

COMSOL Multiphysics Release Notes

© 1998–2020 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; 9,098,106; 9,146,652; 9,323,503; 9,372,673; 9,454,625; 10,019,544; 10,650,177; and 10,776,541. 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 Compiler, 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 5.6

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

Content

Chapter I: Release Notes

| | |
|---|-----------|
| New Products | 8 |
| Battery Design Module | 8 |
| Fuel Cell & Electrolyzer Module | 8 |
| Liquid & Gas Properties Module | 8 |
| Polymer Flow Module | 8 |
| LiveLink [®] for Simulink [™] | 9 |
| | |
| COMSOL Multiphysics | 10 |
| General New Functionality | 10 |
| New Functionality in the Application Builder | 10 |
| New Functionality in COMSOL Compiler [™] | 13 |
| New Functionality in COMSOL Server [™] | 14 |
| New and Improved General Functionality in COMSOL Multiphysics | 14 |
| New Geometry and Mesh Functionality | 17 |
| New Functionality in the Physics Interfaces | 19 |
| New and Updated Operators, Functions, and Definitions | 20 |
| New Functionality in Studies and Solvers | 22 |
| New and Improved Results and Visualization Functionality | 25 |
| New Java [®] API Methods | 28 |
| General Backward Compatibility Considerations | 31 |
| Backward Compatibility with Version 5.5. | 31 |
| Backward Compatibility with Version 5.4. | 31 |
| Backward Compatibility with Version 5.3a | 32 |
| Backward Compatibility with Version 5.3. | 33 |
| | |
| AC/DC Module | 36 |
| New Functionality in Version 5.6. | 36 |
| New and Updated Models in Version 5.6. | 37 |
| Backward Compatibility with Version 5.4 and Earlier | 38 |
| | |
| Acoustics Module | 39 |
| New Functionality in Version 5.6. | 39 |
| New and Updated Model Examples in 5.6 | 45 |
| Backward Compatibility with Version 5.5. | 45 |
| Backward Compatibility with Version 5.4. | 46 |
| Backward Compatibility with Version 5.3a | 46 |
| Backward Compatibility with Version 5.3. | 46 |
| | |
| Battery Design Module | 47 |
| New Functionality in Version 5.6. | 47 |
| New and Updated Applications and Models in Version 5.6 | 51 |
| Backward Compatibility with Version 5.4. | 51 |
| | |
| CAD Import Module, Design Module, and LiveLink[™] Products for CAD | 52 |
| New Functionality in Version 5.6. | 52 |
| New Functionality in the Design Module in Version 5.6 | 52 |
| New Functionality in LiveLink [™] for AutoCAD [®] in Version 5.6 | 53 |
| New Functionality in LiveLink [™] for Inventor [®] in Version 5.6 | 54 |

| | |
|--|-----------|
| New Functionality in LiveLink™ for PTC® Creo® Parametric™ in Version 5.6 | 54 |
| New Functionality in LiveLink™ for Revit® in Version 5.6 | 54 |
| New Functionality in LiveLink™ for Solid Edge® in Version 5.6 | 54 |
| New Functionality in LiveLink™ for SOLIDWORKS® in Version 5.6 | 55 |
| Backward Compatibility with Previous Versions | 55 |
| CFD Module | 56 |
| New Functionality in Version 5.6 | 56 |
| New Models in Version 5.6 | 57 |
| Backward Compatibility with Version 5.5 and Earlier | 58 |
| Backward Compatibility with Version 5.4 and Earlier | 58 |
| Backward Compatibility with Version 5.3a | 59 |
| Chemical Reaction Engineering Module | 60 |
| New Functionality in Version 5.6 | 60 |
| New Applications in Version 5.6 | 64 |
| Updated Applications in Version 5.6 | 65 |
| Backward Compatibility with Version 5.4 and Earlier | 66 |
| Backward Compatibility with Version 5.3a and Earlier | 67 |
| Composite Materials Module | 68 |
| New Functionality in Version 5.6 | 68 |
| New Models in Version 5.6 | 71 |
| Corrosion Module | 73 |
| New Functionality in Version 5.6 | 73 |
| New and Updated Models in Version 5.6 | 76 |
| Backward Compatibility with 5.4 | 77 |
| ECAD Import Module | 78 |
| New Functionality in Version 5.6 | 78 |
| Backward Compatibility with Version 5.3a | 78 |
| Electrochemistry Module | 79 |
| New Functionality in Version 5.6 | 79 |
| Backward Compatibility with Version 5.4 | 82 |
| Electrodeposition Module | 83 |
| New Functionality in Version 5.6 | 83 |
| New and Updated Models in Version 5.6 | 86 |
| Backward Compatibility with Version 5.4 | 87 |
| Fatigue Module | 88 |
| New Functionality in Version 5.6 | 88 |
| Geomechanics Module | 89 |
| New Functionality in Version 5.6 | 89 |
| New Models in Version 5.6 | 90 |
| Heat Transfer Module | 91 |
| New and Improved Functionality in Version 5.6 | 91 |
| New Applications in Version 5.6 | 96 |
| Updated Applications in Version 5.6 | 97 |
| Backward Compatibility with Version 5.5 | 97 |

| | |
|---|------------|
| Backward Compatibility with Version 5.4 | 98 |
| Backward Compatibility with Version 5.3a | 99 |
| Backward Compatibility with Version 5.3 | 101 |
| MEMS Module | 103 |
| New Functionality in Version 5.6 | 103 |
| New Models in Version 5.6 | 106 |
| Metal Processing Module | 109 |
| New Functionality in Version 5.6 | 109 |
| New Models in Version 5.6 | 110 |
| Microfluidics Module | 111 |
| New Functionality in Version 5.6 | 111 |
| Backward Compatibility with Version 5.5 | 111 |
| Backward Compatibility with Version 5.3a | 112 |
| Mixer Module | 113 |
| New Functionality in Version 5.6 | 113 |
| Multibody Dynamics Module | 114 |
| New Functionality in Version 5.6 | 114 |
| New Models in Version 5.6 | 118 |
| Updated Models and Apps in Version 5.6 | 119 |
| Nonlinear Structural Materials Module | 120 |
| New Functionality in Version 5.6 | 120 |
| New Models in Version 5.6 | 122 |
| Optimization Module | 124 |
| New Functionality in Version 5.6 | 124 |
| New and Updated Models and Apps in Version 5.6 | 125 |
| Backward Compatibility with Version 5.4 | 126 |
| Particle Tracing Module | 127 |
| New Functionality in Version 5.6 | 127 |
| New and Updated Applications in Version 5.6 | 130 |
| Backward Compatibility with Version 5.5 | 130 |
| Backward Compatibility with Version 5.4 | 130 |
| Backward Compatibility with Version 5.3 and Earlier | 131 |
| Pipe Flow Module | 132 |
| New Functionality in Version 5.6 | 132 |
| Updated Model in Version 5.6 | 132 |
| Backward Compatibility with Version 5.3 | 132 |
| Plasma Module | 133 |
| New Functionality in Version 5.6 | 133 |
| Backward Compatibility with Version 5.5 and Earlier | 133 |
| Porous Media Flow Module | 134 |
| New Functionality in Version 5.6 | 134 |
| New Models and Apps in Version 5.6 | 137 |

| | |
|---|------------|
| Ray Optics Module | 138 |
| New Functionality in Version 5.6 | 138 |
| New Applications in Version 5.6 | 140 |
| Important Fixes in Version 5.6 | 141 |
| Backward Compatibility with Version 5.5 | 141 |
| Backward Compatibility with Version 5.4 | 142 |
| Backward Compatibility with Version 5.3 and Earlier | 142 |
| | |
| RF Module | 144 |
| New Functionality in Version 5.6 | 144 |
| New and Updated Models in Version 5.6 | 145 |
| | |
| Rotordynamics Module | 147 |
| New Functionality in Version 5.6 | 147 |
| New Models in Version 5.6 | 149 |
| | |
| Semiconductor Module | 151 |
| New Functionality in Version 5.6 | 151 |
| New Models in Version 5.6 | 152 |
| | |
| Structural Mechanics Module | 154 |
| New Functionality in Version 5.6 | 154 |
| New and Updated Models in Version 5.6 | 162 |
| | |
| Subsurface Flow Module | 165 |
| New Functionality in Version 5.6 | 165 |
| New Models in Version 5.6 | 166 |
| | |
| Wave Optics Module | 168 |
| New Functionality in Version 5.6 | 168 |
| New and Updated Models in Version 5.6 | 169 |
| | |
| Material Library | 172 |
| New and Updated Material Data in Version 5.6 | 172 |
| | |
| LiveLink™ for Excel | 173 |
| New Functionality in Version 5.6 | 173 |
| | |
| LiveLink™ for MATLAB | 174 |
| New Functionality in Version 5.6 | 174 |
| Backward Compatibility with Version 5.4 | 175 |
| | |
| The COMSOL API for Use with Java | 176 |
| COMSOL 5.6 API Changes | 176 |
| COMSOL 5.4 API Changes | 176 |
| COMSOL 5.3a API Changes | 177 |
| COMSOL 5.3 API Changes | 177 |
| COMSOL 5.2a API Changes | 178 |
| COMSOL 5.1 API Changes | 179 |
| COMSOL 5.0 API Changes | 181 |



Release Notes

COMSOL Multiphysics® version 5.6 includes the new LiveLink™ *for* Simulink®, Polymer Flow Module, Liquid & Gas Properties Module, Battery Design Module, and Fuel Cell & Electrolyzer Module; extended and improved versions of the Application Builder, COMSOL Compiler™, COMSOL Server™, and COMSOL Multiphysics®; as well as updates and improvements for all COMSOL Multiphysics® add-on products. These *Release Notes* provide information regarding new functionality in version 5.6 for all COMSOL® software products.

New Products

The previous *Batteries & Fuel Cells Module* has been replaced by the *Battery Design Module*. In addition, the 5.6 release adds a dedicated *Fuel Cell & Electrolyzer Module* and the following new products: the *Polymer Flow Module*, *Liquid & Gas Properties Module*, and *LiveLink™ for Simulink®*.

Battery Design Module

The *Battery Design Module* replaces the previous Batteries & Fuel Cells Module and contains functionality for design and analysis of batteries.

Fuel Cell & Electrolyzer Module

The *Fuel Cell & Electrolyzer Module* models the underlying electrochemical behavior in electrodes and electrolytes of fuel cells and electrolyzers. With this module, you can investigate performance for different operating conditions and designs and simulate characteristics such as the transport of ions, current conduction, two-phase gas-liquid fluid flow, heat transfer, and the nature and driving forces of electrochemical reactions at planar electrodes and in porous gas diffusion electrodes.

Liquid & Gas Properties Module

The *Liquid & Gas Properties Module* provides features for modeling of thermodynamic properties of liquids and gases. Use this module to compute the density, viscosity, thermal conductivity, heat capacity and other properties of mixtures as functions of pressure and temperature. These properties can be used for modeling and simulation in CFD, heat transfer, and acoustics.

Polymer Flow Module

The *Polymer Flow Module* includes functionality for modeling of fluids with viscoelastic, thixotropic, shear thickening, or shear thinning properties. You can also account for the properties of the fluids as a function of temperature and composition to model cure and polymerization. In combination with other modules in COMSOL Multiphysics, you can also model fully coupled and time-dependent fluid-structure interactions.

LiveLink™ for Simulink®

LiveLink™ for Simulink® is a new product that enables two-way communication between COMSOL Multiphysics and Simulink.

The most prominent feature is the ability to perform cosimulation. You can take any COMSOL Multiphysics model, define a few inputs and outputs, and then simulate the COMSOL Multiphysics model using the new COMSOL Cosimulation block in Simulink. When the simulation runs, Simulink is in charge of the overall simulation scheme, and Simulink and COMSOL Multiphysics exchange data at so-called communication points. COMSOL Multiphysics then perform its own time stepping in between these communication points.

In order to make it possible to perform control design and to speed up simulation, you can extract state-space models based on a COMSOL Multiphysics model. Using model reduction, it is possible to reduce the number of states from millions to just a few and still be able to capture the dynamics of the input/output system.

COMSOL Multiphysics

General New Functionality

PASSCODE INSTALLATIONS

It is now possible to use a passcode for installation of the COMSOL software when the installation is not a trial.

COMPARING TWO SELECTED NODES

When you select two nodes with settings in the Model Builder tree or Application Builder tree, right-click and choose **Compare Selected Nodes** to compare the contents of the two nodes.

MPI I/O MODE

The `-mpii` cluster command is available for Windows[®] and Linux[®] to set the MPI I/O mode.

New Functionality in the Application Builder

GENERAL IMPROVEMENTS

Generating Numerical Evaluation Code

When a **Derived Values** feature node or an **Evaluation Group** node is selected in the **Model Builder**, the following four options for generating numeric evaluation code are also available:

- **Evaluate real scalar** to evaluate a real scalar value.
- **Evaluate real matrix** to evaluate the values of a real matrix.
- **Evaluate imaginary scalar** to evaluate an imaginary scalar value.
- **Evaluate imaginary matrix** to evaluate the value of an imaginary matrix.

The corresponding functionality is available in the **Editor Tools** window.

Testing Apps in Microsoft Edge[®]

Under the **Test in Web Browser** menu button on the ribbon in the Application Builder, Microsoft Edge[®] has been added as a browser used to test apps.

Color Themes

The desktop color theme in apps is determined by the new **Themes** node, where you can specify color themes for the app and for image export. For example, you can make an app using the dark theme but still export images in the default theme.

IMPROVEMENTS TO THE FORM EDITOR

Layout Templates

The *New Form* wizard now includes a number of layout templates for forms with combinations of ribbons or toolbars, sections, tabs, and graphics as well as layout templates suited for displaying the app on tablets and smartphones. You can also define and add custom layout templates to the wizard.

Knob Form Object

A new Knob form object is available as a way for app users to adjust some value using a knob as an alternative to a slider.

Local Forms

You can now add local forms that can be used by a parent global form (that is, **Form** nodes that appear directly under the **Forms** node). Local forms have a limited scope that consists of the parent global form and its subnodes (for example, local declarations defined by the parent global form).

Editor Previews

The **Preview** window in the Application Builder no longer has a cross for closing it, and the **Preview** button has been removed from the ribbon. Instead, there is a setting on the **Application Builder** page in the **Preferences** dialog box, which you can use to control the preview's presence.

Card Stack Size Adjustments

A card stack can now adjust its size to the selected card using the **Adjust size to selected card** setting. When selected, the card stack will adjust its size to the currently selected card.

Additional Positions for the About Link

The About link in an app can now be placed in the lower-left corner.

Showing and Hiding Column Headers in Results Table Form Objects

There is now an option to show or hide the column headers of Results Table form objects.

Selected Icon Settings For Toggle Buttons and Toggle Items

Toggle Items, used for Toggle Button and Toolbar form objects, now have a Selected Icon setting. The Selected Icon is displayed when the toggle is in the selected state.

Unit Support for Data Display Form Objects

The Data Display form object can now evaluate expressions using an arbitrary unit. The **Unit** setting is located in the **Number Format** section of the **Data Display** node's Settings window.

Improvements to the Selection Input Form Objects

The Selection Input form object now includes support for *On data change* events. The event handling method is triggered when the selection that the selection input is connected to changes. If you use **Create local method** for the *On data change* event, you get a method with an integer array argument. The method is called with the new entities of the selection.

Other improvements to the Selection Input form object:

- It is now possible to click the **Activate Selection** button to deactivate the selection. When doing so, the associated Graphics form object (in apps) shows the geometry without the selection. In Settings forms, the Graphics form object behaves like it does for any other Settings window when deactivating a selection widget.
- The Selection Input form object now has an `active` property, which can be set from method code to change the active state of the selection.
- There is a new *On activated* event, which is triggered when a selection input is activated.

Text Alignment for Array Input Form Objects

The Array Input form object now includes a setting for specifying the text alignment.

IMPROVEMENTS AND CHANGES FOR THE APPLICATION LANGUAGE

See also the *Application Programming Guide* for a complete overview of the application language for programming methods in the Application Builder for use in applications, models, and add-ins.

Running the Zoom to Selection Command

A new utility method for running the Zoom to Selection command has been added.

Example syntax:

Zoom to selection in the main Graphics window in the Model Builder:

```
zoomToSelection();
```

Zoom to selection in the given graphics form object:

```
zoomToSelection("form1/graphics1"); or  
zoomToSelection(app.form("form1").formObject("graphics1"));
```

Commands for Writing Byte Order Marks

New variants of the `writeFile` and `writeCSVFile` commands have been added. The new variants write an optional byte order mark (BOM) to the files. If the append flag is true, the BOM flag is ignored. Example syntax:

Write a double matrix to a CSV file with byte order mark:

```
writeCSVFile("C:\\Temp\\file.csv", new double[][]{new  
double[] {1,2,3}}, false, true);
```

Write a string matrix to a CSV file with byte order mark:

```
writeCSVFile("C:\\Temp\\file.csv", new String[][]{new  
String[] {"1","2","3"}}, false, true);
```

Write a double matrix to file with byte order mark:

```
writeFile("C:\\Temp\\file.txt", new double[][]{new  
double[] {1,2,3}}, false, true);
```

Write a string to file with byte order mark:

```
writeFile("C:\\Temp\\file.txt", "text", false, true);
```

Write a string matrix to file with byte order mark:

```
writeFile("C:\\Temp\\file.txt", new String[][]{new  
String[] {"1","2","3"}}, false, true);
```

New Functionality in COMSOL Compiler™

COMSOL RUNTIME INFORMATION PAGE

When running a compiled application for the first time, the COMSOL Runtime will be installed on your computer in order to run it. To make it easier to get information about what COMSOL Runtime is, a new **COMSOL Runtime Information** button is now displayed in the bottom-left corner of the installer. The button directs you to <https://www.comsol.com/runtime-information>.

DARK THEME FOR APPS

COMSOL Server fully supports apps using the new dark theme, both in COMSOL Client and when using a web browser.

CUSTOM BACKGROUND COLORS

The form background now fills the whole viewport when running apps, allowing for use of custom background colors.

APPLICATION USAGE LOG

A log of all completed application sessions is now available under **Administration>Application Usage**. The information can be exported to a text file or spreadsheet.

COLOR THEMES

The following color themes are now available for the COMSOL Desktop: default, light, and dark. The default theme corresponds to the color theme in earlier versions of COMSOL Multiphysics. The color themes are also available for image export and in plot groups.

You can specify color theme preferences for the desktop, graphics, and image export on the **Color Themes** page in the **Preferences** dialog box.

User-defined theme files now include a section `% Palette`, where you can define colors to use in later sections. For example:

```
% Palette
MyColor=100 120 130

% Graphics colors
key=MyColor
```

For the API, the following methods are available:

- `model.colorTheme(<theme>)` sets the color theme to be used in graphics. `automatic` indicates that the theme specified in the preferences will be used. The other possible theme values are the names of the themes in the `data/color` folder in the COMSOL installation folder.

- `model.colorTheme()` returns the current theme.
- `model.imageColorTheme(<theme>)` sets the color theme for image export.
- `model.imageColorTheme()` returns the current image export color theme.

The color themes that in earlier versions were called Dark and Light are now called DarkSelection and LightSelection.

INTERACTIVE CLIPPING OF 3D GEOMETRIES

To make it easier to select edges, boundaries, and domains that are located inside a surrounding object, you can now use interactive clipping. Add **Clip Plane**, **Clip Box**, **Clip Cylinder**, and **Clip Sphere** clipping features to select what parts of a geometry are shown. The interactive clipping functionality works throughout the Model Builder and is available from a menu button in the Graphics toolbar. When you click the toolbar button, by default, a **Clip Plane** node is added to the **View** node with settings that you can adjust in a Settings window or interactively in the Graphics window. Multiple clip planes can be added and used concurrently.

ENVIRONMENT MAPS

For 3D graphics, you can now include environment mapping by choosing from two environment maps in a 3D **View** node's Settings window: **Indoor** and **Outdoor**. By default, reflections of the environment map appear in the model geometry. You can also select a **Skybox** check box to show the environment map as a background image using a so-called skybox.

GRAPHICS WINDOW CONTEXT MENU

The context menu in the Graphics window has been extended and now provides additional functionality. What appears in the context menu depends on the currently selected node in the Model Builder. There is a menu for the geometry (including a separate menu in the 2D sketch mode), definitions, materials, physics interfaces, multiphysics, and mesh nodes.

The context menu content is different depending on if there is a preselection in the Graphics window or not. When there is a preselection, new features can be added from the menu, taking the preselection as their selection.

The context menu is filtered, for physics for example, so that only the features that can use the preselection appear in the menu.

If the context menu does not contain what you want, there is a **Go To** submenu that you can use to move to another node in the Model Builder while keeping the

preselection if possible. You can then right-click again and select the feature that you want to add.

POWERPOINT® PRESENTATION GENERATOR

You can now generate presentations of COMSOL Multiphysics models as Microsoft® PowerPoint slide presentations. The presentations include information and plots from the model, and you can choose between a default presentation or a detailed presentation. You can also create user-defined presentations with custom content and define and use presentation templates. To add a presentation, right-click the **Reports** node and the choose a presentation type from the **Presentations** submenu.

MISCELLANEOUS NEW AND IMPROVED FUNCTIONALITY

- There is a new preference setting on the **Graphics and Plot Windows** page in the **Preferences** dialog box to control the default view when creating new views and when clicking the **Go to Default View** button in 3D in the Graphics toolbar. Also, some preference settings have moved from this page to the **Graphics Interaction** page. In addition, there are three new buttons on the Graphics toolbar buttons: **Go to YX View**, **Go to ZY View**, and **Go to ZX View**, so that you can quickly move to all six basic 3D views.
- You can add a custom logotype to the Graphics windows. In the **Preferences** dialog box, when the **Show log on canvas** check box is selected on the **Graphics and Plot Windows** page, you can browse and select a custom logo using an image file. If no filename is given, the default COMSOL logo is used.
- A new **State-Space Controller** add-in makes it possible to create and simulate state-space controllers.
- Improved visibility of the unit system setting:
 - The **Unit System** section has been moved up in the root node.
 - The **Same as global system** option now also has an identifier of what the global unit system is in **Component** nodes.
 - The **Unit System** list also shows which unit system is the same as the global unit system in **Component** nodes.
- Large icons can now be used in the regular Graphics toolbar using a new preference setting on the **Graphics Toolbars** page in the **Preferences** dialog box.

- Under **Global Definitions** and **Definitions**, you can now group nodes under another **Group** node so that you can organize those nodes under **Group** nodes at several levels.
- You can now search for applications in the Application Libraries window with a particular identifier in the COMSOL Application Gallery (www.comsol.com/models) using the search prefix **@appdb:** immediately followed by the identifier. This can be useful when communicating with a COMSOL support or sales representative.

IMPROVED RENDERING OF MATERIALS

You can now choose from a much wider selection of materials for a realistic rendering of the materials in a model: brick, car paint, leather, oil, PCB, rubber, soil, textile, and wood, to mention a few. You can also, for a custom material, add noise and brush lines as well as an additional color and control some surface properties.

New Geometry and Mesh Functionality

GEOMETRY FUNCTIONALITY

- The 2D drawing tools have been improved:
 - The drawing tools are now easily accessible in the context menu that appears when right-clicking in a 2D sketch. You can also switch between the drawing tools Polygon, Circular Arc, Interpolation Curve, Quadratic, and Cubic while drawing by using the context menu, which results in a Composite Curve object.
 - When drawing a sequence of polygons, circular arc, interpolation curves, and Bézier curves, you can finish it by clicking the first point again, which creates a closed curve; by right-clicking and selecting **Finish Polygon** (or similar) on the context menu, which does not add a new point; or by double-clicking, which adds a last point.
 - You can cancel the drawing tools by pressing the Esc key.
 - When the **Sketch** button is active in a 2D geometry, you can hold down the Alt key to temporarily toggle snapping to the geometry on and off while drawing or while dragging a vertex.
 - When the **Fillet** button is selected, you can exit the fillet sketching mode by clicking the **Fillet** button again; by right-clicking in the Graphics window and selecting **Fillet**; or by pressing the Esc key.
 - The colors have been adjusted or changed. The most noticeable changes are that auxiliary vertices and edges (for example, in a Circular Arc feature) are now light

gray, and when hovering over an edge or vertex, the red highlighting is weaker than in previous versions.

- When you use the CAD kernel in 3D and the license includes the Design Module, you can now use the Boolean operations from the Design Module kernel (Parasolid kernel).
- In the Work Plane geometry feature, for **Plane type** set to **Edge parallel**, the directions of the xw and yw axes of the local coordinate system were wrong (reversed) in version 5.5 when the edge was created by a Boolean operation as an intersection of two faces using the COMSOL geometry kernel. This issue has been fixed in version 5.6.
- For identity pairs, you can now provide a manual search distance to couple more distant boundaries than the automatic search distance allows. In the new **Advanced** section, select **Manual** from the **Search distance** list.

MESHING FUNCTIONALITY

- Boundary layer meshing is now available on boundaries for creating boundary layer meshes on a selection of sufficiently planar faces (that is, faces that can be projected in a plane without inverting anywhere). In the Settings window for 3D **Boundary Layers** nodes, you can now select **Boundary** as the **Geometry entity level** to select boundaries where you want to create a boundary layer mesh.
- The **Create Edges** feature is now available for creating edges in components without geometry and for imported meshes. You can use this functionality to convert existing mesh edges from an imported mesh into mesh elements or to create new edge elements in empty space.
- For the **Create Edges** and **Create Vertices** features, you can now select vertices and edges by coordinates in the Graphics window. In their Settings window, a new Vertex Specification section is now available, where you can select **Mesh edges** and **Mesh vertices**, respectively. A table then appears in a **Mesh Edge Selection** and **Mesh Vertex Selection** section, respectively, where selected edges and vertices are added. You can also edit the coordinate values in the table manually.
- The new **Intersect with Line** feature in 2D and **Intersect with Plane** feature in 3D make it possible to partition selected edges or domains of a 2D mesh and selected edges or faces of a 3D surface mesh with a specified line in 2D or plane in 3D, respectively. They are available for meshing sequences of import type.
- The handling of NASTRAN mesh files has been improved so that CBEAM entries are now read and edge selections are created for each PID (Property ID) that the CBEAM entries refer to. RBE2 entries are also read, and for each RBE2 entry, a

vertex selection is created with the fixed grid point of the RBE2. In addition, depending on the configuration of the dependent grid points of the RBE2, a boundary selection, an edge selection, or vertex selections are added.

- A new mesh variable is now available, called `meshElemental1`, that for each mesh element gives its number in the entire mesh.
- A new mesh quality measure, *curved skewness*, is now available for mesh statistics and plotting and as a mesh variable, `qualCurvedSkewness`. It can be useful for examining the quality of higher-order meshes.
- The transition smoothing for boundary layer meshing and mesh control has been improved, providing up to 50% reduction in memory requirements and up to 4 times speedup.
- The **Avoid inverted curve elements** check box is now selected by default in the **Free Tetrahedral** node's Settings window. This may change any tetrahedral mesh of a geometry with curved faces, but is most likely to affect low-resolution meshes. Also, the algorithm that avoids the inverted element has been improved.
- Some operations for imported meshes have been renamed in version 5.6:
 - **Ball** is now **Partition with Ball**.
 - **Box** is now **Partition with Box**.
 - **Cylinder** is now **Partition with Cylinder**.
 - **Logical Expression** is now **Partition by Expression**.
 - Also, the menu and ribbon toolbar have been reorganized.

New Functionality in the Physics Interfaces

IMPROVED PRESSURE CONDITION FOR INCOMPRESSIBLE FLUID FLOW

The total (or stagnation) pressure can now be specified on inlet and outlet boundaries for incompressible flow. Both local and averaged values of the total pressure can be imposed. This new feature functionality can, for example, be used in pump simulations.

WEAK INEQUALITY CONSTRAINT

You can now add a **Weak Inequality Constraint** node on a domain, boundary, edge, or point in any physics interface. First you must select **Equation-Based Contributions** in the **Show More Options** dialog box. This constraint feature makes it possible to specify a complementary nonlinear inequality constraint.

IMPROVEMENTS FOR MOVING MESHES AND DEFORMED GEOMETRIES

You can now press Ctrl+Space to add expressions to moving mesh and deformed geometry features.

New and Updated Operators, Functions, and Definitions

IMPROVED EDITING OF USER-DEFINED MATERIAL LIBRARIES

There is a new **Edit** button and context menu option in the **Material Browser** window. You can access the **Edit** button when selecting a user-defined material library. It opens the selected material library so that it can be edited.

OTHER IMPROVEMENTS FOR USER-DEFINED MATERIAL LIBRARIES

The grouping of materials in a material library is changed so that you now define it using node groups in the **Model Builder**. All the built-in material libraries are updated.

Material Library Settings is a new check box in the **Show More Options** dialog box, under **General**. If you select it, it is possible to set a group type of a global **Material** node group. The group type can be **Library folder** to just group materials together, **Phase** to group different phases of the same material, or **Phase and orientation** to group different orientations of the same phase of a material. There is also a button to update the labels of all phases and orientations of the same material.

POROUS MATERIAL

For materials under a model component, you can now add **Porous Material** nodes for defining porous materials by adding **Fluid**, **Solid**, and **Immobile Fluid** subnodes that together form the overall porosity of the materials.

PERFECT VACUUM MATERIAL

A Perfect Vacuum material is now available in the built-in material library for representing a vacuum with a density of zero, relative permittivity of 1, relative permeability of 1, and electrical conductivity of 0.

MERGING MATERIALS

Right-click a material node and choose another material node from the **Merge With** submenu to merge the selected material with the other material.

RECENT MATERIALS

You can now add recently used material from a **Recent Material** menu on the **Materials** ribbon toolbar.

IMPROVEMENTS FOR LAYERED MATERIALS

See [Ply-Based Modeling Approach](#).

NEW AND IMPROVED OPERATORS

The following new operators are available:

- `elemint` — Use `elementint(gporder, expr)` to integrate an expression over each mesh element using the specified Gauss point pattern.
- `elemavg` — Use `elementavg(gporder, expr)` to evaluate the average of an expression using the specified Gauss point pattern.
- `elemgpmn` — Use `elementpmin(gporder, expr)` to evaluate the elementwise minimum of an expression using the specified Gauss point pattern.
- `elemgpmx` — Use `elementpmax(gporder, expr)` to evaluate the elementwise maximum of an expression using the specified Gauss point pattern.
- `laginterp` — A Lagrange interpolation operator is available as `laginterp(order, expr)`. This operator dynamically maps any expression to a Lagrange field of a specified order and then evaluates on that field in each mesh element. The operator also provides spatial derivatives and a complete Jacobian.
- `round` — Use `round(x, p)` to round x to the closest number with precision p (number of decimal digits).

OTHER IMPROVEMENTS

- The support for copying and pasting nodes in the Model Builder between sessions has been extended to include parameters, coordinate systems, selections, nonlocal couplings, probes, mass properties, variables, views, and pairs.
- Curl and divergence elements of the second kind are now available for simplex mesh elements.
- The smoothed functions now accept any unit in their input arguments with the limitation that both arguments must have the same unit. The output is always without unit, except for the special derivative functions of `f1dsms` and `f1dsmsign`, which will get a unit of one over the input argument's unit.
- Import of comma-separated decimals for interpolation functions is now supported.
- You can now define probes for points, in addition to domains, boundaries, and edges.
- For probes, you can now add a new output table for the probe data.

STUDIES AND STUDY STEP FUNCTIONALITY

- You can now right-click a study step and select **Reset Solver to Default for Selected Step** to reset the solver to the default for an individual study step.
- The **Modal Solver** can no longer create a reduced-order model as output. It only computes a solution using a modal method. Modal reduced-order models are now created using the new **Modal Reduction** solver. This solver does not output any solution, only a reduced-order model. In order to run existing Java[®] or MATLAB[®] code creating a reduced-order model, you must therefore change the solver type. Similarly, the AWE Solver now outputs either a solution or a reduced-order model. Existing Java or MATLAB code must therefore be updated, setting the output property to rom, if reduced-model output is desired.
As a consequence, the **Reduced model simulation** option has been removed from the **Modal Reduction** study step. To simulate a reduced model, use a separate study with an ordinary **Stationary** or **Time Dependent** study step.
- For the **Optimization** and **Parametric Sweep** study steps, some of the more advanced settings are now found in new **Advanced** sections in their Settings windows.

SOLVER FUNCTIONALITY

- A new **Changes from Default Settings** section in the solver sequence nodes lists all changes made in the solver sequence with the current value and the default value, so that any changes made to a solver sequence can be easily available.
- The FEAST eigenvalue solver is now available for **Eigenvalue** and **Eigenfrequency** studies and for the **Eigenvalue Solver** as an alternative to ARPACK, which is based on Krylov subspace techniques (the Arnoldi algorithm). FEAST is based on an algorithm inspired by density-matrix representation and contour integration techniques in quantum mechanics. The FEAST algorithm solves generalized eigenvalue problems and can obtain left and right eigenvectors within an interval or a contour in the complex plane.
- The new **Block Navier-Stokes** preconditioner is an efficient preconditioner for the incompressible Navier–Stokes equations in the transient regime. The algorithm is based on block factorization of the Jacobian matrix problem coming from the Newton–Krylov approach to solving the time and space discretization of the incompressible Navier–Stokes equations. You can specify separate solvers from the velocity and pressure as subnodes to the **Velocity Solver** and **Pressure Solver** subnodes, which are added automatically.

- An adaptation of the GMRES iterative solver called GCRO-DR is now available. This method can be useful while solving sequences of linear systems arising from, for example, nonlinear problems. The idea is to retain the subspace determined while solving previous systems and use it to reduce the cost of solving the next system. This process can be referred to as *Krylov subspace recycling*. To activate it, select the **Use GCRO-DR** check box in the Settings window for the **Iterative** solver node.
- New options are available for outputting and interpolating solution data from the **Time-Dependent Solver**, **Plug Flow Solver**, **Time-Discrete Solver**, and **Time-Explicit Solver** nodes:
 - You can now control the stored times as **Output times by interpolation**, which is the default value and behaves as the previous **Specified values; Steps taken by solver**, which stores the steps taken by the solver, so the solution is not interpolated, except at the end time, and has a new option to **Store every N:th step**, which makes it possible to avoid large solution object; **Steps taken by solver closest to output times**, which is a new option where the solution is not interpolated, except at the end time.
 - For **Free**, **Manual**, and **Intermediate** time steps, a new option, **Interpolate solution at end time**, is available and enabled by default. When it is enabled, the solver can step past the last time in the **Output times** list, leading to the end time being interpolated. When it is disabled, the solver will not step past the last time in the **Output times list**, leading to the end time being solved. This option is not available for **Strict** time stepping, because then the end time is always solved. It is also not available for the **Time-Discrete Solver**.
- The time-dependent mesh adaptation feature is now available for the time-explicit Runge–Kutta and Adams–Bashforth methods. In particular, it can be useful for discontinuous Galerkin simulations.
- For transient problems, it is now possible to terminate based on the residual. This can be useful for transient contact problems, for example. Without this termination criterion, the solution-based termination must be set to very restrictive values, which makes the time stepping inefficient. In the **Dependent Variables** and **Field** nodes' settings, a new **Threshold for updating residual scale** setting is available for automatic residual scaling so that, for time-dependent problems, the residual scale is updated when a sufficiently large change in the residual is detected during time stepping.
- A new option for the domain-decomposition solvers — Domain Decomposition (Schwarz) and Domain Decomposition (Schur) — makes it possible to add

absorbing boundary conditions to the domain boundaries created by the solver. These boundary conditions can improve the convergence rate of the linear iterative solver when solving Helmholtz equations, for instance. Some physics, such as pressure acoustics, can automatically add weak contributions that describe the appropriate absorbing boundary conditions. There is also a manual option where you can input an expression of your own. The added contributions do not affect the computed solution.

- The automatic update of the **Dependent Variables** solver nodes at general variable updates has been removed. Such updates were previously triggered, for example, when a new physics interface was added to the model. The automatic update is now postponed until the solver is run or reset. For most solver uses, this change will not affect your modeling at all. For rare cases, where you need to manually edit the dependent variable nodes (for example, with manual scales), you might need to manually refresh the dependent variable list. For such cases, an **Update Variables** operation is available on the context menu and the toolbar for **Dependent Variables** solver nodes. Using the API, there is also a new `updateVariables()` method.
- In the **Automatic Remeshing** node's Settings window, you can now choose **Specified times** and **Specified time steps** to manually control when to remesh. These options to remesh at given time instants or to remesh after a given time step can be robust alternatives to the previous remeshing conditions when the temporal behavior of the moving mesh quality is known.
- You can now manually sweep over outer solution numbers using global parameters in a parametric sweep. To do so, use the solution number inputs to **Index** fields in the **Values of Dependent Variables** sections in study step node Settings windows. This was not possible in earlier versions because you had to choose a fixed parameter value. The index is a single index. This functionality is also available on the solver level (in the **Dependent Variables** node).
- There are new options for the times to store when running the time-dependent solver. You can now choose to store every N th step taken by the solver and the steps taken by the solver closest to the specified output times.
- You can now control the absolute tolerance of the time derivatives in the time-dependent solvers. To do so, set the global **Derivative tolerance method** to **Manual** with the proportionality factor **Time derivative factor**, which acts between the tolerance on the variables and their derivatives. **Automatic** (the default) is the setting used in previous versions. Similarly, you can now control the derivative tolerance individually for the dependent variables with a **Derivative tolerance method** and the **Time derivative factor**. These settings appear in the **Absolute Tolerance** section of the

Time-Dependent Solver node. The Generalized alpha time-dependent solver has also been improved regarding its termination handling for equations featuring second-order time derivatives. The algebraic error for first-order time derivatives is now taken into account for second-order equations.

- You can now solve semiexplicit DAEs of index 1 with explicit time stepping. From the **Algebraic equations** list in the **Time-Explicit Solver** node's settings, you can choose to solve the algebraic equations periodically or at every N th step taken by the solver.

CLUSTER AND BATCH FUNCTIONALITY

- You can now right-click **Solution** nodes and choose **Domain Decomposition>Convert to Domain Decomposition (Schwarz)** and **Domain Decomposition>Convert to Domain Decomposition (Schur)**. This way you can convert a solver into a similar Domain Decomposition solver with settings adapted for cluster computing. These options are mainly intended for running on larger clusters where the domain decomposition strategy can be faster than the usual solvers.
- You can now reset the history for batch runs with the `-resethistory` option for `comsol batch`.
- To control the scalability assembling mode, which can be useful even when running on a single node, you can now select the **Memory scalability optimization for assembling** check box on the **Multicore and Cluster Computing** page in the **Preferences** dialog box. You can then select **Off** (the default, for no scalability mode), **Matrix** for activating scalability mode only for matrix assembling, **Vector** for activating scalability mode only for vector assembling, or **All** for activating scalability mode for all cases. You can also control these options with a `-memoptassem` option, with the supported values `off`, `matrix`, `vector`, and `all`.
- The HYPRE library of linear solvers has been updated to version `hypre-2.17.0` and now supports both OpenMP and MPI.

New and Improved Results and Visualization Functionality

NEW AND IMPROVED DATASET AND EVALUATION FUNCTIONALITY

- New Embed 1D and Embed 2 datasets are now available for embedding 1D data into 2D and 2D data into 3D, respectively.
- A new Partition dataset is available for partitioning a domain with respect to a set of isolevels. The dataset then adds elements corresponding to the isosurfaces or contours. It is possible to create a mesh part from a Partition by choosing a partition dataset in the mesh import node.

- You can now repeat a player animation for a specified number of times by choosing **Number of iterations** from the **Repeat** list.
- For variables from 0D components, you can now access them during postprocessing without a full scope. For example, `re.T` now works, and you no longer have to use `comp1.re.T`.
- The new option `runCommand` in Results nodes can be used to run special functions in the Method Editor. One example is automatic generation of Intersection Point datasets in the image plane by the Spot Diagram plot.
- You can now add a **Touchstone** export node under **Export** to export Touchstone files, which are used to describe data for parameters in port networks from RF simulations.
- In results tables, it is now possible to right-click and delete rows as well as columns from the table.
- In **Data** and **Plot** nodes under **Export**, you can now specify a separator for the exported data: column, space, tab, comma, semicolon, colon, or a vertical bar (pipe).
- It is now possible to specify a dataset in a plot or evaluation feature but take the solution parameters from the parent plot or evaluation group if it refers to the same solution. You can choose **From parent** from the **Solution parameters** list, when applicable. The solution parameters will be taken from the parent.
- When using a **Transformation type** other than **None** in the settings for an **Evaluation Group** node, only the final result is returned. You can now select the new **Keep child nodes** check box to add the results of the child features when using a transformation. This way you are able to, for example, see which boundaries that have the largest flux and total conservation in the same table.
- When exporting data from an impulse response plot using a **Plot** node under **Export**, you can now save it in the WAVE audio file (*.wav) format.
- In Mesh datasets, the **Deduce the geometry shape function order from the mesh** setting has been replaced with an explicit **Geometry shape function** setting for choosing the shape function type and order.

NEW AND IMPROVED GRAPHICS AND PLOT FUNCTIONALITY

- You can now add material appearance to all 2D and 3D plots. By adding a Material Appearance plot attribute, you replace the graphics from the parent node with a material appearance, providing the possibility for a mixed-mode visualization, where some surfaces or volumes are colored according to an expression and others are

plotted using a given material appearance. You can take the material appearance from the defined material or define a custom material appearance.

- A Transparency plot attribute is now available for 3D plots to add transparency for individual surfaces and plots. This way you can create plots with partial transparency.
- An Image plot attribute is now available for embedding images in Surface and Volume plots in 2D and 3D.
- 1D and 2D Table Histogram plots are now available. They are similar to the previous Histogram plots but take their data from a table or evaluation group. Also, the 2D Histogram plots are now available without any license requirements.
- A new Table Annotation plot is available to add annotations from a table to a 1D, 2D, or 3D plot.
- For 1D plots, you can now add Line Segment plots to plot line segments. The lines can be used to indicate some level, for example.
- For 2D and 3D plots, you can now use Point plots for plotting points, torque arrows, and so on. For 2D, you can choose the following point types: Point, Torque, Rectangle, and Ellipse. For 3D, you can choose the following point types: Point, Block, and Ellipsoid.
- You can now add a Marker subnode to Line, Surface, and Volume plots in 2D and 3D to add maximum and minimum markers to such plots.
- It is now possible to add error bars to 1D global and point graph plots.
- Graph markers are now available for 1D plots. For markers and max/min plots, you can prepend the coordinates of their positions.
- There are two new color tables, AuroraAustralisDark and ThermalNarrow, where the colors do not end in full black and white. Compared to the AuroraAustralis and Thermal color tables, these new color tables provide a better contrast when plotted using both the dark and the light desktop modes.
- 1D Directivity plots now support adding labels for the levels in the plots.
- For Histogram plots, you can now add Filter subnodes to filter the histogram data.
- For 1D function preview plots, you can now plot the frequency spectrum using FFT.
- Function plots are now also available as 2D plots for functions of two variables.
- The Point Graph and Global 1D plots now support plotting discontinuous (discrete) functions. You can choose the function type to be continuous or discrete.
- For plot groups, plot nodes, and plot attributes, you can now choose to use their labels as titles in plots.

- It is now possible to right-click 1D plots and choose an action that copies the plot data to a table.
- You can now use vectors of coordinates in annotations.
- In Table Graph plots, you can now sort the data points with respect to x - or y -coordinates.
- For Mesh plots, you can now plot higher-order elements, and color and style settings are available. When the geometry order is greater than 1, you can control the resolution. You can also choose to display node points and control the size of those points. For nonlinear meshes, the normals are now smoothed across element boundaries.
- The Plot All and Plot All in Group commands now update all plots, not only the last plot groups that update each window.
- In tables (Table and Evaluation Group windows) it is now possible to delete rows.
- In Octave Band plots where averaging over bands is performed, you can now plot the data as a graph connecting the centerpoints of the bands using the new line style.
- A **From theme** option is now available for uniform colors in 2D and 3D line plots and 1D graph plots to take the color from the chosen color theme.
- You can now add a Graph Marker subnode to 1D Radiation Pattern plots.
- The order in which color legends are added has changed so that it now follows a left-to-right, top-to-bottom order. For example, when Position is Right, the order of the color legends is now left-to-right instead of right-to-left as in earlier versions.

New Java[®] API Methods

LOADING AND SAVING FILES FOR PARAMETRIC SWEEPS

The `loadFile(String filePath)` and `saveFile(String filePath)` Boolean methods are now available to load and save files for the parametric sweep feature. These functions work if the type of feature is Parametric Sweep and save information concerning either all combinations or specified combinations; otherwise, they return false. The functions also return false if the operation is not successful; otherwise, they return true.

IMAGE OBJECT FOR MESH FEATURES

The `image()` object for mesh features includes a new property to set the selection.

```
model.mesh("foo").feature("bar").image().set("selection", "main")
```

specifies that the feature's main selection, `model.mesh("foo").feature("bar").selection()`, will appear in the plot. This is the default.

```
model.mesh("foo").feature("bar").image().set("selection",  
    <propname>)
```

specifies that a property selection, `model.mesh("foo").feature("bar").selection(<propname>)`, will appear in the plot.

For mesh features that do not have any selection, the selection property is ignored when plotting.

ONLY PLOT WHEN REQUESTED

The following methods have been added to the API for plotting only when requested:

```
void setOnlyPlotWhenRequested(booleanvalue);
```

to set if plots selected in the Model Builder only should be updated when the user explicitly requests it.

```
boolean isOnlyPlotWhenRequested();
```

returns true if plots selected in the Model Builder only should be updated when the user explicitly requests it.

IMPROVED SELECTIONS API

The API for selections has been improved, including the following additions and enhancements:

- `selection.geom(<gtag>)` now sets the selection to be all entities in the geometry `<gtag>`.
- `selection.geom(<gtag>, dim)` now specifies that subsequent calls to all, set, add, and remove refer to entities of dimension `dim` on the geometry `<gtag>`.
- `selection.clear()` removes all entities from the selection.
- `selection.dim()` returns the largest entity dimension on which the selection applies.
- `selection.entities()` returns the geometric entities of the selection on the given geometry at the largest entity dimension as an integer array.

From version 5.6, for a selection of entities, the method `objects()` returns feature tags for objects that are created by a feature that creates exactly one geometry object.

METHODS FOR RETRIEVING THE COLOR IN PLOTS

The following methods are now available:

```
double[] color = plot.getColor(<propertyname>, <useGraphicsTheme>)
```

```
double[] color = plot.getColor(<propertyname>, <colortheme>)
```

These methods are available for plot groups and plots. The return value is an array with the RGB values of the color. The property name has to be a property that defines a color. If the string value of the property is `fromtheme`, the color will be evaluated using the color theme. If the string value is an explicit color, that color will be returned.

In the first method, `useGraphicsTheme = true` means that the model's graphics color theme will be used, and `useGraphicsTheme = false` means that the model's image export color theme will be used.

The second method lets you specify the color theme to use, ignoring the model's theme.

EXTERNAL INTERFACE AND COSIMULATION

There is a new list for the model object, `model.externalInterface()`. This list has the following methods:

```
model.externalInterface().create(<tag>, <type>)  
model.externalInterface(<tag>).set(<property>, <value>)  
model.externalInterface(<tag>).run()
```

The `run()` method exports a file from the feature.

`SimulinkCosimulation` is the only valid type in version 5.6. To use it requires a license for `LiveLink™ for Simulink®`.

SAVING A COPY OF THE MODEL

The syntax `model.save(<path>, <saveCopy>)` saves a copy of the model, if the Boolean argument `<saveCopy>` is true. If it is set to false, this call results in a normal save operation. When saving as a copy, the location of that copy is not remembered, so the previous location for saving models is retained.

GETTING ROW HEADERS AND ADDING ROWS IN TABLES

For table features (`TableFeature`), you can now retrieve row headers using the `getRowHeaders()` access method.

The syntax `TableFeature.addRow()` is now available for adding multiple rows to a table using programming

General Backward Compatibility Considerations

COMSOL Multiphysics version 5.6 can open MPH-files saved from COMSOL Multiphysics versions 4.0–5.5.

COMSOL Multiphysics version 5.6 can run Java® files saved from COMSOL Multiphysics versions 4.0–5.5. 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 5.6.

Backward Compatibility with Version 5.5

The selections imported from Filter datasets in version 5.5 are now given tags by a corrected scheme that makes them independent of the language setting in COMSOL. If you have written API code using such selections, the code might need to be updated to handle the new naming principle.

In version 5.6, the `getAdjSparse` method has been deprecated and may be removed in a future version.

In the Octave Plot, the Solid setting for Type is now called Bar, and the Curve setting is now called Outline. The corresponding API property values are now `bar` and `outline`.

The default value for the compact aggregation setting for the smooth aggregation AMG multigrid solver, from version 5.6, is set to *off*.

Backward Compatibility with Version 5.4

GEOMETRY REPRESENTATION ARRAYS

To reflect the new serialization format for 3D objects represented using the COMSOL kernel, the meaning of the arrays returned from the `GeomInfo` methods `getFace`, `getPVertex`, and `getPEdge` applied to a 3D geometry object has changed slightly in version 5.5. In version 5.4, row 3 of `getFace` contained the index of the surface in the list of all manifolds (curves and surfaces) in the geometry object. In version 5.5, it is instead an index in the list of all surfaces in the geometry object. Similarly, row 5 of `getPVertex` and row 9 of `getPEdge` now contain an index in the list of surfaces instead of an index in the list of manifolds.

INTERPOLATION CURVES

For the **Interpolation Curve** geometry feature, the default end conditions have been changed from **Zero curvature** to **None**. In the API, the default has been changed from `zerocurv` to `none`. In addition, the algorithm has been changed for the case when **Type** is **Open curve** and **Relative tolerance** is 0, which can give a slightly different shape of the curve.

SCALING 2D GEOMETRY OBJECTS

In version 5.4, you could scale and move a set of selected 2D objects in the Graphics window by dragging its bounding box. In version 5.5, you can still move objects by dragging, but scaling requires adding a **Scale** feature in some cases.

FREE QUAD MESHING

Models saved in versions 5.3 to 5.4 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 version 5.5, have the tessellation method or face meshing method set to **Legacy version 5.4** or **Quadrilateral (legacy version 5.4)**, respectively.

PARTICLE TRAJECTORIES STUDY STEP

The Particle Trajectories study step has been removed in version 5.5. Instead, use a Time Dependent study step, which has exactly the same settings.

IMAGE EXPORT PROPERTIES

For image export features, the properties `options`, `title`, `legend`, `logo`, and `axes` have been split into three properties in version 5.5, one for each spatial dimension. For example, `options1d`, `options2d`, and `options3d`. `axes` does not exist in 3D. The same change has been done for all `image()` features.

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.

Backward Compatibility with Version 5.3

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.

AC/DC Module

New Functionality in Version 5.6

NEW MAGNETIC FIELDS, CURRENTS ONLY INTERFACE

The new Magnetic Fields, Currents Only interface is designed to efficiently compute the lumped inductance matrix of complex circuits in 3D.

UPDATED AND IMPROVED STUDY STEPS

- The Time to Frequency Losses study step now uses empirical models (Steinmetz, Bertotti, or user-defined models) to calculate the iron loss caused by magnetic hysteresis, eddy currents, and other effects in laminated cores or yokes.
- The Eigenfrequency study is now supported for most of the AC/DC interfaces. In addition to supporting full-wave cavity mode analysis in the magnetic fields interface, it is possible to do eigenfrequency analysis of models involving electrical circuits.

UPDATED ELECTRICAL CIRCUIT FUNCTIONALITY

- For time-dependent studies, the Electrical Circuit interface has been equipped with an event-based Switch. This allows you to model the more or less instantaneous on-off switching of certain connections in the circuit.
- The Electrical Circuit interface has been equipped with Parameterized Subcircuit Definitions. Together with the Subcircuit Instance, these definitions enable you to create your own “building blocks” containing smaller circuits and use multiple parameterized variants of those in your larger circuit.
- For the Electrical Circuit interface, the state, event, and solver machinery has been improved; especially the transient modeling of nonlinear (semiconductor) devices, which has become more stable.

NEW MAGNETIC MATERIALS

The AC/DC Material Library has been extended with magnetic materials from Bomatec®. The materials include all the NdFeB standard grades; NdFeB (bonded, injection molded, and extruded); Ferrite (isotropic, anisotropic, and injection molded); SmCo (injection molded); SmFeN (injection molded); SmCo5 and Sm2Co17 grades; and AlNiCo (cast and sintered).

OTHER NEW AND UPDATED FUNCTIONALITY

- New default plots have been developed specifically for physics interfaces that support magnetic fields: The Magnetic Fields interface, the Magnetic Fields, No Currents (Boundary Elements) interface, the Rotating Machinery, Magnetic multiphysics interface, the Magnetic and Electric Fields interface, the Magnetic Field Formulation interface, and the new Magnetic Fields, Currents Only interface.
- For the Charge Conservation feature in the Electrostatics interface, the list of dielectric material models has been extended with the option Ferroelectric, for modeling electrostatics in a ferroelectric material.
- The Coil domain feature has been equipped with support for a slanted cut. Selecting the **Slanted cut** check box is needed for cases where the natural direction of the current (or the wires) is at an angle with respect to the input or output normal. A typical example is a helical coil, where the input and output boundaries represent a periodicity plane.
- The Magnetic Field Formulation interface now supports new discretization shape functions referred to as the Nédélec curl element of the second kind (curl type 2). The curl type 2 element provides full polynomial orders in all directions for each field component. The curl type 2 element is available for simplex mesh elements only.

New and Updated Models in Version 5.6

Updates to the AC/DC Module Application Library:

- Several tutorial models have been updated, most notably the Cable Tutorial Series. The Cable Tutorial Series has been updated to include the short-twisted periodic configuration. This configuration makes it possible to solve a twisted, three-phase armored cable model in a matter of seconds rather than hours.

In addition, the following four new models are now available:

- Inductance Matrix Calculation of PCB Coils. This model shows how to compute the inductance matrix of a series of conductors with the Magnetic Fields, Currents Only interface.
- Hysteresis in Ferroelectric Material, which demonstrates how to analyze the nonlinear polarization behavior of ferroelectric materials.

- High-Voltage Insulator. This type of insulator is widely used in high-voltage power transmission lines to support the weight of suspended conductors without allowing the current to flow through the tower to ground.
- Position Optimization of Grading Rings. This model extends the High-Voltage Insulator model and shows how to use the Optimization interface to find the optimum position of the grading rings.

Backward Compatibility with Version 5.4 and Earlier

REMANENT FLUX DENSITY MATERIAL PROPERTY GROUP

With the introduction of the remanent flux density material property group, support for the remanent flux density constitutive relation with the relative permeability given by an effective medium has been dropped. In other words, a linear (hard) magnetic material where \mathbf{B}_r is set and μ_r given by an **Effective medium** is no longer supported. User-defined expressions for μ_r are still possible and may serve as a substitute.

REFACTORING OF MODEL INPUTS

For all AC/DC Module interfaces, some model input functionality has been moved to the **Default Model Inputs** node.

For a temperature-dependent boundary material used in an Impedance boundary condition, for example, this means the default model inputs should be used, rather than the feature's model input section (the model input section in the Impedance boundary condition's Settings window).

For more frequently occurring combinations, like a temperature-dependent domain material and a current conservation law, both the default model inputs as well as the feature's model input section are available as a means of specifying the temperature.

Acoustics Module

New Functionality in Version 5.6

PORT BOUNDARY CONDITION FOR ELASTIC WAVE PROPAGATION

The new **Port** boundary condition, available with the Solid Mechanics interface, is designed to excite and absorb elastic waves that enter or leave solid waveguide structures. A given **Port** condition supports one specific propagating mode. Combining several port conditions on the same boundary allows a consistent treatment of a mixture of propagating waves, for example, longitudinal, torsional, and transverse modes. The combined setup with several Port conditions provides a superior nonreflecting condition for waveguides to, for example, a perfectly matched layer (PML) configuration or the Low-Reflecting Boundary. The port condition supports S-parameter (scattering parameter) calculation, but it can also be used as a source to just excite a system. The power of reflected and transmitted waves is available in postprocessing. To compute and identify the propagating modes, the Boundary Mode Analysis study is available in combination with the port conditions.

NONLINEAR ACOUSTICS, TIME EXPLICIT

The new **Nonlinear Acoustics, Time Explicit** interface is used to model high-amplitude nonlinear pressure waves in fluids. The interface is based on the discontinuous Galerkin (dG or dG-FEM) method and uses a time-explicit solver. The method is very memory efficient and can solve problems with many million degrees of freedom (DOFs). The method is also well suited for distributed computing on clusters. The interface exists in 2D, 2D axisymmetric, and 3D geometries. The formulation used is suited for modeling progressive wave phenomena when the cumulative nonlinear effects surpass the local nonlinear effects, and the model is consistent with the second-order Westervelt equation. The interface is multiphysics enabled and can be coupled with the **Elastic Waves, Time Explicit** interface to model vibroacoustic problems. Absorbing layers are used to set up effective nonreflecting boundary conditions. Consistent handling of material discontinuities is enabled when modeling transitions between materials like, for example, different tissues and fluids. Pair features allow the use of nonconforming or discontinuous meshes at boundaries between materials. For modeling shock formation and propagation, special weighted essentially nonoscillatory (WENO) limiters can be used in combination with linear elements. Transient adaptive mesh refinement can be used to speed up the solution of nonlinear acoustic problems with propagating pulse signals.

NONLINEAR THERMOVISCOUS ACOUSTICS

The **Nonlinear Thermoviscous Acoustics Contributions** feature adds the necessary contributions to the **Thermoviscous Acoustics, Transient** interface in order to model nonlinear effects in a transient thermoviscous simulation. The contributions allow modeling of vortex shedding that may happen at sudden expansions, like in a perforate, a grill, or at a miniature sound port. With the second-order density representation option, the feature can also capture the nonlinear effect associated with high sound pressure levels that require a nonlinear representation of the equation of state. Using this option, cumulative nonlinear effects are also captured. Stabilization has been added to the thermoviscous acoustics interfaces, which is essential when using the **Nonlinear Thermoviscous Acoustics Contributions** feature for solving highly nonlinear problems.

LUMPED PORT BOUNDARY CONDITION IN PRESSURE ACOUSTICS

The new **Lumped Port** boundary condition in **Pressure Acoustics, Frequency Domain** connects a waveguide or duct inlet or outlet to a lumped representation element. This can be an **Electrical Circuit** (with the AC/DC Module), a two-port network defined through a transfer matrix, or a lumped representation of a waveguide. Basically, it couples the end of a waveguide to an exterior system that has a given lumped representation. Several representations and sources exist to describe the lumped system as well as to excite the system. When using the lumped port representation, it is assumed that only plane waves propagate in the acoustic waveguide.

NEWS IN PRESSURE ACOUSTICS, BOUNDARY ELEMENTS (BEM) INTERFACES

For acoustically large models, that is, problems that contain many wavelengths (high frequency or large domains), a new **Stabilized Formulation** option can be enabled to ensure efficient convergence at the cost of some additional degrees of freedom. For small to medium problems, the default formulation should be used. When the new stabilized formulation is enabled, a new dedicated solver suggestion is also used.

In the **Pressure Acoustics, Boundary Elements** interface, a new **Exclude Boundary** feature has been added. The feature can be used to exclude boundaries from the BEM model. The feature is particularly useful when using the interface in half space or infinite baffle setups.

MOIST AIR MATERIAL GENERATED FROM THERMODYNAMICS

Defining moist air is now possible using the **Predefined System** for **Moist air** in the **Thermodynamics** feature. This functionality is available with the new **Liquid & Gas Properties Module**.

THERMOVISCOUS BOUNDARY LAYER IMPEDANCE BOUNDARY CONDITION

The new **Thermoviscous Boundary Layer Impedance** boundary condition, in **Pressure Acoustics, Frequency Domain**, adds the losses due to thermal and viscous dissipation in the acoustic boundary layers at a wall. The losses are included in a locally homogenized manner, where they are integrated through the boundary layers analytically. The condition is applicable in cases where boundary layers are not overlapping. That is, it is not applicable in a very narrow waveguide (with dimensions comparable to the boundary layer thickness) or on highly curved boundaries.

GENERAL NEWS IN THE TIME-EXPLICIT INTERFACES

The time-explicit interfaces that use the discontinuous Galerkin formulation (dG-FEM) come with new functionality and performance improvements:

- General speedup of up to 30% when solving models with the Pressure Acoustics; Convected Wave Equation; and Elastic Waves, Time Explicit interfaces in 3D.
- Reformulation of the Pressure Acoustics, Time Explicit interface for 2D and 2D axisymmetry reduces the number of degrees of freedom (DOFs) solved for with up to 25%.
- The time-explicit dG-FEM method has been extended to work on all mesh types. This, for example, allows the use of a structured mesh in thin elastic structures. In general, using a structured mesh gives a memory reduction and a speedup.
- The Elastic Waves, Time Explicit interface is now also present in a 2D axisymmetric formulation.
- The time-explicit interfaces can now be solved in combination with systems of ODEs (ordinary differential equations). This can, for example, be used to formulate user-defined frequency-dependent impedance boundary conditions in acoustics.
- The time-explicit interfaces can now be solved in combination with an algebraic equation (an equation with no time derivatives).

NEW IN RAY ACOUSTICS

In the **Ray Acoustics** interface, there are several news related to the generation and analysis of the impulse response. The rendering and computation time of the impulse response has been greatly improved. The steps to set up the impulse response are now

also simpler and consistent. It is now also possible to analyze and compute the qualitative room acoustic metrics, like clarity, definition, and reverberation times, based on the computed impulse response. This is done using a new **Energy Decay** subfeature that can be added to an **Impulse Response** plot. It is also possible to export the computed impulse response signal to .wav format for further analysis.

Significant speedup of the computation and the analysis of the impulse response:

- A significant speedup of the initial computation and rendering of the impulse response is achieved in version 5.6 compared to older releases.
- Ray data in the **Impulse Response** plot is now stored in cache. This ensures a very fast rendering time when changing options in the impulse response plot.

New improved impulse response analysis functionality with the **Energy Decay** subfeature:

- More precise computation of the arrival time at the receiver when the source or the last reflection is close to the receiver. Direct sound gets a consistent arrival time computation and summation of power.
- Analysis of the impulse response with room acoustics metrics to compute: Reverberation times (T20, T30, and T60); clarity; definition; EDT; speech transmission index (STI); and the modulation transfer functions.
- Improved filter kernel functionality for the IR computation with user-defined definition of the filter kernel and visualization of the filters.

BACKGROUND FLUID FLOW COUPLING SUPPORTS HIGH MACH NUMBER FLOW

The **Background Fluid Flow Coupling** multiphysics feature now supports **High Mach Number Flow** interfaces as the source for the flow data.

NEWS FOR THE ACOUSTIC PORT CONDITIONS

Automatic Boundary Conditions for Numeric Ports in Thermoviscous Acoustics

When the **Numeric** option is used in the **Port** feature in a **Thermoviscous Acoustics, Frequency Domain** model, it now automatically detects the boundary conditions applied on the adjacent waveguide boundaries. The conditions are then automatically included when computing the mode shape of the propagating acoustic mode. The **Numeric** option supports the use of Wall, No Slip, Isothermal, Adiabatic, and Symmetry conditions.

Frequency Sweep and Numeric Ports

In pressure acoustics, a new option has been added to compute the mode shape and cutoff frequency for numeric modes in lossless systems. For a model without any losses, the new **Computed lossless mode cutoff frequency** option makes it possible to run a frequency sweep using a single **Boundary Mode Analysis** study for each port.

Power Scaling Option for Port Conditions

The port features in **Pressure Acoustics**, **Frequency Domain** and **Thermoviscous Acoustics**, **Frequency Domain** now have the option to use power scaling for the mode shapes. The default is to use amplitude scaling. The power option is equivalent to the default known from RF applications. With the power scaling option, the computed scattering parameters can be directly related to the power carried by a specific mode.

NEW THREE-PARAMETER APPROXIMATION JCAL POROACOUSTIC MODEL IN POROACOUSTICS

A new **Three-parameter approximation JCAL model** option has been added to the **Poroacoustic** feature in **Pressure Acoustics**, **Frequency Domain**. The model is an approximation to the existing Johnson-Champoux–Allard–Lafarge (JCAL) model but only requires three porous parameters to be specified. The three parameters are the **Porosity**, the **Median pore size**, and the **Standard deviation in the pore size distribution**.

IMPROVED STABILIZATION FOR LINEARIZED NAVIER-STOKES PHYSICS

The stabilization in the **Linearized Navier-Stokes** physics interfaces has been improved with a more consistent formulation. Specifically, the balance between the stabilization contributions of the continuity, momentum, and energy equations has been improved.

SOLVER SUGGESTIONS FOR DOMAIN DECOMPOSITION AND SHIFTED LAPLACE METHODS

For pure **Pressure Acoustics**, **Frequency Domain** models, two new iterative solver suggestions have been added; one that uses the **Shifted Laplace** approach (the technology was added in version 5.5) and one that uses **Domain Decomposition**. The iterative solver suggestions now automatically add the necessary equation contributions on domains and at boundaries (between domains) to ensure an efficient solver. The Shifted Laplace solver is efficient for solving large problems on a single machine with a lot of RAM while the Domain Decomposition method is suited for solving very large problems using distributed computation on a cluster.

NEW SETTINGS FOR SOLVING TRANSIENT ELASTIC WAVE PROBLEMS WITH SOLID MECHANICS

New settings have been introduced in the **Solid Mechanics** interface that ensure a correct and efficient solver setup when solving elastic wave problems in the time domain. The settings are similar to the existing settings in the transient acoustics interfaces. At the physics level, a new **Transient Solver Settings** section has been introduced with an option to specify the **Maximum frequency to resolve**.

LICENSE UPDATE FOR PIPE ACOUSTICS

The **Pipe Acoustics**, **Frequency Domain** and **Pipe Acoustics, Transient** interfaces are now available with either the Acoustics Module or the Pipe Flow Module.

IMPORTANT ENHANCEMENTS AND UPDATES IN THE ACOUSTICS MODULE

- Predefined postprocessing variables for the effective speed of sound magnitude and principle directions for **Anisotropic Acoustics**.
- Computation of all intensity variables in the scattered field formulation. Background, scattered, and total field variables are computed using a new option (check box) on the **Background Pressure Field** feature and on the **Incident Pressure Field** feature in **Pressure Acoustics**. This allows for easy postprocessing of incident and scattered acoustic power when modeling scattering problems.
- Postprocessing variables for total, scattered, and background fields have been made consistent across the acoustics interfaces such that variables are defined in all cases. For example, the total, scattered, and background sound pressure levels are now defined as: `phys.Lp_t`, `phys.Lp_s`, and `phys.Lp_b`, respectively.
- It is now possible to compute accurate boundary fluxes in the **Pressure Acoustics** interface. This is an advanced option in the Discretization section at the top physics level in the user interface.
- In the **Acoustic Diffusion Equation** interface, tables defining properties in frequency bands are now imported with “replace” and you can only import a table that has the expected size.
- Consolidation of the stress feature in the Linearized Navier-Stokes and Thermoviscous Acoustics interfaces.
- The predefined power variables of the **Port** boundary conditions have been reformulated such that they can directly be used in optimization models.
- Additional postprocessing variables have been added to the **Elastic Waves, Time Explicit** interface.

- The new function plot preview with FFT option is useful to obtain the frequency content of any signal without the need of setting up a new FFT study step. This is specially relevant for transient models with a known input signal. The frequency content of the signal can be analyzed beforehand and the meshing and solver options can be adjusted according to the frequency content.

New and Updated Model Examples in 5.6

- Propagation of Seismic Waves Through Earth (new)
- Submarine Target Strength (new)
- High-Intensity Focused Ultrasound (HIFU) Propagation Through a Tissue Phantom (new)
- Electric Motor Noise: Analysis of a Permanent Magnet Synchronous Motor (new)
- Nonlinear Slit Resonator (new)
- Topology Optimization and Verification of an Acoustic Mode in a 2D Room (new)
- Nonlinear Propagation of a Cylindrical Wave — Verification Model (new)
- Piezoelectric MEMS Speaker (new)
- Pressure Reciprocity Calibration Coupler with Detailed Moist Air Material Properties (new)
- Loudspeaker Driver - Frequency Domain Analysis (updated)
- Vented Loudspeaker Enclosure (updated)
- Piezoelectric Tonpilz Transducer with a Prestressed Bolt (updated)
- The Brüel & Kjør 4134 Condenser Microphone (updated)

Backward Compatibility with Version 5.5

The handling of tables with data entered in frequency bands in the **Acoustic Diffusion Equation** interface has been improved. Data can now be stored for all three input types. To retrieve the old behavior for user-defined bands, change the parameter names `N_band`, `alphaN`, `JiN`, `QN`, `alphafN`, `TLN`, and `qpn` to `N_band_ud`, `alphaN_ud`, `JiN_ud`, `QN_ud`, `alphafN_ud`, `TLN_ud`, and `qpn_ud` where needed.

For example:

```
model.component("comp1").physics("ade").feature("ps1").set("N_band_ud", new int[][]{{1},{2},{3}});
```

```
model.component("comp1").physics("ade").feature("ps1").set("qpN_u  
d", new int[][]{{1},{0},{0}})
```

Backward Compatibility with Version 5.4

For the Pressure Acoustics interfaces, the following features have been removed from the Fallback feature submenu to the Pair features: Matched Boundary, Exterior Field Calculation, Port, Plane Wave Radiation, Circular Wave Radiation, and Spherical Wave Radiation.

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 ρ , u , and p , remove the Java[®] lines setting $d\rho/dt$, du/dt , and dp/dt (these parameters change names when the field names change).

Battery Design Module

New Functionality in Version 5.6

GENERAL NEW FUNCTIONALITY

- The module name has changed from *Batteries & Fuel Cell Module* to *Battery Design Module*.
- The NMC 811 Electrode Material has been added to the Battery Material Library.

ELECTROCHEMISTRY INTERFACES

- A **Dissolving-Depositing Species** section has been added to the **Separator** node of the Lithium-Ion Battery; Battery with Binary Electrolyte; and Tertiary Current Distribution, Nernst–Planck interfaces.
- The stoichiometric coefficients for the dissolving-depositing species table are now reset to the default value of 0 when duplicated.
- For the **Electrode Reaction** and **Porous Electrode Reaction** nodes of the Tertiary Current Distribution, Nernst–Planck interface, a new option for the linearization of concentration dependence for low concentrations of the Butler–Volmer kinetics expression type is now available when using either mass action law or lumped multistep as the exchange current density type.
- A new **From material** option has been added for the **Local current density** setting in the Electrode Kinetics section in electrode reactions of all electrochemistry interfaces.
- A new property, **Physics vs. Materials Reference Electrode Potential**, has been added to all electrochemistry interfaces.
- The solver defaults have been updated for most Electrochemistry interfaces. Geometric and algebraic multigrid solvers (GMG and AMG) are now automatically generated by the study step nodes. A direct solver will, however, still always be used by default. Minor changes have also been made to default scales and tolerances.
- The Electrode Surface (eebii) boundary feature, applicable to both exterior and interior boundaries of electrolyte domains, is now obsolete. It is replaced with a new Electrode Surface (es) boundary feature, applicable to exterior boundaries only, and a new Perforated Electrode Surface (pes) boundary feature, applicable to interior boundaries of electrolyte domains only.

- The Thin Electrolyte Layer boundary feature is now available for the Electroanalysis charge conservation model in the Tertiary Current Distribution interface.
- A new Highly Conductive Porous Electrode domain node has been added to the Primary/Secondary/Tertiary Current Distribution, Lithium-Ion Battery, and Battery with Binary Electrolyte interfaces.
- The irreversible heat source term now contains a contribution from the film resistance resistive heating.

CHEMICAL SPECIES TRANSPORT INTERFACES

- It is now straightforward to apply diffusion coefficients defined by a Chemistry interface. The input for diffusion coefficients now includes a **Source** list with the options **Material** and **Chemistry**. Selecting the latter, it is then possible to pick up coefficients from any Chemistry interface present in the model.
- The total number of constraints applied on a boundary are now analyzed. When combining concentration constraints with equilibrium reactions on boundaries, a warning is shown when the system is overconstrained. The constraint analysis is performed in the following interfaces: Transport of Diluted Species and Transport of Concentrated Species.
- When solving mass transfer problems, iterative algebraic multigrid (AMG) solvers are now automatically generated in the solver sequences for large models. A direct solver is also added for convenience.

THERMODYNAMICS

- Using the Generate Material Wizard, the resulting material can now be added under global Definitions (as well as in a component of any space dimension).
- The time spent evaluating a function from a thermodynamic system has been significantly reduced.
- The model object for the Thermodynamics node, accessible from the API, has been renamed. It is now accessed in the following manner:

```
model.thermodynamics()
```

instead of

```
model.capeopen()
```

Java[®] files from previous versions will continue to run.

THE CHEMISTRY INTERFACE

- It is now possible to balance the reaction formula. To do so, click the **Balance** button in the upper-right corner of the **Reaction Formula** section.
- The reaction rate is now by default defined from the Mass action law. This was previously referred to as the Automatic definition.
- The overall reaction orders when the Mass action law is used are now displayed in text form in the **Reaction Rate** section. The reaction order for both surface and volumetric species are displayed.
- The overall forward reaction order and the overall reverse reaction order (when present) are reported. For surface reactions, the reaction order for both surface and volumetric species are reported.
- The molar mass is now automatically computed from the species chemical formula. The species name is parsed for elements in the periodic table, and the molar mass is computed using the corresponding elemental masses.
 - The molar mass is automatically defined when a **Species** node is created, as well as when a species is added to a **Species Group** node.
 - When the automatic calculation fails, for example, when descriptive name such as **water** is used, the molar mass is set to 0 g/mol.
 - When computing a model that contains species for which the molar mass remains at zero, a warning node is added in the solver sequence.
- The order of the sections in the Settings window has been changed to reflect their relative importance. The new order, from top to bottom, is:
 - Model Input
 - Mixture Properties
 - Species Matching
 - Calculate Transport Properties
 - Activity
 - Extra Dimension

Note that applications created in previous versions retain the original section order.

- The mixture type (diluted species or concentrated species) is now controlled using the **Type** list at the top of the **Mixture Properties** section. It was previously located in the **Species Matching** section.
- The section Equilibrium Species Vector has been removed from the Settings window of the interface.

- Thermodynamic properties are now always defined. The Calculate Thermodynamic Properties section, with the corresponding check box, has been removed.
 - The **Species Thermodynamic Expressions** section is always available in a **Species** node.
 - The **Reaction Thermodynamic Properties** section is always available in a **Reaction** or **Electrode Reaction** node.
- The **Calculate mixture properties** check box in the **Calculate Transport Properties** section is now enabled by default.
- The ratio of specific heats can now be specified in the Calculate Transport Properties section. The ratio is defined and announced and can be used in other physics interfaces.
- The setting to specify that the concentration, or activity, of species does not change has been renamed. It is now called Constant concentration/activity. It was previously called Locked concentration/activity.
- The nomenclature in the tables in the Species Matching table section has been made more distinct. Species can now be one of the following types: Constant, Variable, Solvent, or Free species.
- The tables in the **Species Matching** section no longer contain a **Reaction rate** column.
- The table at the top of the Species Matching table is now named Bulk species to communicate that these species are present in the mixture phase.

NEW FEATURES FOR POROUS MEDIA TRANSFER

Two new domain features, the Porous Medium and Unsaturated Porous Medium features, are available in the Transport of Diluted Species in Porous Media and Transport of Diluted Species interfaces. The new features include support for using the Porous Material node, also new in version 5.6, for assigning material properties to the multiple phases in a porous medium. The new features have subnodes to define phase properties. The subnodes follow the same structure as the subnodes available for a Porous Material.

The Dispersion and Volatilization settings in partially saturated porous media have been removed from the interface level of the Transport of Diluted Species in Porous Media and Transport of Diluted Species interfaces. This functionality is now available in the subnodes of the new features.

The new domain features replace the previously available Porous Media Transport Properties and Partially Saturated Porous Media features. When opening a model created in a previous version, the previous features are kept in the model.

New and Updated Applications and Models in Version 5.6

- A new *Lithium-Sulfur Battery* model has been added.
- A new *1D Isothermal Nickel-Cadmium Battery* model has been added.
- A new *Lithium Battery Pack Designer* application has been added.

Backward Compatibility with Version 5.4

- The Java API migration code handling the new lithium insertion and lithium metal kinetics in the Lithium-Ion Battery interface and the Single Particle Battery interface assumes that `ka=kc` and `c1_ref` have been manually set in Java; if this is not the case, backward compatibility may be affected.
- The default stoichiometric number for Dissolving-Depositing species has been changed to 0 (from 1). Java API backward compatibility may be affected.
- The Tertiary Current Distribution, Nernst-Planck interfaces have changed default settings to use the Nernst equation option for the equilibrium potential, Butler-Volmer for the kinetics expression type, and Mass action law for the exchange current density type. Java API backward compatibility may be affected.

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

New Functionality in Version 5.6

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 version 5.6 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.

- 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).
- The import of CAD assemblies containing several instances of the same part has been improved and is now faster compared to previous versions.
- You can use the new Delete Interferences tool to detect interference between objects. The tool detects touches, intersections, gaps, and containments.
- The new Check feature can be used to detect faults in CAD objects.
- The Repair feature now has two new check boxes. The **Check resulting objects** check box enables checking for remaining faults in the CAD objects. The **Repair face-to-face inconsistencies in solids** check box has been added to repair these type of faults in solid CAD objects.
- Form assembly has been improved in the CAD kernel case for better imprint and pair generation when overlaps and gaps occur between objects.

New Functionality in the Design Module in Version 5.6

- The Design Module adds support for more advanced Boolean operations that can be enabled in the settings of a 3D geometry by the new **Design Module Boolean operations** check box. The default of this check box is controlled by a preference setting.

- The new Horizontal and Vertical constraints make it easier to constrain straight edges to be parallel to the x - and y -axes.
- Measuring dimensions have been introduced to enable adding dimensions that do not constrain the geometry but instead can be used as a measurement. To change a dimension into a measuring dimension, toggle the padlock icon to the right of the dimension value.
- You can now create a parameter from a dimension by selecting its **Create parameter** check box. The parameter can be used in geometry, mesh, and physics, for example, with some restrictions. When used in the expression of another 2D geometry feature, the expression must depend linearly on the parameter. Parameters can be created from both constraining and measuring dimensions.
- A new context menu displayed when right-clicking in the Graphics window enables a more efficient workflow when adding constraints and dimensions to a drawing in Sketch mode.
- The new **Constrain** button in the Sketch toolbar determines whether constraints can be created automatically when snapping to geometry occurs while drawing or while dragging a vertex. While drawing consecutive segments of a Polygon that snap to being perpendicular, a Perpendicular constraint will be created automatically when the Constrain button is enabled.
- It is now possible to exit the smart Constraint and smart Dimension modes by pressing the Esc key.
- It is now possible to add the Equal Distance constraint to three preselected entities. The constraint will be applied to the distance between the first and second entities and the distance between the second and third entities, according to the order in which the entities were selected.
- To enable a more efficient workflow when adding dimensions to a geometry, the dimension value text field gets focus immediately after a dimension has been added.
- Dragging of vertices, edges, and constraint and dimension symbols is now enabled even when a button in the Constraints or Dimensions group in the Sketch toolbar is selected.

New Functionality in LiveLink™ for AutoCAD® in Version 5.6

The LiveLink™ interface now supports AutoCAD® 2021.

New Functionality in LiveLink™ for Inventor® in Version 5.6

- The LiveLink™ interface now supports Inventor® 2021.
- The new tutorial model *Geometric Parameter Optimization of a Bracket* demonstrates how to set up the parameter optimization of a geometry, which is synchronized by the LiveLink interface when several geometric constraints are required to maintain the design intent.

New Functionality in LiveLink™ for PTC® Creo® Parametric™ in Version 5.6

- The LiveLink™ interface now supports PTC® Creo® Parametric™ 7.0.
- Support for PTC® Creo® Parametric™ 3.0 is no longer available by the LiveLink™ interface. File import of PTC® Creo® Parametric™ 3.0 continues to be supported.
- The new tutorial model *Geometric Parameter Optimization of a Bracket* demonstrates how to set up the parameter optimization of a geometry, which is synchronized by the LiveLink interface when several geometric constraints are required to maintain the design intent.

New Functionality in LiveLink™ for Revit® in Version 5.6

The LiveLink™ interface now supports Revit® 2021.

New Functionality in LiveLink™ for Solid Edge® in Version 5.6

- The LiveLink™ interface now supports Solid Edge® 2021.
- The new *Geometric Parameter Optimization of a Bracket* tutorial demonstrates how to set up the parameter optimization of a geometry, which is synchronized by the LiveLink interface when several geometric constraints are required to maintain the design intent.
- The updated *Electrical Heating in a Busbar Assembly* tutorial 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.

New Functionality in LiveLink™ for SOLIDWORKS® in Version 5.6

- The LiveLink™ interface now supports SOLIDWORKS® 2021.
- The new tutorial model *Geometric Parameter Optimization of a Bracket* demonstrates how to set up the parameter optimization of a geometry, which is synchronized by the LiveLink interface when several geometric constraints are required to maintain the design intent.

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.

CFD Module

New Functionality in Version 5.6

- The total (or stagnation) pressure can now be specified on inlet and outlet boundaries for incompressible flow. Both local and averaged values of the total pressure can be imposed. This new feature functionality can, for example, be used in pump simulations.
- There are three new predefined options for the **Turbulent intensity: Low, Medium, and High**, on inlets and open boundaries. A new **Geometry based** default option has been introduced for the **Turbulence length scale** on inlets.
- The **Fully developed flow** option on inlets and outlets and the **Mass flow** option on inlets have a new default setting, **Apply condition on each disjoint selection separately**. This option sets up a separate equation for each disjoint boundary selection.
- Warnings for disjoint selections have been added in the Fan, Interior Fan, and Grille features.
- The **External Fluid Interface** feature has been replaced by the **Free Surface** feature.
- The Mass Transfer feature in the Euler-Euler Model interfaces has been revised for improved conservation of momentum, and there is also a new predefined **Nonequilibrium model** option available under **Mass transfer model**.
- The Phase Transport interface now supports pseudo time stepping. A Periodic Condition and an Open Boundary condition, available on boundaries exterior to the Phase and Transport Properties feature, have also been added. The Phase Transport interface now also provides improved support for deforming and rotating domains, including the Interior Wall boundary condition.
- The Phase Transport, Mixture Model interfaces have been reformulated and are now based on the mass-averaged velocity field. These interfaces can now be used for compressible, dispersed, multicomponent mixture simulations. It is now also possible to couple separated and dispersed multiphase flow interfaces.
- A new physics area, Nonisothermal Mixture Model, containing physics interfaces for simulating laminar and turbulent nonisothermal n -phase flow, has been added under Multiphase Flow in the Fluid Flow branch. All RANS turbulence models are supported. The multiphysics interfaces couple a laminar or turbulent interface with a heat transfer interface and a phase transport interface using three multiphysics

coupling nodes: Nonisothermal Flow, Mixture Model, and Nonisothermal Mixture Model.

- An Open Boundary condition has been added in the Level Set interface.
- The Two-Phase Flow, Level Set and Two-Phase Flow, Phase Field laminar and turbulent interfaces have been restructured. The Level Set and Phase Field interfaces now have two Initial Values nodes by default, and the previously used Initial Interface nodes have been made obsolete. Instead, the phase interfaces are automatically detected and the initial distribution of the level set or phase field function is solved for in the Phase Initialization study step.
- New options for averaging density and viscosity across phase interfaces have been added in the Two-Phase Flow, Level Set and Phase Field interfaces. In addition to the default volume average, smoothed Heaviside functions and harmonic volume averages can be used for both density and viscosity averaging. For viscosity averaging, mass average and harmonic mass average are also available.
- The new physics interface, Shallow Water Equations, Time Explicit, for modeling depth-averaged free surface flows, has been added under Shallow Water Equations in the Fluid Flow branch.
- The Porous Media Transport Properties feature in the Transport of Diluted Species in Porous Media interface has been replaced by a Porous Medium feature containing two subnodes: Fluid and Porous Matrix. The Porous Matrix node includes support for Porous Material, which is also new in version 5.6.
- In the Heat Transfer in Fluids interface, the **Heat transfer in porous media** check box has been removed, and the Porous Medium feature is now always available. It has been restructured and now contains two subnodes: Fluid and Porous Matrix.

New Models in Version 5.6

There are two new dispersed multiphase flow models under Multiphase Flow in the Application Library. The model *Droplet Rising Through a Suspension* simulates an oil droplet rising through an initially stratified suspension, with a dense layer between two clear layers. This model demonstrates the combination of a dispersed multiphase flow model with a level set model. The model *Polymer Electrolyte Membrane Electrolyzer* simulates dispersed multiphase flow, where liquid water is fed to the anode side, forming oxygen gas on the anode and hydrogen gas on the cathode side, respectively.

Two new models have also been added under Shallow Water Equations in the Application Library. The model *Dam Breaking on a Column* simulates the impact of

a water wave on a column. The pressure force on the column is calculated and compared with experimental results. The model *Tsunami Runup onto a Complex 3D Beach, Monai Valley* is an established benchmark case that models a scaled laboratory experiment of the tsunami runup in the Monai Valley in Japan.

Backward Compatibility with Version 5.5 and Earlier

TURBULENT INTENSITY AND TURBULENCE LENGTH SCALE

The new default value for Turbulent intensity is the same as the old default value, whereas the new **Geometry based** default value in general will differ from the old one. Old Java files with default values for **Turbulence length scale** should add the line,

```
model.component("comp1").physics("spf").feature("in11").  
set("LT_list", "user_defined");
```

in the model Java file to retrieve the old length scale.

EULER-EULER MODEL

Results from models using the Mass Transfer feature in the Euler-Euler Model interfaces may differ in version 5.6 due to the improved momentum conservation.

PHASE TRANSPORT, MIXTURE MODEL

Since the Phase Transport, Mixture Model interfaces have been revised using a mass-averaged formulation to support compressible mixture flows, models built in version 5.5 can return slightly different results when run in version 5.6.

TWO-PHASE FLOW

In the Level Set and Phase Field interfaces, the Initial Interface feature is now obsolete. It does not need to be replaced because the phase interfaces are automatically detected from the selection of the Initial Values features, and the initial distribution of the level set or phase field function is solved for in the Phase Initialization study step.

Backward Compatibility with Version 5.4 and Earlier

Models created using the Brinkman Equations and Free and Porous Media Flow interfaces need to be modified if they apply Forchheimer drag. The previous subfeature has been replaced by a new setting in the Porous Matrix Properties section.

Old Java files need to replace the line

```
model.component("comp1").physics("fp").feature("fmp1").  
create("fd1", "Forchheimer", 2);
```

with

```
model.component("comp1").physics("fp").feature("fmp1").  
set("ktype", "nonDarcianModel");
```

Forchheimer is the default. Also replace

```
model.component("comp1").physics("fp").feature("fmp1").  
feature("fd1").set("betaF", "value");
```

with

```
model.component("comp1").physics("fp").feature("fmp1").  
set("cf", "value");
```

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 5.5 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.

Chemical Reaction Engineering Module

New Functionality in Version 5.6

GENERAL NEW FUNCTIONALITY

- It is now straightforward to apply diffusion coefficients defined by a Chemistry interface. The input for diffusion coefficients now includes a **Source** list with the options **Material** and **Chemistry**. Selecting the latter, it is possible to pick up coefficients from any Chemistry interface present in the model.
- The total number of constraints applied on a boundary are now analyzed. When combining concentration constraints with equilibrium reactions on boundaries, a warning is shown when the system is overconstrained. The constraint analysis is performed in the following interfaces: Transport of Diluted Species, Transport of Concentrated Species, and Nernst–Planck Equations.
- The Fully developed flow option (in Inlet nodes) is now available in single-phase flow interfaces when licensed to the Chemical Reaction Engineering Module.
- When solving mass transfer problems, iterative algebraic multigrid (AMG) solvers are now automatically generated in the solver sequences for large models. A direct solver is also added for convenience.

THERMODYNAMICS

- Three predefined systems are now available in Thermodynamics: dry air, moist air, and water-steam. These common systems can be set up using a reduced number of steps.
- Using the Generate Material Wizard, the resulting material can now be added under global Definitions (as well as in a component of any space dimension).
- The time spent evaluating a function from a thermodynamic system has been significantly reduced.
- The model object for the Thermodynamics node, accessible from the API, has been renamed. It is now accessed in the following manner:

```
model.thermodynamics()
```

instead of

model.capeopen()

Java[®] files from previous versions will continue to run.

REACTIVE PELLET BED FEATURES

- A new Reactive Pellet Bed feature is now available in the Transport of Concentrated Species interface.
- The Reactive Pellet Bed feature in the Transport of Diluted Species interface has been improved in the following manners:
 - A new **Porous Media Diffusion** node is now used for defining the diffusion coefficients for species transport in pellets. When a Chemistry interface is present in the model, diffusion coefficients defined in this interface can be used.
 - The equations solved for have been updated and are now displayed in the main node, in the **Reactions** subnode, and in the **Porous Media Diffusion** subnode.
 - The **Reactions** subnode now includes a **Reacting Volume** setting (**Total volume** or **Pore volume**) for controlling the basis of the pellet reactions.
 - The bed porosity is no longer defined in the **Reactive Pellet Bed** node. Instead, this value needs to be defined in a **Porous Medium** node or in an **Unsaturated Porous Medium** node that solves for the transport in the bed pores. When none of the porous medium domain features are present, a warning node is shown in the solver sequence when computing the model.
 - When using multiple pellet sizes, the porosity for each pellet can be defined.
 - When using multiple pellet sizes, the bed volume fraction of each size is given instead of the volume percentage.

NEW REACTION NODE FUNCTIONALITY IN THE REACTION ENGINEERING AND CHEMISTRY INTERFACES

- It is now possible to balance the reaction formula. To do so, click the **Balance** button in the upper-right corner of the **Reaction Formula** section.
- The reaction rate is now by default defined from the Mass action law. This was previously referred to as the Automatic definition.
- The overall reaction orders when the Mass action law is used are now displayed in text form in the **Reaction Rate** section. The reaction order for both surface and volumetric species are displayed.
- The overall forward reaction order and the overall reverse reaction order (when present) are reported. For surface reactions, the reaction order for both surface and volumetric species are reported.

AUTOMATIC DEFINITION OF THE MOLAR MASS IN THE REACTION ENGINEERING INTERFACE AND THE CHEMISTRY INTERFACE

- The molar mass is now automatically computed from the species chemical formula. The species name is parsed for elements in the periodic table, and the molar mass is computed using the corresponding elemental masses.
 - The molar mass is automatically defined when a **Species** node is created, as well as when a species is added to a **Species Group** node.
 - When the automatic calculation fails, for example, when a descriptive name such as **water** is used, the molar mass is set to 0 g/mol.
 - When computing a model that contains species for which the molar mass remains at zero, a warning node is added in the solver sequence.

THE REACTION ENGINEERING INTERFACE

- It is no longer necessary to use fully scoped variables when postprocessing Reaction Engineering results in 0D components. For example, you can now plot a reaction rate variable `re.R_OH` instead of using the fully scoped `comp1.re.R_OH`.
- The mass balance settings have been moved to the **Reactor** section in the interface settings. The settings for the reactor volume and volumetric rate of change are now given below the Reactor type setting. The Mass Balance section has been removed.
- The order of the sections in the Settings window has been changed to reflect their relative importance. The new order, from top to bottom, is:
 - Reactor
 - Energy Balance
 - Mixture Properties
 - Species Matching (visible when Thermodynamics has been enabled)
 - Calculate Transport Properties
 - Equilibrium Species Vector (visible when equilibrium reactions are used)
 - Activity
 - CHEMKIN Import for Species Properties
 - Advanced Settings (visible when Advanced Physics Options is enabled)

Note that applications created in previous versions retain the original section order.

- Reaction rates for all species are now available as postprocessing variables.
- A new setting, Surface area to volume ratio, is now available when modeling surface reactions.

- When overconstrained equilibrium reactions occur, a **Show Conflicts** button is now shown in the **Equilibrium Species Vector** section on the interface level. Clicking that button, a message indicating the overconstrained reaction is shown.
- The mass conservation equation solved for is now shown when selecting a **Species** node. The equation is updated in accordance with the reactor type used.

THE CHEMISTRY INTERFACE

- The order of the sections in the Settings window has been changed to reflect their relative importance. The new order, from top to bottom, is:
 - Model Input
 - Mixture Properties
 - Species Matching
 - Calculate Transport Properties
 - Activity
 - CHEMKIN Import for Species Transport
 - Extra Dimension

Note that applications created in previous versions retain the original section order.

- The mixture type (diluted species or concentrated species) is now controlled using the **Type** list at the top of the **Mixture Properties** section. It was previously located in the **Species Matching** section.
- The section Equilibrium Species Vector has been removed from the Settings window of the interface.
- Thermodynamic properties are now always defined. The Calculate Thermodynamic Properties section, with the corresponding check box, has been removed.
 - The **Species Thermodynamic Expressions** section is always available in a **Species** node.
 - The **Reaction Thermodynamic Properties** section is always available in a **Reaction** or **Electrode Reaction** node.
- The **Calculate mixture properties** check box, in the **Calculate Transport Properties** section, is now enabled by default.

- The ratio of specific heats can now be specified in the Calculate Transport Properties section. The ratio is defined and announced and can be used in other physics interfaces.
- The setting to specify that the concentration, or activity, of species does not change has been renamed. It is now called Constant concentration/activity. It was previously called Locked concentration/activity.
- The nomenclature in the tables in the Species Matching table section has been made more distinct. Species can now be one of the following types; Constant, Variable, Solvent, or Free species.
- The tables in the Species Matching section no longer contain a Reaction rate column.
- The table at the top of the Species Matching table is now named Bulk species to communicate that these species are present in the mixture phase.

NEW FEATURES FOR POROUS MEDIA TRANSFER

Two new domain features, the Porous Medium and Unsaturated Porous Medium features are available in the Transport of Diluted Species in Porous Media and Transport of Diluted Species interfaces. The new features include support for using the Porous Material node, also new in version 5.6, for assigning material properties to the multiple phases in a porous medium. The new features have subnodes to define phase properties. The subnodes follow the same structure as the subnodes available for a Porous Material.

The settings Dispersion and Volatilization in partially saturated porous media have been removed from the interface level of the Transport of Diluted Species in Porous Media and Transport of Diluted Species interfaces. This functionality is now available in the subnodes of the new features.

The new domain features replace the previously available Porous Media Transport Properties and Partially Saturated Porous Media features. When opening a model created in a previous version, the previous features are kept in the model.

New Applications in Version 5.6

HEAT PIPE WITH ACCURATE LIQUID AND GAS PROPERTIES

Heat pipes are designed to transfer heat efficiently through vaporization, mass transfer, and condensation of a working fluid. They are found in a wide variety of applications

where thermal control is of importance, with cooling of electronics being a prominent example.

Inside a heat pipe, the temperature difference between the hot and cold sides together with the temperature dependence of the vapor pressure, induce a pressure difference across the vapor chamber. The pressure difference, in turn, drives the vapor from the hot to the cold side. The vaporization acts as a heat sink at the vapor–wick interface at the hot side, and conversely, the condensation as a heat source, at the cold side. This model demonstrates how the laminar flow in the vapor chamber of the heat pipe can be coupled to the liquid phase transport through the porous wick, and how thermodynamic properties of water can be obtained from the thermodynamics database. The importance of vapor transport is compared to the conductive heat transfer in the pipe wall. The former dominates the latter by several orders of magnitude.

FERMENTATION IN BEER BREWING

During the beer fermentation process, alcohol, carbon dioxide, and various flavors are formed when sugars are consumed by yeast. In this example, the fermentation process is first modeled in a perfectly mixed tank using the Reaction Engineering interface. In the next step, a space-dependent tank model is set up, accounting for mass transfer, heat transfer, and natural convection.

The model in this example can be used to optimize the beer fermenting process and improve the taste of your beer. The model allows you to study the effect of, for example, initial sugar content, temperature, and yeast type on the fermentation process.

Updated Applications in Version 5.6

IMPROVED MODELS

The following applications have been updated and significantly improved:

- Analysis of NO_x Reaction Kinetics
- Chemical Reactions and Soot Build-Up in a Diesel Filter
- Engine Coolant Properties, under Thermodynamics in the Application Library.
- Finding Kinetic Arrhenius Parameters Using Parameter Estimation
- Hydrocarbon Dehalogenation in a Tortuous Microreactor
- NO_x Reduction in a Monolithic Reactor

- Parameter Estimation for Nonideal Reactor Models
- Steam Reformer

FULLY DEVELOPED FLOW

The following applications have been updated to use the fully developed flow at inlets:

- Chemical Vapor Deposition of GaAs
- Dissociation in a Tubular Reactor
- Fine Chemical Production in a Plate Reactor
- Steam Reformer
- Thermal Decomposition

AUTOMATIC DEFINITION OF MOLAR MASS

The following applications have been updated to use molar masses deduced from the species chemical formula:

- Carbon Deposition in Heterogeneous Catalysis
- Chemical Reactions and Soot Build-Up in a Diesel Filter
- Packed Bed Reactor

Backward Compatibility with Version 5.4 and Earlier

SPECIFYING VOLUMETRIC FLOW RATE OF A PLUG FLOW REACTOR IN REACTION ENGINEERING

When using the Reaction Engineering interface to model gas flow in a plug flow reactor, you must now specify either the pressure or the volumetric flow rate, not both. Running Java® files from previous versions may fail if the volumetric flow rate is set while also prescribing the pressure. In this case, specify to instead compute the pressure from the ideal gas law in the following manner:

```
model.component("comp1").physics("re").prop("mixture").set("psource",  
"idealGas");
```

This selects **Ideal gas law** in the **Reactor pressure** list in the **Mixture Properties** section. When this is done, the **User defined** option is available in the **Volumetric rate** list in the **Mass Balance** section.

WALL FUNCTION VARIABLES OMITTED IN LAMINAR REACTING FLOW

Dependent variables for the mass fraction wall functions are no longer added when using the Reacting Flow multiphysics interface for laminar flow. Due to the now-omitted variables, running Java® files from previous versions may fail during the solver sequence generation.

When specifying the content of a segregated group in the manner of

```
model.sol("sol1").feature("s1").feature("se1").feature("ss1")
.set("segvar", new String[]
{"comp1_w1", "comp1_w2", "comp1_tcs_wWall_w1", "comp1_tcs_wWall_w2"});
```

remove entries for the wall functions variables of the kind comp1_tcs_wWall_name to result in

```
model.sol("sol1").feature("s1").feature("se1").feature("ss1")
.set("segvar", new String[]{"comp1_w1", "comp1_w2"});
```

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.

Composite Materials Module

New Functionality in Version 5.6

POROELASTICITY, LAYERED SHELL MULTIPHYSICS INTERFACE

The new **Poroelasticity, Layered Shell** multiphysics interface is intended for modeling Darcy's law coupled with structural layered shells. This interface will add Layered Shell and Layered Darcy's Law interfaces, together with the new **Layered Poroelasticity** multiphysics coupling. This new multiphysics interface allows the modeling of multilayer domains, such as paperboard and composites having different material properties per layer.

The Layered Darcy's Law interface requires the Porous Media Flow Module.

PLY-BASED MODELING APPROACH

Support for the ply-based modeling has been added in order to define complex layouts much easier compared to the zone-based modeling approach. This is achieved by adding a **Selection** option on the **Layered Material Link, Layered Material, Single Layer Material** nodes under **Layered Material Stack**.

If the selection of all subnodes representing different plies is the same as that of a parent node, no zones are created. If the subnodes (or plies) are assigned different selections, the **Layered Material Stack** automatically creates zones, and the information about each zone is displayed in the **Stack Zone Definition** table. Each zone is treated as a different layered material and can be used accordingly in the physics and the postprocessing.

You can also create **Layer Cross Section Preview** plots and **Create Layer Cross Section Plot** from all layered materials and layered features.

PERFORMANCE IMPROVEMENT IN THE FSDT SHELL

The performance of FSDT shells, implemented in the **Layered Linear Elastic Material** of the Shell interface, is improved significantly for some simpler use cases. This is achieved by integrating the energy expressions only on the base surface using NMQ formulation instead of integrating energy expressions in the product geometry. The new formulation is quite useful in composite sizing problems and is used automatically,

except in the following cases, where energy expressions are integrated on the product geometry:

- If there is any subnode present
- If there is any layered multiphysics coupling present
- If out-of-plane strain DOF is present
- If mixed formulation DOF is present

CONNECTION TYPE OPTION IN THE LAYERED MATERIAL CONTINUITY

A new option of **Connection type** has been added in the **Continuity** node in the Layered Shell interface. The new option is available with **Advanced Physics Options** enabled. The two choices in the connection type input are **Straight** and **Twisted**. The twisted connection type connects the two layered materials meeting side by side in reversed order by connecting the top of one layered material to the bottom of the other layered material. The twisted connection is particularly useful when the normal vector is flipped on one of the sides.

CHANGE IN DEFAULT SOLID MODEL IN THE LAYERED SHELL INTERFACE

The default **Solid model** of the **Linear Elastic Material** in the Layered Shell interface is switched from isotropic to orthotropic. The orthotropic solid model seems like a more natural choice for composite materials. In case any isotropic material is used from the library, its equivalent orthotropic properties are automatically computed using the elastic properties synchronization functionality.

LAYERED MATERIAL PREVIEW PLOTS IN AN APPLICATION

Support for including layered material preview plots in an application has been added. This is achieved by adding a support for creating a preview plot in the postprocessing. The two new buttons, **Create Layer Cross Section Plot** and **Create Layer Stack Plot**, are added on different layered material nodes for this purpose. In addition to using it in an application, the new functionality can also be used for storing different preview plots in the model.

DAMAGE IN LAYERED SHELLS

All the **Damage** models available in the Solid Mechanics interface are now available in **Layered Linear Elastic Material** in the Shell interface, and to the **Linear Elastic Material** model in the Layered Shell interface.

The **Damage** feature requires the Nonlinear Structural Materials Module.

LAYERED HYPERELASTIC MATERIAL IN THE SHELL INTERFACE

All the **Hyperelastic Material** models available in the Solid Mechanics interface are now available in the **Layered Hyperelastic Material** node in the Shell interface. If the Composite Materials Module is available, the material models can also be used in multilayered shells, and the individual layers can have different material models.

The **Hyperelastic Material** feature requires the Nonlinear Structural Materials Module.

LARGE STRAIN PLASTICITY IN THE LAYERED SHELL INTERFACE

The **Linear Elastic Material** in the Layered Shell interface now includes the possibility to model **Large Strain Plasticity**. The same set of yield functions and isotropic hardening models as in the Solid Mechanics interface are available. The formulation assumes large plastic strains, which gives a good approximation above 10% strain.

It is now possible to add **Plasticity** to the **Hyperelastic Material** in the Layered Shell interface. The new subnode uses the **Large Strain Plasticity** formulation. The same set of yield functions and isotropic hardening models as in **Hyperelastic Material** models for the Solid Mechanics interface are available.

The **Plasticity** feature requires the Nonlinear Structural Materials Module.

SET VARIABLES NODE UNDER PLASTICITY

A new subnode **Set Variables** is added on the **Plasticity** node, which is available under **Linear Elastic Material** in the Layered Shell interface and **Layered Linear Elastic Material** in Shell and Membrane interfaces. The purpose of this node is to reset plastic DOFs to model cases like stress relieving and annealing.

The **Plasticity** feature requires the Nonlinear Structural Materials Module.

MIXED FORMULATION IN LAYERED LINEAR ELASTIC MATERIAL

In **Layered Linear Elastic Material** in the Shell interface, a mixed formulation has been added. Mixed formulations can improve accuracy for materials with low compressibility.

VISCOUS DAMPING IN LAYERED LINEAR ELASTIC MATERIAL

The possibility to add viscous damping has been added to the **Damping** subnode under the **Layered Linear Elastic Material** in the Shell interface.

PORE PRESSURE OPTION IN EXTERNAL STRESS

In the **External Stress** subnode of **Linear Elastic Material** in the Layered Shell interface, a new option of **Pore Pressure** has been added.

MISCELLANEOUS

- Improvements and corrections in handling variable thickness layers in the Layered Shell, Shell, and Membrane interfaces.
- Improvements and corrections in computing total reaction forces and moments in the Layered Shell interface.
- Default continuity is imposed on the common edges of **Rigid Domain** and **Layered Linear Elastic Material** (or similar) in the Shell interface.
- New operator `xdintopallint` has been added for integrating a quantity in all interfaces of all the layered materials.
- Support for slitting the DOFs in the extra dimension with partial base surface selection has been added. This allows the **Delamination** node to be available on only a few interfaces with the partial base surface selection.
- The default plots for **Layered Linear Elastic Material** in the Shell and Membrane interfaces have been improved.

New Models in Version 5.6

PLY DROP-OFF IN A COMPOSITE PANEL

Ply drop-off often occurs in composite laminates to tailor the laminate thickness based on the loading requirements. This example illustrates the modeling of ply drop-off in a composite panel. The panel considered for the analysis has symmetric angle-ply layup with three sections: thick, taper, and thin. The thick section has sixteen plies divided into the core, top-bottom belts, and dropped plies. Out of sixteen plies in the thick section, eight plies are dropped gradually in the taper section with specified stagger distance. The layerwise-theory-based approach, together with the Stack Zone modeling and variable thickness layer functionality, is used for the detailed representation of the ply drop scenario. A stationary analysis is performed to compute stresses in different plies of various sections of the composite panel under the applied external load.

STACKING SEQUENCE OPTIMIZATION

This example illustrates the optimization of the stacking sequence in a composite laminate. The composite laminate considered for the analysis has six layers with symmetric layup. The Carbon-Epoxy material having orthotropic material properties is used as a lamina material. The optimization analysis is performed to find the optimum fiber orientation in each layer under specified loading conditions with an objective of minimizing the maximum stress value in the laminate. The layup is

assumed symmetric, thus three ply angles are the control variables and the BOBYQA optimization method is applied to find the optimum stacking sequence.

HEATING CIRCUIT: LAYERED SHELL VERSION

This multiphysics example simulates the electrical heat generation, heat transfer, and mechanical stresses and deformations of a heating circuit device. The model uses the Heat Transfer in Shells interface of the Heat Transfer Module in combination with the Electric Currents, Layered Shell interface from the AC/DC Module and the Layered Shell interface from the Composite Materials Module.

Corrosion Module

New Functionality in Version 5.6

- A new Corrosion Material Library has been added.
- A new Phase Field interface has been added.

ELECTROCHEMISTRY INTERFACES

- A **Dissolving-Depositing Species** section has been added to the **Separator** node of the Tertiary Current Distribution, Nernst–Planck interface.
- The stoichiometric coefficients for the dissolving-depositing species table are now reset to the default value of 0 when duplicated.
- The **Electrode Reaction** and **Porous Electrode Reaction** nodes of the Tertiary Current Distribution, Nernst–Planck interface now include a new option for linearization of concentration dependence for low concentrations of Butler–Volmer kinetics expression type when using either a mass action law or lumped multistep as the exchange current density type.
- A new **From material** option has been added for the **Local current density** setting in the Electrode Kinetics section in electrode reactions of all electrochemistry interfaces.
- A new property, **Physics vs. Materials Reference Electrode Potential**, has been added to all electrochemistry interfaces.
- The solver defaults have been updated for most Electrochemistry interfaces. Geometric and algebraic multigrid solvers (GMG and AMG) are now automatically generated by the study step nodes. A direct solver will, however, still always be used by default. Minor changes have also been made to default scales and tolerances.
- The Electrode Surface (eebii) boundary feature, applicable to both exterior and interior boundaries of electrolyte domains, is now obsolete. It is replaced with a new Electrode Surface (es) boundary feature, applicable to exterior boundaries only, and a new Perforated Electrode Surface (pes) boundary feature, applicable to interior boundaries of electrolyte domains only.
- The Thin Electrolyte Layer boundary feature is now available for the Electroanalysis charge conservation model in the Tertiary Current Distribution interface.

- A new Highly Conductive Porous Electrode domain node has been added to the Primary/Secondary/Tertiary Current Distribution, Lithium-Ion Battery, and Battery with Binary Electrolyte interfaces.
- The irreversible heat source term now contains a contribution from the film resistance resistive heating.

CHEMICAL SPECIES TRANSPORT INTERFACES

- It is now straightforward to apply diffusion coefficients defined by a Chemistry interface. The input for diffusion coefficients now includes a **Source** list with the options **Material** and **Chemistry**. Selecting the latter, it is then possible to pick up coefficients from any Chemistry interface present in the model.
- The total number of constraints applied on a boundary are now analyzed. When combining concentration constraints with equilibrium reactions on boundaries, a warning is shown when the system is overconstrained. The constraint analysis is performed in the following interfaces: Transport of Diluted Species and Transport of Concentrated Species.
- When solving mass transfer problems, iterative algebraic multigrid (AMG) solvers are now automatically generated in the solver sequences for large models. A direct solver is also added for convenience.

THE CHEMISTRY INTERFACE

- It is now possible to balance the reaction formula. To do so, click the **Balance** button in the upper-right corner of the **Reaction Formula** section.
- The reaction rate is now by default defined from the Mass action law. This was previously referred to as the Automatic definition.
- The overall reaction orders when the Mass action law is used are now displayed in text form in the **Reaction Rate** section. The reaction order for both surface and volumetric species are displayed.
- The overall forward reaction order and the overall reverse reaction order (when present) are reported. For surface reactions, the reaction order for both surface and volumetric species are reported.

- The molar mass is now automatically computed from the species chemical formula. The species name is parsed for elements in the periodic table, and the molar mass is computed using the corresponding elemental masses.
 - The molar mass is automatically defined when a **Species** node is created, as well as when a species is added to a **Species Group** node.
 - When the automatic calculation fails, for example, when descriptive name such as water is used, the molar mass is set to 0 g/mol.
 - When computing a model that contains species for which the molar mass remains at zero, a warning node is added in the solver sequence.
- The order of the sections in the Settings window has been changed to reflect their relative importance. The new order, from top to bottom, is:
 - Model Input
 - Mixture Properties
 - Species Matching
 - Calculate Transport Properties
 - Activity
 - Extra Dimension

Note that applications created in previous versions retain the original section order.

- The mixture type (diluted species or concentrated species) is now controlled using the **Type** list at the top of the **Mixture Properties** section. It was previously located in the **Species Matching** section.
- The Equilibrium Species Vector sector has been removed from the settings window of the interface.
- Thermodynamic properties are now always defined. The Calculate Thermodynamic Properties section, with the corresponding check box, has been removed.
 - The **Species Thermodynamic Expressions** section is always available in a **Species** node.
 - The **Reaction Thermodynamic Properties** section is always available in a **Reaction** or **Electrode Reaction** node.
- The **Calculate mixture properties** check box, in the **Calculate Transport Properties** section, is now enabled by default.

- The ratio of specific heats can now be specified in the **Calculate Transport Properties** section. The ratio is defined and announced and can be used in other physics interfaces.
- The setting to specify that the concentration, or activity, of species does not change has been renamed. It is now called Constant concentration/activity. It was previously called Locked concentration/activity.
- The nomenclature in the tables in the **Species Matching** table section has been made more distinct. Species can now be one of the following types: Constant, Variable, Solvent, or Free species.
- The tables in the Species Matching section no longer contain a Reaction rate column.
- The table at the top of the **Species Matching** table is now named **Bulk species** to communicate that these species are present in the mixture phase.

NEW FEATURES FOR POROUS MEDIA TRANSFER

Two new domain features, the Porous Medium and Unsaturated Porous Medium features, are now available in the Transport of Diluted Species in Porous Media and Transport of Diluted Species interfaces. The new features include support for using the Porous Material node, also new in version 5.6, for assigning material properties to the multiple phases in a porous medium. The new features have subnodes to define phase properties. The subnodes follow the same structure as the subnodes available for a Porous Material.

The Dispersion and Volatilization settings in partially saturated porous media have been removed from the interface level of the Transport of Diluted Species in Porous Media and Transport of Diluted Species interfaces. This functionality is now available in the subnodes of the new features.

The new domain features replace the previously available Porous Media Transport Properties and Partially Saturated Porous Media features. When opening a model created in a previous version, the previous features are kept in the model.

New and Updated Models in Version 5.6

- The *Localized Corrosion Using the Level Set Method* model has been updated by evaluating the local current density expression only in the microstructure domain.
- A new *Localized Corrosion Using the Phase Field Method* tutorial model is now available.

- The following updated models use the new Corrosion Material library: crevice_corrosion_fe, crevice_corrosion_with_deformation, galvanic_corrosion_mg_alloys, galvanic_corrosion_with_deformation, isolator_thickness, ship_hull, stray_current, atmospheric_corrosion, and under_deposit_corrosion.
- The *Crevice Corrosion in Iron in an Acetic Acid/Sodium Acetate Solution* model has been updated to use the Highly Conductive Porous Electrode domain node.

Backward Compatibility with 5.4

- Default stoichiometric number for Dissolving-Depositing species changed to 0 (from 1). Java API backward compatibility may be affected.
- The Tertiary Current Distribution, Nernst Planck interfaces have changed default settings to use the Nernst equation for equilibrium potential, Butler-Volmer for kinetics expression type, and Mass action law for exchange current density type. Java API backward compatibility may be affected.

ECAD Import Module

New Functionality in Version 5.6

The ECAD Import Module brings a significant performance improvement for the import of copper layers from IPC-2581 and ODB++ files. Large copper layers may import several times faster when compared to previous versions.

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.

Electrochemistry Module

New Functionality in Version 5.6

ELECTROCHEMISTRY INTERFACES

- For the **Electrode Reaction** and **Porous Electrode Reaction** nodes of the Tertiary Current Distribution, Nernst–Planck interface, a new option for the linearization of concentration dependence for low concentrations of the Butler–Volmer kinetics expression type is now available when using either mass action law or lumped multistep as the exchange current density type.
- A new **From material** option has been added for the **Local current density** setting in the Electrode Kinetics section in electrode reactions of all electrochemistry interfaces.
- A new property, **Physics vs. Materials Reference Electrode Potential**, has been added to all electrochemistry interfaces.
- The solver defaults have been updated for most Electrochemistry interfaces. Geometric and algebraic multigrid solvers (GMG and AMG) are now automatically generated by the study step nodes. A direct solver will, however, still always be used by default. Minor changes have also been made to default scales and tolerances.
- The Electrode Surface (eebii) boundary feature, applicable to both exterior and interior boundaries of electrolyte domains, is now obsolete. It is replaced with a new Electrode Surface (es) boundary feature, applicable to exterior boundaries only, and a new Perforated Electrode Surface (pes) boundary feature, applicable to interior boundaries of electrolyte domains only.
- The Thin Electrolyte Layer boundary feature is now available for the Electroanalysis charge conservation model in the Tertiary Current Distribution interface.
- A new Highly Conductive Porous Electrode domain node has been added to the Primary/Secondary/Tertiary Current Distribution, Lithium-Ion Battery, and Battery with Binary Electrolyte interfaces.
- The irreversible heat source term now contains a contribution from the film resistance resistive heating.

CHEMICAL SPECIES TRANSPORT INTERFACES

- It is now straightforward to apply diffusion coefficients defined by a Chemistry interface. The input for diffusion coefficients now includes a **Source** list with the

options **Material** and **Chemistry**. When selecting the latter, it is then possible to pick up coefficients from any Chemistry interface present in the model.

- The total number of constraints applied on a boundary are now analyzed. When combining concentration constraints with equilibrium reactions on boundaries, a warning is shown when the system is overconstrained. The constraint analysis is performed in the following interfaces: Transport of Diluted Species and Transport of Concentrated Species.
- When solving mass transfer problems, iterative algebraic multigrid (AMG) solvers are now automatically generated in the solver sequences for large models. A direct solver is also added for convenience.

THE CHEMISTRY INTERFACE

- It is now possible to balance the reaction formula. To do so, click the **Balance** button in the upper-right corner of the **Reaction Formula** section.
- The reaction rate is now by default defined from the Mass action law. This was previously referred to as the Automatic definition.
- The overall reaction orders when the Mass action law is used are now displayed in text form in the **Reaction Rate** section. The reaction order for both surface and volumetric species are displayed.
- The overall forward reaction order and the overall reverse reaction order (when present) are reported. For surface reactions, the reaction order for both surface and volumetric species are reported.
- The molar mass is now automatically computed from the species chemical formula. The species name is parsed for elements in the periodic table, and the molar mass is computed using the corresponding elemental masses.
 - The molar mass is automatically defined when a **Species** node is created, as well as when a species is added to a **Species Group** node.
 - When the automatic calculation fails, for example, when descriptive name such as `water` is used, the molar mass is set to 0 g/mol.
 - When computing a model that contains species for which the molar mass remains at zero, a warning node is added in the solver sequence.

- The order of the sections in the Settings window has been changed to reflect their relative importance. The new order, from top to bottom, is:
 - Model Input
 - Mixture Properties
 - Species Matching
 - Calculate Transport Properties
 - Activity
 - Extra Dimension

Note that applications created in previous versions retain the original section order.

- The mixture type (diluted species or concentrated species) is now controlled using the **Type** list at the top of the **Mixture Properties** section. It was previously located in the **Species Matching** section.
- The Equilibrium Species Vector section has been removed from the settings window of the interface
- Thermodynamic properties are now always defined. The Calculate Thermodynamic Properties section, with the corresponding check box, has been removed.
 - The **Species Thermodynamic Expressions** section is always available in a **Species** node.
 - The **Reaction Thermodynamic Properties** section is always available in a **Reaction** or **Electrode Reaction** node.
- The **Calculate mixture properties** check box in the **Calculate Transport Properties** section is now enabled by default.
- The ratio of specific heats can now be specified in the **Calculate Transport Properties** section. The ratio is defined and announced and can be used in other physics interfaces.
- The setting to specify that the concentration, or activity, of species does not change has been renamed. It is now called Constant concentration/activity. It was previously called Locked concentration/activity.
- The nomenclature in the tables in the Species Matching table section has been made more distinct. Species can now be one of the following types: Constant, Variable, Solvent, or Free species.
- The tables in the Species Matching section no longer contain a Reaction rate column.

- The table at the top of the Species Matching table is now named Bulk species to communicate that these species are present in the mixture phase.

NEW FEATURES FOR POROUS MEDIA TRANSFER

Two new domain features, the Porous Medium and Unsaturated Porous Medium features, are now available in the Transport of Diluted Species in Porous Media and Transport of Diluted Species interfaces. The new features include support for using the Porous Material node, also new in version 5.6, for assigning material properties to the multiple phases in a porous medium. The new features have subnodes to define phase properties. The subnodes follow the same structure as the subnodes available for a Porous Material.

The settings Dispersion and Volatilization in partially saturated porous media have been removed from the interface level of the Transport of Diluted Species in Porous Media and Transport of Diluted Species interfaces. This functionality is now available in the subnodes of the new features.

The new domain features replace the previously available Porous Media Transport Properties and Partially Saturated Porous Media features. When opening a model created in a previous version, the previous features are kept in the model.

Backward Compatibility with Version 5.4

- The Tertiary Current Distribution, Nernst–Planck interfaces have changed default settings to use the Nernst equation for equilibrium potential, Butler-Volmer for kinetics expression type, and Mass action law for exchange current density type. Java API backward compatibility may be affected.

Electrodeposition Module

New Functionality in Version 5.6

GENERAL NEW FUNCTIONALITY

A new Phase Field interface has been added.

ELECTROCHEMISTRY INTERFACES

- A **Dissolving-Depositing Species** section has been added to the **Separator** node of the Tertiary Current Distribution, Nernst–Planck interface.
- The stoichiometric coefficients for the dissolving-depositing species table are now reset to the default value of 0 when duplicated.
- For the **Electrode Reaction** and **Porous Electrode Reaction** nodes of the Tertiary Current Distribution, Nernst–Planck interface, a new option for the linearization of concentration dependence for low concentrations of the Butler–Volmer kinetics expression type is now available when using either mass action law or lumped multistep as the exchange current density type.
- A new **From material** option has been added for the **Local current density** setting in the Electrode Kinetics section in electrode reactions of all electrochemistry interfaces.
- A new property, **Physics vs. Materials Reference Electrode Potential**, has been added to all electrochemistry interfaces.
- The solver defaults have been updated for most Electrochemistry interfaces. Geometric and algebraic multigrid solvers (GMG and AMG) are now automatically generated by the study step nodes. A direct solver will, however, still always be used by default. Minor changes have also been made to default scales and tolerances.
- The Electrode Surface (eebii) boundary feature, applicable to both exterior and interior boundaries of electrolyte domains, is now obsolete. It is replaced with a new Electrode Surface (es) boundary feature, applicable to exterior boundaries only, and a new Perforated Electrode Surface (pes) boundary feature, applicable to interior boundaries of electrolyte domains only.
- The Thin Electrolyte Layer boundary feature is now available for the Electroanalysis charge conservation model in the Tertiary Current Distribution interface.

- A new Highly Conductive Porous Electrode domain node has been added to the Primary/Secondary/Tertiary Current Distribution, Lithium-Ion Battery, and Battery with Binary Electrolyte interfaces.
- The irreversible heat source term now contains a contribution from the film resistance resistive heating.

CHEMICAL SPECIES TRANSPORT INTERFACES

- It is now straightforward to apply diffusion coefficients defined by a Chemistry interface. The input for diffusion coefficients now includes a **Source** list with the options **Material** and **Chemistry**. Selecting the latter it is possible to pick up coefficients from any Chemistry interface present in the model.
- The total number of constraints applied on a boundary are now analyzed. When combining concentration constraints with equilibrium reactions on boundaries, a warning is shown when the system is overconstrained. The constraint analysis is performed in the following interfaces: Transport of Diluted Species and Transport of Concentrated Species.
- When solving mass transfer problems, iterative algebraic multigrid (AMG) solvers are now automatically generated in the solver sequences for large models. A direct solver is also added for convenience.

THE CHEMISTRY INTERFACE

- It is now possible to balance the reaction formula. To do so, click the **Balance** button in the upper-right corner of the **Reaction Formula** section.
- The reaction rate is now by default defined from the Mass action law. This was previously referred to as the Automatic definition.
- The overall reaction orders when the Mass action law is used are now displayed in text form in the **Reaction Rate** section. The reaction order for both surface and volumetric species are displayed.
- The overall forward reaction order and the overall reverse reaction order (when present) are reported. For surface reactions, the reaction order for both surface and volumetric species are reported.

- The molar mass is now automatically computed from the species chemical formula. The species name is parsed for elements in the periodic table, and the molar mass is computed using the corresponding elemental masses.
 - The molar mass is automatically defined when a **Species** node is created, as well as when a species is added to a **Species Group** node.
 - When the automatic calculation fails, for example, when descriptive name such as water is used, the molar mass is set to 0 g/mol.
 - When computing a model that contains species for which the molar mass remains at zero, a warning node is added in the solver sequence.
- The order of the sections in the Settings window has been changed to reflect their relative importance. The new order, from top to bottom, is:
 - Model Input
 - Mixture Properties
 - Species Matching
 - Calculate Transport Properties
 - Activity
 - Extra Dimension

Note that applications created in previous versions retain the original section order.

- The mixture type (diluted species or concentrated species) is now controlled using the **Type** list at the top of the **Mixture Properties** section. It was previously located in the **Species Matching** section.
- The section Equilibrium Species Vector has been removed from the Settings window of the interface.
- Thermodynamic properties are now always defined. The Calculate Thermodynamic Properties section, with the corresponding check box, has been removed.
 - The **Species Thermodynamic Expressions** section is always available in a **Species** node.
 - The **Reaction Thermodynamic Properties** section is always available in a **Reaction** or **Electrode Reaction** node.
- The **Calculate mixture properties** check box in the **Calculate Transport Properties** section is now enabled by default.

- The ratio of specific heats can now be specified in the **Calculate Transport Properties** section. The ratio is defined and announced and can be used in other physics interfaces.
- The setting to specify that the concentration, or activity, of species does not change has been renamed. It is now called Constant concentration/activity. It was previously called Locked concentration/activity.
- The nomenclature in the tables in the **Species Matching** table section has been made more distinct. Species can now be one of the following types: Constant, Variable, Solvent, or Free species.
- The tables in the Species Matching section no longer contain a Reaction rate column.
- The table at the top of the **Species Matching** table is now named **Bulk species** to communicate that these species are present in the mixture phase.

NEW FEATURES FOR POROUS MEDIA TRANSFER

Two new domain features, the Porous Medium and Unsaturated Porous Medium features, are now available in the Transport of Diluted Species in Porous Media and Transport of Diluted Species interfaces. The new features include support for using the Porous Material node, also new in version 5.6, for assigning material properties to the multiple phases in a porous medium. The new features have subnodes to define phase properties. The subnodes follow the same structure as the subnodes available for a Porous Material.

The settings Dispersion and Volatilization in partially saturated porous media have been removed from the interface level of the Transport of Diluted Species in Porous Media and Transport of Diluted Species interfaces. This functionality is now available in the subnodes of the new features.

The new domain features replace the previously available Porous Media Transport Properties and Partially Saturated Porous Media features. When opening a model created in a previous version, the previous features are kept in the model.

New and Updated Models in Version 5.6

- The *Electroplating of Multiple Components in a Rack* model has been updated by solving for charge transport only in the electroplating bath, excluding pump cover domains.

- A new *Copper Deposition in a Trench Using the Phase Field Method* tutorial model is now available.
- The *Copper Deposition in a Trench Using the Level Set Method* model has been updated to use the Highly Conductive Porous Electrode feature.

Backward Compatibility with Version 5.4

- The default stoichiometric number for Dissolving-Depositing species changed to 0 (from 1). Java API backward compatibility may be affected.
- The Tertiary Current Distribution, Nernst Planck interfaces have changed default settings to use the Nernst equation for equilibrium potential, Butler-Volmer for kinetics expression type, and Mass action law for exchange current density type. Java API backward compatibility may be affected.

Fatigue Module

New Functionality in Version 5.6

FATIGUE COMPUTATIONS IN GEOMETRICALLY NONLINEAR PROBLEMS

In the Fatigue study step, the **Include geometric nonlinearity** option has been added. This makes it possible to set up fatigue computations for problems with geometric nonlinearities, such as large rotations.

Geomechanics Module

New Functionality in Version 5.6

PLASTICITY IMPROVEMENTS

A new subnode, called **Set Variables**, has been added under **Plasticity**, **Soil Plasticity**, **Concrete**, **Rocks**, and **Elastoplastic Soil Material** nodes. These new nodes make it possible to reset plastic variables based upon boolean conditions, for both small-strain and large-strain plasticity options.

INPUT OF IN-SITU STRESS

A new option to enter **In-situ stress** is available in the **External Stress** subnode for the **Linear Elastic Material**, **Nonlinear Elastic Material**, and **Elastoplastic Soil Material** models.

ELLIPTICAL CAP IMPROVEMENT

In the **Soil Plasticity** node, the transition zone between the elliptical cap and Mohr–Coulomb or Drucker–Prager yield surfaces is modified to account for a smooth interval.

MODIFICATIONS TO ELASTOPLASTIC SOIL MODELS

For the **Extended Barcelona Basic** soil model, Poisson’s ratio is a new input besides the shear modulus.

The initial void ratio is added as a material input for the **Modified Cam-Clay**, **Modified Structured Cam-Clay**, and **Extended Barcelona Basic** soil models.

For the **Hardening Soil** model, the input for the void ratio at reference pressure is replaced by the initial void ratio.

DEFAULT CONTOUR PLOTS FOR NONLINEAR MATERIAL MODELS

When using the **Elastoplastic Soil Material** in the Solid Mechanics interface, default contour plots of the equivalent plastic strain are created inside stress plot groups. Additionally, a separate contour plot showing the void ratio has been added.

BRITTLE FRACTURE OF A HOLED PLATE

In this example, the brittle fracture of a holed plate with a notch is modeled using a phase field damage model. The geometry of the plate is set up so that a mixed-mode fracture is induced. The dimensions and material properties of the plate are taken from experimental data. The example shows how to set up an efficient and stable solver configuration for this kind of problem.

MECHANICAL MODELING OF BENTONITE CLAY

In this example, the Extended Barcelona Basic Model (BBMx) is used to model oedometer, uniaxial swelling, triaxial, and constrained swelling tests on bentonite clays in order to recover the hydromechanical characteristics of the soil samples.

There is a good qualitative agreement between the simulations and the experimental results used to validate the implementation of the BBMx soil model.

STRENGTH REDUCTION METHOD FOR SLOPE STABILITY

The strength reduction technique is a tool to find the factor of safety (FOS) in geotechnical problems, particularly in slope stability. To determine the factor of safety of the slope, the strength properties of the Mohr–Coulomb model are gradually reduced until the failure occurs. The effect of nonassociative plasticity is demonstrated for a range of friction angles.

SHALLOW FOUNDATION ON UNSATURATED SOIL

In this example, a settlement and heave analysis of a shallow foundation resting on an unsaturated soil stratum is investigated with the Modified Cam-Clay and Extended Barcelona Basic soil models. The effect of pore suction due to movements in the phreatic line and the resulting settlement and heave on the clay stratum are shown.

Heat Transfer Module

New and Improved Functionality in Version 5.6

PHASE CHANGE INTERFACE BOUNDARY CONDITION

The new **Phase Change Interface** feature is now available to capture the position of a phase change interface between two solids, two fluids, two porous media, or between a solid and a fluid. The Stefan condition sets the temperature at the interface and defines the front velocity from the heat transfer properties across the interface. Used together with a **Deforming Domain** node, this method provides an accurate computation of the phase change front when no topology changes are expected for the interface. See the *Tin Melting Front* tutorial for an example that uses this feature.

SEMITRANSSPARENT SURFACE FOR RADIATION IN PARTICIPATING MEDIA

The new **Semitransparent Surface** feature is available in the Radiation in Participating Media interface. On exterior boundaries, it allows users to specify an external radiation intensity and to account for the part of this incoming intensity that is diffusively transmitted through the surface, along with the intensities emitted, reflected, and specularly transmitted by the surface. On interior boundaries, the radiation intensities upside and downside the surface are considered instead. It should be used instead of the **Opaque Surface** feature when modeling semitransparent surfaces.

DIRECTIONAL DEPENDENT SURFACE PROPERTIES FOR SURFACE-TO-SURFACE RADIATION

It is now possible to define a directional dependence for the surface material properties in the **Opaque Surface** and **Semitransparent Surface** features of the Surface-to-Surface Radiation interface. The **Polar** or **Polar and Azimuthal** options of the **Directional dependence** list make it possible to set functions of the polar and azimuthal angles to be used in the definition of the surface emissivity and transmissivity. Any analytical, interpolation, or piecewise function with one argument, already defined in the component, may be selected when the **Polar** option is selected. For the **Polar and Azimuthal** option, you can select among the analytical and interpolation functions with two arguments. Other dependencies, such as on temperature, can also be defined in the **Surface emissivity** and **Surface transmissivity** user inputs.

VIEW FACTORS UPDATE FOR SURFACE-TO-SURFACE RADIATION

The option to control the view factor update for time-dependent simulations is now available for **Hemicube** and **Direct area integration** methods. The **View Factors Update** section is available with the **Advanced Physics Options** in the Surface-to-Surface Radiation interface. In this section, you can define a criterion based on a user-defined **Expression**. Doing so may considerably reduce the computation time of time-dependent simulations.

NEW MATERIAL AND FEATURES FOR POROUS MEDIUM MODELS

In order to achieve a better clarity and usability of the user interface, the **Porous Medium** feature is now always available without having to select, at the interface level, the **Heat transfer in porous media** check box, which has been removed.

The **Porous Medium** feature itself has been refactored. Each phase is now handled in dedicated subfeatures, namely **Porous Matrix** and **Fluid**. The effective material properties are handled by the **Porous Medium** node, whereas the solid properties are defined in the **Porous Matrix** subnode, and the fluid properties and velocity field are defined in the **Fluid** node. It is also possible to account for an additional fluid phase, provided that this phase is immobile, by adding an **Immobile Fluids** subnode under the **Porous Medium** feature. The **Geothermal Heating**, **Thermal Dispersion**, and **Optically Thick Participating Medium** subnodes remain available under the **Porous Medium** feature. However, the **Pressure Work**, **Viscous Dissipation**, and **Phase Change Material** subfeatures are now available as subnodes of the **Fluid** subnode, as they affect the fluid variables in particular.

In the user interface of the **Porous Matrix** subfeature, the settings have been homogenized with those of the porous media features in the fluid flow and chemical species transport interfaces. In particular, the **Porosity** is specified instead of the **Volume fraction** of the solid phase. In addition, it is possible to set either the **Dry bulk properties** (which are measured including the empty pores) or the **Solid phase properties**, depending on the availability of these measurements.

Using this new design, you can take advantage of the new **Porous Material** functionality under the **Materials** node, in which the porosity and volume fractions can be set, and under which the material properties are set in subnodes dedicated to each phase: **Solid**, **Fluid**, and **Immobile Fluid**.

HEAT AND MOISTURE TRANSPORT IN POROUS MEDIA

A set of new interfaces and features is available for modeling coupled heat and moisture transport in porous media filled with moist air and liquid water. They complete the

features already available in version 5.5 for the modeling of heat and moisture transport in moist air and building materials.

Moisture transport in porous media, by vapor convection and diffusion, and liquid water convection and capillary flow, is handled by the new Moisture Transport in Porous Media interface, which adds the **Hygroscopic Porous Medium** node as the default model. It extends the capabilities of the **Building Material** node by modeling convection in both phases due to total pressure variations, by modeling the liquid capillary flux either with a Darcy's Law or a diffusion model, and by adding support for gravity force handling. This new formulation is more generic than the **Building Material** formulation, and should be applied for porous media staying in the hygroscopic region regarding their amount of moisture. This new version of the Moisture Transport interface benefits from the features already available for moisture transport in building materials and moist air (**Moisture Flux**, **Moisture Content**, **Outflow**, and so on).

The new **Moist Porous Media** version of the Heat and Moisture Transport interface is found (along with the **Moist Air** and **Building Materials** versions) under the **Heat and Moisture Transport** group in the **Heat Transfer** branch in the model wizard. It adds the Heat Transfer in Moist Porous Media interface, the Moisture Transport in Porous Media interface, and the **Heat and Moisture** multiphysics node.

The Heat Transfer in Moist Porous Media interface is a new version of the Heat Transfer interface, only available with the **Moist Porous Media** version of the Heat and Moisture Transport interface and having the new **Moist Porous Medium** domain feature as the default model.

The new **Moist Porous Medium** domain feature of the Heat Transfer interface is similar to the new version of the **Porous Medium** node, but with **Porous Matrix**, **Moist Air**, and **Liquid Water** subnodes. The effective material properties defined in the **Moist Porous Medium** node account for the solid, liquid water, and moist air properties. The **Moist Air** subnode defines the material properties by taking into account the moisture content, and computes the convective flux and diffusive enthalpy flux in moist air. The **Liquid Water** subnode defines the liquid water saturation and velocity field, which may be automatically set by the **Heat and Moisture** multiphysics coupling if available.

The new **Porous Media Flow** version of the Moisture Flow interface is found (along with the **Laminar Flow** and **Turbulent Flow** versions) under the **Moisture Flow** group in the **Chemical Species Transport** branch in the model wizard. It adds the Brinkman Equations interface, the Moisture Transport in Porous Media interface, and the **Moisture Flow** multiphysics node.

The new **Porous Media Flow** version of the Heat and Moisture Flow interface is found (along with the **Laminar Flow** and **Turbulent Flow** versions) under the **Heat and Moisture Flow** group in the **Heat Transfer** branch in the model wizard. It adds the Brinkman Equations interface, the Moisture Transport in Porous Media interface, the Heat Transfer in Moist Porous Media interface, and the **Moisture Flow** multiphysics node.

The definition of the moisture variables has been updated in the Heat Transfer interface to improve numerical accuracy in concentrated species conditions when the air concentration is small. This update affects the vapor mass fraction, vapor concentration, moisture content, and relative humidity variables.

Finally, the diffusive flux of thermal enthalpy is now accounted for when modeling coupled heat and moisture transport in moist air with the **Heat and Moisture** multiphysics coupling node.

THERMAL CONTACT, SYMMETRY, AND ADDITIONAL IMPROVEMENTS FOR LAYERED MATERIAL FEATURES

The new **Thermal Contact, Interface** feature is available in the Heat Transfer in Shells interfaces. It allows you to represent the surface asperities and the fluid gap at the internal interfaces between the layers of a layered shell. It is available when the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent physics interface.

The new **Symmetry** feature is available in the Heat Transfer in Shells interfaces. It allows you to set a no flux boundary condition for symmetry edges, adjacent to the layered boundaries selected in the parent interface.

In the **Thermoelectric Effect, Layered Shell** coupling node, the conversion of the coupling variables from the spatial frame to the material frame has been improved.

The handling of the heat fluxes and sources, added by the subfeatures available under the **Thin Layer, Thin Film**, and **Fracture** features, has been improved on external boundaries, when an **Isothermal Domain Interface** is also active.

AUTOMATIC DETECTION OF IDEAL GAS MATERIAL IN HEAT TRANSFER IN FLUIDS

The **Fluid** feature, available within the Heat Transfer interfaces, has been updated to take advantage of the ideal gas assumption to improve computational efficiency. The new **From material** option of the **Fluid type** list automatically detects whether the material applied on each domain selection is an ideal gas or not, and uses the relevant properties for either case. This may speed up computation when computing pressure

work in compressible nonisothermal flows, for example. The **Moist air** option has been removed from the **Fluid type** list as it is handled in the dedicated **Moist air** feature.

When the Heat Transfer interface is coupled to a Fluid Flow interface through the **Nonisothermal Flow** coupling node, the density may be taken from the coupled Fluid Flow interface or from the coupling node. In this case, the user interface of the **Fluid** feature has been simplified to show only the relevant user inputs.

HEAT AND ENERGY BALANCE

New and updated variables are available for heat and energy balance control in postprocessing. Among them:

- Updated definition of the `ht.muJT` variable, accounting for compressibility of the flow and taking advantage of the ideal gas assumption when applicable.
- Updated definition of the `ht.WInt` variable, using the new `ht.Wtot`, `ht.WBndTot_u`, and `ht.WBndTot_d` variables and accounting for the work of volume forces and boundary stresses.
- New `ht.Wstr` and `ht.WstrInt` (replacing `ht.WnsInt`) variables, accounting for the total stress power (including viscous dissipation and pressure work).
- New `ht.nthflux`, `ht.nthflux_u`, and `ht.nthflux_d` variables to account for enthalpy flux on boundaries.
- Updated definition of the `ht.nthfluxInt` variable on boundaries adjacent to an **Isothermal Domain** feature, not relying on the gradient of temperature.
- Updated definition of the `ht.ndflux_u` and `ht.ndflux_d` variables in Infinite Elements and axisymmetric domains.
- Update of the `ht.Qoop` variable to account for out-of-plane heat source in the energy balance and residuals.
- New `ht.Q1tot`, `ht.Q1rtot`, and `ht.Qitot` variables and use of these variables in the expression of `ht.QInt`, to account for point and line heat sources in the energy balance.
- Update of the `ht.q0` variable.

In the Heat Transfer in Shells interfaces, there is an improved definition of most for the heat and energy balance variables.

MISCELLANEOUS

- The **Nonisothermal Flow** multiphysics coupling now supports partial domain selection among the coupled fluid flow and heat transfer interfaces. In particular, it makes it possible to couple the Heat Transfer in Fluids interface to a Laminar Flow

interface in one part of the domain, and to a Turbulent Flow interface in another part, by adding two multiphysics coupling nodes, one for each part of the domain. If the variables that are meant to be synchronized between the coupled interfaces (velocity, pressure, temperature) are not available on some part of the domain, the **Common Model Input** value is used instead.

- In the Lumped Thermal System interface, all the two-pin components and terminal nodes now have an **Initial Values** section that provides one or two user inputs for the initial temperature at nodes. This allows the use of, for example, the output of another study or a user-defined value for the temperature in the thermal circuit.
- In the **Irreversible Transformation** subfeature, the new **Initial Values** section provides an input to specify a user-defined value for the **Initial irreversible transformation indicator**.
- In the **Thermal Damage** subfeature, the new **Initial Values** section provides inputs to specify user-defined values for the **Initial damage tissue indicator** and **Initial instant necrosis indicator**.
- The scaling of the equations in the **Electromagnetic Heating**, **Electrochemical Heating**, and **Thermoelectric Effect** multiphysics coupling features has been improved when a **Scaling System** node is added under the **Definitions** node.
- The computation of heat sources on boundaries adjacent to both a solid and a porous medium domain has been improved.
- In the **Heat Flux** feature, the definition of the heat transfer coefficient for **Internal forced convection** in an **Isothermal tube** has been improved, with the evaluation of the fluid material properties at the average between the surface and the external temperatures.

The **Inflow** and **Outflow** features are now available as fallback features for the pair features.

New Applications in Version 5.6

FREEZE DRYING

This example simulates primary freeze drying of skim milk in a vial under vacuum-chamber conditions. This process, also known as lyophilization, can be used to preserve high-value products in the pharmaceutical and food industries. It relies on sublimation of the product. The tutorial computes the phase change front from a solid frozen state to a gaseous state and the vapor flow in the pores of the dried product by solving for heat transfer in both phases and applying the Stefan condition at the phase

change front. The Deformed Geometry interface is used to compute the coupled heat and mass transfer on a moving mesh tracking the ice surface.

Updated Applications in Version 5.6

The tutorials have been updated to take advantage of the latest geometry, mesh, and solver features in order to have simplified step-by-step instructions for the model construction. In addition to these updates, some tutorials have been updated regarding physical modeling:

- The *Tin Melting Front* tutorial has been updated with the use of the new **Phase Change Interface** feature to capture the phase change front.
- The *Evaporation in Porous Media with Large Evaporation Rates* tutorial has been updated with the use of the new Moisture Transport in Porous Media interface to model the transport of vapor and liquid water through the porous medium.
- The *Inline Induction Heater* tutorial now uses boundary layer meshes on faces.

The *Thermal Performances of Roller Shutters* tutorial now uses a corrected equivalent thermal conductivity for the slightly ventilated cavity.

Backward Compatibility with Version 5.5

- The **Heat and Moisture** multiphysics coupling feature is no longer applicable to domains where the **Fluid** (Heat Transfer interface) feature is active. Models built in version 5.5 and before, where the **Heat and Moisture** multiphysics feature and the **Fluid** feature with the **Moist air** option have been used, are updated when opened in version 5.6. The **Fluid** feature is then replaced by the **Moist air** feature so the multiphysics coupling is still active. In the unexpected case where the **Heat and Moisture** multiphysics feature and the **Fluid** feature with an option other than **Moist air** were combined, the multiphysics coupling is no longer active in version 5.6, which is likely to modify the model solution.
- In the Surface-to-Surface Radiation interface, the **Opaque Surface** and **Semitransparent Surface** features not supporting directional dependence of surface properties are now obsolete. When opening a model created in version 5.5 or earlier in the user interface, these features are not migrated. They should be manually replaced by the new **Opaque Surface** and **Semitransparent Surface** features to account for directional dependence of surface properties. Support for features marked as obsolete is not ensured in future versions; thus it is strongly recommended to transfer all these features to the new features.

- In the Heat Transfer interfaces, the **Porous Medium** feature not supporting the new **Porous Material** functionality is now obsolete. When opening a model created in version 5.5 or earlier in the user interface, this feature is not migrated. It should be manually replaced by the new **Porous Medium** feature (which comes with **Porous Matrix** and **Fluid** subfeatures) to allow an updated definition of the material properties of the porous medium. Support for features marked as obsolete is not ensured in future versions; thus it is strongly recommended to transfer this feature to the new feature.
- The **Temperature** feature is no longer applicable on boundaries adjacent to a domain where the **Isothermal Domain** feature is active. When opening a model built in version 5.5 and before, where the **Temperature** feature is applied in this way, the node is present but the feature is not active, which is likely to modify the model solution.
- The **Open Boundary** feature is no longer applicable on boundaries adjacent to a nonfluid domain. When opening a model built in version 5.5 and before, where the **Open Boundary** feature is applied in this way, the node is present but the feature is not active, which is likely to modify the model solution.
- The **Surface-to-surface** radiation interface is no longer applicable on boundaries that belong to infinite element domains, perfectly matched layer domains or absorbing domains since the view factor computation does not account for the corresponding coordinate system change.

Backward Compatibility with Version 5.4

- In the **Inflow** feature (Heat Transfer interface), the **Upstream absolute pressure** user input is now inactive, unless the new **Specify upstream absolute** pressure check box is selected. Scripts using the COMSOL API that used to fill this input field need to add an instruction for selecting the **Specify upstream absolute pressure** check box beforehand.
- The **Open Boundary** feature now comes with a new default formulation based on the Danckwerts inflow condition to specify the incoming temperature. The former default formulation is still available by choosing the **Discontinuous Galerkin constraints** option in the **Inflow Condition** section. To keep the former default formulation in scripts using the COMSOL API, you need to add the corresponding instruction to set the **Discontinuous Galerkin constraints** option in the **Inflow Condition** section.

- When the **Relative humidity** option is used in the **Moisture Content** boundary condition, the associated temperature now has to be defined. In version 5.4 and before, the moisture transport local temperature was used instead. In order to reproduce previous results, models created in version 5.4 or earlier will use the moisture transport interface temperature when opened in the GUI. The scripts using the COMSOL API will use the default temperature 293.15 K instead and may need to be updated to reproduce existing results.
- In the **Heat Flux, Interface** and **Heat Source, Interface** features, when the **Heat Rate** option is selected in the **Heat Flux** and **Boundary Heat Source** sections, respectively, the surface definition has been corrected in order to take into account several interfaces in the **Interface Selection**. For example, if two interfaces are selected, the surface definition is now multiplied by two.
- The **Deposited Beam Power, Interface** and **Surface-to-Ambient Radiation, Interface** features are now applicable on the interior interfaces. In the scripts using the COMSOL API in which the **All interfaces** option was selected, the interior interfaces were selected but no contribution was added on these interfaces. Changes in the numerical results are expected now that the contribution is added also on the interior interfaces.

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>Shared Properties** 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_{pInt} , 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.

MEMS Module

New Functionality in Version 5.6

FERROELECTROELASTICITY MULTIPHYSICS INTERFACE

The new **Ferroelectroelasticity** multiphysics interface is intended for analysis of ferroelectric materials. These materials exhibit nonlinear piezoelectric properties. This interface will add Solid Mechanics and Electrostatics interfaces, together with the new **Electrostriction** multiphysics coupling. In Electrostatics, the new **Charge Conservation, Ferroelectric** material model is used. In this material, it is possible to model, for example, hysteresis using a Jiles–Atherton model.

ELECTROSTRICTION MULTIPHYSICS INTERFACE

With the new **Electrostriction** multiphysics interface, you can model electrostriction phenomena; that is, a situation where a strain proportional to the square of the polarization is induced by an electric field. This interface will add Solid Mechanics and Electrostatics interfaces, together with the new **Electrostriction** multiphysics coupling. In Electrostatics, the standard **Charge Conservation** material model is used.

SPRINGS AND DAMPERS CONNECTING POINTS

In the Solid Mechanics interfaces, a new feature called **Spring-Damper** has been added. With it, you can connect two points with a spring and/or a damper. The points can be geometrical points, but they can also be abstract through the use of, for example, attachments or direct connections to rigid bodies. The spring can either be physical, with a force acting along the line between the two points, or described by a full matrix, connecting all translational and rotational degrees of freedom in the two points. The feature also makes it possible to connect a spring between points in two different physics interfaces.

PORT BOUNDARY CONDITION FOR ELASTIC WAVE PROPAGATION

The new **Port** boundary condition, available with the Solid Mechanics interface, is designed to excite and absorb elastic waves that enter or leave solid waveguide structures. A given **Port** condition supports one specific propagating mode. Combining several port conditions on the same boundary allows a consistent treatment of a mixture of propagating waves; for example, longitudinal, torsional, and transverse modes. The combined setup with several Port conditions provides a superior nonreflecting condition for waveguides to, for example, a perfectly matched layer

(PML) configuration or the Low-Reflecting Boundary. The Port condition supports S-parameter (scattering parameter) calculation, but it can also be used as a source to just excite a system. The power of reflected and transmitted waves is available in postprocessing. To compute and identify the propagating modes, the Boundary Mode Analysis study is available in combination with the Port conditions.

GENERALIZED PLANE STRAIN WITHOUT BENDING

When using generalized plane strain for 2D Solid Mechanics, it is now possible to choose that the out-of-plane strain is limited to be constant over the cross section. You control this behavior through the new **Enable out-of-plane bending** check box in the **2D Approximation** section in the settings for the Solid Mechanics interface.

NEW OPTION FOR PRESCRIBING ROTATING FRAME SPEED

In the **Rotating Frame** node in the Solid Mechanics and Multibody Dynamics interfaces, the new option **Rigid body** has been added. With this option, you enter a time-dependent torque around the axis of rotation, and the rotational velocity is computed by integration of the rigid body equation of motion.

IMPROVED DYNAMIC CONTACT

New algorithms have been introduced in order to improve the conservation of momentum and energy during transient contact events. These can be activated by selecting either the **Penalty, dynamic** or the **Augmented Lagrangian, dynamic** formulation in the **Contact** node.

MORE OPTIONS FOR PENALTY FACTORS IN CONTACT

When using the penalty method for contact analysis, there are three new options in the **Contact** and **Friction** nodes for controlling the value of the penalty factor: **Automatic, soft, Manual tuning,** and **Nonlinear.**

When using the augmented Lagrangian method with a preset penalty factor, there is one new option: **Bending.** The purpose is to provide a softer penalty factor that can stabilize the iterations for bending dominated problems.

FULLY COUPLED SOLUTION METHOD FOR AUGMENTED LAGRANGIAN METHOD IN CONTACT

It is now possible to also use a fully coupled solver together with the augmented Lagrangian contact algorithm. This makes it easier to set up solver sequences, and may also improve stability and convergence for some problems. The new formulation,

allowing a fully coupled solver, is selected in the **Solution method** list in the settings for the **Contact** node.

USER-DEFINED FRICTION MODEL

In the **Friction** node under **Contact**, you can now also select **User defined** as the **Friction model**. In this case, you directly enter an expression for the tangential force that causes sliding in terms of any other variables.

NEW VISCOELASTICITY MODELS

Two new viscoelasticity models have been added: Maxwell and Generalized Kelvin–Voigt.

The Maxwell material can be considered as a type of liquid, since its long-term deformation under a constant stress is unbounded.

The Generalized Kelvin–Voigt model has a Prony series representation with several time constants. Conceptually, it consists of a set of Kelvin elements (spring and dashpot elements in parallel) connected in series.

FRACTIONAL DERIVATIVE VISCOELASTICITY

For frequency-domain analysis, all the viscoelasticity models (Generalized Maxwell, Generalized Kelvin–Voigt, Maxwell, Kelvin–Voigt, Standard linear solid, and Burgers) have been augmented by a fractional derivative representation.

Using a fractional time derivative representation makes it easier to fit material data to experiments for some materials.

NEW TEMPERATURE SHIFT FUNCTION IN VISCOELASTICITY

The Tool–Narayanaswamy–Moynihan shift function is commonly used to describe the glass transition temperature in glasses and polymers. It has been added to the set of shift functions in the Viscoelasticity node.

PERFORMANCE IMPROVEMENTS FOR VISCOELASTICITY

For time-domain analysis using the Generalized Maxwell and Standard Linear Solid viscoelastic models, performance has been improved by up to one order of magnitude. The viscoelastic stresses are no longer represented by ordinary shape functions, but with state variables in the Gauss points.

ANALYSIS OF LARGE DEFORMATION PROBLEMS IN AN UPDATED FRAME

When analyzing problems with large deformations, it is now possible to do so in a manner where the material frame is updated between each parameter value. A

deformed geometry approach is used, and the deformation implicit in the transformation between the two frames is injected as a prestrain in the material model in the updated geometry. To enable this functionality, you add an **Elastic Predeformation** node.

STRAIN FIELD CONSISTENT INTERPOLATION OF INELASTIC STRAIN CONTRIBUTIONS

Several physical interactions, for example, thermal expansion and hygroscopic swelling, cause inelastic strains that are incorporated in the material models. If such strain fields vary too much over an element, spurious artifacts can be seen in the resulting stress fields. In version 5.6, this is taken care of by a reinterpolation so that the external strain field has a polynomial order that matches the total strains computed from the displacements. In particular, this means that in a multiphysics analysis, the discretization orders in the two participating interfaces can be chosen independently, without considering consistency.

As an effect, the Thermal Stress, Solid and Joule Heating and Thermal Expansion multiphysics interfaces no longer reduce the discretization order of the Heat Transfer in Solids interface.

HARMONIC PERTURBATION FOR GRAVITY

The **Gravity** node in the structural mechanics interfaces now has the **Harmonic Perturbation** option. This is useful when a support excitation is transformed into a frame acceleration.

SOLVER SETTINGS CONTROL FOR ELASTIC WAVES

In the settings for the Solid Mechanics interface, a new section **Transient Solver Settings** has been added. The purpose is to generate appropriate solver settings when analyzing wave propagation problems in the time domain.

New Models in Version 5.6

A MICROMACHINED COMB-DRIVE TUNING FORK RATE GYROSCOPE

This tutorial example of a comb-drive tuning fork gyroscope is kindly provided by Dr. James Ransley at Veryst Engineering, LLC. The model demonstrates fully parameterized geometry, extensive use of selection features, implementation of analytic formulas for the electromechanical forces and response estimation, and comparison of numerical results with analytical estimations. In particular, extrusion operators are used to compute the distances between electrodes for force calculation.

MANUFACTURING VARIATION EFFECTS IN A MICROMACHINED COMB-DRIVE TUNING FORK RATE GYROSCOPE

This tutorial example is kindly provided by Dr. James Ransley at Veryst Engineering, LLC. This model continues from the base model “A Micromachined Comb-Drive Tuning Fork Rate Gyroscope”, which is also provided by Dr. Ransley. The model demonstrates how to accurately compute the effects of manufacturing variations of MEMS devices without the need of very fine mesh. This highly efficient modeling approach is based on the unique Deformed Geometry functionality of COMSOL Multiphysics, which implements the device shape change due to fabrication imperfections using the same mesh, thus eliminating the error introduced if different meshes are used for different geometries.

A PIEZOELECTRIC MICROPUMP

This tutorial model is kindly provided by Riccardo Vietri, James Ransley, and Andrew Spann at Veryst Engineering, LLC. Piezoelectric micropumps are frequently used in medical applications because of their ability to precisely control the metering of very small volumes of fluids or gases. This example shows how to simulate a simple, nonresonant micropump, suitable for low-flow-rate applications. The model demonstrates the combination of piezoelectric materials with a fluid-structure interaction, and also illustrates the use of a simple velocity-dependent formula to account for the presence of valves on the inlet and outlet boundaries.

HYSTERESIS IN PIEZOELECTRIC CERAMICS

Many piezoelectric materials are ferroelectric. Ferroelectric materials exhibit nonlinear polarization behavior such as hysteresis and saturation at large applied electric fields. In addition, the polarization and mechanical deformations in such materials can be strongly coupled due to the electrostriction effect. This model uses the Ferroelectroelasticity interface to analyze a simple actuator made of PZT piezoelectric ceramic material, which is subjected to an applied electric field and mechanical load.

MECHANICAL MULTIPOINT SYSTEM: ELASTIC WAVE PROPAGATION IN A SMALL ALUMINUM PLATE

In this tutorial, the vibrational behavior of a small aluminum plate with four waveguide structures is analyzed. This is an example of a structural component located in a device where elastic waves are propagating, like a smart speaker, an electric motor, or a MEMS device. The plate can be thought of as a mechanical multiport system. The model uses the Port boundary condition at the inlet/outlet of the waveguide structures. The port conditions consistently capture and treat the different propagating elastic modes like longitudinal, transverse, and torsional waves. The transmission and reflection of the

various modes is characterized through the scattering matrix of the system, which is computed automatically. This allows the vibrational behavior of the component to be characterized in great detail, for example, for subsequent use in a system simulation.

Metal Processing Module

New Functionality in Version 5.6

THE CARBURIZATION INTERFACE

A new interface has been added to the Metal Processing Module. The interface can be used to study the heat treatment process of carburization. The carbon potential of the surrounding atmosphere can be defined as a boost-diffuse cycle or as user defined. The carbon exchange between a component and its surroundings can be modeled either by prescribing the carbon concentration directly or by applying a carbon flux boundary condition.

IMPORT OF MATERIAL PROPERTIES

It is now possible to import material properties from JMatPro[®] for steel quenching purposes. Material properties are imported for the metallurgical phases austenite, ferrite, pearlite, bainite, and martensite. The imported properties are temperature dependent and can also depend on plastic strain and plastic strain rate.

MISCELLANEOUS

- The Metal Phase Transformation and Austenite decomposition interfaces can now be used in 0D. This allows for easier setup of phase transformation model parameter calibrations.
- A **Transformation times** functionality has been added to the **Metallurgical Phase** nodes in 0D. This functionality is provided to more easily compute data for transformation diagrams.
- It is now possible to define lower and upper temperature limits for the **Leblond–Devaux** and **Johnson–Mehl–Avrami–Kolmogorov** phase transformation models.
- A **Transformation Condition** subnode has been added under the **Phase Transformation** node. The purpose of this is to be able to add additional conditions on a given phase transformation, such as a condition on cooling rate.

CALIBRATION AGAINST TTT DATA

This model shows how to calibrate the parameters of a phase transformation model to experimentally obtained time temperature transformation (TTT) data. The model requires the Optimization Module.

Microfluidics Module

New Functionality in Version 5.6

- The total (or stagnation) pressure can now be specified on inlet and outlet boundaries for incompressible flow. Both local and averaged values of the total pressure can be imposed. This new functionality can, for example, be used in pump simulations.
- The **Fully developed flow** option on inlets and outlets and the **Mass flow** option on inlets have a new default setting, **Apply condition on each disjoint selection separately**. This option sets up a separate equation for each disjoint boundary selection.
- The **External Fluid Interface** feature has been replaced by the **Free Surface** feature.
- The Two-Phase Flow, Level Set and Two-Phase Flow, Phase Field interfaces have been restructured. The Level Set and Phase Field interfaces now have two **Initial Values** nodes by default, and the previously used **Initial Interface** nodes have been made obsolete. Instead, the phase interfaces are automatically detected and the initial distribution of the level set or phase field function is solved for in the Phase Initialization study step.
- New options for averaging density and viscosity across phase interfaces have been added in the Two-Phase Flow, Level Set and Phase Field interfaces. In addition to the default volume average, smoothed Heaviside functions and harmonic volume averages can be used for both density and viscosity averaging. For viscosity averaging, mass average and harmonic mass average are also available.
- The Porous Media Transport Properties feature in the Transport of Diluted Species in Porous Media interface has been replaced by a Porous Medium feature containing the two subnodes Fluid and Porous Matrix. The Porous Matrix node includes support for Porous Material, which is also new in version 5.6.
- An **Open Boundary** condition has been added in the Level Set interface.

Backward Compatibility with Version 5.5

In the Level Set and Phase Field interfaces, the Initial Interface feature is now obsolete. It does not need to be replaced, since the phase interfaces are automatically detected from the selection of the Initial Values features and the initial distribution of the level set or phase field function is solved for in the Phase Initialization study step.

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.

Mixer Module

New Functionality in Version 5.6

SUPPORT FOR FROZEN ROTOR STUDY STEP, CONTINUITY PAIR FEATURE AND INTERIOR WALL IN PHASE TRANSPORT, MIXTURE MODEL INTERFACES

Rotating machinery simulations of dispersed n-phase flow modeled with the Phase Transport, Mixture Model interfaces can now be set up manually by adding a Rotating Domain under Moving Mesh in Definitions and using a Continuity pair feature (for sliding mesh simulations) between the rotating and stationary domains. In addition, quasistationary frozen rotor simulations can also be performed due to added support for this study type.

Multibody Dynamics Module

New Functionality in Version 5.6

RIGID BODY CONTACT AND FRICTION

Rigid body contact and friction functionality has been added. The new **Rigid Body Contact** node in the Multibody Dynamics interface can be used to model meshless contact between standard-shaped rigid bodies. In this version, it allows **Spherical-Spherical** and **Spherical-Arbitrary** types of contact between the two rigid bodies. In case of **Spherical-Spherical** contact, the destination can also be a hollow sphere having inside concave boundaries as the contact boundaries. The contact formulation can be chosen as either **Penalty** or **Penalty, dynamic**. Frictional effects can be included by adding a **Friction** subnode. The friction force is modeled using a continuous friction law, which is smooth and numerically stable but at the same time restricts the use of friction to time-dependent studies.

CLEARANCE JOINT

Joints between components of real-life systems do not always have a perfect fit. For the ease of assembly and movement between parts, a small gap, called a clearance, is often provided between the joined parts. The presence of a clearance can sometimes adversely affect the performance of the system by generation of impact forces, vibration, and noise. In order to model this effect, the new **Clearance Joint** node is added in the Multibody Dynamics interface. This joint essentially allows a clearance distance between the source and destination components. The clearance is an input in this node. This joint allows all rotational degrees of freedom and behaves as a ball joint in 3D and a hinge joint in 2D. A penalty method is used to enforce that the two objects are within the specified clearance.

AUTOMATIC JOINT CREATION FROM GEOMETRY

This functionality makes the model setup easier for large multibody systems. A new button **Create Joints** has been added in the **Automated Model Setup** section of the Multibody Dynamics interface. This button creates joints automatically by finding the objects that are in assembly state and forming **Identity Pairs** among them. By default, **Prismatic Joint** is created for **Planar Boundaries**, **Hinge Joint** is created for **Cylindrical Boundaries**, and **Ball Joint** is created for **Spherical Boundaries**. It is also possible to choose any other appropriate joint for a particular type of boundary. In general, the joints are created for all identity pairs in the component, but it is possible to create joints only

for some identity pairs by disabling the others. The joints created by this operation will also be assigned appropriate information about source, destination, center, and axis.

NEW LUMPED-STRUCTURE CONNECTION COUPLING

The new multiphysics coupling **Lumped-Structure Connection** has been added to connect lumped systems modeled using the Lumped Mechanical System interface to finite element systems modeled using the Multibody Dynamics or any other structural mechanics interface. The connection in the Lumped Mechanical System interface can be made either using the **External Source** node or **Displacement Node**, depending on whether the finite element component is connected to the lumped model at both ends or one end only.

IMPROVEMENTS IN LUMPED MECHANICAL SYSTEM MODELING

The lumped mechanical system modeling functionality has been improved:

- Lumped mechanical systems now support nested subsystems. This makes it possible to model more advanced and complicated systems.
- Support for parametric subsystems has been added. More than one subsystem instance can be added with different input parameters. This makes it possible to create more general subsystem models.

NONLINEAR SPRING FOR JOINTS

A new option **Spring type** has been added in the **Spring and Damper** subnode of all **Joint** nodes in the Multibody Dynamics interface. The **Spring type** option has the option to specify **Force as function of extension** in addition to the existing **Spring constant**. This new option makes it possible to model nonlinear springs in a joint by writing the spring force as a function of extension. In joints having rotational degrees of freedom, **Moment as a function of rotation** is the corresponding option.

PHYSICS SYMBOLS FOR JOINTS

Support for **Physics Symbols** has been added for all the **Joint** nodes in the Multibody Dynamics interface. The physics symbol used is different for each joint type. It is placed at the center of the joint and connectors are drawn from the joint center to all source and destination boundaries.

IMPROVEMENTS IN CHAIN DRIVE MODELING

The chain drive modeling functionality has been improved in the following aspects:

- The response is significantly faster for large models.
- The geometry part selection identification has been improved, making it more robust.

CHANGES IN ENFORCING GEOMETRIC NONLINEARITY

There are several nodes in the Multibody Dynamics interface that have been enforcing geometric nonlinearity in all types of study steps. Some such nodes are **Joints**, **Cam-follower**, and **Prescribed rotation** in **Rigid Connector**, **Rigid Domain**, and **Gears**. There are some situations where this may not be desired. From version 5.6, geometric nonlinearity is enforced only in time-dependent and stationary study steps.

NEW OPTION FOR PRESCRIBING ROTATING FRAME SPEED

In the **Rotating Frame** node in the Solid Mechanics and Multibody Dynamics interfaces, the new option **Rigid body** has been added. With this option, you enter a time-dependent torque around the axis of rotation, and the rotational velocity is computed by integration of the rigid body equation of motion.

RIGID CONNECTOR IMPROVEMENTS

The **Rigid Connector** features have a set of improvements:

- When the center of rotation is defined by a point selection, the point no longer has to be part of the physics interface itself.
- It is possible to couple rigid connectors from different physics interfaces, thus defining a new type of virtual rigid object. This selection resides in the **Advanced** section of the settings for the rigid connector.

IMPROVED DYNAMIC CONTACT

New algorithms have been introduced in order to improve the conservation of momentum and energy during transient contact events. These can be activated by selecting either the **Penalty, dynamic** or the **Augmented Lagrangian, dynamic** formulation in the **Contact** node.

MORE OPTIONS FOR PENALTY FACTORS IN CONTACT

When using the penalty method for contact analysis, there are three new options in the **Contact** and **Friction** nodes for controlling the value of the penalty factor: **Automatic**, **Soft**, **Manual tuning**, and **Nonlinear**. When using the augmented Lagrangian method

with a preset penalty factor, there is one new option: **Bending**. The purpose is to provide a softer penalty factor that can stabilize the iterations for bending-dominated problems.

USER-DEFINED FRICTION MODEL

In the **Friction** node under **Contact**, you can now also select **User defined** as the **Friction** model. In this case, you directly enter an expression for the tangential force that causes sliding in terms of any other variables.

FRICTION WITH A KNOWN SLIDING VELOCITY

In some important applications, the direction of the friction force is known, since two objects are sliding on each other with a given relative velocity. In this case, it is not necessary to solve for the friction forces. Such an approach can speed up the solution significantly. The new **Slip Velocity** node, added under **Contact**, can be used when modeling such cases. The inputs for this node are similar to those of the **Friction** node.

WEAR MODELING

Methods for modeling mechanical wear have been added. This is controlled from the new **Wear** node, a subnode under **Contact** available in the Solid Mechanics and Multibody Dynamics interfaces.

There are two algorithms for wear modeling. In the more general, allowing arbitrary geometry changes due to the wear, a deformed geometry approach is used. There is also a simplified approach in which the wear is incorporated into the offset in the contact gap.

COMPUTING ROTATION FROM DISPLACEMENTS

In physics interfaces that have displacements as degrees of freedom, it is sometimes necessary to study the rotation of a region. To do that, you can add the new **Average Rotation** node, which uses the displacements of a set of points to determine a rotation. If more than three points are given, a least squares scheme is applied to make the best approximation of a rigid body rotation for the entire set of points. In addition, angular velocities and accelerations can be computed.

HARMONIC PERTURBATION FOR GRAVITY

The **Gravity** node in the structural mechanics interfaces now has the **Harmonic Perturbation** option. This is useful when a support excitation is transformed into a frame acceleration.

ITEMS MOVED IN CONTEXT MENUS

The context menu items **Initial Values** and **Initially Rigid** have been moved into the **More** submenu in order to reduce the size of the menu.

New Models in Version 5.6

DYNAMICS OF A CYLINDRICAL ROLLER BEARING

This model simulates the dynamics of a cylindrical roller bearing. Here, multiple cylindrical rollers are inserted between the outer and inner race, and held in place with the help of the cage. The inner race is connected to a rotating shaft and the outer race to a fixed foundation. All components of the bearing are assumed rigid. Frictional contact is modeled between rollers and races using rigid body contact with friction. The connection between the rollers and the cage is simplified using hinge joints. An external load is applied on the inner race and a transient study is performed to analyze the lateral dynamics of the inner race, dynamics of rollers and cages, contact and friction forces, and energy dissipation due to friction.

SLIDER CRANK MECHANISM WITH JOINT CLEARANCE

Joints between two components of a mechanical system are not always perfectly fitting. For ease of assembly and to allow relative movement between the components, a small gap called a clearance is provided that can sometimes adversely affect the performance of the system by generating impact forces, thus giving rise to noise and vibrations. This model compares the performance of a slider crank mechanism with and without a joint clearance. All components of the mechanism are assumed to be rigid. A **Hinge Joint** is used when there is no clearance, whereas a **Clearance Joint** is used to include clearance. A transient analysis is performed to analyze the effect of joint clearance on slider velocity, slider acceleration, and crank moment. In addition to this, the dynamics of the journal within the bearing and the reaction force in the clearance joint are also analyzed.

MODELING VIBRATION AND NOISE IN A GEARBOX: BEARING VERSION

This example is an extension of the model *Modeling Vibration and Noise in a Gearbox*. In this version of the model, a detailed representation of the roller bearings is used to connect the shafts with the housing, rather than a hinge joint. Due to the finite stiffness of the bearings, the vibration response of the system can be significantly different from that obtained using the rigid bearing assumption. In this model, a 5-speed synchromesh gearbox of a manual transmission vehicle is studied. First, the gearbox vibration is computed using a time-dependent analysis for the specified engine

speed and external load. The normal acceleration of the gearbox housing is converted to the frequency domain and used as a source of noise. Finally, an acoustics analysis is performed to obtain the sound pressure levels in the near, far, and exterior fields.

Updated Models and Apps in Version 5.6

UPDATED MODELS

The following models have been updated significantly:

- Three-Cylinder Reciprocating Engine: Uses the new automatic joint creation functionality.
- Reciprocating Engine with Hydrodynamic Bearings: Uses the new automatic joint creation functionality.
- Lumped Model of a Vehicle Suspension System: Uses the new **Lumped-Structure Connection** node.

UPDATED APPS

The following apps have been updated significantly:

- Centrifugal Governor Simulator: The layout of this app has been changed significantly. It also now uses many local forms instead of global forms.
- Truck-Mounted Crane Analyzer: The layout of this app has been changed significantly. It also now uses the new dark theme available in version 5.6.

Nonlinear Structural Materials Module

New Functionality in Version 5.6

PLASTICITY IMPROVEMENTS

A new subnode, called **Set Variables**, has been added under **Plasticity** and **Porous Plasticity** nodes in the Solid Mechanics, Shell, Layered Shell, Membrane, and Truss interfaces. These new nodes make it possible to reset plastic variables based upon boolean conditions for both small-strain and large-strain plasticity options.

POROUS PLASTICITY IMPROVEMENTS

The Capped Drucker–Prager model has been added to the **Porous Plasticity** feature available for the **Linear Elastic Material** and **Nonlinear Elastic Material** models.

The **Large Strain Plasticity** option has been added to all the material models under the **Porous Plasticity** node. The formulation uses a multiplicative decomposition of deformation gradients, giving a good approximation also at high compressive strains.

NONLINEAR ELASTIC MATERIAL IMPROVEMENTS

A new material model called **Shear data** has been added. It is similar to the **Uniaxial data** model, but intended for situations where shear stress vs. shear angle measurements are available.

Hardening data from the Material Library can now be used in the **Uniaxial data** material model.

VISCOELASTICITY IMPROVEMENTS

Two more built-in viscoelastic models have been added to the **Nonlinear Elastic Material**: the Maxwell model and the Generalized Kelvin model. For frequency-domain analysis, it is possible to add fractional derivatives for all the built-in viscoelastic models.

As with the other viscoelastic models, it is accessed by adding a **Viscoelasticity** subnode under the **Nonlinear Elastic Material** node.

When using a **Viscoelasticity** subnode under the **Hyperelastic Material** node, a new leaner implementation for the Generalized Maxwell and Standard Linear Solid models for large-strain viscoelasticity is available, giving significant speedup.

NEW HYPERELASTIC MATERIAL MODELS

Three new hyperelastic material models have been added: The **Extended Tube** model for modeling rubber-like materials and the **Delfino** and anisotropic **Fung** material models to simulate large deformations in biological tissue.

All the **Hyperelastic Material** models available in the Solid Mechanics interface are now available in the **Layered Hyperelastic Material** node in the Shell interface. If the Composite Materials Module is available, the material models can also be used in multilayered shells, and the individual layers can have different material models.

NONLINEAR MATERIAL MODELS IN THE LAYERED SHELL INTERFACE

The **Linear Elastic Material** in the Layered Shell interface now includes the possibility to model **Large Strain Plasticity**. The same set of yield functions and isotropic hardening models as in the Solid Mechanics interface are available. The formulation assumes large plastic strains, which gives a good approximation above 10% strain.

It is now possible to add **Plasticity** to the **Hyperelastic Material** in the Layered Shell interface. The new subnode uses the **Large Strain Plasticity** formulation. The same set of yield functions and isotropic hardening models as in **Hyperelastic Material** models for the Solid Mechanics interface are available.

The Layered Shell interface requires the Composite Materials Module.

DAMAGE IMPROVEMENTS

A new viscous regularization method for time-dependent analysis has been added to the **Damage** feature.

New options, **Polynomial strain softening** and **Multilinear strain softening**, are available for the damage evolution laws for **Scalar damage** and **Mazars damage for concrete** models.

A new **Phase field damage** model has been added as an option to the **Damage** feature. This new model is available with the **Linear Elastic Material** model in the Solid Mechanics interface.

The **Damage** feature has been added to the **Layered Linear Elastic** material in the Shell interface, and to the **Linear Elastic Material** model in the Layered Shell interface. The Layered Shell interface requires the Composite Materials Module.

IMPROVEMENTS TO SHAPE MEMORY ALLOY

The user inputs for the **Shape Memory Alloy** material models are now available as material properties.

For the **Lagoudas** model, it is now possible to enter the material data in terms of transition temperatures or transition stresses.

The **Souza–Auricchio** model is modified to account for the reference temperature instead of the martensite finish temperature. The radius of the elastic domain is entered instead of the initial yield stress.

The **Lagoudas** and **Souza–Auricchio** models are now available in the Truss interface. The **Shape Memory Alloy** node in the Truss interface also has the following subnodes:

- Thermal Expansion
- External Stress
- Initial Stress and Strain
- Phase Transformation

NEW DEFAULT PLOTS

New default contour plots, showing the inelastic strains, have been added for the **Plasticity**, **Viscoplasticity**, **Creep**, **Viscoelasticity**, and **Shape Memory Alloy** features. Similar plots that were overlaid on the default stress plots have been removed.

New Models in Version 5.6

IMPACT ANALYSIS OF A GOLF BALL

In this example, the mechanical impact of a golf club hitting a golf ball is studied. The contact between the two parts is modeled using a viscous penalty formulation to stabilize the dynamic event. The material of the ball is defined using a hyperelastic material model so that the large deformations can be described. The core material is viscoelastic and dissipates the elastic energy imposed by the impact. Results of the simulation are compared to typical golf metrics such as ball speed and spin rate.

BRITTLE FRACTURE OF A HOLED PLATE

In this example, the brittle fracture of a holed plate with a notch is modeled using a phase field damage model. The geometry of the plate is set up so that a mixed-mode fracture is induced. The dimensions and material properties of the plate are taken from experimental data. The example shows how to set up an efficient and stable solver configuration for this kind of problem.

INFLATION OF A SPHERICAL RUBBER BALLOON — SHELL AND MEMBRANE VERSION

This version of the balloon inflation example demonstrates how the Shell and Membrane interfaces can be used to model thin structures made of hyperelastic materials.

PLASTIC STRAIN MAPPING

This example shows the analysis of a perforated plate loaded into the plastic regime. The purpose of the analysis is to demonstrate how plastic strains can be mapped between dissimilar meshes.

POWDER COMPACTION OF A ROTATIONAL FLANGED COMPONENT

The powder compaction process is common in the manufacturing industry, thanks to its potential to produce components of complex shape and high strength. In this example, the compaction of iron powder to form an axisymmetric rotational flanged component is analyzed using the Capped Drucker–Prager plastic model. The friction between the metal powder and die is taken into account.

Optimization Module

New Functionality in Version 5.6

IPOPT OPTIMIZATION SOLVER

The IPOPT optimization solver has properties similar to SNOPT. The main authors are Andreas Waechter and Carl Laird from IBM, but the code is still developed as an open-source project with many contributors.

SAVE INTERMEDIATE SOLUTIONS FOR GRADIENT-BASED OPTIMIZATION

It is now possible to save the last N or every N th solution for gradient-based optimization. This simplifies troubleshooting and makes it possible to create animations of the optimization history for presentation purposes.

SCALE OBJECTIVE FUNCTION WITH INITIAL VALUE

Most optimization solvers work best with scaled objectives, if there are active constraints. Now the initial value for the objective function can be used as the scaled objective function. This simplifies the setup of many constrained optimization problems.

PARAMETER OPTIMIZATION

It is now possible to choose **Copy Selected Rows to New Parameter Case** for parameter optimization. This is available by right-clicking a row in a table produced by one of the derivative-free optimization solvers.

SHAPE OPTIMIZATION

The **Symmetry/Roller** feature now supports specifying a normal vector associated with an edge. This makes it simpler to set up some shape optimization problems for shells. The behavior of the feature has improved in the context of axisymmetry.

TOPOLOGY OPTIMIZATION

The **Topology Link** material is a new feature that combines a structural material with a **Density Model** feature. It simplifies the workflow for structural topology optimization because you no longer have to enter special expressions for the Young's modulus or the density.

The MMA optimizer has gained support for move limits, and it is no longer mandatory to use the globally convergent MMA solver.

A **Prescribed Boundary Density** feature has been introduced. It can be used to impose constraints on density models that enable regularization with a Helmholtz filter.

Finally, the **Parametric Sweep** now has an option for reusing the solution from the previous step. This simplifies the setup of parameter continuation where a sequence of optimization problems are solved; the solution of one problem being used as the initial condition for the next optimization problem.

You can use the graphics context menu for adding shape optimization and topology optimization features.

New and Updated Models and Apps in Version 5.6

In general, the models and apps in the Application Library have been updated to use shape- and topology optimization study steps, the IPOPT solver, the Topology Link, and Model Wizard support for structural topology optimization.

- A new *Optimization of a Photonic Crystal for Signal Filtering* model (photonic_crystal_filter_optimization) changes the position of the pillars in the photonic crystal in order to maximize the ratio of the transmission at the desired frequency to the transmission at the undesired frequency. This model requires the Wave Optics Module in addition to the Optimization Module.
- The *Optimization of a Flywheel Profile* model (flywheel_profile) has been completely revised and updated and is now a 2D axisymmetric model.
- The *General Parameter Estimation* app (general_parameter_estimation) as well as the *Optimization Tutorials* app in the Application Gallery have been updated to support the IPOPT solver.
- The *Shape Optimization of a Shell* model (shell_shape_optimization) has been updated to use the new edge selection for the Symmetry/Roller shape optimization feature.
- The *Bracket — Topology Optimization* model (bracket_topology_optimization_stl) takes advantage of the fact that parametric sweeps now support reusing the solution from the previous step, so the model no longer has to use the withsol operator.
- Three new optimization models are available via the Application Gallery: *Shape and Topology Optimization of Compliant Pliers*, *Shape and Topology Optimization of an Extruded Beam*, and *Optimization of a Truss Tower*.

Backward Compatibility with Version 5.4

The computed solution in models using the Nelder-Mead, Coordinate search, or Monte-Carlo method should not change with *Enforce design constraints* strictly enabled, compared to the solution obtained in the previous version. The number of optimization iterations should either decrease or remain unchanged depending on the number of infeasible iterations. The computed solution in models using COBYLA can differ with the *Enforce design constraints* strictly enabled.

Particle Tracing Module

New Functionality in Version 5.6

PARTICLE PROPERTIES FROM MATERIAL

In the Particle Tracing for Fluid Flow interface, the particle material properties can now be taken from a **Material** node instead of being specified directly. This change allows the Material Libraries to be used more effectively in particle tracing models. It also gets rid of the redundancy of having to specify the same property several times if it is used by several different forces.

By default, every Particle Tracing for Fluid Flow model requires the particle density to be specified. Depending on what additional forces and other features are added to the model, other material properties might also be required. For example, the **Dielectrophoretic Force** requires the particle relative permittivity and electrical conductivity. These will automatically be taken from the same material as the particle density. You still have the option to change any material input back to **User defined**.

When using the new **Droplet Evaporation** node (see below) to treat the model particles as evaporating droplets, you can also take the properties of the vapor phase from another **Material** node.

Backwards Compatibility

When opening any Particle Tracing for Fluid Flow model built in a version prior to 5.6, all particle material properties will be **User defined**.

NEW FORMULATION FOR TRACING SMALL PARTICLES IN FLUIDS

A new formulation is available for the Particle Tracing for Fluid Flow interface. Called the **Newtonian, ignore inertial terms** formulation, it solves first-order equations for the particle position while assuming that the drag force counterbalances all other forces on the particles. In essence, this ignores the particle acceleration when particles are first inserted into the fluid.

Ordinarily, the time step size you need in order to resolve particle acceleration in a fluid scales with the square of the particle diameter. As a result, a full inertial treatment of very small particles (usually submicron size) requires extremely small time steps, and the study can take a very long time to finish. The new **Newtonian, ignore inertial terms** formulation allows much larger time steps to be taken without incurring any additional numerical instability.

DROPLET EVAPORATION

A built-in feature is now available to model evaporation of liquid droplets. The evaporation rate is automatically computed based on the saturation vapor pressure at the droplet surface and the diffusion coefficient of the droplet vapor in the surrounding gas. The **Droplet Evaporation** node supports a simplified **Maxwell** diffusion model, a more detailed **Stefan–Fuchs** model, and an option to specify the evaporation constant directly.

NUMBER DENSITY CALCULATION

The new **Number Density Calculation** feature can be used to compute the number density of particles within the simulation domain. The density is averaged over each domain mesh element.

IMPROVED CONVECTIVE HEATING AND COOLING

When solving for particle temperature, there are now two different ways to apply convective heating or cooling to the particles. In the settings for the **Convective Heat Losses** node, you can now either specify the heat transfer coefficient h directly, or you can specify the particle Nusselt number Nu and the thermal conductivity of the fluid k . The heat transfer coefficient will then be computed automatically.

RANDOM SAMPLING OF MASS, TEMPERATURE, AND OTHER VARIABLES

When you specify the initial values of an **Auxiliary Dependent Variable** for released particles, you can now sample the initial values deterministically or randomly. In previous versions, the sampling was always deterministic. You can sample from built-in normal, lognormal, or uniform distributions.

For the Particle Tracing for Fluid Flow interface, you can also sample the initial particle mass or diameter from these distributions, deterministically or randomly. When sampling the diameter, there is a built-in option to enter the Sauter mean diameter, a common way to describe the size distribution of aerosol particles. The Sauter mean diameter, along with other new variables for describing the particle size distribution, are also available in postprocessing.

EASIER SAMPLING FROM UNIFORM DISTRIBUTIONS

When a particle tracing model includes an **Auxiliary Dependent Variable**, or in the Particle Tracing for Fluid Flow interface when solving for particle mass or diameter, you have the option of sampling the initial values of these extra variables from a **Uniform** distribution. In previous versions, you had to specify the mean and standard deviation, but now you can simply specify the minimum and maximum values.

Backwards Compatibility

When an older model is loaded into version 5.6, if the initial value of an **Auxiliary Dependent Variable**, particle mass, or particle diameter was sampled from a **Uniform** distribution, the minimum and maximum are automatically defined to give the same result as in the older model:

$$\begin{aligned}\text{New minimum} &= (\text{Old mean}) - (\text{Old standard deviation}) * \text{sqrt}(3) \\ \text{New maximum} &= (\text{Old mean}) + (\text{Old standard deviation}) * \text{sqrt}(3)\end{aligned}$$

IMPROVEMENTS TO SPACE CHARGE LIMITED EMISSION

The **Space Charge Limited Emission** Multiphysics coupling node, used with the Charged Particle Tracing interface, has seen significant stability improvements and performance upgrades in version 5.6.

Previous iterations of this feature computed the electric potential in a thin buffer region between the particle emission surface and the cathode by attaching a 1D extra dimension to the selected boundaries, essentially solving Poisson's equation in the outer product space of this extra dimension and the selected boundaries. In version 5.6, this feature no longer defines an extra dimension, instead just applying a mixed boundary condition on the electric potential at the emission surface that is consistent with Child's law (that is, the four-thirds power law).

This change reduces the overall number of degrees of freedom in models using the **Space Charge Limited Emission** node. It also makes postprocessing easier because there is no need to evaluate variables in an extra dimension. In addition, some bugs related to the operation of this feature in 2D axisymmetric geometries were fixed.

VELOCITY OFFSET FOR THERMAL RE-EMISSION

The **Thermal Re-Emission** feature, which lets molecules be adsorbed onto a surface and then released back into the simulation domain with a thermal velocity distribution, now allows you to set a wall velocity. When tracing particles in a rotating frame of reference (using the **Rotating Frame** node), there is also a built-in option to offset the wall velocity by the frame velocity, effectively making the wall stationary with respect to the inertial (or laboratory) frame.

IONIZATION NODE IMPROVEMENTS

The **ionization** node, which can be added to the **Collision** parent node in the Charged Particle Tracing interface, has been improved. You can now separately control whether the primary electron, secondary electron, and ionized species are released following each ionization reaction.

New and Updated Applications in Version 5.6

PIERCE ELECTRON GUN (NEW MODEL)

An electron gun must be able to draw a sufficient current and accelerate the electrons to the desired speed. The first part of an electron gun geometry presents unique design challenges because the emitted electron speeds are usually lowest there, and therefore the space charge density is quite high. The Pierce electron gun design uses electrodes with a particular shape to counteract the Coulomb repulsion between electrons in the beam. As a result, the electrons in the beam propagate in straight lines. The emitted electrons at the cathode are assumed to be space charge limited; the initial thermal distribution of electron velocities is neglected.

TURBOMOLECULAR PUMP (UPDATED MODEL)

The updated turbomolecular pump has a simplified geometry and an updated selection of boundary conditions, including a more realistic treatment of the tip wall that subtracts the rotating frame velocity from the velocity of the reflected particles. The resulting compression ratios agree much better with published literature, compared to previous versions of the model.

Backward Compatibility with Version 5.5

TEMPERATURES MOVED TO MODEL INPUT SECTION

In the **Convective Heat Losses** and **Radiative Heat Losses** nodes (available for the Particle Tracing for Fluid Flow interface when the **Compute particle temperature** check box is selected), the input for the ambient temperature has been moved to the **Model Input** section.

Backward Compatibility with Version 5.4

REMOVED THE PARTICLE TRAJECTORIES STUDY STEP

The **Particle Trajectories** study step is obsolete. All instances of this study step have been replaced with Time Dependent study steps with the same settings.

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.

Pipe Flow Module

New Functionality in Version 5.6

FLUID-PIPE INTERACTION, FIXED GEOMETRY MULTIPHYSICS INTERFACE

The new **Fluid-Pipe Interaction, Fixed Geometry** interface connects the Pipe Flow and Pipe Mechanics interfaces in order to transfer fluid loads to the structural analysis. The new **Fluid-Pipe Interaction** multiphysics coupling is used. Internal pressure, drag forces, loads on bends, and loads on junctions can be taken into account.

LICENSE UPDATE FOR PIPE ACOUSTICS

The **Pipe Acoustics, Frequency Domain** and **Pipe Acoustics, Transient** interfaces are now available with either the Pipe Flow Module or the Acoustics Module.

Updated Model in Version 5.6

COUPLED ANALYSIS OF FLOW AND STRESS IN A PIPE

The model has been updated to make use of the new **Fluid-Pipe Interaction, Fixed Geometry** interface.

Backward Compatibility with Version 5.3

The old **Pipe Connection** nodes, which could be added directly to the Pipe Flow interface and Laminar Flow Interface, respectively, are now obsolete. The same functionality is available in a more convenient manner using the **Pipe Connection** multiphysics coupling.

Plasma Module

New Functionality in Version 5.6

Several interfaces in the Plasma Module were updated to reflect internal advances in the COMSOL Multiphysics software. The affected interfaces are Plasma; Drift Diffusion; Heavy Species Transport; Inductively Coupled Plasma; Microwave Plasma; and Boltzmann Equation, Two-Term Approximation. In general, the user interface, functionality, and operation of the new interfaces are the same as in older versions. However, there are fundamental internal changes that impede the possibility to have full backward compatibility. Therefore, when opening a model created with an interface before version 5.6, the model will still use the old interface. A warning sign will appear in the interface indicating that the interface is obsolete. It is encouraged to recreate the model using version 5.6.

The finite volume formulation was completely reformulated and is now much more stable. This is a major advancement and makes it possible to have better performance in drift-dominated problems.

Backward Compatibility with Version 5.5 and Earlier

There are new versions of the following interfaces: Plasma; Drift Diffusion; Heavy Species Transport; Inductively Coupled Plasma; Microwave Plasma; and Boltzmann Equation, Two-Term Approximation. When opening models created with version 5.5 and earlier, a warning sign appears saying “This physics interface is obsolete and will be removed in a future version. A new version of the physics interface is available from the Add Physics window”. These models can still be changed and will run as before, but it is encouraged to recreate the models using version 5.6.

Porous Media Flow Module

New Functionality in Version 5.6

LAYERED DARCY'S LAW INTERFACE

The Layered Darcy's Law interface is used to simulate fluid flow through interstices in layered porous media, such as paperboard, composites, or plywood. It can be used to model low-velocity flows and porous media where the permeability and porosity are very small, and for which the pressure gradient is the major driving force.

POROELASTICITY, LAYERED SHELL MULTIPHYSICS INTERFACE

The new **Poroelasticity, Layered Shell** multiphysics interface allows the modeling of multilayer domains such as paperboard and composites having different material properties per layer. The interface adds the **Layered Shell** and the **Layered Darcy's Law** interfaces, together with the new **Layered Poroelasticity** multiphysics node.

The Layered Shell interface requires the Composite Materials Module.

REVAMPED POROUS MEDIA FEATURES FOR TRANSPORT OF DILUTED SPECIES

The **Transport of Diluted Species in Porous Media** interface, used to study the flow and chemical composition of fluids moving through the interstices of a porous medium, is revamped to use the new **Porous Material** node.

Two new domain features, the **Porous Medium** and the **Unsaturated Porous Medium** nodes, are available in the **Transport of Diluted Species in Porous Media** and **Transport of Diluted Species** interfaces and include support for using the new **Porous Material** node for assigning material properties to the multiple phases in a porous medium. The new nodes have subnodes to define the properties for the **Liquid, Gas, and Porous Matrix**. The subnodes follow the same structure as the subnodes available in the **Porous Material**.

The **Dispersion** and **Volatilization in partially saturated porous media** check boxes have been removed from the **Transport Mechanisms** settings. This functionality is now available in the subnodes of the new **Porous Medium** and **Unsaturated Porous Medium** nodes.

The new nodes replace the previously available **Porous Media Transport Properties** and **Partially Saturated Porous Media** nodes. These nodes are kept when opening a model created in a previous version.

REVAMPED POROUS MEDIA FEATURES FOR HEAT TRANSFER

The **Porous Medium** feature is now always available without selecting the **Heat transfer in porous media** check box, which has been removed.

The **Porous Medium** node has been revamped. The solid and fluid phases are now handled in dedicated subnodes, namely the **Porous Matrix** and **Fluid** nodes. The computation of the effective material properties is handled in the **Porous Medium** node, whereas the solid and fluid properties are defined in the **Porous Matrix** and **Fluid** subnodes. It is also possible to account for an additional immobile fluid phase by adding the **Immobile Fluids** subnode under the **Porous Medium** node.

In the new **Porous Medium** node, the settings are homogenized with those of the porous media features in the fluid flow and chemical species transport interfaces. In particular, the **Porosity** is specified instead of the **Volume fraction** of the solid phase. In addition, it is possible to set either the **Dry bulk properties** or the **Solid phase properties**, depending on the availability of these material properties.

The new design allows to take advantage of the new **Porous Material** functionality under the **Materials** node, in which the porosity and volume fractions may be set, and where material properties are set in subnodes dedicated to each phase: **Solid**, **Fluid**, and **Immobile Fluid**.

MOISTURE TRANSPORT IN POROUS MEDIA

A set of new features are available for modeling coupled heat and moisture transport in porous media, saturated or partially saturated with moist air and liquid water.

The new **Moisture Transport in Porous Media** interface is found along with the **Moisture Transport in Air** and **Moisture Transport in Building Materials** interfaces under the **Moisture Transport** group in the **Chemical Species Transport** branch in the Model Wizard. The new **Moisture Transport in Porous Media** interface can handle moisture transport in porous media by convection and diffusion of vapor, and by convection and capillary flow of liquid water. The new **Hygroscopic Porous Medium** node extends the capabilities of the **Building Material** node, by including convection in both gas and liquid phases due to total pressure variations, and by modeling the liquid capillary flux with either Darcy's law or a diffusion model. This new formulation is more general than the one implemented in the **Building Material** node, and it is intended for

applications where the porous medium stays in the hygroscopic region regarding the amount of moisture.

The new **Porous Media Flow** multiphysics interface is available along with the **Laminar Flow** interface under the **Moisture Flow** group in the **Chemical Species Transport** branch in the Model Wizard. It adds the **Brinkman Equations** interface, the new **Moisture Transport in Porous Media** interface, and the **Moisture Flow** multiphysics node.

HEAT AND MOISTURE TRANSPORT IN POROUS MEDIA

The **Heat Transfer in Moist Porous Media** interface is a new version of the Heat Transfer interface, available with the **Moist Porous Media** version of the Heat and Moisture Transport interface, which adds the new **Moist Porous Medium** node as default. The new **Moist Porous Medium** node is similar to the new **Porous Medium** node, but it includes the **Porous Matrix**, **Moist Air**, and **Liquid Water** subnodes as default. The effective material properties defined in the **Moist Porous Medium** node account for the solid, liquid water, and moist air properties. The **Moist Air** subnode defines the material properties from the moisture content, and it computes the convective flux and diffusive enthalpy flux in moist air. The **Liquid Water** subnode defines the liquid water saturation and velocity field, which is automatically set by the **Heat and Moisture** multiphysics node if available.

The new **Moist Porous Media** multiphysics interface is found along with the **Moist Air** and **Building Materials** interfaces under the **Heat and Moisture Transport** group in the **Heat Transfer** branch in the Model Wizard. It adds the new **Moisture Transport in Porous Media** interface, the **Heat Transfer in Moist Porous Media** interface, and the **Heat and Moisture** multiphysics node.

The new **Porous Media Flow** multiphysics interface is found along with the **Laminar Flow** multiphysics interface under the **Heat and Moisture Flow** group in the **Heat Transfer** branch in the Model Wizard. It adds the **Brinkman Equations** interface; the new **Moisture Transport in Porous Media** interface; the **Heat Transfer in Moist Porous Media** interface; and the **Moisture Flow**, **Heat and Moisture**, and **Nonisothermal Flow** multiphysics nodes.

The definition of the moisture variables has been updated in the **Heat Transfer** interface to improve numerical accuracy in concentrated species conditions, when the air concentration is small. This update affects the vapor mass fraction, the vapor concentration, the moisture content, and the relative humidity variables.

The diffusive flux of thermal enthalpy is now accounted for when modeling coupled heat and moisture transport in moist air with the **Heat and Moisture** multiphysics coupling node.

IMPROVEMENTS FOR PHASE TRANSPORT

The Phase Transport interface now supports pseudo time stepping. A Periodic Condition and an Open Boundary condition, available on boundaries exterior to the Phase and Transport Properties feature, have also been added. The Phase Transport interface now also provides improved support for deforming and rotating domains, including the Interior Wall boundary condition.

New Models and Apps in Version 5.6

ANALYZING POROUS STRUCTURES ON THE MICROSCOPIC SCALE

Modeling flow through realistic porous structures is difficult due to the complexity of the structure itself. Resolving the flow field in detail is not feasible in real-life applications. Therefore, macroscopic approaches are used, which utilize averaged quantities of the porous structure, such as porosity and permeability. This example analyzes the flow field at the pore scale in detail.

OPTIMIZATION OF A POROUS MICROCHANNEL HEAT SINK

In this example, the performance of a porous microchannel heat sink is optimized using the Optimization Module. The optimum value of the thickness of the porous substrate is calculated. This example is an extension of the model “Performance of a Microchannel Heat Sink” and the results are compared with the results of the parameter study of the original model.

Ray Optics Module

New Functionality in Version 5.6

IMPROVED RENDERING OF RAY TRAJECTORIES

A new option for displaying a large number of rays in a **Ray Trajectories** plot is now available. In the **Extra Time Steps** section, you can now select **All** from the **Maximum number of extra time steps rendered** list. This new option will clearly display the intersection point of each ray, just as it reaches a boundary. The time to display the plot also scales much better with the total number of rays in the plot and the total number of intersection points, compared to the other options.

The new option **All** can also be selected for the **Intersection Point 3D** and **Intersection Point 2D** datasets, allowing for fast and accurate evaluation of ray quantities at the intersection points with a sphere or plane.

OPTICAL SCATTERING IN DOMAINS

You can now model attenuation of rays due to scattering and absorption in a domain containing small particles. Some typical examples include dust, smoke, and water droplets in air, or small bubbles in water.

You can choose from different methods of computing the scattering and absorption cross sections, depending on the size of the particles relative to the electromagnetic wavelength. For very small particles, the Rayleigh theory can be used. A more general approach for somewhat larger particles is the Mie theory. Finally, some asymptotic solutions for very large scattering particles are also available. The functions that generate the scattering and extinction efficiency factors from the Mie scattering theory are now built-in functions, called `miescattering(n,x)` and `mieextinction(n,x)`, that can be used with any physics interface.

Optical extinction can either be modeled deterministically, with the ray intensity and power falling off exponentially or randomly, with each ray having a probability of extinction in any given time step. Built-in accumulated variables are also available to collect the deposited energy due to absorption or the total energy lost due to extinction.

SCATTERING AT RANDOM ROUGH SURFACES

A dedicated feature for modeling reflection and refraction at rough surfaces is now available. You may specify diffuse or isotropic reflection and refraction at the surface, where the probability distribution of reflected and refracted ray direction does not depend on the incident ray direction. Alternatively, you can specify a Rayleigh roughness parameter, or surface slope error, that applies a random perturbation to the surface normal and then applies Snell's law and the Fresnel equations to the reflected and refracted rays based on this perturbed normal direction. This allows for more realistic modeling of lens surfaces that might not be perfectly smooth.

IDEAL LENS BOUNDARY CONDITION

A dedicated boundary condition is now available for treating flat surfaces as if they were thin lenses of a specified optical power. This allows for some very quick, approximate ray optics simulations that don't require a very detailed geometry or material properties to be specified.

NEW OPTIONS FOR SPECIFYING REFRACTIVE INDEX

The options for specifying a refractive index in the **Medium Properties** node have been reorganized. You can now choose to specify either the absolute index (with respect to vacuum) or the relative index (with respect to air) directly. For the relative index, you specify the reference temperature and reference pressure at which the refractive index of air is computed.

Backwards Compatibility

The old **Medium Properties** node settings (before version 5.6) have been mapped to the new settings so as to give the same solution as in older versions. For example, the old option **Specify refractive index** is now **Specify absolute refractive index**.

NEW CONE-BASED RELEASE: FLAT CONE IN 3D

When releasing rays in a conical distribution in 3D, you now have the option to release a flat cone, essentially requiring all rays to lie in the same plane. You can control the orientation of the plane that the rays are held in.

In addition, for some other conical distributions, you now have finer control over the definition of the transverse directions to the cone axis. This can make a significant difference when rays are released at specific azimuthal angles to this transverse direction.

RESET VACUUM WAVELENGTH OR FREQUENCY

At some boundary conditions, such as the **Wall**, you can now reset the vacuum wavelength or frequency of rays. The new values can be specified directly or sampled from a normal, lognormal, or uniform distribution.

RANDOM SAMPLING OF WAVELENGTH AND FREQUENCY

When releasing polychromatic rays, you can now sample the vacuum wavelength or frequency of the rays deterministically or randomly. In addition, if the model includes any **Auxiliary Dependent Variable** nodes, you can sample the initial values of these extra variables deterministically or randomly. In previous versions, these variables were always sampled deterministically.

EASIER SAMPLING FROM UNIFORM DISTRIBUTIONS

When releasing polychromatic rays, if you sample the vacuum wavelength or frequency from a **Uniform** distribution, you can now specify the maximum and minimum values in the distribution. Previously, you had to specify the mean and standard deviation. This change also applies to any **Auxiliary Dependent Variable** nodes in the model.

Backwards Compatibility

When an older model is loaded into version 5.6, if the initial value of an **Auxiliary Dependent Variable**, vacuum wavelength, or ray frequency was sampled from a **Uniform** distribution, the minimum and maximum are automatically defined to give the same result as in the older model:

$$\begin{aligned}\text{New minimum} &= (\text{Old mean}) - (\text{Old standard deviation}) * \text{sqrt}(3) \\ \text{New maximum} &= (\text{Old mean}) + (\text{Old standard deviation}) * \text{sqrt}(3)\end{aligned}$$

NEW GEOMETRY PART: SPHERICAL POLYGONAL LENS

The Spherical Polygonal Lens can now be selected from the Part Library for the Ray Optics Module. The lens surfaces can either be concave or convex, and the outer edge can be a regular polygon with any number of vertices.

New Applications in Version 5.6

DOUBLE GAUSS LENS IMAGE SIMULATION

This example extends the basic Double Gauss Lens tutorial by illustrating how the lens system creates an image. A bitmap is first loaded into the model and positioned at the object plane. Rays are released from the object plane, and as they reach the corresponding image plane, they reconstruct an image of the original object.

PETZVAL LENS GEOMETRIC MODULATION TRANSFER FUNCTION

The example demonstrates how to use model methods to compute and plot the line spread function (LSF) and the geometric modulation transfer function (MTF) of a Petzval lens.

Important Fixes in Version 5.6

GRATINGS CONTRIBUTE TO THE REFLECTION COUNT

When using the built-in option to **Count reflections** in a model that uses the **Grating** or **Cross Grating** node, the reflection counter is also incremented for the reflected rays of all diffraction orders.

RELEASE FROM FAR FIELD WITH A WAVELENGTH DOMAIN STUDY

The **Release from Far-Field Radiation Pattern** node can now be used to release rays when the far-field radiation pattern was computed using a **Wavelength Domain** study step. Previously, only a **Frequency Domain** study step was supported.

PHASE SHIFT OF P-POLARIZED RAYS IN 2D

Previously a nonphysical phase shift was observed for reflected rays at a **Material Discontinuity** with in-plane (p) polarization in 2D models. This has been corrected.

ILLUMINATED SURFACE WITH PHASE

Previously, the **Illuminated Surface** ray release feature would give an error if it was used while the **Compute phase** check box was selected in the physics interface **Additional Variables** section. This has been fixed.

In addition, the initialization of ray Stokes parameters from the **Illuminated Surface** has been updated to be more consistent with the behavior of rays incident on walls and material discontinuities.

Backward Compatibility with Version 5.5

ILLUMINATED SURFACE SETTINGS

Many of the settings for the **Illuminated Surface** ray release feature have been renamed. However, in older models, the values of the variables with the old names will be mapped to the variables with the new names. In addition, the old names can still be used in model methods in the Application Builder or in Model Java[®] files.

Backward Compatibility with Version 5.4

HANDLING OF GRATING ORIENTATION DIRECTION

The handling of grating orientation was made more robust and consistent in version 5.5 compared to previous versions. Occasionally, this means that the directions associated with positive and negative diffraction orders will appear reversed when opening an older model in version 5.5 or newer.

To assist in the reassignment of any grating orientation directions, the direction associated with positive diffraction orders will be indicated by an arrow in the Graphics window.

RENAMED RAY RELEASE FEATURES

In version 5.5 the **Inlet** feature was renamed to **Release from Boundary** and the **Inlet on Axis** feature was renamed to **Release from Symmetry Axis**.

RENAMED RAY STATISTICS

Since version 5.5 some variables for the ray statistics have been renamed. Here, <phys> is the physics identifier and <feat> is the tag of a ray release feature.

- <phys>.rmaxall has been renamed to <phys>.rmax.
- <phys>.rmidrms has been renamed to <phys>.rnrms.
- <phys>.<feat>.rmaxrel has been renamed to <phys>.<feat>.rmax.
- <phys>.<feat>.rmidrms has been renamed to <phys>.<feat>.rnrms.

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.

RF Module

New Functionality in Version 5.6

ANALYZING THE RCS OF A CONDUCTIVE CONVEX-SHAPED OBJECT QUICKLY

When the scattered shape is mostly convex, such as a sphere, this new physics interface can be used for quick studies of the far-field response of a 3D or 2D object to a given background field. The physics interface sets up a surface electric background field for the far-field transformation using the Stratton–Chu formula, performed in the postprocessing.

SOLVING FOR FAST PORT SWEEP

Use a new study step, Frequency Domain Source Sweep, to solve a frequency-domain study that is sweeping among ports and lumped ports calculating a full S-parameter matrix. The settings for this study step are similar to those for the Frequency Domain study step and much simpler than the traditional port sweep, which requires a parametric sweep step.

PORT ENHANCEMENT

Numeric TEM impedance calculation can now be performed without defining the magnetic field integration line for the current. The port impedance is computed using the average power flow on the port boundary and voltage calculated by the electric field line integration. A transverse electromagnetic (TEM) type port supporting the user-defined electric potential and ground boundaries is now available in 2D and 2D axisymmetric models. Lumped port excitation can be configured by input power.

ADDITIONAL RF PART LIBRARY

Four new edge-launch connectors from Signal Microwave have been added in the RF Module Part Library.

POLARIZATION PLOT TYPE

The Polarization plot type depicts the polarization state for the different diffraction orders. It is available as a plot type in postprocessing and is used for default plots when periodic ports are included in the simulation.

SCATTERING BOUNDARY CONDITION CAN USE OBLIQUE ANGLE OF INCIDENCE IN MODE ANALYSIS

The Scattering boundary condition can now, for mode analysis, efficiently absorb waves having a wave vector composed of the mode's propagation constant directed tangentially to the boundary and a remaining normal component.

INPUT POWER TO CONTROL GAUSSIAN BEAM AMPLITUDE

For Gaussian beam background fields and input fields to the Scattering boundary condition, the amplitude of the beam can be specified by providing the input power.

REFERENCE POINT CAN BE DEFINED BY A GENERAL EXPRESSION

The Reference point subfeature to the Scattering boundary condition can now also be specified from a general vector expression.

SYNCHRONIZATION OF MATERIAL PARAMETERS BETWEEN DIFFERENT RELATED MATERIAL PROPERTY GROUPS

The relative permittivity, refractive index, loss tangent, and dielectric loss material models can synchronize the material parameters between the groups. Thus, if a material is added and specified by the refractive index material property group, the Electric displacement field setting in the Wave equation node can be any of the mentioned material models. If the required parameters are not directly available in the material, the parameters are created using a synchronization rule.

NEW CURL SHAPE FUNCTION

The Nédélec element of the second kind is implemented, having full polynomial orders in all directions for each field component. This can, for some simulations, make the problem solve for lower shape orders or with coarser meshes, and it can also make the resulting fields look smoother in postprocessing.

New and Updated Models in Version 5.6

THERMOSTRUCTURAL EFFECT OF A CAVITY FILTER

This example investigates the electrical performance of a cascaded cavity filter operating in the millimeter-wave 5G band with temperature changes. The thermal variations result in structural deformations of the structure. Thus, the resonant frequencies of the filter elements (cavities) are affected by thermostructural phenomena. S-parameters are computed with different thermal expansion configurations.

BASIC EMISSION AND IMMUNITY ANALYSIS OF A CIRCUIT BOARD

This example provides a walkthrough of how to simulate the basic radiated emission of a printed circuit board and its immunity response from outside noise. First, when one of the microstrip lines is excited, the crosstalk to an adjacent printed line and the radiated field, through an incomplete shield of a plastic enclosure, are computed for the emission analysis. The second study reviews the impact on the circuit board when there is the radiation from an antenna located outside the enclosure.

FAST ASYMPTOTIC RADAR CROSS-SECTION ANALYSIS OF A CONDUCTIVE SPHERE

This example uses asymptotic techniques to study the radar cross-section (RCS) response of a conductive sphere. The selected physics interface transforms the incident plane-wave field on the boundaries to the far field using the Stratton–Chu formula. The computed results are compared to the well-known asymptotic RCS value of a conductive sphere in the optical region.

MODELING OF A PHASED ARRAY ANTENNA

This example shows how to design a phased array with beam scanning functionality based on the arithmetic phase difference between the array elements. The initial complicated model can be reduced to a simple single unit cell model with periodic conditions to make the analysis faster and more efficient. Two-phased array designs built with microstrip patch antennas are studied and they are in good agreement of the antenna gain.

Rotordynamics Module

New Functionality in Version 5.6

MULTI-SPOOL BEARING

Using the new feature **Multi-Spool Bearing**, it is possible to model vibrations in coaxial rotors running at different speeds. This feature models the intershaft bearing between two coaxial shafts. Such rotors are common in steam turbines in power plants, where the system consists of a sequence of turbines having coaxial rotors running at different speeds. Another example of multispool rotors is in turbofan engines, where an inner shaft having a low-pressure turbine drives the fan, and an outer shaft having a high-pressure turbine drives the compressor.

SQUEEZE FILM DAMPER

Squeeze film dampers are components that provide additional damping to rotating systems through squeezing of the fluid film. Often, these components are used together with rolling element bearings, which do not offer enough damping on their own. Fluid film dampers are, however, also used together with hydrodynamic bearings. There are two ways to model squeeze film dampers:

- By solving Reynolds equation for the pressure distribution in the film, thus computing the net reaction forces and moments of the damper. For this type of modeling, the new feature **Squeeze Film Damper** is provided in the Hydrodynamic Bearing interface.
- By using an analytical expression for the forces and moments of the damper obtained using the short length approximation. For this type of modeling, you can add a **Squeeze Film Damper** subnode to the **Journal Bearing** and **Radial Roller Bearing** nodes in the Solid Rotor and Beam Rotor interfaces.

DYNAMIC COEFFICIENTS CALCULATION FOR HYDRODYNAMIC BEARINGS

Dynamic coefficients (that is, equivalent linear stiffness and damping coefficients about an equilibrium location of the journal) can now be computed. The computation is enabled by selecting **Calculate dynamic coefficients** in the settings for the Hydrodynamic Bearing interface. This is useful for bearing design where it is necessary to limit the cross-coefficients in the bearing to avoid instability. Also, dynamic coefficients can be used to run simplified rotor simulations to speed up the process.

DEFAULT GEOMETRY PLOT FOR BEAM ROTOR INTERFACE

A new default plot for the geometry of the rotor has been added in the Beam Rotor interface. This helps in visualizing the location of the various components while analyzing the results.

- A disk is shown as a thin circular plate.
- A radial bearing is shown as a cone in the radial direction pointing toward the rotor.
- Axial bearings at the end of the rotor are represented by cones in the axial direction. If an axial bearing is located at the interior of the rotor, a small disk representing the collar of the bearing is drawn together with two cones pointing toward the collar from both sides.

There is also an option in the interface settings to include this graphical representation in the stress and whirl plots.

CHANGE ROTOR SPEED FEATURE IN BEAM ROTOR INTERFACE

A new feature, **Change Rotor Speed**, has been added to the Beam Rotor interface to allow modeling multispool rotors that are running at different angular speeds.

FILM BOUNDARY CONDITION IN HYDRODYNAMIC BEARING

When cavitation is not modeled in detail, a new section, **Film Boundary Condition**, is added for **Hydrodynamic Journal Bearing**, **Floating Ring Bearing**, and **Hydrodynamic Thrust Bearing** nodes. There are two options: **Sommerfeld** and **Gümbel**. The **Sommerfeld** condition works as before, so that the full film is used for the net force computation. The **Gümbel** condition considers only the part of the film where the pressure is positive for the net force computation. This facilitates approximate modeling of the cavitation effect in the film.

AUTOMATIC SETTINGS FOR CAMPBELL PLOT

In earlier versions, to get the proper Campbell plot, it was necessary to select the **Axis Source Data** as **Outer solution** for all the nodes in the Campbell plot manually. This is now handled automatically and no further manual settings are needed to get a proper Campbell plot.

FATIGUE ANALYSIS IN SOLID ROTOR INTERFACE

In the Fatigue interface, it is now also possible to perform fatigue analysis based on results from the Solid Rotor interface. Fatigue analysis requires the Fatigue Module.

NEW POSTPROCESSING VARIABLES IN HYDRODYNAMIC BEARING INTERFACE

Several new postprocessing variables have been added in the Hydrodynamic Bearing interface and are available in the results menus. These variables include minimum film thickness, maximum pressure, total viscous heat loss, journal eccentricity, journal relative eccentricity, and attitude angle.

New Models in Version 5.6

CRITICAL SPEED OF A DUAL ROTOR SYSTEM

An eigenfrequency analysis of a dual rotor system connected through a multispool bearing is performed in this model. Cross-exciting vibrations through the multi-spool bearings make the dynamic behavior of one shaft dependent on the dynamic behavior of the other. Computed critical speeds are compared with values given in the literature.

EVALUATION OF THE DYNAMIC COEFFICIENTS OF A PLAIN JOURNAL BEARING

When analyzing rotors, it is common that bearings are modeled through their effective dynamic coefficients about a static equilibrium position. This example demonstrates how to compute such coefficients for a plain journal bearing. Computed coefficients are also compared with analytical values obtained from the solution of the Reynolds equation for a short bearing approximation.

DAMPING COEFFICIENTS OF A SQUEEZE FILM DAMPER

Squeeze film dampers are components that provide additional damping to rotating machines. To simplify the modeling of a rotor assembly, squeeze film dampers are often modeled in terms of their damping coefficients. In this model, damping coefficients are computed for a short squeeze film damper and compared to analytical values.

SHAFT VIBRATION DUE TO GEAR RATTLE AND BEARING MISALIGNMENT

In a gearbox, vibrations due to rattling of the gears and bearing misalignments are well-known sources of noise. In this example, two shafts connected through a pair of gears are considered. The driver rotates at varying speed. Due to backlash in the gears, the teeth meshing becomes intermittent, thus causing vibration in the shafts. After some time, the driven shaft is loaded with a torque, making the teeth meshing in gears smooth. In order to analyze the effect of the misalignment on the rotor vibrations, a time-dependent analysis is performed for two cases. In the first case, all bearings are aligned with the shafts, and in the second case, one of the bearings has a small angular

misalignment. The results show larger torsional and axial vibrations in the shaft in the presence of misalignment.

Semiconductor Module

New Functionality in Version 5.6

MULTICOMPONENT WAVE FUNCTION

The Schrödinger Equation interface has been expanded to support multicomponent wave functions. This new capability allows the straightforward modeling of multiband systems and particles with spins.

Tensor Effective Mass

The Schrödinger Equation interface now supports tensor effective masses.

LORENTZ FORCE

This new feature for the Schrödinger Equation interface adds the contribution to the kinetic momentum from the magnetic vector potential.

ROTATING FRAME

This new feature for the Schrödinger Equation interface adds the additional contribution to the Hamiltonian in a rotating frame.

DISSIPATION

This new feature for the Schrödinger Equation interface adds dissipation for time-dependent systems.

FINITE VOLUME FORMULATION

- Improved accuracy of models with curved boundaries by switching from geometry normal vectors to mesh normal vectors.
- Improved electrostatic terminal boundary condition with nonuniform applied voltage by using the circumcenter operator on the boundary.
- Improved charge carrier constraint values on metal contacts by using the circumcenter operator on the boundary.

DRIFT-DIFFUSION FORMULATION

Expanded the drift-diffusion formulation for graded materials to include the effect of continuously varying effective density of states. (The quasi-Fermi level formulation already included this.)

IMPACT IONIZATION

Impact ionization: A new option of indirectly updating the generation rate is added, to combine with the **Previous Solution** node in the solver sequence to improve convergence by delaying the update of the generation rate. This allows the use of stationary study steps instead of the time-dependent study step, which was required in previous versions.

MODEL UPDATES

- The model `gaas_pin_photodiode` has been updated, using global evaluations to demonstrate the consistency between the input optical power and the output electrical current (conservation of particles).
- The model `pin_reverse_recovery` has been updated to take advantage of the improved circuit interface.
- A fully analytic formula for the Thomas–Fermi approximation is added to the model `gross_pitaevskii_equation_for_bose_einstein_condensation`.

CORRECTIONS AND BACKWARD COMPATIBILITY

The safeguard against square roots of negative numbers is updated.

New Models in Version 5.6

K • P METHOD FOR STRAINED WURTZITE GAN BAND STRUCTURE

This benchmark model computes the valence band structure of an unstrained and a strained bulk GaN wurtzite crystal, as a tutorial for users who wish to set up multiple wave function components with the Schrödinger Equation interface. The model follows the formulation given in the reference paper by Chuang and Chang. The diagonal and off-diagonal elements of the Hamiltonian matrix are entered using built-in features, with detailed instructions in the model documentation. The computed eigenvalues agree well with the analytic solution and Fig. 5 in the paper.

A SILICON QUANTUM DOT IN A UNIFORM MAGNETIC FIELD

This tutorial model solves a two-component Schrödinger equation for the eigenstates of a simple silicon quantum dot in a uniform magnetic field, based on the paper by Jock and others on the topic of spin-orbit qubits. The built-in domain condition Lorentz Force for the Schrödinger Equation interface is used to account for the contribution to the kinetic momentum from the vector potential. The coupling of the spin-up and spin-down components is implemented using the built-in domain condition Zeroth Order Hamiltonian. Together with the benchmark model `k dot p Method for Strained`

Wurtzite GaN Band Structure, these examples show how to set up multiple wave-function components with the Schrödinger Equation interface. The computed probability density and kinetic momentum density of the ground state compare well with Supplementary Figure 1 in the paper. In addition, the computed energy difference between the first two eigenstates agrees well with the expected value from an intuitive analytic calculation.

VORTEX LATTICE FORMATION IN A ROTATING BOSE-EINSTEIN CONDENSATE

This tutorial model solves the Gross–Pitaevskii equation for the vortex lattice formation in a rotating Bose–Einstein condensate bound by a harmonic trap. The equation is essentially a nonlinear single-particle Schrödinger equation, with the interparticle interaction represented by a potential energy contribution in proportion to the local particle density. The time evolution in the rotating frame with phenomenological damping is configured with built-in features of the Schrödinger Equation physics interface. Nucleation of vortices is seen starting at the periphery of the condensate. Subsequently, the system goes through a period of spectacular dynamical instability before settling down in the low-energy state of a vortex lattice. The Optimization Module is used for parameter estimation from the numerical results. The time scales of the initial oscillation and the eventual collapse of the ellipticity parameter agree well between simulation and the experimental data published by Madison and others.

Structural Mechanics Module

New Functionality in Version 5.6

EMBEDDED REINFORCEMENTS

A new multiphysics coupling **Embedded Reinforcement** has been added. It can be used to insert edge elements from the Truss or Beam interfaces, as well as surface elements from the Membrane interface, into a domain modeled with Solid Mechanics. The mesh in the lower-dimensional interface is independent of the domain mesh. The purpose is to model structures like embedded reinforcements, sensors, etc.

The connection can be fully rigid or flexible. You can also model debonding between the embedded element and the surrounding solid.

FERROELECTROELASTICITY MULTIPHYSICS INTERFACE

The new **Ferroelectroelasticity** multiphysics interface is intended for analysis of ferroelectric materials. These materials exhibit nonlinear piezoelectric properties. This interface will add Solid Mechanics and Electrostatics interfaces together with the new **Electrostriction** multiphysics coupling. In Electrostatics, the new **Charge Conservation, Ferroelectric** material model is used. In this material, it is possible to model, for example, hysteresis using a Jiles–Atherton model.

This multiphysics interface requires the MEMS Module or the AC/DC Module.

ELECTROSTRICTION MULTIPHYSICS INTERFACE

With the new **Electrostriction** multiphysics interface, you can model electrostriction phenomena; that is, a situation where a strain proportional to the square of the polarization is induced by an electric field. This interface will add Solid Mechanics and Electrostatics interfaces together with the new **Electrostriction** multiphysics coupling. In Electrostatics, the standard **Charge Conservation** material model is used.

This multiphysics interface requires the MEMS Module or the AC/DC Module.

TRANSITION MULTIPHYSICS COUPLINGS FOR PIPE MECHANICS

The new multiphysics coupling **Structure-Pipe Connection** has been added for facilitating transitions from the Pipe Mechanics interface to the Shell and Solid Mechanics physics interfaces. The purpose is to switch between the line representation

in the Pipe Mechanics interfaces to a more accurate 3D representation where there are transitions in the pipe geometry.

FLUID-PIPE INTERACTION, FIXED GEOMETRY MULTIPHYSICS INTERFACE

The new Fluid-Pipe Interaction, Fixed Geometry interface connects the Pipe Flow and Pipe Mechanics interfaces, in order to transfer fluid loads to the structural analysis. The new **Fluid-Pipe Interaction** multiphysics coupling is used. Internal pressure, drag forces, loads on bends, and loads on junctions can be taken into account.

This multiphysics interface requires the Pipe Flow Module.

SPRINGS AND DAMPERS CONNECTING POINTS

In all structural mechanics interfaces, a new feature called **Spring-Damper** has been added. With it, you can connect two points with a spring and/or a damper. The points can be geometrical points, but they can also be abstract through the use of, for example, attachments or direct connections to rigid bodies. The spring can either be physical, with a force acting along the line between the two points, or described by a full matrix, connecting all translational and rotational degrees of freedom in the two points. The feature also makes it possible to connect a spring between points in two different physics interfaces.

CRACK MODELING

The new **Crack** node in Solid Mechanics can be used for different types of fracture mechanics simulations. When applied to an internal boundary, the two sides become disconnected. A crack can either be ideal or modeled with separate faces in the geometry. Through the addition of the **J-integral** subnode, it is possible to compute J-integrals and stress intensity factors in 2D, 2D axisymmetry, and 3D. A special **Face Load** subnode is available for prescribing a load on the crack faces.

PORT BOUNDARY CONDITION FOR ELASTIC WAVE PROPAGATION

The new **Port** boundary condition, available with the Solid Mechanics interface, is designed to excite and absorb elastic waves that enter or leave solid waveguide structures. A given **Port** condition supports one specific propagating mode. Combining several Port conditions on the same boundary allows a consistent treatment of a mixture of propagating waves; for example, longitudinal, torsional, and transverse modes. The combined setup with several Port conditions provides a superior nonreflecting condition for waveguides to, for example, a perfectly matched layer (PML) configuration or the Low-Reflecting Boundary. The Port condition supports

S-parameter (scattering parameter) calculation, but it can also be used as a source to just excite a system. The power of reflected and transmitted waves is available in postprocessing. To compute and identify the propagating modes, the Boundary Mode Analysis study is available in combination with the Port conditions.

GENERALIZED PLANE STRAIN WITHOUT BENDING

When using generalized plane strain for 2D Solid Mechanics, it is now possible to choose that the out-of-plane strain is limited to be constant over the cross section. You control this behavior through the new **Enable out-of-plane bending** check box in the **2D Approximation** section in the settings for the Solid Mechanics interface.

NEW OPTION FOR PRESCRIBING ROTATING FRAME SPEED

In the **Rotating Frame** node in the Solid Mechanics and Multibody Dynamics interfaces, the new option **Rigid body** has been added. With this option, you enter a time-dependent torque around the axis of rotation, and the rotational velocity is computed by integration of the rigid body equation of motion.

ACTIVATION OF MATERIAL IN THE TRUSS INTERFACE

It is now possible to activate and deactivate a material in the Truss interface by adding the **Activation** subnode under **Linear Elastic Material**. This can be used to simulate construction processes where bars are added at different stages.

RIGID CONNECTOR IMPROVEMENTS

The **Rigid Connector** features have a set of improvements:

- In the Shell and Beam interfaces, the selection alternatives have been extended also to the top level (that is, boundaries and edges, respectively).
- When the center of rotation is defined by a point selection, the point no longer has to be part of the physics interface itself.
- It is possible to couple rigid connectors from different physics interfaces, thus defining a new type of virtual rigid object. This selection resides in the **Advanced** section of the settings for the rigid connector.
- In the Solid Mechanics, Shell, and Beam interfaces, it is possible to automatically generate rigid connectors from RBE2 elements in an imported file in the Nastran® format. This is controlled from a section named **Automatic Modeling** in the settings for these interfaces. Rigid connectors created can belong to several physics interfaces in order to mimic the connections in the imported file.

UNIDIRECTIONAL CONSTRAINTS FOR BEAMS AND PIPES

In the Beam and Pipe Mechanics interfaces, the **Prescribed Displacement/Rotation** node has been augmented with an option called **Limited Displacement**. When this option is selected, you can define an upper and lower limit for the displacement. In particular, this facilitates the modeling on beams resting on unidirectional supports.

This option is available for constraints on points, as well as for constraints on edges.

IMPROVED DYNAMIC CONTACT

New algorithms have been introduced in order to improve the conservation of momentum and energy during transient contact events. These can be activated by selecting either the **Penalty, dynamic** or **Augmented Lagrangian, dynamic** formulation in the **Contact** node.

MORE OPTIONS FOR PENALTY FACTORS IN CONTACT

When using the penalty method for contact analysis, there are three new options in the **Contact** and **Friction** nodes for controlling the value of the penalty factor: **Automatic, soft, Manual tuning**, and **Nonlinear**.

When using the augmented Lagrangian method with a preset penalty factor, there is one new option: **Bending**. The purpose is to provide a softer penalty factor that can stabilize the iterations for bending-dominated problems.

FULLY COUPLED SOLUTION METHOD FOR AUGMENTED LAGRANGIAN METHOD IN CONTACT

It is now possible to use a fully coupled solver together with the augmented Lagrangian contact algorithm. This makes it easier to set up solver sequences and may also improve stability and convergence for some problems. The new formulation, allowing a fully coupled solver, is selected in the **Solution method** list in the settings for the **Contact** node.

USER DEFINED FRICTION MODEL

In the **Friction** node under **Contact**, you can now also select **User defined** as the **Friction model**. In this case, you directly enter an expression for the tangential force that causes sliding in terms of any other variables.

FRICTION WITH A KNOWN SLIDING VELOCITY

In some applications, the direction of the friction force is known, because two objects are sliding on each other with a given relative velocity. In this case, it is not necessary to solve for the friction forces. Such an approach can speed up the solution

significantly. The new **Slip Velocity** node, added under **Contact**, can be used when modeling such cases. The inputs for this node are similar to those of the **Friction** node.

WEAR MODELING

Methods for modeling mechanical wear have been added. This is controlled from the new **Wear** node, a subnode under **Contact** available in the Solid Mechanics and Multibody Dynamics interfaces.

There are two algorithms for wear modeling. In the more general, allowing arbitrary geometry changes due to the wear, a deformed geometry approach is used. There is also a simplified approach in which the wear is incorporated into the offset in the contact gap.

COMPUTING ROTATION FROM DISPLACEMENTS

In physics interfaces that have displacements as degrees of freedom, it is sometimes necessary to study the rotation of a region. To do that, you can add the new **Average Rotation** node, which uses the displacements of a set of points to determine a rotation. If more than three points are given, a least squares scheme is applied to make the best approximation of a rigid body rotation for the entire set of points. In addition, angular velocities and accelerations can be computed.

EVALUATING LOCAL BUCKLING IN TRUSS STRUCTURES

A set of variables, representing the safety margin against local buckling in truss structures, has been added. You provide input data for such an evaluation in the new section **Local Buckling** in the settings for the **Cross-Section Data** node.

NEW VISCOELASTICITY MODELS

Two new viscoelasticity models have been added: Maxwell and Generalized Kelvin–Voigt.

The Maxwell material can be considered as a type of liquid, since its long-term deformation under a constant stress is unbounded.

The Generalized Kelvin–Voigt model has a Prony series representation with several time constants. Conceptually, it consists of a set of Kelvin elements (spring and dashpot elements in parallel) connected in series.

FRACTIONAL DERIVATIVE VISCOELASTICITY

For frequency-domain analysis, all the viscoelasticity models (Generalized Maxwell, Generalized Kelvin-Voigt, Maxwell, Kelvin-Voigt, Standard linear solid, and Burgers) have been augmented by a fractional derivative representation.

Using a fractional time derivative representation makes it easier to fit material data to experiments for some materials.

NEW TEMPERATURE SHIFT FUNCTION IN VISCOELASTICITY

The Tool–Narayanawamy–Moynihan shift function is commonly used to describe the glass transition temperature in glasses and polymers. It has been added to the set of shift functions in the Viscoelasticity node.

PERFORMANCE IMPROVEMENTS FOR VISCOELASTICITY

For time-domain analysis using the Generalized Maxwell and Standard Linear Solid viscoelastic models, performance has been improved by up to one order of magnitude. The viscoelastic stresses are no longer represented by ordinary shape functions but with state variables in the Gauss points.

DIRECT INPUT OF SECTION STIFFNESS IN BEAM

A new material model, **Section Stiffness**, has been introduced in the Beam interface. Here, you directly enter data such as axial stiffness and bending stiffness. The purpose of this material model is to facilitate modeling of complex beam cross sections where the stiffness properties cannot be deduced from a single elastic material and the geometrical properties of the cross section. This is the case for composite beams, for example.

TIMOSHENKO THEORY AS DEFAULT FORMULATION IN PIPE MECHANICS

The default beam theory formulation used in the Pipe Mechanics interface has been changed from Euler–Bernoulli to Timoshenko. The latter formulation, which includes shear deformations, is more relevant for the majority of piping systems.

ANALYSIS OF LARGE DEFORMATION PROBLEMS IN AN UPDATED FRAME

When analyzing problems with large deformations, it is now possible to do so in a manner where the material frame is updated between each parameter value. A deformed geometry approach is used, and the deformation implicit in the transformation between the two frames is injected as a prestrain in the material model in the updated geometry. To enable this functionality, you add an **Elastic Predeformation** node.

STRAIN FIELD CONSISTENT INTERPOLATION OF INELASTIC STRAIN CONTRIBUTIONS

Several physical interactions, for example, thermal expansion and hygroscopic swelling, cause inelastic strains that are incorporated in the material models. If such strain fields vary too much over an element, spurious artifacts can be seen in the resulting stress fields. In version 5.6, this is taken care of by a reinterpolation, so that the external strain field has a polynomial order that matches the total strains computed from the displacements. In particular, this means that in a multiphysics analysis, the discretization orders in the two participating interfaces can be chosen independently, without considering consistency.

As an effect, the Thermal Stress, Solid and Joule Heating and Thermal Expansion multiphysics interface no longer reduce the discretization order of the Heat Transfer in Solids interface.

NEW WEAK EQUATION FOR GEOMETRIC NONLINEARITY

The weak contribution used for building the stiffness matrices in a geometrically nonlinear study step has been changed for some of the material models in the Solid Mechanics interface. The new formulation, which is based on 1st Piola–Kirchhoff stresses and variation of the deformation gradients, results in shorter assembly times.

HARMONIC PERTURBATION FOR GRAVITY

The **Gravity** node in the structural mechanics interfaces now has the **Harmonic Perturbation** option. This is useful when a support excitation is transformed into a frame acceleration.

SOLVER SETTINGS CONTROL FOR ELASTIC WAVES

In the settings for the Solid Mechanics interface, a new section **Transient Solver Settings** has been added. The purpose is to generate appropriate solver settings when analyzing wave propagation problems in the time domain.

DEFAULT PLOT IMPROVEMENTS

There are some changes in the default plot generation:

- When **Safety** nodes are present in one of the structural mechanics interfaces, default plots showing failure indices will automatically be created.

- The default plots in the Plate interface make use of the new **Embed 2D** dataset. This makes it possible to show a 3D representation of, for example, the out-of-plane displacements. Also, default load plots have been added using this technique.
- The default plots for **Layered Linear Elastic Material** in the Shell and Membrane interfaces have been improved.

ONE-WAY FLUID-STRUCTURE INTERACTION STUDIES REMOVED

Studies of the one-way fluid-structure interaction type can no longer be added from the **Add Study** wizard. This does not affect studies already added in existing models. The documentation has been augmented with instructions for how to set up such studies manually for different use cases. This allows for more flexibility than the previous predefined studies could provide.

DISABLING OF WARPING EQUATIONS IN SOLID-BEAM CONNECTION

When the **Solid-Beam Connection** multiphysics coupling is used with the **Solid boundaries to beam points, transition** option, it is now possible to suppress the equations used for computing the warping displacement on the boundary of the solid. This can simplify solver setup for cases when the warping is negligible or nonexistent, for example, on circular sections.

IMPROVED PRESSURE LOAD FOR SHELLS

When using a pressure load in the Shell interface under geometric nonlinearity, changes in area due to local stretching are now taken into account.

VISCOUS DAMPING IN LAYERED LINEAR ELASTIC MATERIAL

The possibility to add viscous damping has been added to the **Damping** subnode under the **Layered Linear Elastic Material** in the Shell interface.

IMPROVED SYMMETRY AND ANTISYMMETRY CONDITIONS IN MEMBRANE

The **Symmetry** and **Antisymmetry** conditions in the Membrane interface now provide the possibility to select a coordinate system for more exact determination of the symmetry plane. The options are identical to what is already present in the corresponding nodes in the Shell interface.

CHANGES IN GEOMETRIC NONLINEARITY FOR BEAM AND PIPE MECHANICS

In the Beam and Pipe Mechanics interfaces, there is no formulation for prestressed dynamic analysis. In previous versions, an attempt to do such an analysis was stopped with an error message. This has been changed to a warning, since there are situations

where such an analysis is meaningful. This is the case when the only source of stress stiffening comes from another physics interface, and it is thus now possible to conduct such studies.

ITEMS MOVED IN CONTEXT MENUS

Some less used features in the structural mechanics interfaces have been moved down into the **More** submenus in the context menus in order to reduce the size of the menus. These items are:

- **Initial Values**
- **Change Thickness**
- **Cell Periodicity**
- **Low Reflecting Boundary**

In addition, some new features reside in the **More** submenus:

- **Elastic Predeformation**
- **Crack**
- **Port**

OPTION FOR SOLVING FOR OUT-OF-PLANE STRAINS IN SHELL MOVED

The check box **Solve for out-of-plane strain components** in the Shell and Plate interfaces has been moved from the interface level to the settings for the individual material models.

CHANGED SHAPE FUNCTIONS IN AXISYMMETRIC SHELL

In the Shell interface in 2D axisymmetry, the shape functions for the out-of-plane strains are now of the type Discontinuous Lagrange.

POINT SELECTION CHANGED FOR BEAM END RELEASE

The point selection in **Beam End Release** has been changed so that it only allows selection of a single point. This will not introduce a limitation, since a selection of multiple points was not meaningful.

New and Updated Models in Version 5.6

BRACKET — GENERAL PERIODIC DYNAMIC ANALYSIS

This is a new example in the tutorial series showing various types of analyses of a bracket. In this case, the loading is periodic, but not harmonic. The approach used

consists of computing a Fourier series representing the excitation, and then performing a frequency sweep using the coefficients of the Fourier series as the frequency-dependent loads. The set of harmonic solutions are superimposed to give the time-domain solution for one period of the excitation. Finally, a modal time-dependent analysis over a large number of cycles is also performed, and the results of the two approaches are compared.

DISC BRAKE PAD WEAR

In this example, wear of the friction material in a disc brake is studied. Quasistatic friction forces are prescribed from simple kinematic considerations. The geometry of the brake pad is continuously updated to account for the material removal due to wear following the well-known Archard equation. Both the contact pressure and the wear rate changes over time as an effect of the updated pad geometry.

IMPACT BETWEEN TWO SOFT RINGS

In this conceptual example, the soft impact between two elastic rings is modeled using the Solid Mechanics interface. One of the rings is given an initial velocity to initiate the impact event. Both rings are unconstrained and not subjected to any external forces. Contact is modeled using a viscous formulation with both the penalty and augmented Lagrangian techniques. The conservation of linear and angular momentum as well as energy is verified.

ANGLE CRACK EMBEDDED IN A PLATE

This example reproduces a NAFEMS benchmark in which a plate with an angle crack is subjected to tensile loading. The J-integral is calculated and the stress intensity factors for Mode I and Mode II are compared to the benchmark results for several values of the angle.

SURFACE CRACKED CYLINDER

In this benchmark example, a semi-elliptical crack at the inner surface of a cylinder is studied. The inside of the cylinder and the crack faces are subjected to a pressure load. The J-integral is calculated along the crack front, and the stress intensity factor is then compared with the reference values.

MECHANICAL MULTIPOINT SYSTEM: ELASTIC WAVE PROPAGATION IN A SMALL ALUMINUM PLATE

In this tutorial, the vibrational behavior of a small aluminum plate with four waveguide structures is analyzed. This is an example of a structural component located in a device where elastic waves are propagating, like a smart speaker, an electric motor, or a MEMS

device. The plate can be thought of as a mechanical multiport system. The model uses the Port boundary condition at the inlet/outlet of the waveguide structures. The Port conditions consistently capture and treat the different propagating elastic modes like longitudinal, transverse, and torsional waves. The transmission and reflection of the various modes are characterized through the scattering matrix of the system, which is computed automatically. This allows the vibrational behavior of the component to be characterized in great detail, for example, for subsequent use in a system simulation.

HYSTERESIS IN PIEZOELECTRIC CERAMICS

Many piezoelectric materials are ferroelectric. Ferroelectric materials exhibit nonlinear polarization behavior such as hysteresis and saturation at large applied electric fields. In addition, the polarization and mechanical deformations in such materials can be strongly coupled due to the electrostriction effect. This model uses the Ferroelectroelasticity interface to analyze a simple actuator made of PZT piezoelectric ceramic material, which is subjected to an applied electric field and mechanical load.

SINGLE EDGE CRACK

This is a significantly updated version of an existing model, where an edge crack in a plate is studied. The new **Crack** and **J-integral** nodes are used, and different evaluation options are compared.

Subsurface Flow Module

New Functionality in Version 5.6

NEW SHALLOW WATER EQUATIONS, TIME EXPLICIT INTERFACE

The **Shallow Water Equations, Time Explicit** interface is used for modeling flow below a free surface under the condition that the horizontal length scale in the model is much greater than the vertical depth between the free surface and the bottom topography. The physics interface solves the shallow water equations obtained by depth averaging the Navier–Stokes equations. The dependent variables are the water depth and water flux. The interface is based on the discontinuous Galerkin method (dG-FEM) and uses a time-explicit solver.

NEW ADD-IN FOR CREATING DISCRETE FRACTURE NETWORKS (DFN)

Fracture networks are important for groundwater flow, geothermal and petroleum applications, nuclear waste storage, and more. A new add-in method is available for creating discrete fracture networks (DFN) in realistic geometries, such as of reservoirs and rock formations. The add-in works for both 2D and 3D geometries.

REVAMPED POROUS MEDIA FEATURES FOR TRANSPORT OF DILUTED SPECIES

The **Transport of Diluted Species in Porous Media** interface, used to study the flow and chemical composition of fluids moving through the interstices of a porous medium, is revamped to use the new **Porous Material** node.

Two new domain features, the **Porous Medium** and the **Unsaturated Porous Medium** nodes, are available in the **Transport of Diluted Species in Porous Media** interface and **Transport of Diluted Species** interfaces and include support for using the new **Porous Material** node for assigning material properties to the multiple phases in a porous medium. The new nodes have subnodes to define the properties for the **Liquid**, **Gas**, and **Porous Matrix**. The subnodes follow the same structure as the subnodes available in the **Porous Material**.

The check boxes **Dispersion** and **Volatilization in partially saturated porous media** have been removed from the **Transport Mechanisms** settings. This functionality is now available in the subnodes of the new **Porous Medium** and **Unsaturated Porous Medium** nodes.

The new nodes replace the previously available **Porous Media Transport Properties** and **Partially Saturated Porous Media** nodes. These nodes are kept when opening a model created in a previous version.

REVAMPED POROUS MEDIA FEATURES FOR HEAT TRANSFER

The **Porous Medium** feature is now always available without having to select the **Heat transfer in porous media** check box, which has been removed.

The **Porous Medium** node has been revamped. The solid and fluid phases are now handled in dedicated subnodes, namely the **Porous Matrix** and **Fluid** nodes. The computation of the effective material properties is handled in the **Porous Medium** node, whereas the solid and fluid properties are defined in the **Porous Matrix** and **Fluid** subnodes. It is also possible to account for an additional immobile fluid phase by adding the **Immobile Fluids** subnode under the **Porous Medium** node.

In the new **Porous Medium** node, the settings are homogenized with those of the porous media features in the fluid flow and chemical species transport interfaces. In particular, the **Porosity** is specified instead of the **Volume fraction** of the solid phase. In addition, it is possible to set either the **Dry bulk properties** or the **Solid phase properties**, depending on the availability of these material properties.

The new design allows to take advantage of the new **Porous Material** functionality under the **Materials** node, in which the porosity and volume fractions may be set, and where material properties are set in subnodes dedicated to each phase: **Solid**, **Fluid**, and **Immobile Fluid**.

IMPROVEMENTS FOR PHASE TRANSPORT

The Phase Transport interface now supports pseudo time stepping. Periodic Condition and Open Boundary conditions, available on boundaries exterior to the Phase and Transport Properties feature, have also been added. The Phase Transport interface now also provides improved support for deforming and rotating domains, including the Interior Wall boundary condition.

New Models in Version 5.6

ANALYZING POROUS STRUCTURES ON THE MICROSCOPIC SCALE

Modeling flow through realistic porous structures is difficult due to the complexity of the structure itself. Resolving the flow field in detail is not feasible in real-life applications. Therefore, macroscopic approaches are used, which utilize averaged

quantities of the porous structure, such as porosity and permeability. This example analyzes the flow field at the pore scale in detail.

DAM BREAKING ON A COLUMN

This transient model solves the shallow water equations to model the impact of a water wave on a column. A body of water with a height of 0.3 meters is initially contained behind a gate. At the start of the simulation, the gate is suddenly released and the body of water forms a wave moving toward the structure. After impacting on the structure, the water continues until it is reflected at the wall of the tank and impinges a second time on the column. The pressure force on the column is calculated and can be compared with experimental results.

Tsunami RUNUP ONTO A COMPLEX 3D BEACH, MONAI VALLEY

This example is an established benchmark case that models a scaled laboratory experiment of the tsunami runup in the Monai Valley in Japan. The benchmark focuses on a region near the shoreline for which detailed experimental data is available. The tank is initially filled with still water, and a known incident wave is imposed at one of the boundaries. The wave makes the shoreline move back and forth, eventually covering the small island in the middle of the domain.

FLOW IN A FRACTURED RESERVOIR

This model demonstrates how to model flow through a fractured reservoir. The reservoir is modeled with a discrete fracture network (DFN) where the fractures have a random distribution of position, size, orientation, and aperture.

Wave Optics Module

New Functionality in Version 5.6

SOLVING FOR FAST PORT SWEEPS

Use a new study step, Frequency Domain Source Sweep, to efficiently solve a frequency-domain study that is sweeping among ports calculating a full S-parameter matrix. The settings for this study step are similar to those for the Frequency Domain study step and much simpler than the traditional port sweep, which requires a parametric sweep step.

TIME TO FREQUENCY FFT STUDY FOR THE ELECTROMAGNETIC WAVES, TRANSIENT INTERFACE

Use the Time to Frequency FFT study type to first make a broadband transient study. Then the time-domain data is transformed to the frequency domain using a fast Fourier transform (FFT). This study type is now available for the Electromagnetic Waves, Transient interface.

POLARIZATION PLOT TYPE

The Polarization plot type depicts the polarization state for the different diffraction orders. It is available as a plot type in postprocessing and is used for default plots when periodic ports are included in the simulation.

SCATTERING BOUNDARY CONDITION CAN USE OBLIQUE ANGLE OF INCIDENCE IN MODE ANALYSIS

The Scattering boundary condition can now, for mode analysis, efficiently absorb waves having a wave vector composed of the mode's propagation constant directed tangentially to the boundary and a remaining normal component.

INPUT POWER TO CONTROL GAUSSIAN BEAM AMPLITUDE

For Gaussian beam background fields and input fields to the Scattering and Matched boundary conditions, the amplitude of the beam can be specified by providing the input power.

REFERENCE POINT CAN BE DEFINED BY A GENERAL EXPRESSION

The Reference point subfeature to Scattering and Matched boundary conditions can now also be specified from a general vector expression.

SYNCHRONIZATION OF MATERIAL PARAMETERS BETWEEN DIFFERENT RELATED MATERIAL PROPERTY GROUPS

The relative permittivity, refractive index, loss tangent, and dielectric loss material models can synchronize the material parameters between the groups. Thus, if a material is added and specified by the refractive index material property group, the Electric displacement field setting in the Wave Equation node can be any of the mentioned material models. If the required parameters are not directly available in the material, the parameters are created using a synchronization rule.

NEW CURL SHAPE FUNCTION

The Nédélec element of the second kind is implemented, having full polynomial orders in all direction for each field component. This can, for some simulations, make the problem solve for lower shape orders or with coarser meshes, and it can also make the resulting fields look smoother in postprocessing.

New and Updated Models in Version 5.6

THRESHOLD GAIN CALCULATIONS FOR VERTICAL-CAVITY SURFACE-EMITTING LASERS (VCSELS)

An eigenfrequency study is used to find the resonance frequency and threshold gain for an oxide-confined, GaAs-based, vertical-cavity surface-emitting laser (VCSEL).

The simulations are performed in two steps. A regular eigenfrequency analysis is first performed to find good initial values for the subsequent nonlinear eigenfrequency analysis.

The resonance frequencies and threshold gain, calculated for different device geometries, compare well to values from a paper collecting the results from different computational methods on this benchmark problem.

LEAKY MODES IN A MICROSTRUCTURED OPTICAL FIBER

A mode analysis study is used to find the complex effective indices for a microstructured optical fiber (MOF), consisting of air holes in a silica host. As the effective index is smaller than the refractive index of the silica background material, the modes are leaky.

The example demonstrates both the use of a perfectly matched layer (PML) and the use of the Scattering boundary condition, using an oblique angle of incidence, to truncate the simulation domain.

The real and imaginary parts of the effective index compare well to values from a published paper.

TIME DOMAIN TO FREQUENCY FFT ANALYSIS OF A DISTRIBUTED BRAGG REFLECTOR

This example demonstrates how to setup a Time Domain to Frequency FFT study for a distributed Bragg reflector (DBR) structure.

The results agree well with the results of a regular Frequency Domain study.

FABRY-PEROT RESONATOR

A Fabry–Perot resonator is a fundamental optical device and has a broad range of applications. Examples include measuring length, frequency, or wavelength and filtering specific spatial modes. This example uses the Electromagnetic Waves, Beam Envelopes interface in the bidirectional formulation to efficiently compute the mode shape and reflection/transmission properties of a macroscopic Fabry–Perot resonator.

This model replaces the Fabry–Perot Cavity model in the Wave Optics Module’s Application Library.

OPTIMIZATION OF A PHOTONIC CRYSTAL FOR SIGNAL FILTERING

This example builds on the Photonic Crystal example, where a photonic crystal structure is studied. This structure has a band gap, so only waves within a specific frequency range will propagate through the outlined guide geometry. This example changes the position of the pillars in order to maximize the ratio of the transmission at the desired frequency to the transmission at the undesired frequency.

SINGLE MODE FIBER-TO-FIBER COUPLING

Focusing a laser beam onto the tip of a single-mode fiber is a common way to couple light. To achieve a good coupling efficiency, the spatial mode of the light field has to match the spatial mode of the fiber. In this example, we use the Electromagnetic Waves, Beam Envelopes interface to compute a small free-space fiber-fiber coupling. The coupling efficiency depends on the position of the coupling lenses.

UPDATED MODELS

Application Library Path for Model Using the New Gaussian Beam Input Power Option

- Wave_Optics_Module/Nonlinear_Optics/self_focusing

Application Library Path for Models Using the New Curl Shape Function

- Wave_Optics_Module/Beam_Propagation/orbital_angular_momentum

Material Library

New and Updated Material Data in Version 5.6

In version 5.6, the following improvements have been made:

- The following materials are now available: CMSX-4 Plus (SLS); Cu-8Al; Cu-26Cr; Cu-8Cr-1Al; Cu-17Al-5Al; Cu-23Cr-5Al; P355N; P355NH; P355NL1; P355NL2; 15-15HS; AerMet 340; CarTech ACUBE 100; Hypocore alloy; MagneDur 20-4 alloy; H19; CBS-223 VIM-VAR; CarTech No. 1 JR alloy type 1, 2, 4; and Ti - 6 Al - 4 V, Grade 5.
- Stress-strain curves in compression have been added.
- S-N data has been added for 4140, 51SiCr7, 1008, 1020, 1040, 2315, 2330, 4330V, 8630, 8640, HY-TUF, Super HY-TUF, H23, Stellite 31, ZK60, AZ81, maple, and birch.
- Data has been added for DIROS 500 S, P500QL2, P690Q, A537 (S185), AerMet 100, AF1410, 1008, 1020, 1040, 4135, 4037, 5140, 9310, 1080, 4130, 300M, and glassy Se.
- More data for molten salts has been added.
- Isochronous creep data and creep data for several aluminum alloys have been added.
- The stress-rupture data for UNS C90500 was incorrect and has been removed.
- The test temperatures for the stress-rupture data for “Inconel 600 - solution treated” were incorrect and have been corrected.
- The stress-strain and % elongation data for magnesium alloy WE54 were incorrect and have been corrected.
- The data for Inconel C-276 and Hastelloy C-276 were combined, and Hastelloy C-276 was removed.
- The reference for the expansion of 2205 steel was changed. The new values are 5% to 10% higher than the old values.

LiveLink™ for Excel®

New Functionality in Version 5.6

MODEL PARAMETERS

Direct support for models that have more than one parameter node is added to the toolbar. This makes it easier to work with models that have a lot of parameters.

LiveLink™ for MATLAB®

New Functionality in Version 5.6

LiveLink™ for MATLAB® now supports MATLAB® 2020a and 2020b.

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

MPHNAVIGATOR

The `model.capeopen` node has been renamed to `model.thermodynamics`.

An extra node, `model.externalInterface`, has been added. It contains any settings for the Simulink® cosimulation export (which requires the new product LiveLink™ for Simulink®).

LAUNCHERS

COMSOL with MATLAB and *COMSOL with Simulink* are sharing the same launchers, which set the static Java class path as opposed to setting the dynamic class path in previous releases. This means that you can only use the `mphstart` command when working with LiveLink™ for MATLAB® and not when using LiveLink™ for Simulink®, which requires a static class path.

MPHSAVE AND MPHLOAD

These functions now support the use of tags of the model on the server instead of the model variable. This makes it easier to work with multiple models on the server.

You can save a model with the tags `Model` using this command

```
mphsave('Model')
```

You can retrieve the model variable for a model on the server using this command

```
model = mphload('Model')
```

MPHPLOT

Plots with transparency on some of the plot features are supported. Plot with images and material appearance are not supported in MATLAB. Themes and clipping are not supported.

MPHGEOM

Using

```
h = mphgeom(model, 'geom1')
```

you get a handle to the plotted objects, which can later be used for customization in MATLAB.

MATERIAL PARAMETERS

Material parameters from another MPH-file can now be loaded using a single line as opposed to having code that sets each material property line by line.

MPHINTERP

The new error message “Argument COORD has incorrect orientation or size” makes it easier to detect when coordinates are of the wrong size (or have been transposed).

Backward Compatibility with Version 5.4

The `mphreduction` function has been changed in version 5.5 such that it no longer can be used to define model reduction studies. You have to manually add the necessary studies to the model. The `mphreduction` function can then be used to extract the defined matrices for use in MATLAB.

The COMSOL API for Use with Java[®]

COMSOL 5.6 API Changes

MESH PART UNITS

The unit of the mesh part created by `createMeshPart` is now set to the unit of the source geometry. In earlier versions, it was created without a unit.

THE ADAPT FUNCTION — THE EXPRTYPE PROPERTY

For the `adapt` API function, the default for the `exprtype` property has changed: It is `error` if added in a component and a solution exists; otherwise, it is `size`. In previous versions, the default was always `error`.

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`.

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");
```

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-1: 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> |

TABLE I-1: DEPRECATED METHODS AND THEIR REPLACEMENTS

| DEPRECATED METHOD | NEW METHOD |
|---|--|
| <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> |
| <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. |

TABLE I-1: DEPRECATED METHODS AND THEIR REPLACEMENTS

| DEPRECATED METHOD | NEW METHOD |
|---|---|
| <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. |

This concludes the release notes for COMSOL Multiphysics version 5.6.

I n d e x

- A** AC/DC Module
 - new and updated models in 37
 - new functionality in 36
- Acoustics Module
 - backward compatibility 46
- Application Builder, new functionality in 10
- B** backward compatibility, general considerations 31
- Battery & Fuel Cells Module
 - backward compatibility 51
 - new and updated models 51
 - new functionality in 47
- C** CAD Import Module
 - new functionality in 52
- CFD Module
 - backward compatibility 59
 - new functionality in 56
 - new models in 57
- Chemical Reaction Engineering Module
 - backward compatibility 58, 66–67
 - new applications in 64
 - new functionality in 60
- Composite Materials Module
 - new functionality in 68
- COMSOL API changes 176
- COMSOL Compiler
 - new functionality in 13
- COMSOL Multiphysics, new functionality in 14
- COMSOL Server, new functionality in 14
- Corrosion Module
 - backward compatibility 77
 - new and updated models in 76
 - new functionality 73
- D** Design Module
 - new functionality in 52
- E** Electrochemistry Module
 - backward compatibility 82
 - new functionality in 79
- Electrodeposition Module
 - backward compatibility 87
 - new and updated models in 86
 - new functionality in 83
- F** Fatigue Module
 - new functionality in 88
- G** general new functionality 10
- Geomechanics Module
 - new functionality in 89
 - new models in 90
 - geometry and mesh, new functionality for 17
- H** Heat Transfer Module
 - backward compatibility 97–99, 101, 111
 - new applications in 96
 - new functionality in 91
- L** LiveLink for Excel
 - new functionality in 173
- LiveLink for MATLAB
 - backward compatibility 175
 - new functionality in 173–174
- LiveLink™ products for CAD
 - new functionality in 52
- M** Material Library
 - new and updated material data in 172
 - new material data in 172
- MEMS Module
 - new functionality in 103
- Metal Processing Module

- new functionality in 109
- Microfluidics Module
 - backward compatibility 112
 - new functionality in 111
 - new models in 110
- Mixer Module
 - new functionality in 113
- Multibody Dynamics Module
 - new functionality in 114
- N** Nonlinear Structural Materials Module
 - new functionality in 120
 - new models in 122
- O** operators, functions, and definitions,
 - new and updated 20
- Optimization Module
 - new functionality in 124
- P** Particle Tracing Module
 - backward compatibility 130
 - new applications in 130
 - new functionality in 127
- Pipe Flow Module
 - new functionality in 132
- Plasma Module
 - backward compatibility 133
 - new functionality in 133
- Porous Media Flow Module
 - new functionality in 134
 - new models in 137
- R** Ray Optics Module
 - backward compatibility 141
 - new applications in 140
 - new functionality in 138
- results and visualization, new functionality in 25
- RF Module
 - new and updated apps and models in 145
 - new functionality in 144
- Rotordynamics Module
 - new functionality in 147
 - new models in 149
- S** Semiconductor Module
 - new applications in 152
 - new functionality in 151
- Structural Mechanics Module
 - new functionality in 154
 - new models in 162
- studies and solvers, new functionality in 22
- Subsurface Flow Module
 - new functionality in 165
 - new models in 166
- W** Wave Optics Module
 - new functionality in 168
 - new models in 169