changed from being a boundary feature with a boundary selection only, to a domain feature with a domain selection. It also has a subfeature — a boundary selection.
- The dependent variable associated with gauge fixing in the Magnetic Fields, Magnetic and Electric Fields, Electromagnetic Waves, and Transient Electromagnetic Waves interfaces is now unique to each interface. It is no longer available in the model scope, for example, `mod1.psi`. Instead, the gauge fixing field is only available in the interface scope, for example, as `mod1.mf.psi`.
- In the scattered field formulation in the Electromagnetic Waves interface, the scattered field is no longer available in the model scope (for example, `mod1.relEx`). Instead, the scattered field is only available in the interface scope as, for example, `mod1.emw.relEx`.
- In the Solid Mechanics interfaces (`SolidMechanics`, `PiezoelectricDevices`, `AcousticStructure`, `Poroelasticity`, `ThermallyInducedStresses`, `JouleHeatingThermalExpansion`, `TransientAcousticStructure`, and `FluidStructureInteraction`), tensors in local coordinate systems (`e1`, `ee1`, `S1`, `si`, and `ei`) have new names. The coordinates of the local system (for example, `x1`, `x2`, and `x3`) are no longer used. Instead 1, 2, and 3 are used together with double indexing for all components. As an example, `e1X2` is replaced by `e122` and `e1x2x3` is replaced by `e123`. The tensors `si` and `ei` are now called `S11` and `e11`.
- In the Darcy's Law interface and the Richards' Equation interface in the Subsurface Flow Module, fluid compressibility is now a material parameter and no longer has a default value. If the default value was used, you now have to set the value. The following example sets the permeability to the old default value:

```
model.physics("d1").feature("smm1").set(chif_mat,userdef);
model.physics("d1").feature("smm1").set(kappa,4e-10);
```
- In the Poroelasticity interface in the Subsurface Flow Module, the fluid compressibility and the Biot-Willis coefficient are now material parameters and no longer have default values. If the default values were used, you now have to set the value. The following example sets the permeability to the old default value:

```
model.physics("d1").feature("smm1").set(chif_mat,userdef);
model.physics("d1").feature("smm1").set(kappa,4e-10);
```


- The Level Set and Phase Field interfaces now include the **Initial Interface** feature by default. If you have a model that was created in an earlier version of COMSOL Multiphysics, it will fail to create a feature with the same tag name.

COMSOL 4.1 API Changes

The following changes were made to the COMSOL API between versions 4.0a and 4.1:

- In version 4.0a, the property `nonlin` in the stationary solver could have the values `auto`, `on`, `off`, and `linearized`. The default in most cases was `auto`, but in some cases, specifically in a frequency-domain study, the default was `linearized`. In version 4.1, the possible values are `auto`, `on`, `off`, and `linper`, with `auto` as default, except in the Frequency Domain, Linearized study step where the default is `linper`. The value `linearized` can still be set, but this is treated as `auto`. If the problem is linear and the linearization point is zero, `auto` should give the same solution as `linearized`. For nonlinear problems where the linearization point is zero, the solution given by `auto` might be slightly different, and probably more accurate, while changing to `off` should give exactly the same solution as `linearized`. When there is a nonzero linearization point, it is relevant to use the `linper` option. This option works like `linearized`, except that source terms that contribute to the linearized problem must be enclosed by the `linper` operator.
- In the Darcy's Law interface, Brinkman interface, Richards' Equation interface, and Poroelasticity interface (in the CFD Module and Subsurface Flow Module), the permeability and porosity are now material parameters and no longer have default values. If the default values were used in a model file for Java[®], you now have to set them. The following example sets the permeability to the old default value:


```
model.physics("d1").feature("d1m1").set(kappa_mat,userdef);
model.physics("d1").feature("d1m1").set(kappa,3e-11);
```
- The pressure variable in solid mechanics, typically `solid.pw`, now only gets allocated degrees of freedom for incompressible materials. If you have referred to it, for example, during segregation in the solver, the code will have to be changed to remove the field.
- The solution to eigenvalue and eigenfrequency problems may now appear in a different order.
- The Batteries & Fuel Cells Module's boundary feature, `BoundaryReactionCoefficient`, is tagged by `rc` instead of `brc` by default. This

means that files that explicitly refer to the old default tag name must be modified to refer to the new tag name.

COMSOL 4.0a Java API Changes

The following changes were made to the COMSOL API between versions 4.0 and 4.0a:

- The units of the load face parameter (Fph) of the **Phase** feature in `AcousticStructure`, `TransientAcousticStructure`, `ThermallyInducedStresses`, `SolidMechanics`, `JouleHeatingThermalExpansion`, `Poroelasticity`, and `FluidStructureInteraction` have changed from degrees to radians.
- The `physics.field()` operator for `Beam` and `Truss` are not backward compatible with 4.0. Scalar fields are now vector fields.
- The variables for strain and stress in the `Shell` interface have been renamed. Variable names that began with a lowercase `s` now begin with an uppercase `S`. For variables names that ended in lowercase `l` (`L`), the letter was moved to the third position. For example, `emXXl` is now `emlXX`.
- Force loads on solids and trusses have been moved from the material to the spatial frame. This means that the index notation of some variables has changed from uppercase `XYZ` to lowercase `xyz`.
- The acoustics interfaces (`PressureAcoustics`, `TransientPressureAcoustics`, `AcousticStructure`, and `TransientAcousticStructure`) have been moved from the material to the spatial frame. This means that the index notation of vector variables has changed from uppercase `XYZ` to lowercase `xyz`.

This concludes the release notes for COMSOL Multiphysics version 5.4.

I n d e x

- A** AC/DC Module
 - backward compatibility 37
 - new applications in 37
 - new functionality 36
- Acoustics Module
 - backward compatibility 42
 - new functionality in 40
- Application Builder, new functionality in 13
- B** backward compatibility, general considerations 24
- Battery & Fuel Cells Module
 - new functionality in 44
- C** CAD Import Module
 - new functionality in 48
- CFD Module
 - backward compatibility 54–55
 - new functionality in 53
 - new models in 54
- Chemical Reaction Engineering Module
 - backward compatibility 64
 - new functionality in 62
- COMSOL API changes 166
- COMSOL Multiphysics, new functionality in 15
- COMSOL Server, new functionality in 15
- Corrosion Module
 - backward compatibility 70
 - new and updated models in 69
 - new functionality 68
- D** Design Module
 - new functionality in 48
- E** ECAD Import Module
 - backward compatibility 71
- Electrochemistry Module
 - new and updated models in 74
 - new functionality in 72
- Electrodeposition Module
 - new and updated models in 77
 - new functionality in 75
- F** Fatigue Module
 - backward compatibility 78
 - new functionality in 78
- G** general new functionality 13
- Geomechanics Module
 - new functionality in 79
 - new models in 80
- geometry and mesh, new functionality for 18
- H** Heat Transfer Module
 - backward compatibility 89, 91, 93–94
 - new applications in 87
 - new functionality in 81
- L** LiveLink for MATLAB
 - backward compatibility 165
 - new functionality in 164
- LiveLink™ products for CAD
 - new functionality in 48
- M** Material Library
 - new and updated material data in 163
 - new material data in 163
- MEMS Module
 - backward compatibility 108
 - new functionality in 103
- Microfluidics Module
 - backward compatibility 109
 - new functionality in 109
- Mixer Module
 - new functionality in 111

- Molecular Flow Module
 - backward compatibility 112
- Multibody Dynamics Module
 - new functionality in 113
- N** Nonlinear Structural Materials Module
 - new functionality in 115
 - new models in 116
- O** operators, functions, and definitions,
 - new and updated 19
- Optimization Module
 - new functionality in 118
- P** Particle Tracing Module
 - backward compatibility 120
 - new applications in 120
 - new functionality in 120
- Pipe Flow Module
 - backward compatibility 124
- Plasma Module
 - backward compatibility 126
- R** Ray Optics Module
 - backward compatibility 131
 - new applications in 131
 - new functionality in 129
 - results and visualization, new functionality in 22
- RF Module
 - new and updated apps and models in 138
 - new functionality in 136
- Rotordynamics Module
 - new functionality in 141
 - new models in 142
- S** Semiconductor Module
 - new applications in 147
 - new functionality in 144
- Structural Mechanics Module
 - backward compatibility 155
 - new functionality in 149
 - new models in 154
 - studies and solvers, new functionality in 19
- Subsurface Flow Module
 - backward compatibility 158
 - new functionality in 157
 - new models in 158
- W** Wave Optics Module
 - new functionality in 159
 - new models in 162