COMSOL Multiphysics

Release Notes

COMSOL

COMSOL Multiphysics Release Notes

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries 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/forum
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/videos
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM010001

Contents

Chapter 1: Release Notes

I

Release Notes

Cᴏᴍꜱᴏʟ Multiphysics® version 6.0 includes the all-new Model Manager, which provides simulation process and data management (SPDM) tools for managing your COMSOL modeling projects. There is also a new Uncertainty Quantification Module for performing uncertainty quantification (UQ) of parameters in COMSOL Multiphysics simulations. These *Release Notes* provide information regarding new functionality in version 6.0 for all COMSOL® software products.

New Products

Model Manager

The Model Manager provides all-new functionality for working with COMSOL projects. You can save drafts of model files, save versions of models with version control and comments, compare versions, add auxiliary files in project databases, search and tag models, and much more. This functionality makes the COMSOL software work as a part of SPDM or product lifecycle management (PLM).

The *Model Manager* is included in COMSOL Multiphysics version 6.0. The *Model Manager server* requires a floating network license and makes it possible to share database content. The Model Manager is not available with a class kit license.

See the new *Model Manager Reference Manual* and *Model Manager Server Manual* in the COMSOL documentation set for more information.

Uncertainty Quantification Module

The new Uncertainty Quantification Module adds functionality for studying UQ using the following analysis types:

- Screening, MOAT for Morris one-at-a-time screening
- Sensitivity analysis
- Uncertainty propagation
- Reliability analysis

Furthermore, you specify quantities of interest and input parameters for the UQ analysis. For the input parameters, you can choose from a number of distributions and specify parameters for those distributions, such as mean and standard deviation, and also add optional lower and upper bounds. For the results analysis, a number of UQ results quantities are available in tables, and a new Sobol Index plot is available for visualizing Sobol indices.

COMSOL Multiphysics

General New Functionality

UPDATED LICENSE SERVER

The FlexNet license server software has been upgraded to version 11.18. For that reason, update the COMSOL license server before you install and run version 6.0.

UPDATED SYSTEM REQUIREMENTS

For up-to-date information about the system requirements for version 6.0, including supported versions of Windows®, Linux®, and macOS operating systems, see https://www.comsol.com/system-requirements.

New Functionality in the Application Builder

Also see the *Introduction to the Application Builder*, *Application Builder Reference Manual*, and *Application Programming Guide* for more information.

GENERAL IMPROVEMENTS

- The **Main Window** feature includes a new **Prefer inner scrollbars** check box, selected by default. This setting controls where scrollbars appear when shrinking the height of the app window. If you run an app with the default setting and shrink the window, you will get a vertical scrollbar to the right of the input sections. If you clear the **Prefer inner scrollbars** check box and run the app again, then, when you shrink the window, you will instead get a vertical scrollbar to the very right.

- The icons included by default in the **Images** library have changed, and there are new icons (on disk) that can be used when making apps.

IMPROVEMENTS TO THE FORM EDITOR

Editor for the Main Window
You can now right-click the **Main Window** node and choose Edit to open the Main Window contextual toolbar. It contains menus and menu items for creating main window menus:

- The **Classic Menu** and **Ribbon** buttons in the **Menu Type** section can be used to switch the menu type of the application. The corresponding options in the **Main Window** settings window are updated accordingly. Depending on the chosen menu type, you

can add **Menu Bar** and **Toolbar** items in the **Classic Menu** section or **File Menu**, **Ribbon Tab**, and **Ribbon Section** items in the **Ribbon** section.

- **Open Menu** opens the selected menu, and **Close Menu** closes the selected menu if open or the selected item's parent menu otherwise. After closing, the menu that was closed is selected, if not selected already. This makes it possible to close a "tree of menus" by clicking the button multiple times.

- Click the **Item** and similar buttons in the **Items** section to create items of the correct type depending on the current selection (for example, toolbar items if a toolbar is selected and menu items if a menu is selected).

Gauge Form Object

A new Gauge form object is available as a way to display a scalar value in an app using a circular gauge with an arrow or needle pointing to the current value. Optional tick marks and labels indicate the range of values that the gauge can display. There are also color ranges along the scale. You can configure the colors.

More Customization to Buttons

- Outlined and flat button styles are now also available for small buttons.

- Toggle buttons can now have "radio button behavior".

- A new **Selected style** list with the options **Color highlight** and **None**. This setting makes it possible to skip the highlight in cases where the icon itself indicates the state. It is only available for large buttons using the flat or outlined style and is not available on Linux and macOS because apps use built-in buttons there.

Visibility for Forms in Form Collections

It is now possible to specify which form is visible in form collections. You can also change the visible and expanded states at runtime. The Settings window for **Form Collection** objects has also been updated and improved.

Error and Warning Column in Table of Toolbar Items

The table of toolbar items in the Settings windows of **Toolbar**, **Graphics**, and **Table** form objects now has a column that shows error or warning icons for items that have any error. For example, a toggle item that is missing a source.

Player Buttons in the Graphics Toolbar

When an animation is the source of a Graphics form object in an app, the toolbar will include player buttons for playing, stopping, and stepping in the animation.

IMPROVEMENTS TO THE METHOD EDITOR

Improved Debugger

The debugging tools have been significantly expanded and improved in version 6.0:

- A new **Variables** window that opens by default when a method is paused at a breakpoint. In the **Variables** window you can inspect and modify local variables, declarations, and parameters.

- A new **Call Stack** window that opens by default when a method is paused at a breakpoint. In that window, it is possible to select a stack frame to see the corresponding local variables in the **Variables** window.

- A new **Breakpoints** window that contains a list of all breakpoints in all methods in the application. You can open this window from the **Method** ribbon toolbar or from a new **Windows** menu in the **Home** ribbon toolbar.

- It is possible to enable and disable individual breakpoints, either from the **Breakpoints** window or by right-clicking on a breakpoint in the left margin in any method editor window.

- Two new debugger actions have been added:
  - **Step Out**: Step until the current method returns.
  - **Break**: Suspend method execution at the next reached line in the method code, as if there had been a breakpoint there.

Improved Method Support for Events

It is now possible to pass arguments to methods that are used in the **Events** section for form objects. You can now choose **Create Global Method**, **Create Form Method**, **Create Local Method**, and **Create Command Sequence** next to an **On data change** list, for example.

- Clicking directly on the plus sign corresponds to **Create Global Method**.

- Clicking **Create Global Method** or **Create Form Method** opens a dialog box where you can enter the name of the method to create. A method of the corresponding type is then created. It has a first argument called `newValue`, which is based on the form object type in the same manner that the `newValue` argument type is determined for local methods. The argument added to the method contains the new value in the form object when the event is triggered. The newly created method is also selected in the Application Builder tree and opened in the editor area.

- Clicking **Create Command Sequence** opens a **Create Command Sequence** dialog box where a command sequence to run when the event is triggered can be specified. The **Go to Method** button in the dialog box first performs an "OK operation" and then

selects the method in the Application Builder tree and opens the method in the editor area. When **OK** is clicked in the **Create Command Sequence** dialog box, a **Command sequence** item is added to the **On data change** list after **None**, and that item is selected.

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.

Methods for Calling Go to View Actions

The first method parameter in the following methods is a view orientation string from the set {"xy", "xz", "yx", "yz", "zx", "zy"}.

```
// Go to view in main graphics
goToView("xy");
// Go to view in the given graphics form object.
goToView("xz", "form1/graphics1");
// Go to view in the given graphics form object.
goToView("yz", app.form("form1").formObject("graphics1"));
```

Methods for Clearing the Logs

In apps with multiple **Log** or **Message Log** form objects, all **Log** and **Message Log** form objects are cleared of text. These methods are also supported in methods run from the Model Builder.

```
// Clears all logs
clearLog();
// Clears all message logs.
clearMessageLog();
```

New FormObject Methods

The following methods are now available for a `FormObject`:

```
void visible(String form, boolean visible)
```

sets the form's visible state, and

```
void expanded(String form, boolean expanded)
```

sets the form's expanded state when using the Sections type. There are also new Boolean `formvisible` and `sectionexpanded` properties for the visible state of forms in form collections and the expanded state of forms in form collection when using sections type, respectively.

New Functionality in COMSOL Compiler™

It is now possible to sign compiled apps on Windows.

New and Improved General Functionality in COMSOL Multiphysics

BETTER FILE SELECTION TOOLS

For opening ans saving files, you can now choose **Open From** and **Save To** on the **File** menu; **Insert Components From**, **Insert Sequence From**, **Insert Physics From**, **Import Materials From** from context menus for component and material nodes; and choose **Browse From** when choosing files to insert or import. All these actions open one of these new fullscreen windows for managing files:

- The **Open** window for opening files. You can choose among recently used files, add databases, and browse for files on the file system.

- The **Save** window for saving files. You can add databases and browse for a location to save the file on the file system.

- The **Select Model** window for selecting files for inserting components, physics, and materials. You can add databases and browse for model files on the file system.

- The **Select File** window for selecting a file to insert of import. You can choose among available files of different types (such as text files), add databases, and browse for files on the file system.

When insert or importing files, you can also choose (when applicable) **Show in Auxiliary Data**, to add it to the **Auxiliary Data** window, and **Copy File Location** and **Paste File Location** for copying the file location to the clipboard and pasting an already copied file location in any text field or document.

IMPROVED GRAPHICS AND RENDERING

- It is now easier to determine the number or name of a domain, boundary, edge, point, or geometry object. By default, the number or name now appears in the upper-left corner of the Graphics window when hovering over the geometry. There is also information about what is the up and down sides of interior boundaries in assembly geometries (requires OpenGL rendering). You can turn off this display by clearing the **Show information on hover** check box on the **Graphics Interaction** page in the **Preferences** dialog box.

- You can also control the amount of glow when hovering over the geometry. On the **Graphics Interaction** page in the **Preferences** dialog box, choose **Full glow effect** (the default), **Only glow lines**, **No glow**, or **Only lines** from the **Primary hover effect type** list.

- You can now add screen-space ambient occlusion (SSAO), which can improve the quality of images. Select the **Ambient occlusion** check box in the 3D **View** node's settings to include SSAO in the view. Ambient occlusion adds soft shadows from ambient and indirect lighting to the scene.

- For transparency plots, Fresnel transmittance is now available. In the F**resnel transmittance** field in the Transparency section of the 3D **View** node's settings, or using the slider underneath, you can specify the Fresnel transmittance as a value between 0 (no transmittance) and 1 (full transmittance). Fresnel transmittance results in that the transparency depends on the angle of incidence, which can improve realism.

- When clipping solid domains, you can now show a cross section as well as colorize it. It is also possible to highlight overlapping domains and to make the cross section transparent.

- By default, antialiasing is now enabled for transparency rendering. You can turn the antialiasing on or off using the **Enable antialiasing for on-screen transparency effects** check box on the **Graphics and Plot Windows** page in the **Preferences** dialog box.

- You can now zoom by scrolling. Either rotate the scroll wheel while holding down the CTRL key or rotating the scroll wheel while keeping it pressed.

IMPROVED RENDERING OF MATERIALS

Four new material types are now available for rendering of materials in the Graphics window: Brass, Aluminum (anodized), Steel (anodized), and Silicon. A **Color** list for quickly changing the color of a material is now also available for Aluminum (anodized), Steel (anodized), Car paint, and Leather. There are also settings for specifying a custom basis for the material coordinates and new options for brush lines.

NEW XML FORMAT FOR IMPORTING AND EXPORTING MATERIALS

There is a new XML format for importing materials into COMSOL Multiphysics, either using a dedicated import tool or as a material library in the **Material Browser**. The main purpose of this format is to provide a well-defined and documented XML syntax, simplifying collaboration. A material supplier, for example, can generate material libraries only with the aid of the documentation and examples that you generate by the export tool.

Use the new **Import Materials** and **Import Materials From** options to import materials from an MPH-file or a model XML file and insert it into the open model. Use the **Export Materials** option to export the materials under the selected node to a Model XML-file. You can use the export to see the correct XML syntax for the materials.

MERGING CHANGES IN COMPARISON RESULTS

For nodes that correspond to a node in the Model Builder tree, right-click the node and choose **Override Difference in Opened** to overwrite that difference in the open model. After this operation, the open model is altered to match the other model as closely as possible. When comparing two models that both can be altered, for example, two models saved on your file system, you can also right-click and choose **Override Difference in Remote File** to overwrite the difference in the other model instead. See the Model Manager documentation for information on merging model files with version information.

SAVING WITHOUT BUILT, COMPUTED, OR PLOTTED DATA

In the Settings window for the root node, there is a new **Save** section where you can specify if the model should be saved without built, computed, or plotted data. There are separate settings for saving to an MPH-file and to a database. If you choose to exclude such data, the model will be saved without the binaries for the geometry, mesh, solution, and postprocessing data.

There are also settings on the new **Save** page in the **Preferences** dialog box to handle the default for new models or old models that are opened.

IMPROVED DEFAULTS FOR PLOT GENERATION AND DATASETS

You can now right-click a **Study** node and choose **Show Default Plots** to create default plots for a study and **Reset Default Plots** to reset the plots related to the study to the defaults. These commands are also available in the **Solver** section on the **Study** toolbar. In both cases, nothing is computed. There is also a new **Generate default plots for intermediately stored solutions** check box in the Settings window for **Study** nodes. Select it to generate a plot group for each **Solution Store** node in the solver sequence of the study if such nodes appear after a **Compile Equations** node.

There is also a new setting on the **Results** page in the **Preferences** dialog box. Under **Dataset generation policy**, select the **Add datasets only for components solved for** check box to change the behavior of the dataset creation functionality and avoid creation of unnecessary datasets.

You can also create plots automatically when running COMSOL batch jobs from the command line using the new `-createplots` argument.

AUXILIARY DATA WINDOW

The **Auxiliary Data** window is a new window that contains a table of all files referenced by the current model with their properties, source locations, and status. It is available from the **Windows** menu in the **Main** toolbar. This window also includes auxiliary data files stored in a database when using the Model Manager.

VERSIONS WINDOW

The new **Versions** window is intended for managing versions of files in your modeling projects when using the Model Manager. You can open, run, compare, and restore versions of files in the **Versions** window. It is available from the **Windows** menu in the **Main** toolbar.

OTHER IMPROVEMENTS AND CHANGES

- There is a new preference for what unit system COMSOL will use for new models. In the **General** section of the **Preferences** dialog box, under **Unit system**, you will find the new **In new models** list. The unit system selected here will be used when a new model is created.

- In the Windows version, you can now insert ASCII characters in text fields by holding down ALT and then typing the character code using the number pad.

- The ribbon toolbar section previously called Physics Variables is now called **Physics Utilities**. **Physics Utilities** also now appears as a separate submenu in the context menu for **Definitions** nodes. They are no longer in the Variables Utilities submenu. The physics utilities are always visible (provided your license includes them) and no longer require that you activate **Variable Utilities** in the **Show More Options** dialog box.

- In the Model Builder toolbar, the **Expand All** and **Collapse All** buttons now include a menu that includes those actions and the new actions **Expand Selected** and **Collapse Selected**.

- The **Save recovery file** preference setting has been changed to **Keep recovery files when the application terminates unexpectedly** for clarity.

- Some new actions are available for the Quick Access Toolbar. They appear if you choose **More Commands** and open the **Quick Access Toolbar** dialog box. The new actions are **Model Manager**, **Model Builder**, **Application Builder** (to move to those workspaces), and **Refresh Add-ins**.

- The **Messages** window has got a new **Show on New Message** toolbar button. When selected, the **Messages** window will be brought on top when there is a new message. Previously, the **Messages** window was sometimes brought on top when there was a new message, but not in a way that you could control.

COMSOL API ADDITIONS

Setting and Reading the Model's Title and Description
New API commands for setting and reading the model's title and description:

```
model.title(title)
title = model.title()
model.description(description)
description = model.description()
```

Previously, the title and description were part of `model.comments()`.

Saving Without Computed Data
There are two new methods

```
model.excludeComputedDataInMph(<exclude>)
model.excludeComputedDataInMph()
```

With the first method, you can specify whether to exclude the computed data by providing a Boolean input variable. The second method returns the current value.

Blocking Other Clients
There is a new API command

```
ModelUtil.blockOtherClients(boolean blockOtherClients)
```

that blocks other clients from a server. This command is only meaningful when there are multiple clients connected to the server. Calling `ModelUtil.blockOtherClients(true)` will block other clients. The blocking is lifted when clients request the block call `ModelUtil.blockOtherClients(false)` or when it disconnects from the server. While the server is blocked, the COMSOL Multiphysics desktop will show the **Server Busy** window.

Creating Default Plots When Running Study Steps
The `runAll`, `runFrom`, and `runFromTo` methods can all now take an extra Boolean input argument (`boolean createPlots`) to create default plots as part of running the study steps when it is set to `true`.

Trigger ModelChangedHandler When the Model Changes
The `ModelChangedHandler` interface in the COMSOL API now includes a new method. It was originally intended to be used when there are multiple clients

connected to a COMSOL Multiphysics server. Now it can also be triggered when the client registering it makes changes to the model using the new method

```
boolean triggeredByChangesFromThisClient()
```

It has a default implementation returning `false`, which gives the behavior in earlier versions. In this case, the method

```
handleModelChangeOnServer(<modelChangeInfo>)
```

is only called when another client changes the model.

If `triggeredByChangesFromThisClient()` returns `true` then `handleModelChangeOnServer(<modelChangeInfo>)` is also called when the client registering the `ModelChangedHandler` changes the model.

New Geometry and Mesh Functionality

GEOMETRY FUNCTIONALITY

- Faster geometry build: After building the geometry sequence to a certain node, building to a node above it is now very fast. Also, building a node below is very fast, if it has been built before.
- You can now group nodes in the Geometry sequence, which can be useful for long sequences. Note that the order in which the nodes appear in the sequence is still important.
- The **Selection List** now includes a section for named selections, which gives an overview of all available named selections.
- You can mark a 2D or 3D geometry object as **Construction geometry**. The construction objects are used only to help with the geometry construction and are not included in the geometry once **Form Union** or **Form Assembly** has been built. This is mainly useful to draw construction lines (visualized as dashed lines) and points in a 2D sketch, for use in constraints and dimensions.
- For the **Difference** operation, there is finer control of which objects to keep with the **Keep objects to add** and **Keep objects to subtract** options.
- **Mirror** (in 2D) now supports specifying a straight edge as the line of reflection.
- The **Rotate** operation now supports specifying a straight edge (3D) or a point (2D) as the center of rotation.

- In the settings for **Intersection**, there is a new option, **Keep lower-dimensional intersections**, which can make the intersection of two surfaces become a curve (instead of an empty result).

- Use the new **Offset** operation to offset the edges of a 2D curve or solid object.

- You can use the new **Extract** operation to extract entities of a geometry object. The default is to create a new object containing the extracted entities while keeping the original object. It is also possible to create one object for the extracted entities and one for the remainder entities.

- Use the new **Thicken** operation to create a thickened solid object of a curve object in 2D.

- Many of the primitives for creating curves in 2D are now located under the **More Primitives** menu on the Geometry toolbar and in the Model Builder menu.

MESHING FUNCTIONALITY

- The **Mesh** toolbar and the **Model Builder** menu have been reorganized for better ease of use.

- Some mesh **Warnings** have been converted into **Information** nodes to better indicate the severity of the reported problems. For all **Information**, **Warning**, and **Error** nodes that contain problem coordinates, it is possible to zoom in on the coordinate and also add a **Clip Sphere** to inspect the region more closely.

- The Settings window of all mesh operations now contains an **Information** section with information about building time, build version, and more.

- The **Adjust edge** mesh option for **Mapped** has been improved to better reduce element skewness.

- The new functionality for **Mesh Plots** include:
  - A new default color table, **Traffic Flow**, designed to highlight elements of poor quality more clearly. The color table also works better for people with color blindness.
  - The possibility to color the elements by their type. Set **Element color: Type** in the Settings window of the **Mesh** plot node.

- You can now do componentwise copies from the entire source to each destination component. Select **Copy method: Array copy** in the Settings window of the operation to specify this. The option **Copy method: Automatic** allows for a mixture of single and array copy.

- For the **Distribution** attribute, some names have been updated when **Distribution type** is set to **Predefined**. The change also affects similar options for the **Create Edges** operation:
  - **Growth formula** is now **Growth rate**.
  - **Arithmetic sequence** is now **Linear**.
  - **Growth formula** is now **Exponential**.
- The **Boundary Layers** operation is updated with new functionality:
  - Specification of the **Total thickness** for the layers.
  - No trimming is done where isolated boundary-layer boundaries connect to exterior boundaries.
  - Support to add boundary layers for imported 2D and 3D meshes.
- The **Import domain elements** option is now available for the **Import** operation. Clear this option to import the surface mesh only. This allows for remeshing of the faces or other types of editing of the imported mesh.
- Curvature of imported linear 2D and 3D meshes is now taken into account. The software estimates the shape of the faces and edges from its linear elements. This curved representation is used to:
  - Place higher-order nodes.
  - Export of second-order elements for an imported linear mesh.
  - Place new or moved mesh vertices when modifying imported meshes with the **Refine** or **Adapt** operations.
- From a **Filter** dataset, it is now possible to select **Create Mesh in New Component** in addition to the **Create Mesh Part** option.
- Intersecting and uniting surface meshes and repairing self-intersecting surface meshes is now possible using the new **Union** operation.
- The new **Transform** attribute is now available to position, rotate, or scale an imported mesh. The attribute is only available for the **Import** feature.
- You can now use the **Fill Holes** operation to fill slit-like holes in an imported surface mesh. This is done by merging the two sides of the hole, introducing new mesh edges when needed.
- It is now possible to combine imported mesh (STL) with CAD/geometry using one component only. The updated tutorial *STL Import 2 — Remeshing an Imported Mesh* shows this workflow.

- The **Build automatically when leaving mesh** option controls if the mesh operations are automatically built when leaving the meshing sequence. This option is only available for imported meshes on which the physics is defined.

New Functionality in the Physics Interfaces

FLUID-SOLID INTERACTION, FIXED GEOMETRY

The Fluid-Solid Interaction, Fixed Geometry interface is now available without any additional module license.

FULLY DEVELOPED FLOW

The **Fully developed flow** option for the Inflow and Outflow boundary conditions in the Laminar Flow interface is now available without any additional module license.

INCLUDE GRAVITY AND OPTION TO USE THE REDUCED PRESSURE

The **Include gravity** option and the associated **Use reduced pressure** option in the Settings window for the main **Laminar Flow** physics interface node are now available without any additional module license.

MASS FLOW OPTION FOR THE PERIODIC FLOW CONDITION

In the Settings window for the Periodic Flow Condition in the Laminar Flow interface, you can now choose **Mass flow** from the **Flow condition** list without any additional module license.

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. You must first 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

NEW IMPROVED PAIR FUNCTIONALITY

The handling of pairs — identity pairs and contact pairs — has been improved in the following ways:

- The fallback features under each pair feature have been removed. For pairs, there can be parts of the boundaries that do not overlap, which typically represent exterior boundaries outside of the overlapping area. For those parts, the other boundary conditions under the physics interface will contribute with proper conditions, acting as fallback boundary conditions on the nonoverlapping parts. These boundary conditions have an indicator in the lower-left corner displaying their status as fallback boundary conditions.

- Most physics interfaces now have a default pair feature, and the default boundary condition will act as the default fallback condition for all pairs. If you do not want to use one of the pairs under **Definitions** in your physics interface, you can select the Disconnect pair check box to disconnect all the pairs. Then, the default pair feature will not contribute to any conditions, and the fallback conditions from the other boundary condition will apply to the selection of the default pair feature.

- Pair features can override each other.

See also Pair Features under *COMSOL 6.0 API Changes.*

NEW NORMAL DISTRIBUTION FUNCTION

The new **Normal Distribution** function represents a normal distribution with a bell-shaped (Gaussian) curve. It supports the following statistical functions related to a normal distribution: density function, cumulative distribution function, quantile function, and randomly generated function.

IMPROVED POROUS MATERIAL FUNCTIONALITY

The **Porous Material** node, used for defining and sharing properties for modeling porous media processes, has been significantly improved.

- The **Phase-Specific Properties** section lists all properties requested by physics features with support for a porous material.

- For each phase-specific property, the phase is indicated in the right-most column.
- When requested, multiple entries of the same quantity, one for each phase, can be listed.
- Tables listing properties can be sorted using any of the columns.
- By right-clicking any property, it is now possible to select **Go to Requesting Mode** to navigate to where it is used. In the same manner, it is also possible to select **Go to Source** to inspect how the property is defined.
- The **Homogenized Properties** section lists all properties that are not phase specific, like the permeability. It also lists properties requested by interfaces without support for a porous material. The homogenized properties can be entered directly by selecting **Locally defined** in the **Material** list.
- When defining properties in a **Fluid**, **Solid**, or **Pellet** subnode to **Porous Material**, **Locally defined** can be selected, in the **Material** list, to enter the properties directly.

A new subnode, **Pellet**, is available for **Porous Material**. The new subnode is used to provide properties for solving mass and energy transport inside porous pellets in a packed bed.

IMPROVED UNIT SETTINGS IN FUNCTIONS

You now define units of input arguments in **Analytic** functions using a table with a row for each argument. A new **Unit** column is also available in the **Plot Parameters** section for **Analytic** function nodes.

The **Step**, **Ramp**, **Random**, and **Rectangle** functions now support unit syntax in their settings.

SELECTION INFORMATION

The **Override and Contribute** option in the **Show More Options** dialog box has been renamed **Selection Information**. You would typically use it to turn this functionality off to improve performance in complex models with a lot of physics features.

OTHER NEW FUNCTIONALITY

- For **Interpolation** functions, **None** is now available in the **Extrapolation** list. It can be useful to prevent extrapolation of interpolation functions outside the range of the given data.
- The **Base Vector System** and **Rotated System** coordinate systems now include settings for specifying an origin.

- A new **Events Sequence** node is now available in the **Events** interface. This node defines an ordered sequence of events, as defined by the added **Sequence Member** subnodes.

- The `sqrteps` and `poweps` built-in functions are now available. They are similar to `sqrt` and `pow`, but with an added, very small `eps` value in their derivatives, which can be needed when the derivatives of the expressions need to be evaluated near zero.

- You can now control the identification method for identifying meshes in the source and destination in the Settings windows for **Period Boundary Conditions** and **Identity Pair** nodes, if desired. Choose one of the following alternatives from the **Elementwise mapping for compatible meshes** list: **On**, **Off**, or **Require compatible meshes**. Choose **On** if you expect compatible meshes and want to use an elementwise mapping, but you want to get a mapping even if source and destination cannot be identified. Choose **Off** if you do not expect compatible meshes or if you want to use elemental constraints for continuous fields. Choose **Require compatible meshes** if you expect compatible meshes and you would rather fail than use noncompatible meshes.

- Import of images on the TIFF format (`*.tif` and `*.tiff` files) is now available in **Image** functions.

- A fast alternative for coupling operators is now available by selecting **Walk in connected component** from the **Mesh search method** list.

- In the settings for **Expression Operator** nodes, there is a new **Argument type** column, where you can choose **Expression** (the default) or **Value**. Use **Value** to pass expression operator arguments by value instead of as an expression.

- In the Settings window for **State Variables** and **Expression Operator** nodes, you can now use Ctrl+Space to add parameters, built-in mathematical constants and functions, operators, and physical constants to define expressions.

- In the settings for the **State Variables** node, **Only initialization** is now available in the **Update** list. It can sometimes be useful to ensure that a complicated or expensive expression is evaluated only once.

- **Material** nodes now have a **Name** field to change their identifier. This setting also changes the tag at the same time and makes it possible to control the scope of variables and functions that the material defines. There is also a new **Name** field available for property groups under materials that change the identifier of the property group.

STUDIES AND STUDY STEP FUNCTIONALITY

- For each individual study step you can alter the equation form of a physics interface if the study step shows the tree view in the **Physics And Variables Selection** section. When the study step shows the table view, you can see what equation form applies to a physics interface in a new **Equation form** column, and an icon representing the type of changes made for the study step is available. The **Discretization** column available in earlier versions has been removed. Also, when you have selected the **Modify model configuration for study step** check box, you can now modify state variables and matrix and vector utilities in the model tree.

- The **Model Reduction** study step, **Modal Solver** solver node, and generated reduced-order model nodes under **Global Definitions>Reduced-Order Modeling** have all been updated and extended to support component mode synthesis (CMS).

    - For the reduced-order modeling, you can choose a *stateful* or *stateless* interface in the **Time-Dependent Modal Reduced-Order Model** and **Frequency-Domain Modal Reduced-Order Model** nodes, which appear under **Global Definitions>Reduced-Order Modeling**. A reduced-order model with a stateless interface is a black box with a number of inputs and outputs. A reduced-order model with a stateful interface, in contrast, relies on the main COMSOL solvers to solve a set of reduced-order equations in terms of states (degrees of freedom), which are exposed to the main solver and subsequently stored in the main solution vector.

    - In the settings for the Model Reduction study step, you can now choose a training study and a study step for constraint modes. There is a new **Use extra Compile Equations for Results** check box. This setting controls whether an extra, active **Compile Equations** node will be added last in the default solver sequence. It then includes the newly created reduced-order model feature in the final solution output by the study so that the reduced-order model's outputs can be evaluated directly using this solution.

    - The **Modal Reduction** solver node's settings includes a new Constraint Modes section with settings for specifying which solution to be used when constructing the reduced model and which of the constraint modes present in the solution to include when constructing the reduced model.

    - In the **Compile Equations** node's settings, there is a new Geometric Entity Selection section. This section contains a **Use entities** list, where you can choose **All** (the default) and **Selected**. When **Selected** is chosen, you can choose a named selection in the **Selections** list. Solving on a subset of the entire geometry can be

useful for component mode synthesis (CMS) and general submodeling simulations.

- In the settings for the **Parametric Sweep** node, you can now select the **Keep reduced-order models** check box. If selected, such models are not discarded, and you can specify what to add to the tag for those models (the parameter name and value, for example). You can also choose to clear previous models so that you can update all reduced-order models by running the study again and to keep the generating reduced-order models, so that they are saved in addition to the reduced-order model copies.

- In the settings under **Adaptation and Error Estimates** in study step nodes, when you choose **Functional** from the **Error estimate** list, a new **Interpolation** option is available in the **Adjoint solution error estimate** list. It provides an estimation of the interpolation error in an expression to get a better weight related to the dual solution in the error estimate based on a functional.

- The new **Auxiliary-Space AMG** iterative solver is suitable for solving complex-valued RF problems and other electromagnetics simulations that use vector elements (Nédélec elements). Compared to the Auxiliary-Space Maxwell (AMS) solver, the Auxiliary-Space AMG solver supports complex arithmetic. More intermediate smoothers are available, which makes it possible to, for example, use Auxiliary-Space AMG when a gauge feature is used or, in general, to use it combined with Vanka smoothers.

- You can now use automatic remeshing for stationary parametric studies that use one continuation parameter. If **Auxiliary sweep** is enabled in the **Study Extension** section for a stationary study step, you can now select an **Automatic remeshing** check box and select in which geometry to remesh.

- The linearization point settings have been removed from the study level for the following study types: Frequency Domain, Frequency Domain Modal, Eigenvalue, Eigenfrequency, Linear Buckling, Boundary Mode Analysis, and Mode Analysis.

- Equation-related settings now have a separate **Equations** section in domain decomposition and multigrid solvers. You no longer have to activate the **Prefer the matrix free format** option when using absorbing boundary conditions and shifted Laplace contributions. You can now use the Schwarz method for domain decomposition to solve Helmholtz equation problems.

- In the **Domain Decomposition (Schwarz)** solver node's settings, you can now choose **Nicolaides** from the **Use coarse level** list. The Nicolaides coarse space is a simple way of constructing a coarse space for Schwarz domain decomposition. It does not need

any additional information, such as a coarser mesh, except the local system matrices from the local domains.

- There are new options for filtering lines in the SCGS smoother to provide slimmer line construction for use with the AMG solver.

- A local Schur complement approximation for the Vanka and SCGS smoothers is available as a more "lightweight" version of the Vanka solver that can store the data permanently without using too much memory. For **Vanka**, choose **Direct, stored factorization** from the **Block solver** list. For **SCGS**, select the **Vanka** check box to access these settings.

- In the settings for parameters, for example, in the **Values of Dependent Variables** section in the settings for study step nodes, the following improvements and changes have been made:

  - The previous **Automatic** and **All** options have been renamed to **Automatic (single solution)** and **Automatic (all solutions)**, respectively.

  - The **Automatic (all solutions)** option is now also available in the **Initial values of variables solved for** list. When selected, all solutions are passed down to the solver and the solver selects the initial values based on if there is a match to the current parameter tuple; otherwise, the logic from previous versions is used.

- To improve time-dependent adjoint sensitivity analysis for DAEs, new backward time stepping is available. In the **Sensitivity** node's settings, a new **Backward Time Stepping** section is available when you have selected **Adjoint** as the sensitivity method. Use the new **From forward** or **Manual** options if needed.

- A new **Drop tolerance for local matrix contributions** field is available in the **Advanced** solver node's settings. It is primarily useful for semiconductor simulations.

- In the settings for the Model Reduction study step node, the **Compute** list in the **Model Reduction Settings** section has been removed because it is no longer transparent, useful, or safe to use. The corresponding API property, `trainingRecompute`, has also been removed. The behavior in 6.0 corresponds to the previous **Initially** option, and you will have to compute or recompute any dependent studies manually.

CLUSTER AND BATCH FUNCTIONALITY

- A new **Parallel modified independent set** parallel coarsening method for the algebraic multigrid solver is now available. It makes the algebraic multigrid solver more efficient in distributed computing. The old coarsening method is still available under the name **Classical**. You can select these methods in the **Coarsening method** list for the Algebraic multigrid solver: **Parallel modified independent set** (the new default)

and **Classical** (the previous implementation). New options are also available to further reduce memory consumption and computation time per iteration. The **Sparser prolongator** option (enabled by default) gives better performance per iteration (less memory and computation time) but the solver may require more iterations to converge. You can use the **Prolongator truncation factor** to control the fill-in of the prolongator.

- Improved file handling for batch jobs in the settings for **Batch Sweep** and **Cluster Sweep** nodes:

  - Under **Before sweep**: The **Clear meshes** and **Clear solutions** check boxes are now available and selected by default.

  - Under **During sweep**: The **Clear synchronization data** and **Save model to file after synchronization is completed** check boxes are now available when you select the **Synchronize solutions** check box.

  - There is also a new **Clear All Data** option in the settings for the **Batch Data** and **External Process** nodes.

- There are new settings for controlling the use of METIS for DOF numbering on cluster. In the settings for the **Solution** and **Compile Equations** nodes under **Solver Configurations**, a new **Partitioning method for distributed computing** list is now available. This setting affects the partitioning of the mesh data on cluster for the purpose of creating a DOF enumeration optimized for the particular cluster configuration (number of nodes).

- There are new options for running Hydra when initializing the MPI environment on Linux. See *COMSOL Commands on Linux* in the *COMSOL Multiphysics Reference Manual* for more options.

New and Improved Results and Visualization Functionality

GENERAL NEW RESULTS FUNCTIONALITY

In version 6.0, only the relevant postprocessing features, dependent on the physics in the current COMSOL Multiphysics simulation model, are visible. To show all features under **Results**, select the **All Dataset Types**, **All Plot Types**, **All Numerical Evaluation Types**, and **All Export Types** check boxes in the **Show More Options** dialog box.

ROTATION CENTER IMPROVEMENTS

Setting the rotation center now works on all platforms, including macOS. The following functionality has also been improved:

- The automatic placement of the rotation center has been improved so that it snaps to the center of the axis-aligned bounding box of whatever geometrical entity (geometry objects, domains, boundaries, or edges) that is found at the center of the screen (if the bounding box is found suitable; that is, it is mostly seen on screen and its center is seen on screen).

- Moreover, the algorithm for the automatic placement of the rotation center now searches around the position where you zoom into or out from after an interactive zoom with the mouse.

- You can Ctrl+middle-click or Ctrl+click the scroll wheel to set the rotation center at the center of the axis-aligned bounding box of the geometrical entity that you currently hover over. As before, middle-clicking only will explicitly set the rotation center on the surface of the geometry over which you have positioned your cursor.

NEW AND IMPROVED DATASET AND EVALUATION FUNCTIONALITY

- New **Transformation 3D** and **Transformation 2D** datasets are available for rotations and other affine transformation of data.

- Under **Derived Values>Measure**, you can now add the following nodes for computing various geometric measurements: **Volume Measurement**, **Surface Measurement**, **Line Measurement**, and **Distance Measurement**.

- The **Global Matrix Evaluation** feature has been extended to handle the following cases:
  - A parameter sweep where the innermost two parameter levels both are inner.
  - A parameter sweep where the innermost two parameter levels both are outer.

  The **Inner solutions** and **Outer solutions** lists in previous versions have been renamed to **Inner parameters** and **Outer parameters**, respectively, or show a dynamic level for structured sweeps where the two innermost columns are both inner and both outer.

- The new **Kernel Density Estimation** dataset takes a table and computes the kernel density estimation (KDE), which is a nonparametric way to estimate the probability density function of a random variable.

- You can now add a **Selection** subnode to **Shell** dataset nodes, making it easier to postprocess large shell models. Selection subnodes are now also available for **Cut Plane** datasets.

- It is now possible to specify rotations in 2D and 3D Array datasets.

- There is a new Spatial FFT dataset for use in applications where the spatial resonance and harmonics of a scalar variable or expression are of interest.

- In Grid datasets, the point distribution for the grids can now be uniform, exponential, or mixed uniform and exponential.

- For the following numerical evaluation features under **Derived Values** and evaluation groups, it is now possible to normalize the values within each column:
  - Average
  - Global
  - Integration
  - Maximum
  - Measure
  - Minimum

  From the **Normalization** list, you can choose to normalize values with respect to the first, last, and maximum absolute value within each column. Also, the **Operation** list in earlier versions has been renamed as **Transformation** in version 6.0.

- For Filter and Partition datasets, you can now right-click and choose **Create Mesh in New Component** to add a new **Component** node and import the modified geometry from the dataset as a mesh in that component. You can also right-click and choose **Mesh Import Parameters** to control which solution parameters to use for the data transferred from the dataset to a mesh part.

- You can now use a transparent background for exported animations.

- Through-thickness evaluation options have been added to Cut Point datasets that refer to Layered Material datasets.

- Area and line scale factors are now available in Shell datasets. Also, a Through-Thickness section has been added to Shell datasets that are chosen in a numerical evaluation feature that evaluates on a geometry.

- Shell datasets now include a location setting to specify where in the Shell dataset that evaluation takes place.

- It is now possible to create 2D contour plots from the data in a Table or Evaluation Group. This could be useful in, for example, AC/DC models for generating efficiency map plots.

- For Extrusion datasets, you can now specify the extrusion direction.

- You can now snap cut points in 3D to the closest edges in addition to the closest boundaries (surfaces).

- Edge and Surface datasets are now supported as inputs to Join datasets.

- For Cut Line and Parametric Curve datasets, there are now variables with unit length that evaluate to the tangent direction.

- In the Integration data series transformation, there is a new **Cumulative** option that makes it possible to compute the cumulative sum or integral.

- When exporting plots on the PLY and 3MF file formats, there is now an option to include vertex colors, which can make duplicated vertices unique.

- The Touchstone export capability is now available with the MEMS Module and AC/DC Module.

- For WAV file export, there is now an option to add a reference value for normalization of the WAV data.

- Tables can now be exported to Microsoft® PowerPoint®, maintaining a link to COMSOL Multiphysics so that you can modify the layout of the tables and update them using a new **COMSOL Table** ribbon in PowerPoint.

- When you are evaluating a minimum or maximum under **Derived Values**, there is now an option in the **Configuration** section of the Settings window to include the position of that maximum or minimum by adding the coordinate values for that position to the table for the evaluated quantities.

NEW AND IMPROVED GRAPHICS AND PLOT FUNCTIONALITY

- With the new **Plot Array** functionality, you can easily arrange the plots in a 2D or 3D plot group in an array (linear or square), with options to exclude some of the plots and to control the amount of padding (empty space) between each plot in the plot array. Such a plot array can be useful for comparing and presenting multiple solutions with different physical quantities, time steps, parameter values, or eigenmodes.

- A new **Translation** subnode can also be used to shift a plot a given distance to put two plots side by side, for example. You no longer need to use an "artificial deformation" in a **Deformation** subnode for this purpose.

- A new **Streamline Multislice** plot simplifies the creation of plots that combine streamlines with slices and for which you previously had to combine multiple Cut Plane datasets and as many Streamline Surface plots.

- A new 1D **Sobol Index** plot is available with the Uncertainty Quantification Module to visualize Sobol indices for an uncertainty quantification study.

- A number of new and updated color tables have been added:
  - New Dipole and DipoleDark color tables, which are useful for electromagnetic fields and other field quantities with large variations.
  - New Prism and PrismDark color tables, which are alternatives to the Rainbow color scale. Prism is now used as the default color table for solid mechanics models.
  - New Cividis and Viridis color tables, which are suitable for users with red-green color vision deficiency.
  - New Inferno, Magma, and Plasma color tables, which all use a blue-red-yellow sequence. They are also relatively friendly to common forms of color vision deficiency.
  - New Gaia and GaiaLight color tables span colors that make them suitable for plots related to topography and bathymetry.
  - The new TrafficFlow color table is specifically designed for visualizing mesh quality. The green half has a very soft gradient, so all good mesh elements have about the same color when using a reversed TrafficFlow color table. This makes it much easier to focus on the mesh elements with a low quality.
  - DiscoDark and RainbowDark are new variations of the Disco and Rainbow color tables, respectively.
  - The Cyclic, Disco, DiscoLight, Rainbow, RainbowLight, Spectrum, Traffic, TrafficLight, Wave, and WaveLight color tables have been updated.
  - New GrayBody and GrayBodyLight color tables are based on the Planckian locus for blackbody radiation. They are useful within the context of metal processing, for example.
  - New SpectrumLight color table that complements the Spectrum color table.
  - The Thermal color tables have been revised. There is a new ThermalDark color table, and the Thermal and ThermalLight color tables have been revised. The ThermalNarrow and ThermalEquidistant color tables from earlier versions have been removed and are now replaced with the new Thermal and ThermalDark color tables, respectively. There are also new ThermalWave and ThermalWaveDark color tables that are designed for wave phenomena of a thermal nature, such as thermoviscous acoustics and linearized Navier–Stokes physics.

    The color tables have been revised to, for example, have less saturated colors, have smoother luminance curves, or be more symmetric. For the color tables that have

been revised, the corresponding color tables from earlier versions are available with a Classic suffix (for example, RainbowClassic).

A small plot of the color table is now visible when choosing a color table from a **Color table** list to make it easier to choose a suitable color table.

- There is now a **Color table transformation** list in the plot nodes' Settings windows. You can choose **None**, **Reverse**, **Nonlinear**, and **Nonlinear symmetric**. For the nonlinear options, you can reverse the color table and control the nonlinearity using a color calibration parameter. For the **None** and **Reverse** options, you can now use a logarithmic color scale in the plots. Also, the previous **Symmetrize color range** check box has been moved to a new **Scale** list as **Linear symmetric** together with the new **Logarithmic** option and the default **Linear** option. In connection with this change, the new API property `colorscalemode` replaces the `colortablesym` property in earlier versions of COMSOL Multiphysics.

- Color tables can now span from the minimum mesh quality to the maximum mesh quality for mesh plots.

- There is a new **Information** section in the Settings window for all plots and graphs to display different quantities useful for benchmarking against different settings and setups, such as the time that it took to prepare the data for the plot. The information appears in a table that is saved with the model and updated each time the plot had to be redrawn. This section is hidden by default, and you can enable it in the **Show More Options** dialog box by selecting the **Plot Information Section** check box.

- For Isosurface plots, you can now choose to plot the isosurfaces as closed volumes using a new **Filled** isosurface type.

- You can now choose **Discrete Fourier transform** from the **Transform** list for the x-axis data to use a discrete Fourier transform (DFT) in **Table Graph**, **Impulse Response**, and **Global** 1D plots.

- You can now add transparency to 2D plots with a height attribute.

- A **From theme** option for colors is now available in streamline plots.

- There are new filtering options in 1D Global and Point graph plots that make it possible to draw noncontinuous lines when the x-axis data expression is nonmonotonic. They are available in the **Line Segment Selection** section in the Settings window for the **Filter** subnode.

- For streamline plots, you can now use mesh-based streamline starting points on selected boundaries.

- It is now possible to use the element type as element color in 2D and 3D mesh plots.

- In plot groups, there is a new option for dataset edges to use the same frame as the underlying dataset.
- Point Graph and Line Graph plots now support Layered Material datasets and include a setting for specifying the through-thickness location.
- The layer/interface settings in Layered Material datasets have been extended to handle models containing multiple shells.
- It is now possible to filter line segments with respect to direction for Global, Line Graph, and Point Graph 1D plots to remove unwanted edges in such plots.
- There is now an option to add a grid in the Matrix Histogram plot for use in fatigue simulations.
- It is now possible to use evaluations of expressions in legends in graph plots.
- When evaluating variable expressions in annotations in plots, the `eval()` operator supports a new third input argument for the numerical precision.
- It is now possible to reverse the direction of Edge datasets.
- You can now choose **Dataset space variable** from the **Parameter** list for the x-axis data in Line Graph plots, which can be useful for line graphs of Spatial FFT datasets, for example.
- The Table Graph plots now also support row-based tables (that is, tables with data on rows and sample points on columns).
- You can now add padding to Table Annotation plots, which makes it possible to get better-looking results when zooming.

NEW AND IMPROVED REPORT AND PRESENTATION FUNCTIONALITY

- For PowerPoint presentations, you can now use two-content PowerPoint slide layouts.
- Presentations now include link information for the COMSOL add-in for PowerPoint for images and tables in presentations.
- You can now use a custom view for general report images.
- Reports can now include styled text area toolbar buttons and highlighting for bulleted and numbered lists.
- Reports now have full support for Microsoft Word templates, including background (for example, colors, gradients, and graphics); header (for example, watermarks); and footer settings.

New Java® API Methods

The following methods are now available:

GENERAL GEOMETRY COMMANDS

```
model.component(<tag>).geometricModel();
```

Returns the tag of the component's geometric model on which the physics is defined.

```
model.component(<tag>).geometricModel(<sequencetag>);
```

This method specifies the geometric model used in the component.

GEOMETRY OBJECT INFORMATION

```
Boolean assem = model.component(<ctag>).geom(<tag>).isAssembly();
```

Use this method to check if this is a geometry of assembly type (originating from a Form Assembly feature).

```
Boolean virt = model.component(<ctag>).geom(<tag>).isVirtual();
```

Use this method to check if the geometry is a virtual geometry.

```
Boolean gM = model.component(<ctag>).geom(<tag>).isMesh();
```

Returns true if the component's geometric model is represented by a mesh (rather than a geometry).

MESH SETTINGS

```
String gM = model.component(<ctag>).mesh(<tag>).geometricModel();
```

This method returns a string with the tag of the geometry or meshing sequence, which defines the geometric model of the mesh.

```
model.component(<ctag>).mesh(<tag>).geometricModel(boolean);
```

Use this method to specify if the meshing sequence defines its own geometric model.

```
model.component(<ctag>).mesh(<tag>).autoRebuild(boolean);
```

This is a user interface setting that controls if the mesh sequence is automatically rebuilt when clicking a node in the model tree outside the mesh sequence.

```
model.component(<ctag>).definedByMesh(<tag>);
```

Use this method to control on which geometric model to define the physics.

General Backward Compatibility Considerations

COMSOL Multiphysics version 6.0 can open MPH-files saved from COMSOL Multiphysics versions 4.0–5.6.

COMSOL Multiphysics version 6.0 can run Java® files saved from COMSOL Multiphysics versions 4.0–5.6. However, the Java® files may need to be modified in accordance with information in this document and may need to be recompiled with the `comsol compile` command in version 6.0.

Backward Compatibility with Version 5.6

PAIR FEATURES

The new framework for pair features can affect old models, although the goal is that the effect is as small as possible. When opening an old model in the **Model Builder**, pair features are migrated using the following main steps:

1 Add a default pair feature with the **Disconnect pair** check box selected to ensure that the default pair feature does not contribute any new constraints or equations to the model.

2 Move all fallback features that were located under the pair features in 5.6 to the level below the physics interface. The selection of these moved features will typically get the same active selection as the pair feature they belonged to in 5.6. Because those moved features now have an overlapping selection with that pair feature, they will act as an equivalent fallback feature to the pair feature.

3 Optimize all moved fallback features by merging exact duplicates into one feature, including existing features such as the default features. In many situations, the result will be that only the default feature of the physics will remain because most fallback features under pair features in 5.6 were identical to the default feature of the physics interface.

Potential Issues

If the optimization in Step 3 above cannot merge all features, some moved fallback features will remain and be placed before their pair feature. Normally, a pair feature in 5.6 overrode all features before it, so if the moved fallback also overrides these features, the selections should not change for them. However, there can be configurations where the selection may change because of special schemes for overriding and contributing between features. One example is shell interfaces that have pair features on the same level as the physics interface. The pair feature can then contribute with a

feature that the moved feature will override. Such cases are difficult to detect during migration and can create a discrepancy in some selections between the versions.

PHYSICS AND VARIABLES SELECTIONS

The **Discretization** column is removed from the table to the more advanced tree option, which is shown when the **Modify model configuration for study step** check box is selected.

USER DOCUMENTATION NODES IN THE PHYSICS BUILDER

The User Documentation nodes in the Physics Builder are no longer available in version 6.0.

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.

AC/DC Module

MAGNETOMECHANICS MULTIPHYSICS INTERFACES

Two new physics interfaces for analysis of coupled magnetic and mechanical effects have been added: Magnetomechanics and Magnetomechanics, No Currents. When you add such an interface, two physics interfaces are added to the model: Solid Mechanics and either Magnetic Fields or Magnetic Fields, No Currents. The new Magnetomechanical Forces multiphysics coupling is also added. You can find these interfaces under the **Electromagnetics and Mechanics** branch in the **AC/DC** folder in the **Add Physics** tree.

Magnetomechanics is demonstrated in the new *plate_deflected_by_magnet* tutorial model. In addition to the AC/DC Module, these interfaces require either the MEMS Module, Structural Mechanics Module, or Acoustics Module.

PERIODIC PAIR FOR THE MAGNETIC FIELDS INTERFACE

The new **Periodic Pair** feature in the Magnetic Fields interface simplifies the modeling of linear machines in 2D (machines such as motors, generators, actuators, and sensors). Instead of modeling the entire device, periodicity is assumed: Typically one or a couple of unit cells are included, ignoring the end effects.

The **Periodic Pair** feature is demonstrated in the new *linear_motor_2D* tutorial model.

ARKKIO TORQUE CALCULATION FOR THE ROTATING MACHINERY, MAGNETIC INTERFACE

Rotating machines typically have a cylindrical air gap between rotor and stator, where electromagnetic fields are exchanged and forces and torque are exerted. The new **Arkkio Torque Calculation** feature in the Rotating Machinery, Magnetic interface analyzes the fields in this gap and makes it possible to easily evaluate the total torque exerted on the rotor and stator in both 2D and 3D. Also, several postprocessing quantities important to motor design are now available. These include the radial force density, azimuthal force density, radial magnetic flux density, and azimuthal magnetic flux density in the air gap.

The **Arkkio Torque Calculation** feature is demonstrated in the following tutorial models: *pm_motor_2d_introduction*, *pm_motor_3d*, and *electric_motor_parameter_optimization*.

MAGNETOSTRICTION FOR THE ROTATING MACHINERY, MAGNETIC INTERFACE

Magnetostrictive materials exhibit free strain under the influence of a magnetic field. This is the effect causing the low-pitched humming sound coming from a transformer, for instance. While previously only available for the Solid Mechanics and Magnetic Fields interfaces, the **Magnetostriction** multiphysics coupling can now also be used to connect the Solid Mechanics and Rotating Machinery, Magnetic physics interfaces. It is available in 3D, 2D, and 2D axisymmetry.

Magnetostriction is demonstrated in the *nonlinear_magnetostriction* tutorial model. In addition to the AC/DC Module, this multiphysics coupling requires either the MEMS Module, Structural Mechanics Module, or Acoustics Module.

FREQUENCY DOMAIN SUPPORT FOR THE MAGNETIC FIELDS, CURRENTS ONLY INTERFACE

The Magnetic Fields, Currents Only interface, designed to efficiently compute the lumped inductance matrix of complex circuits in 3D, has now been extended to include frequency-domain support. This is done using the new **Frequency Domain Source Sweep with Initialization** study step. It can be used to calculate the partial contributions from magnetic fields generated by open conductors, such as those commonly found on printed circuit boards (PCBs).

The main difference compared to the Stationary Source Sweep, introduced in version 5.6, is that the frequency-domain solution considers skin and proximity effects. This makes it possible to extract inductance matrices at low to medium frequencies (in the range where capacitive effects are not yet dominant). Also, the automated postprocessing for resistance and inductance matrices has been significantly improved.

The **Frequency Domain Source Sweep with Initialization** study step is demonstrated in the *inductance_matrix_pcb_coils* tutorial model.

SPLIT BY CONNECTIVITY OPTION FOR TERMINALS, CONDUCTORS, AND THE LUMPED PORT

The new **Split by Connectivity** option is available on the context menu for **Terminal** nodes in (for example) the Electrostatics and the Magnetic and Electric Fields

interface, for **Conductor** nodes in the Magnetic Fields, Currents Only interface, and for the **Lumped Port** in the Magnetic Fields interface.

This option is useful when a model contains many conductors — as is the case for a PCB or a touchscreen. In those cases, having a single selection containing all conductors is enough: You can add a single **Terminal** feature to your model, choose all conductors as its selection, and then choose **Split by Connectivity** to automatically generate a separate **Terminal** feature for each disconnected part of the selection. Especially for terminal sweep studies and inductance matrix extraction, this will greatly speed up your workflow.

This functionality is demonstrated in the *inductance_matrix_pcb_coils* and *capacitive_position_sensor* tutorial models.

LAYERED TRANSITION BOUNDARY CONDITION FOR THE MAGNETIC FIELDS INTERFACE

At low to medium frequencies, the new **Layered Transition Boundary Condition** in the Magnetic Fields interface can be used to model composite electromagnetic shielding materials such as those containing mu-metal and copper layers or, for instance, gold-plated copper on a circuit board. At higher frequencies, this feature can even be used for modeling RF metamaterials and antireflection coatings, although for those applications, you would typically use the same feature in the RF Module or the Wave Optics Module. The feature is used in combination with a **Layered Material** in the global **Materials** node, and a **Layered Material Link** in the **Materials** node. Together with the extended **Electromagnetic Heating, Layered Shell** multiphysics coupling, the Layered Transition Boundary Condition supports fully coupled induction heating in the layer stack.

FULLY COUPLED ELECTROMAGNETIC HEATING FOR THE LAYERED TRANSITION BOUNDARY CONDITION

The **Electromagnetic Heating, Layered Shell** multiphysics coupling has been extended to support the **Layered Transition Boundary Condition** in the Magnetic Fields interface for frequency-stationary and frequency-transient study types in 2D and 3D. The Layered Transition Boundary Condition takes the layer-averaged (temperature dependent) material properties for each individual layer, and uses them to compute the electromagnetic field propagation and power dissipation in the layer stack. The layer-averaged heat sources are then fed back into the thermal simulation. Together with the **Thin Layer** feature in the Heat Transfer in Solids interface, this allows you to investigate the temperature distribution in the layer stack. Typical use cases include the induction welding of layered composite materials, for example.

SYMMETRY PLANE FEATURES FOR VARIOUS INTERFACES

The new **Symmetry Plane** feature is available for the following physics interfaces: Electrostatics; Electric Currents; Magnetic Fields; and Magnetic Fields, No Currents. The symmetry planes for Electrostatics and Electric Currents provide symmetry and antisymmetry conditions for the electric field. For the Magnetic Fields and Magnetic Fields, No Currents interfaces, the symmetry type can be set for the magnetic flux density B and the magnetic field H, respectively.

The Symmetry Plane features are demonstrated in several tutorial models, including: *vector_hysteresis_modeling*, *permanent_magnet*, and *dielectric_shielding_comparison*.

DEBYE DISPERSION MODELS FOR DIELECTRICS

New damping models have been added for dielectric materials. Under **Charge Conservation**, when the material type is set to **Solid**, you can now use the **Dispersion** dielectric material model. In the **Dispersion** subnode, you can choose between the **Debye** and **Multipole Debye** dispersion models.

NEW ALGEBRAIC MULTIGRID PRECONDITIONER FOR LINEAR CURL ELEMENTS

The **Auxiliary-Space Algebraic Multigrid** (ASAMG) preconditioner is an implementation of the AMS (Auxiliary-Space Maxwell) solver. For the AC/DC Module, it is now available as an alternative preconfigured solver in the default solver configuration for the Magnetic Fields interface (when the discretization of that interface is set to linear in 3D).

GEOMETRY PART LIBRARY FOR ELECTRIC MOTORS

A new Geometry Part Library has been added specifically for modeling rotating machines in 2D. It contains three internal permanent magnet rotors and one external stator (equipped with coils). The rotor designs include the following types: embedded magnet, embedded magnet (V-shape configuration), and surface-mounted magnet.

The geometry parts are demonstrated in the following tutorial models: *pm_motor_2d_introduction*, *generator_2d*, *sector_generator_3d*, and *electric_motor_parameter_optimization*.

IMPROVED PLOT DEFAULTS

As an extension to the default plot improvements included in version 5.6, the AC/DC Module's plot defaults now use new color tables and new plot features such as the **Streamline Multislice** plot. A new **Prism** color table has been designed specifically for

vector field norms with singular tendencies such as the electric field norm or the magnetic flux density norm. Multislice and streamline plots are combined to indicate both field strength and direction. For physics interfaces that use a scalar potential formulation — Electrostatics; Electric Currents; Magnetic and Electric Fields; and Magnetic Fields, No Currents — extra plot groups are added that use the symmetric **Dipole** color table.

TOUCHSTONE EXPORT SUPPORT IN POSTPROCESSING

The Electrostatics, Electric Currents and Magnetic Fields interfaces can generate S-parameter matrices through port or terminal sweeps. In addition to the old Touchstone export functionality, these physics now support the **Touchstone Export** functionality available during postprocessing.

ELECTRIC CIRCUIT IMPROVEMENTS

The Electric Circuit interface has received a number of fixes and extensions. These improvements include the support for importing and exporting the mutual inductance element K and more robust and general SPICE import.

New and Updated Models in Version 6.0

The following models are new in version 6.0:

ELECTRICAL HEATING IN A BUSBAR WITH TERMINALS

This example demonstrates how to use a terminal fed by a current to compute the resistive heating in a busbar.

PERMANENT MAGNET MOTOR 2D

In this tutorial, a PM motor with 10 rotor poles and 12 stator slots is modeled in 2D to capture the torque ripple over an electrical period and map the volumetric loss density in the rotor and stator iron.

LINEAR MOTOR IN 2D

In this model, a synchronous motor with a three-phase winding on the stationary part and permanent magnets on the moving part is investigated. A moving mesh and periodic conditions are utilized in order to model a periodic part of a complete motor. It is also explained how to use this model for a 2D representation of an axial flux motor (rotational motion).

PARAMETER OPTIMIZATION OF AN ELECTRIC MOTOR

This example demonstrates how to use parameter optimization to identify the optimal position and shape of the permanent magnets in an electric motor. The torque is used as the objective function, but different designs are found depending on whether the minimum or average torque is considered.

SHAPE OPTIMIZATION OF COILS

This example shows how to use the Optimization Module to find a coil geometry giving a uniform magnetic field on axis and minimal field near the axis ends.

DEFORMATION OF AN IRON PLATE BY MAGNETIC FORCE

This example studies the elastic deformation and stress in a ferromagnetic plate when deflected by a permanent magnet. The deformation of the plate has an influence on the distribution of the magnetic field. This effect is accounted for using a moving mesh in the air surrounding the plate.

UPDATED MODELS

In addition, the following models have been updated:

- Magnetic Damping of Vibrating Conducting Solids
- Inductance Matrix Calculation of PCB Coil
- Topology Optimization of a Magnetic Circuit
- Inductance of a Power Inductor

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 6.0

PIEZOELECTRIC WAVES, TIME EXPLICIT PHYSICS INTERFACE

With the **Piezoelectric Waves, Time Explicit** multiphysics interface, there are new capabilities for modeling piezoelectric phenomena in the time domain for wave propagation. Both the direct and inverse piezoelectric effects can be modeled, and the piezoelectric coupling can be formulated using the strain-charge or stress-charge forms. The new interface couples the **Elastic Waves, Time Explicit** interface with the **Electrostatics** interface using the new **Piezoelectric Effect, Time Explicit** multiphysics coupling. The interface is based on the discontinuous Galerkin (dG or dG-FEM) method and uses a time-explicit solver. The electrostatics problem is solved at every time step through an algebraic system of equations solved with classical FEM. This ensures a very computationally efficient hybrid method that can solve very large models with many millions of degrees of freedom (DOFs). The method is well suited for distributed computing on cluster architectures.

PHYSICS-CONTROLLED MESH FOR PRESSURE ACOUSTICS

Physics-Controlled mesh is introduced for the **Pressure Acoustics, Frequency Domain** and **Pressure Acoustics, Transient** interfaces. The generated mesh follows best practices to resolve the wave problem with an adequate number of mesh elements. Perfectly matched layers are meshed with a structured mesh, periodic conditions use copy mesh operations, and a single boundary-layer mesh is used for exterior field calculations. For the frequency domain, the maximum frequency is automatically picked up from the study. For time-domain models, the maximum frequency is taken from physics, ensuring a consistent spatial resolution with the mesh and temporal resolution of the transient solver. For all other acoustic interfaces and for **Solid Mechanics**, only PMLs and Periodic Conditions are handled in this version.

INTRODUCING FLOW-INDUCED NOISE

With this release, a hybrid computational aeroacoustic (CAA) method is introduced for modeling flow-induced noise. It is based on a one-way coupling between the turbulent flow sources and the acoustic problem. It is assumed that no back-coupling exists from the acoustics field to the flow. The computational method is based on the finite element method (FEM) discretization of Lighthill's acoustic analogy (wave equation). This formulation of the equations ensures that any solid (fixed or vibrating)

boundaries are implicitly taken into account. Both the Lighthill and the simpler aeroacoustic wave equation (AWE) analogies can also be selected.

The new functionality relies on coupling a large eddy simulation (LES) fluid flow model to the **Aeroacoustic Flow Source** domain feature in **Pressure Acoustics, Frequency Domain**. The coupling is achieved by using the **Aeroacoustic Flow Source Coupling** multiphysics coupling and the dedicated **Transient Mapping** study.

NEW HIGH-FREQUENCY PRESSURE ACOUSTICS PHYSICS INTERFACES

Two new dedicated physics interfaces are based on the Kirchhoff–Helmholtz integral, so they both rely on a high-frequency assumption. The **Pressure Acoustics, Asymptotic Scattering** interface is useful for modeling scattering problems, while the **Pressure Acoustics, Kirchhoff-Helmholtz** interface is primarily for modeling radiation problems.

Pressure Acoustics, Asymptotic Scattering

The **Pressure Acoustics, Asymptotic Scattering** interface is used to model scattering problems at high frequencies using the Kirchhoff–Helmholtz integral formulation. The acoustic field is assumed to be a locally plane field such that the scattered field can be expressed analytically. The scattering object surface can be treated as perfectly reflecting or having absorbing properties by defining a surface normal impedance, a reflection coefficient, or an absorption coefficient. The latter two can depend on the angle of incidence. The interface can model scattering from spherical and plane waves. The interface has built-in functionality to compute the visibility factor using both a simple angle consideration and a more advanced hemicube method.

Pressure Acoustics, Kirchhoff–Helmholtz

The **Pressure Acoustics, Kirchhoff-Helmholtz** interface is used to model radiation problems at high frequencies using the Kirchhoff–Helmholtz integral formulation. The acoustic field is assumed to be a locally plane field. This technique is also often referred to as high-frequency BEM or simply HFB. The method is often used for computing the radiated acoustic field from vibrating structures at high frequencies, without the need to model the surrounding fluid. The method is valid as long as the acoustic wavelength in the fluid is smaller than the structure and the structural modes. For a flat vibrating surface, the method reduces to solving the Rayleigh integral. A user-defined option gives access to the full Kirchhoff–Helmholtz integral formulation for defining both pressure and its normal gradient.

LUMPED SPEAKER BOUNDARY AND INTERIOR LUMPED SPEAKER BOUNDARY

For many electroacoustics applications, speakers can be efficiently modeled by combining a lumped Thiele–Small representation and the finite element method. The electromagnetic components of the motor are modeled with an **Electric Circuit**, while the acoustics are solved with the **Pressure Acoustics, Frequency Domain** interface. This approach assumes piston motion of the speaker diaphragm, so it is of particular interest for microspeakers. The **Interior Lumped Speaker Boundary** includes the effects of the air on both sides of the diaphragm. The **Lumped Speaker Boundary** has options to include the compliance of a back volume through an impedance.

SECTOR SYMMETRY OPTIONS FOR EXTERIOR FIELD CALCULATION

The **Exterior Field Calculation** feature has been extended with new options to handle models with sector symmetry. Two options extend the existing symmetry planes functionality: the **Sector symmetry** and **Sector symmetry with one symmetry plane** options. The latter is of particular interests when modeling loudspeaker drivers placed in an infinite baffle. The analysis of the exterior field can also be extended with an **Azimuthal mode number** for advanced sector-symmetric problems.

OPTIMIZATION WITH NEW EXTERIOR FIELD VARIABLES

In 3D, new dedicated exterior field variables exist for use in gradient-based optimization problems, such as shape optimization or topology optimization. The optimization objective can now be defined as a variable evaluated in the exterior field, defining, for example, values on a radiation pattern or an off-axis response. The variable exists only for the **Symmetry planes** option in the **Exterior Field Calculation** feature. The new variables are defined with _opt appended to the existing exterior field variables. The operator for the pressure is `pext_opt(x,y,z)`, and for the sound pressure level it is `Lp_pext_opt(x,y,z)`.

PERFECTLY MATCHED BOUNDARY (PMB) RADIATION CONDITION

The new **Perfectly Matched Boundary** is effectively a perfectly matched layer (PML) that is applied to the open boundary as a radiation condition, without the need to define a domain (a layer in the geometry). The condition automatically applies the PML formulation using the extra dimension functionality in COMSOL Multiphysics. This approach also simplifies the requirements to the radiating boundary as it can, in principle, have any convex shape. Different options exist for controlling the attenuation direction. The condition exists for the **Pressure Acoustics, Frequency Domain** physics interface in all relevant space dimensions.

LINEARIZED NAVIER-STOKES, BOUNDARY MODE PHYSICS INTERFACE

The **Linearized Navier-Stokes, Boundary Mode** interface is used to compute and identify propagating and nonpropagating modes in waveguides and ducts in the presence of any stationary isothermal or nonisothermal background mean flow. It performs a boundary mode analysis on a boundary, inlet, or cross section of the waveguide. The interface considers all thermal and viscous loss effects and the interaction with the background flow. This includes the acoustic boundary layer losses, if necessary.

LINEARIZED EULER, BOUNDARY MODE PHYSICS INTERFACE

The **Linearized Euler, Boundary Mode** interface is used to compute and identify propagating and nonpropagating modes in waveguides and ducts in the presence of a stationary background mean-flow that is well approximated by an ideal gas flow. The interface performs a boundary mode analysis on a boundary, inlet, or cross section of the waveguide.

OUT-OF-PLANE AND BOUNDARY MODE ANALYSIS FOR LINEARIZED EULER AND LINEARIZED NAVIER-STOKES

In both the Linearized Euler and the Linearized Navier-Stokes interfaces, it is now possible to add optional out-of-plane and circumferential wave numbers in 2D and 2D axisymmetry, respectively. In the same space dimensions, there is now also support for the mode analysis study. This functionality makes it possible to set up so-called 2.5D simulations in these physics.

COMPUTE DISPLACEMENT POSTPROCESSING FEATURE IN ELASTIC WAVES, TIME EXPLICIT

A new postprocessing feature, **Compute Displacement**, has been added to the **Elastic Waves, Time Explicit** physics interface. The feature allows the computation of the displacement in points, along edges, on boundaries, or in domains by solving auxiliary ODEs. The new features are added as subfeatures to a material model such as the **Elastic Waves, Time Explicit Model** or **Piezoelectric Material** nodes. The feature is a pure postprocessing type, and the variables can be used to visualize displacements. This feature adds and solves an ODE and thus requires additional computational resources.

MINIMUM AND MAXIMUM PRESSURE POSTPROCESSING FEATURE IN NONLINEAR PRESSURE ACOUSTICS, TIME EXPLICIT

A new postprocessing feature, **Compute Minimum and Maximum Pressure**, has been added to the **Nonlinear Pressure Acoustics, Time Explicit** interface. This feature computes the maximum and minimum pressure over time and space in a domain or on a

boundary. Two variables are automatically created, `nate.p_min` and `nate.p_max`, which can be used in postprocessing to, for example, evaluate the size of a focal zone.

PAIR MULTIPHYSICS COUPLINGS FOR VIBROACOUSTIC SIMULATIONS

Two new pair multiphysics couplings have been added to the Acoustics Module for coupling acoustic and solid domains in an assembly. These couplings make it possible to use nonconforming meshes at the interface between the two domains.

- **Pair Acoustic-Structure Boundary** multiphysics coupling for coupling the **Pressure Acoustics, Frequency Domain** or the **Pressure Acoustics, Transient** interface to the **Solid Mechanics** interface, in an assembly geometry where identity pairs have been created.

- **Pair Thermoviscous Acoustic-Structure Boundary** multiphysics for coupling **Thermoviscous Acoustics, Frequency Domain** or the **Thermoviscous Acoustics, Transient** interface to the **Solid Mechanics** interface, in an assembly geometry where identity pairs have been created. The condition uses a penalty formulation for more efficient computations in the time domain.

UPDATES TO THERMOVISCOUS ACOUSTICS

- For ports, a new **Plane Wave** option for handling slip and adiabatic cases or models that are not set in a waveguide context. The names of the existing waveguide modes have been updated to **Numeric (0,0)-mode** and **Circular (0,0)-mode**.

- The default discretization in all thermoviscous interfaces has been changed from Lagrange to serendipity elements. This results in important performance gains when solving models that use a structured mesh (for example, a mapped or extruded mesh).

- Improved solver settings when solving models with **Nonlinear Thermoviscous Acoustics Contributions**. The settings are especially important for highly nonlinear problems. If you open a version 5.6 model, remember to reset the solvers to default.

- Improved coupling between Thermoviscous Acoustics and structures that have curved surfaces by using a Nodal constraint type.

- In the **Boundary Layers** mesh, a new **Thickness specification** option, **All layers**, has been added. This allows to more easily set up a mesh that spans the entire thermoviscous acoustic boundary layer.

- The material properties for the second-order density approximation option in **Nonlinear Thermoviscous Acoustics Contributions** can now be evaluated if the air material (moist or dry) is specified using a **Thermodynamics** system.

- In the **Thermoviscous Acoustics, Frequency Domain** interface, it is now possible to add an optional circumferential wave number for 2D axisymmetric models. This allows for advanced 2.5D analysis of complex propagation patterns.

SOURCE WITH DIRECTIVITY

In the **Ray Acoustics** interface, the new **Source with Directivity** node can be used to release a distribution of rays with initial intensity or power based on a user-defined spatial directivity function. This is of particular interest when defining loudspeaker sources in ray acoustic simulations.

NEW RAY ATTENUATION MODELS

In the **Ray Acoustics** interface, the attenuation of rays can now be specified by entering any of the following:

- Pressure amplitude attenuation coefficient in nepers per meter
- Intensity attenuation coefficient in nepers per meter
- Pressure amplitude attenuation coefficient in decibels per meter
- Pressure amplitude attenuation coefficient in decibels per wavelength

All of these options are supported in domains as well as the exterior void domain outside the geometry.

SIMPLIFIED NONLOCAL COUPLING NAMES IN RAY ACOUSTICS

The built-in couplings that evaluate the sum, average, maximum, and minimum of expressions over all rays have been renamed. In an instance of the **Ray Acoustics** interface with the default name `rac`, the old name for the sum was `rac.racop1()`; the new name is `rac.sum()`. Similarly, the average, maximum, and minimum are `rac.ave`, `rac.max`, and `rac.min`, respectively.

Backward Compatibility
The old coupling names, such as `rac.racop1()` in the example above, will be labeled as deprecated but they will still be supported in version 6.0.

TRANSFORMATIONS OF LOADED RAY COORDINATES

For the Ray Acoustics interface, when loading the ray position and direction from a file using the **Release from Data File** node, the distribution of initial ray positions can now be dilated, rotated, and translated. If a rotation is applied, you can also choose to apply the same rotation to the initial ray directions if they are also loaded from the same file.

RELEASE FROM EXTERIOR FIELD CALCULATION

In the Ray Acoustics interface, the **Release from Exterior Field Calculation** node can be used to launch a distribution of rays with initial intensity or power proportional to the solution of an **Exterior Field Calculation** from a previous study. Several copies of the exterior field calculation can be released, each with its own translation and rotation.

POSTPROCESSING NEWS FOR ACOUSTICS

- New **Discrete Fourier transform** (DFT) and **Moving average** options have been added when analyzing an **Impulse Response** plot in **Ray Acoustics**.

- A new **Information** section is available on plots when **Plot Information Section** is enabled in the **Show More Options** dialog box. The new section displays the plot time, which can be considerable in **Ray Acoustic** models or models using the new Kirchhoff–Helmholtz-based high-frequency interfaces.

- In the **Octave Band** plot, the plotted **Quantity** can now be selected as **Continuous power spectral density**, **Band power**, or **Band average power spectral density**.

- A warning message has been added in the **Energy Decay** subfeature when the behavior of the level decay deviates from normal.

- New Thermal Wave and Thermal Wave Dark color schemes have been added. The Thermal Wave color table is the new default for the temperature variation default plot in the thermoviscous acoustics and linearized Navier–Stokes physics interfaces.

- New updated Wave and Wave Light color tables have been added and are used as default colors for all plots that represent the acoustic pressure.

- There is an important speedup of postprocessing of large surface plots.

- The preview of **Receiver** datasets is now much faster.

IMPORTANT ENHANCEMENTS AND UPDATES IN THE ACOUSTICS MODULE

- Improved formulation of damping for the dG-FEM-based time-explicit interfaces. The new formulation is more stable, more accurate, and results in faster simulations.

- Better handling of lumped circular approximations in thermoviscous acoustics models (Bessel functions) for narrow region acoustics and for interior perforated plates.

- The robustness of the mapping performed by the **Mapping** study, in combination with the **Background Fluid Flow Coupling** multiphysics feature, has been improved. This is particularly useful for models with an imported geometry and for models that have a CFD boundary layer mesh on curved boundaries.

- The **Compressible Potential Flow** physics interface is now applicable in PML domains, which simplifies setting up the background flow for the subsequent analysis in the **Linearized Potential Flow** interface.

Note 1

The default discretization has changed for the Thermoviscous Acoustics interfaces. Add the following lines to obtain the old behavior of a physics interface `ta`:

```
model.component("comp1").physics("ta").prop("ShapeProperty").set(
"shapeorder_u", 2);
model.component("comp1").physics("ta").prop("ShapeProperty").set(
"shapeorder_T", 2);
```

Note 2

In the Linearized Navier–Stokes physics interfaces, the default value for the Stabilization constant has changed. Add the following line to obtain the old behavior:

```
model.component("comp1").physics("lnsf").prop("Stabilization").se
t("alpha", "1e-2");
```

New and Updated Model Examples in 6.0

- Chamber Music Hall

- Schroeder Diffuser 2D

- Loudspeaker Spider Optimization

- Tweeter Dome and Waveguide Shape Optimization

- Angle Beam Nondestructive Testing

- Submarine High-Frequency Asymptotic Scattering

- Several models have been updated to use the new Physics-Controlled mesh available for pressure acoustics.

- Updated Ultrasonic Flowmeter with Piezoelectric Transducers to use the new Piezoelectric Waves, Time Explicit multiphysics interface.

Important Fixes in Version 6.0

ACCURACY IMPROVEMENTS FOR BOUNDARY ACCUMULATORS

For the Ray Acoustics interface, the accuracy of accumulated variables defined on curved boundaries has been significantly improved.

COUNT REFLECTIONS CHECK BOX NO LONGER CAUSES AN ERROR

In the Ray Acoustics interface, an error message was given if the intensity formulation for graded media was selected in a 3D model and the **Count reflections** check box was selected. In version 6.0, these features can be used together without causing an error message to appear.

ERROR MESSAGES FROM INTERFACES NOT SOLVED FOR

In version 5.6, an error message was given when trying to run any study if an instance of the Ray Acoustics interface had no ray release features, or if the ray release features had no selection. In version 6.0, such error messages will be ignored when running a study in which the Ray Acoustics interface is not solved for.

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`, `JinN_ud`, `QN_ud`, `alphafN_ud`, `TLN_ud`, and `qpn_ud` where needed.

For example:

```
model.component("comp1").physics("ade").feature("ps1").set("N_ban
d_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.

Battery Design Module

New Functionality in Version 6.0

GENERAL NEW FUNCTIONALITY IN THE ELECTROCHEMISTRY INTERFACES

- The Highly Conductive Porous Electrode domain node is no longer available when using a Primary Current Distribution type in the Primary Current Distribution and Secondary Current Distribution interfaces.

- The Thermodynamic Equilibrium electrode kinetics option is no longer available in the **Highly Conductive Porous Electrode** domain node in the Primary Current Distribution, Secondary Current Distribution, and Tertiary Current Distribution interfaces as well as for the Lithium-Ion Battery and Battery with Binary Electrolyte interfaces.

- A new **Adsorbing-Desorbing Species** section has been added to the **Electrode Surface**, **Porous Electrode**, and related nodes in the electrochemistry interfaces. The section can be used to model adsorption-desorption and the resulting surface coverage.

- The implementation of the **Concentration-Dependent Kinetics** kinetics type in the **Electrode Reaction** and **Porous Electrode Reaction** (and related) nodes has changed in order to improve accuracy and numerical convergence for well-posed problems. Prior to version 6.0, the `max()` operator was used on the CO and CR parameter expressions before applying them internally in the definition expression for the `iloc` variable. The usage of the `max` operator has been removed in version 6.0. To get old models to behave exactly as before, rewrite the CO and CR parameter expressions as `max(CO,eps^2)` and `max(CR,eps^2)`, respectively.

- New Adsorption-Desorption Voltammetry tutorial added to the Electrochemistry Module Application Library.

NEW BATTERY-SPECIFIC FUNCTIONALITY

- Battery Material Library: NMC 622 was added as an electrode material. The solid diffusion coefficient value and reference information was updated for the electrode material NMC 111. The volumetric intercalation strain, Young's modulus, and Poisson's ratio data have been added for a number of electrode materials. Lithium metal was added.

- A **Porous Conductive Binder** domain node has been added to the Lithium-Ion Battery and Battery with Binary Electrolyte interfaces.

- A new **Intercalation Strain** subnode to Linear Elastic Material in Solid Mechanics interface is now available with a Battery Design Module license.

- A new rate constant option for specifying the exchange current density in the **Lithium Insertion** kinetics option has been added to the Lithium-Ion Battery and Single Particle Battery interfaces.

- A new postprocessing variable, Cell State-of-Health, has been added to the **Capacity Loss** feature of the Lumped Battery interface.

- A new **Lithium-Ion Battery, Deformed Geometry** model wizard entry has been added.

- The **Nondeforming Boundary** and **Deforming Electrode Surface** multiphysics coupling features are now available with a Battery Design Module license.

GENERAL NEW FUNCTIONALITY IN THE CHEMICAL TRANSPORT INTERFACES

- The **Porous Material** node, used for defining and sharing properties for modeling porous media processes, has been significantly improved.
  - The **Phase-Specific Properties** section lists all properties requested by physics features with support for a porous material.
  - For each phase-specific property, the phase is indicated in the right-most column.
  - When requested, multiple entries of the same quantity, one for each phase, can be listed.
  - Tables listing properties can be sorted using any of the columns.
  - By right-clicking any property, it is now possible to select **Go to Requesting Node** to navigate to where it is used. In the same manner, it is also possible to select **Go to Source** to inspect how the property is defined.
  - The **Homogenized Properties** section lists all properties that are not phase specific, like the permeability. It also lists properties requested by interfaces without support for a porous material. The homogenized properties can be entered directly by selecting **Locally defined** in the **Material** list.
  - When defining properties in a **Fluid** or **Solid** subnode to **Porous Material**, **Locally defined** can be selected, in the **Material** list, to enter the properties directly.

- A new **Reacting Flow** multiphysics coupling feature has been introduced. In addition to isothermal flow, it also supports nonisothermal flow, provided that both a **Chemistry** interface and a **Heat Transfer** interface are included in the coupling.

Opening a model created in a previous version, the old coupling feature is migrated into a new one.

- Postprocessing variables for the resulting flow rates (mass or molar) on boundaries are now available in all mass transport interfaces and are listed under the respective boundary conditions used. They can, for example, be used to evaluate the mass conservation.

- The out-of-plane geometry extension can now be specified in all mass transport interfaces. In 2D, the out-of-plane thickness is available, while in 1D, it is the cross-sectional area. The out-of-plane dimension is accounted for in the governing equations and in postprocessing variables for the flow rate.

- A new **Out-of-Plane Flux** feature is available in the Transport of Diluted Species and Transport of Concentrated Species interfaces. It can be used to model flux out of slender geometries in 2D and 1D.

- The **Partition Condition** feature now includes the possibility to specify a user-defined relation, possibly nonlinear, between the concentration on the two sides.

- A new **Porous Medium** feature is available in the Transport of Concentrated Species interface. The new feature replaces the Porous Media Transport Properties feature. When opening a model created in an older version, the previous features are kept in the model.

- A new **Transport of Concentrated Species in Porous Media** interface is available in the **Chemical Species Transport** branch when adding a physics interface.

REACTING FLOW, DILUTED SPECIES NOW SUPPORTS TURBULENT FLOW

- The **Reacting Flow, Diluted Species** coupling feature can now be used to model transport of dilute solutes in a fluid subjected to turbulent flow. The coupling feature applies turbulence modeling for the species transport. Turbulent mixing is accounted for, and mass transfer close to walls is modeled using wall functions.

- When licensed to use models for turbulent flow, multiphysics interfaces are available in the **Turbulent Flow, Diluted Species** group under **Reacting Flow** in the Chemical Species Transport branch.

- The **Turbulent Mixing** node is now obsolete. It was previously available under the **Transport Properties** feature in the Transport of Diluted Species and Transport of Concentrated Species interfaces.

CHEMISTRY INTERFACE IMPROVEMENTS

- It is no longer needed to explicitly type in variable names to specify the mixture composition in the **Species Matching** section. Instead, use the new **Species solved for** list to first select a mass transport interface solving for concentrations. Then, for each of the bulk species in the Chemistry interface, select which of the concentrations to use.

- Using the Chemistry interface, it is no longer possible to define a species as solvent when modeling a concentrated mixture (when **Type** is set to **Concentrated Species** in the **Mixture Properties** section).

THERMODYNAMICS

- The **Thermodynamics** wizards can now be started from the **Physics** toolbar.

- The filter search used when including species in a thermodynamic system has been improved. Species whose molecular formula includes any combination of the given atoms are now shown. When searching for ammonia, NH3 or H3N can hence be used. It is also possible to filter species using aliases defined in the database. For example, "dihydrogen oxide" can be used to find water.

- The Davidson model (including pressure correction) can now be used for the dynamic viscosity.

- Second derivatives, with respect to temperature and pressure, are now available for the density and the specific volume.

New and Updated Applications and Models in Version 6.0

- Electrochemical Capacitor with Porous Electrodes
- Electrochemical Capacitor with Side Reactions
- Heterogeneous Electrode Geometry Generation
- Heterogeneous NMC Electrode
- Homogenizing a Heterogeneous Electrode Model
- Lithium Plating with Deformation
- Updated *li_battery_1d* and *li_battery_single_particle* tutorials to use the new rate constant option for specifying the exchange current density in lithium insertion kinetics. Also, a few parameters for these two models were updated for consistency with the reference paper.

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

New Functionality in Version 6.0

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 6.0 includes an upgraded version of the Parasolid kernel. As a result, a number of stability issues have been fixed, which makes geometry modeling operations and the import of CAD models more robust.

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

- The **Detect Interferences** tool has been improved to better present and visualize the detected touches, intersections, gaps, and containments.

- The new **Projection** feature can be used to project 3D objects and entities to work planes and 2D geometries.

New Functionality in the Design Module in Version 6.0

- All buttons for constraints and dimensions on the **Sketch** toolbar are now sticky, so that after adding a constraint or dimension, it is possible to continue with adding constraints or dimensions of the same kind without the need to select the button again. Previously only the smart **Constraint** and **Dimension** buttons were sticky.

- When a constraint or dimension button is selected from the **Sketch** toolbar, the mouse cursor is decorated with the icon for the constraint or dimension to indicate which button is selected.

New Functionality in LiveLink™ for AutoCAD® in Version 6.0

- The LiveLink™ interface now supports AutoCAD® 2022.

- The new offline synchronization functionality enables synchronization when COMSOL Multiphysics and the CAD software cannot be run side by side on the same computer. A synchronization file, which can contain one or several versions of the geometry, is prepared in advance using the LiveLink add-in inside the CAD software. The synchronization file is then loaded by the LiveLink feature in a COMSOL model. Synchronization files can also be saved in response to a request saved to file from the LiveLink feature. Offline synchronization supports parametric sweeps and can be used on all supported operating systems. In addition to being installed by the COMSOL installer, the LiveLink add-in is also installed by the COMSOL Client installer, available for download from https://www.comsol.com. Such an installation enables the functionality for preparing designs for synchronization, saving files for offline synchronization, and running simulation apps that synchronize geometry.

New Functionality in LiveLink™ for Inventor® in Version 6.0

- The LiveLink™ interface now supports Inventor® 2022.

- The new offline synchronization functionality enables synchronization when COMSOL Multiphysics and the CAD software cannot be run side by side on the same computer. A synchronization file, which can contain one or several versions of the geometry, is prepared in advance using the LiveLink add-in inside the CAD software. The synchronization file is then loaded by the LiveLink feature in a COMSOL model. Synchronization files can also be saved in response to a request saved to file from the LiveLink feature. Offline synchronization supports parametric sweeps and can be used on all supported operating systems. In addition to being installed by the COMSOL installer, the LiveLink add-in is also installed by the COMSOL Client installer, available for download from https://www.comsol.com. Such an installation enables the functionality for preparing designs for synchronization, saving files for offline synchronization, and running simulation apps that synchronize geometry.

- The new tutorial *Stress Analysis of a Pipe Fitting* has been added to demonstrate how to set up a simulation on a 2D cross section of a 3D geometry that has been synchronized using the LiveLink™ interface.

New Functionality in LiveLink™ for PTC® Creo® Parametric™ in Version 6.0

- Support for PTC® Creo® Parametric™ 4.0, 5.0, and 6.0 by the LiveLink™ interface will be removed in the next major release of COMSOL Multiphysics®. File import of PTC® Creo® Parametric™ 4.0, 5.0, and 6.0 files will continue to be supported.

- The LiveLink™ interface now supports PTC® Creo® Parametric™ 8.0.

- The new offline synchronization functionality enables synchronization when COMSOL Multiphysics and the CAD software cannot be run side by side on the same computer. A synchronization file, which can contain one or several versions of the geometry, is prepared in advance using the LiveLink add-in inside the CAD software. The synchronization file is then loaded by the LiveLink feature in a COMSOL model. Synchronization files can also be saved in response to a request saved to file from the LiveLink feature. Offline synchronization supports parametric sweeps and can be used on all supported operating systems. In addition to being installed by the COMSOL installer, the LiveLink add-in is also installed by the COMSOL Client installer, available for download from https://www.comsol.com. Such an installation enables the functionality for preparing designs for synchronization, saving files for offline synchronization, and running simulation apps that synchronize geometry.

- The new tutorial *Stress Analysis of a Pipe Fitting* has been added to demonstrate how to set up a simulation on a 2D cross section of a 3D geometry that has been synchronized using the LiveLink™ interface.

New Functionality in LiveLink™ for Revit® in Version 6.0

The LiveLink™ interface now supports Revit® 2022.

New Functionality in LiveLink™ for Solid Edge® in Version 6.0

- The LiveLink™ interface now supports Solid Edge® 2022.

- The new offline synchronization functionality enables synchronization when COMSOL Multiphysics and the CAD software cannot be run side by side on the same computer. A synchronization file, which can contain one or several versions of the geometry, is prepared in advance using the LiveLink add-in inside the CAD software. The synchronization file is then loaded by the LiveLink feature in a COMSOL model. Synchronization files can also be saved in response to a request saved to file from the LiveLink feature. Offline synchronization supports parametric

sweeps and can be used on all supported operating systems. In addition to being installed by the COMSOL installer, the LiveLink add-in is also installed by the COMSOL Client installer, available for download from https://www.comsol.com. Such an installation enables the functionality for preparing designs for synchronization, saving files for offline synchronization, and running simulation apps that synchronize geometry.

- The new tutorial *Stress Analysis of a Pipe Fitting* has been added to demonstrate how to set up a simulation on a 2D cross section of a 3D geometry that has been synchronized using the LiveLink™ interface.

New Functionality in LiveLink™ for SOLIDWORKS® in Version 6.0

- The LiveLink™ interface now supports SOLIDWORKS® 2022.

- The new offline synchronization functionality enables synchronization when COMSOL Multiphysics and the CAD software cannot be run side by side on the same computer. A synchronization file, which can contain one or several versions of the geometry, is prepared in advance using the LiveLink add-in inside the CAD software. The synchronization file is then loaded by the LiveLink feature in a COMSOL model. Synchronization files can also be saved in response to a request saved to file from the LiveLink feature. Offline synchronization supports parametric sweeps and can be used on all supported operating systems. In addition to being installed by the COMSOL installer, the LiveLink add-in is also installed by the COMSOL Client installer, available for download from https://www.comsol.com. Such an installation enables the functionality for preparing designs for synchronization, saving files for offline synchronization, and running simulation apps that synchronize geometry.

CFD Module

New Functionality in Version 6.0

- The Rotating Machinery, High Mach Number Flow interfaces are now available for laminar flow and for turbulent flow using either the k-epsilon or the Spalart–Allmaras turbulence models.

- Boundary-level interfaces for Phase Field Thin-Film Flow, Shell and Phase-Field Thin-Film Flow, Edge have been added under the Moving Interface section in the Mathematics branch, and predefined Two-Phase Thin-Film Flow, Phase Field multiphysics couplings to the corresponding Thin-Film Flow interfaces have been added under the Multiphase Flow section in the Fluid Flow branch.

- A Fluid-Solid Interaction, Fixed Geometry interface for Two-Phase Flow, Phase Field has been added under the Fluid-Structure Interaction section in the Fluid Flow branch.

- The Brinkman Equations, Free and Porous Media Flow and Darcy's Law interfaces have a new **Porous Medium** feature that replaces the old **Fluid and Matrix Properties** feature. The new feature contains two subfeatures, **Porous Matrix** and **Fluid**, in which the characteristic properties can be specified directly or by linking to a **Material** node.

- A predefined multiphysics coupling between the Brinkman Equations and Heat Transfer in Porous Media interfaces has been added under the Nonisothermal Flow section in the Fluid Flow branch.

- The Brinkman Equations and Free and Porous Media Flow interfaces have a new option, **Porous treatment of no slip condition**, with two options, **Standard no slip formulation** and **Porous slip**, where the latter one is suitable for models with under-resolved boundary layers.

- Automatic wall treatment has been added for the LES interfaces. For nonisothermal LES, thermal wall functions are automatically added when **Wall treatment** is set to **Automatic** in the LES interface.

- Source terms for mass and momentum have been added in the Shallow Water Equations interfaces.

- The **Phase and Porous Media Transport Properties** node in the Phase Transport in Porous Media interface has a new **Capillary diffusion** option in the **Capillary pressure** setting.

- The Phase Transport Mixture Model interfaces have a new **Low dispersed phase concentration** option in the **Mixture Model** multiphysics coupling node.

- The **Flow condition** setting in the **Grille** feature has a new **Loss coefficient** option for quadratic losses.

- The **Interior Fan** feature has a new **Prescribed flux** option in the **Flow condition** setting.

- A warning for missing **Wall Distance Initialization** study steps will be displayed when this step is missing for RANS turbulence models requiring the wall distance.

New Model in Version 6.0

DAM BREAKING ON A COLUMN, LEVEL SET

This is a transient model for the impact of a water wave on a column. A body of water 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.

Backward Compatibility with Version 5.6 and Earlier

BRINKMAN EQUATIONS, FREE AND POROUS MEDIA FLOW AND DARCY'S LAW

When opening a model created in version 5.6 or earlier, the old **Fluid and Matrix Properties** feature is not migrated, but a warning message is prompted. This feature should be manually replaced by the new **Porous Medium** feature (which adds **Porous Matrix** and **Fluid** subfeatures) to allow an updated definition of the material properties. Support for obsolete features is not ensured in future versions; thus it is strongly recommended to transfer the old feature to the new feature.

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("inl1").
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

Because 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");
```

Chemical Reaction Engineering Module

New Functionality in Version 6.0

GENERAL NEW FUNCTIONALITY

- The **Porous Material** node, used for defining and sharing properties for modeling porous media processes, has been significantly improved.

    - The **Phase-Specific Properties** section lists all properties requested by physics features with support for a porous material.

    - For each phase-specific property, the phase is indicated in the right-most column.

    - When requested, multiple entries of the same quantity, one for each phase, can be listed.

    - Tables listing properties can be sorted using any of the columns.

    - By right-clicking any property, it is now possible to select **Go to Requesting Mode** to navigate to where it is used. In the same manner, it is also possible to select **Go to Source** to inspect how the property is defined.

    - The **Homogenized Properties** section lists all properties that are not phase specific, like the permeability. It also lists properties requested by interfaces without support for a porous material. The homogenized properties can be entered directly by selecting **Locally defined** in the **Material** list.

    - When defining properties in a **Fluid**, **Solid**, or **Pellet** subnode to **Porous Material**, **Locally defined** can be selected, in the **Material** list, to enter the properties directly.

- A new subnode **Pellet** is available for **Porous Material**. The new subnode is used to provide properties for solving mass and energy transport inside porous pellets in a packed bed.

- A new **Reacting Flow** multiphysics coupling feature has been introduced. In addition to isothermal flow, it now also supports nonisothermal flow, provided that both a **Chemistry** interface and a **Heat Transfer** interface are included in the coupling. Opening a model created in a previous version, the old coupling feature is migrated into a new one.

- Postprocessing variables for the resulting flow rates (mass or molar) on boundaries are now available. These are available in all mass transport interfaces, and are listed

under the respective boundary conditions used. They can, for example, be used to evaluate the mass conservation.

- The out-of-plane geometry extension can now be specified in all mass transport interfaces. In 2D, the out-of-plane thickness is available, while in 1D, it is the cross-sectional area. The out-of-plane dimension is accounted for in the governing equations and in postprocessing variables for the flow rate.

- A new **Out-of-Plane Flux** feature is available in the Transport of Diluted Species and Transport of Concentrated Species interfaces. It can be used to model flux out of slender geometries in 2D and 1D.

- The **Partition Condition** feature now includes the possibility to specify a user-defined relation, possibly nonlinear, between the concentration on the two sides.

REACTING FLOW, DILUTED SPECIES NOW SUPPORTS TURBULENT FLOW

- The **Reacting Flow, Diluted Species** coupling feature can now be used to model transport of dilute solutes in a fluid subjected to turbulent flow. The coupling feature applies turbulence modeling for the species transport. Turbulent mixing is accounted for, and mass transfer close to walls is modeled using wall functions.

- When licensed to use models for turbulent flow, multiphysics interfaces are available in the **Turbulent Flow, Diluted Species** group under **Reacting Flow** in the **Chemical Species Transport** branch.

- The **Turbulent Mixing** node is now obsolete. It was previously available under the **Transport Properties** feature in the Transport of Diluted Species and Transport of Concentrated Species interfaces.

CHEMISTRY INTERFACE IMPROVEMENTS

- It is no longer needed to explicitly type in variable names to specify the mixture composition in the **Species Matching** section. Instead, use the new **Species solved for** list to first select a mass transport interface solving for concentrations. Then, for each of the bulk species in the Chemistry interface, select which of the concentrations to use.

- Using the Chemistry interface, it is no longer possible to define a species as solvent when modeling a concentrated mixture (when **Type** is set to **Concentrated Species** in the **Mixture Properties** section).

- To create variables available in pellets of a packed bed, select the **Define variables for porous pellets** check box in the **Pellet Chemistry** section (seen in the Settings window when selecting the interface). The check box was formerly called **Define variables in extra dimension** and was available in a section named **Extra Dimension**.

- New **Packed Bed** features are available in the Transport of Diluted Species and Transport of Concentrated Species interfaces. The features can be used to study mass transport in beds filled with reactive pellets and solves for interparticle transport (surrounding the pellets) as well as intraparticle transport (inside the pellets). The properties of the bed-filling porous pellets are defined using a **Porous Material**, including one or several **Pellet** subnodes (one per pellet type). The **Packed Bed** features replace the previous **Reacting Pellet Bed** features. Opening a model created in an older version, the previous features are kept in the model.

- The **Heat Transfer in Packed Beds** interface can be used to solve for the temperature in a bed filled with porous pellets. Similar to the mass transport feature, both interparticle energy transport (surrounding the pellets) as well as intraparticle energy transport (inside the pellets) is solved for.

- New **Porous Catalyst** features are available in the Transport of Diluted Species and Transport of Concentrated Species interfaces. The features can be used to study adsorption and desorption as well as heterogeneous reactions in porous media.

- A new **Porous Medium** feature is available in the Transport of Concentrated Species interface. The new feature replaces the Porous Media Transport Properties feature. Opening a model created in an older version, the previous features are kept in the model.

- A new **Transport of Concentrated Species in Porous Media** interface is available in the Chemical Species Transport branch when adding a physics interface.

- Two new multiphysics interfaces, **Transport of Diluted Species in Porous Catalysts** and **Transport of Concentrated Species in Porous Catalysts**, are available in the **Reacting Flow in Porous Media** group of the **Chemical Species Transport** branch when adding physics interfaces.

- Two new multiphysics interfaces, **Transport of Diluted Species in Packed Beds** and **Transport of Concentrated Species in Packed Beds**, are available in the **Reacting Flow in Porous Media** group of the **Chemical Species Transport** branch when adding physics interfaces.

- Physics interfaces using any of the **Porous Medium**, **Porous Catalyst**, or **Packed Bed** features can be created and automatically set up using the **Generate Space-Dependent Model** node in the **Reaction Engineering** interface.

- The **Porous Catalyst** and **Packed Bed** features include the possibility to solve for surface species adsorbed to the solid substrate in a porous medium domain.

  - Selecting an interface containing any of these features in a **Generate Space-Dependent Model** node, the surface reaction kinetics defined in the **Reaction Engineering** interface will be transferred to the space-dependent model.

  - When none of these domain features are created by the **Generate Space-Dependent Model** feature, the surface reaction kinetics are instead implemented in a **Surface Reactions** interface and solved for on boundaries adjacent to the model geometry.

  - The **Surface reactions on** setting is no longer present in the **Generate Space-Dependent Model** node.

THERMODYNAMICS

- The **Thermodynamics** wizards can now be started from the **Physics** toolbar.

- The filter search used when including species in a thermodynamic system has been improved. Species whose molecular formula includes any combination of the given atoms are now shown. When searching for ammonia, NH3 or H3N can hence be used. It is also possible to filter species using aliases defined in the database. For example, "dihydrogen oxide" can be used to find water.

- The Davidson model (including pressure correction) can now be used for the dynamic viscosity.

- Second derivatives, with respect to temperature and pressure, are now available for the density and the specific volume.

New Models in Version 6.0

CUBIC AUTOCATALYSIS: EXPLORING THE GRAY–SCOTT MODEL

A deceptively simple system of two species and two reactions, describing the autocatalytic conversion of a substance, is shown to display a surprisingly exotic behavior. Starting from a 0D CSTR description, a 2D reaction-diffusion problem is formulated showing rich patterns stemming from the multistability of the underlying reaction network.

GOLD RECYCLING THROUGH OXIDATIVE DISSOLUTION

This model studies the oxidative dissolution of this noble metal in an air-saturated cyanide solution. The system encompasses three phases: a gaseous phase (air), an aqueous phase, and a solid gold phase. The system is assumed to be homogeneous on

a macroscopic scale — for example, fine particulate matter dispersed in water, which is continuously agitated by a stream of air bubbles.

MASS TRANSFER FROM A THIN DOMAIN

This model demonstrates how mass transfer out from a thin domain can be approximated with a 2D model using the Out-of-Plane Flux feature. To check the validity of the 2D approximation, the equivalent 3D model is also solved. Additionally, this model illustrates how the out-of-plane thickness affects the relative error of the 2D approximation and by how much the computational time can be decreased.

OSCILLATIONS IN METABOLIC REACTION NETWORKS

Under certain conditions, reaction rates involved in the glycolysis may exhibit a limit cycle, where the concentrations vary periodically in a manner not usually seen in chemical kinetics. Here, a lumped kinetic model from the literature is used to study how the rate of consumption of glucose varies along with the coenzyme NADH and the energy carrier ATP. The time evolution of the swings in concentration is studied in a homogeneous system, and transport processes across the cell membrane is treated in an implicit manner.

TURBULENT MIXING IN A STIRRED TANK

This example shows how the mixing capability of a stirred vessel can be evaluated. To efficiently reach the steady operational conditions, the turbulent flow is solved for using a frozen rotor analysis followed by a time-dependent simulation. When a quasisteady flow field has developed, the turbulent mixing of a trace species is simulated, and the time until it is fully mixed is evaluated.

Updated Applications in Version 6.0

STEAM REFORMER

The model has been extended to include a water-gas shift reaction in order to study carbon monoxide formation. Three different setups involving cocurrent and countercurrent flow of the heating media are modeled, and the impact on the carbon monoxide production is evaluated.

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"});
```

Composite Materials Module

New Functionality in Version 6.0

IMPROVEMENTS TO SAFETY FACTOR COMPUTATIONS

Some of the models for safety factor computation in the **Safety** node have been updated.

- In the Tsai–Hill criterion, there is now a separate input for the compressive strengths.

- The **Modified Tsai-Hill** criterion, intended for 2D analysis, has been removed from the list of failure models. To obtain the same effect, use the **Tsai-Hill** criterion, and select the new **Use plane stress formulation** check box. When you open an old model that contained the **Modified Tsai-Hill** criterion, it will automatically be updated to the new settings.

- 2D formulations of the Tsai–Wu and Hashin criteria have been added. To use them, choose **Tsai-Wu orthotropic** or **Hashin** as the failure criterion and then select the new **Use plane stress formulation** check box.

- In the Layered Shell interface, several new criteria were added or extended: The Norris and Azzi–Tsai–Hill criteria are new. The Tsai–Wu and Tsai–Hill criteria have been extended with a 2D formulation.

NEW TYPE OF FOLD-LINE CONSTRAINTS IN LAYERED SHELL

A new type of constraint, called a *simplified fold-line constraint*, has been implemented in the Layered Shell interface. It is computationally more efficient than the standard full fold-line constraint but is less accurate for thick shells. The new constraint type is particularly advantageous for large geometrically linear problems because it has a special linear formulation in that case.

You select the type of fold-line constraint in the **Fold-Line Settings** section of the interface settings.

NEW COUPLING BETWEEN SHELL AND LAYERED SHELL

The new **Layered Shell-Shell Connection** multiphysics coupling makes it possible to mix the Shell and Layered Shell interfaces by defining different stack members within a single layup using either of the interfaces. This modeling approach is sometimes called the *multiple model method*.

STACK MEMBER SELECTION

In the **Layered Linear Elastic Material** in the Shell interface, it is now possible to select one single stack member from a **Layered Material Stack** by clearing the **Use all layers** check box. The main use for this facility is when mixing different shell formulations on the same boundary in the *multiple model method*.

NEW OPERATORS FOR LAYERED MATERIALS

For evaluation at a certain location in the extra dimension (the thickness direction), two new physics scoped operators have been added: `<phys>.atxd0` (for evaluation in points) and `<phys>.atxd1` (for evaluation on edges). These operators take two arguments, the through-thickness location and the expression to be evaluated — for example, `lshell.atxd1(2*th,mean(lshell.mises))`.

The new operators are available in the Layered Shell interface as well as for the layered materials in the Shell and Membrane interfaces.

New Models in Version 6.0

ANALYSIS OF A COMPOSITE BLADE USING MULTIPLE MODEL METHOD

This example demonstrates the modeling of a sandwiched composite blade made up of carbon-epoxy, glass-vinylester, and PVC foam materials. To model the sandwich composite structure, three different methods are used: layerwise theory, equivalent single layer (ESL) theory, and a multiple model method. Modal as well as frequency response analyses are performed for the specified loading, and the multiple model method is compared with the traditional layerwise and ESL theory in terms of performance and accuracy in the results, including the through-thickness stress distribution.

Corrosion Module

New Functionality in Version 6.0

GENERAL NEW FUNCTIONALITY IN THE ELECTROCHEMISTRY INTERFACES

- The Highly Conductive Porous Electrode domain node is no longer available when using a Primary Current Distribution type in the Primary Current Distribution and Secondary Current Distribution interfaces.

- The **Thermodynamic Equilibrium** electrode kinetics option is no longer available in the **Highly Conductive Porous Electrode** domain node in the Primary Current Distribution, Secondary Current Distribution, and Tertiary Current Distribution interfaces.

- A new **Adsorbing-Desorbing Species** section has been added to the **Electrode Surface**, **Porous Electrode**, and related nodes in the electrochemistry interfaces. The section can be used to model adsorption-desorption and the resulting surface coverage.

- The implementation of the **Concentration Dependent Kinetics** kinetics type in the **Electrode Reaction**, **Porous Electrode Reaction**, and related nodes has changed in order to improve accuracy and numerical convergence for well-posed problems. Prior to version 6.0, the `max()` operator was used on the CO and CR parameter expressions before applying them internally in the definition expression for the `iloc` variable. The usage of the `max` operator has been removed in version 6.0. To get old models to behave exactly as before, rewrite the CO and CR parameter expressions as `max(CO,eps^2)` and `max(CR,eps^2)`, respectively.

NEW CORROSION MODULE-SPECIFIC FUNCTIONALITY

- The **Corrosion, Deformed Geometry** model wizard entries now make use of the new Deformed Geometry formulation. The new multiphysics interfaces also use new **Nondeforming Boundary** and **Deforming Electrode Surface** multiphysics features.

- A Seawater material entry was added to the Corrosion material library. The new material contains temperature- and salinity-dependent electrolyte conductivity data.

- The **Phase Initialization** study node for Level Set and Phase Field interfaces is now available.

- The Corrosion material library polarization data has been cleansed of nonmonotonic polarization data, when deemed caused by noise.

- A new **Cathodic Protection** model wizard entry has been added.

- A number of new Cathodic Protection feature nodes have been added to the Primary and Secondary Current Distribution interfaces and are active in the new Cathodic Protection model wizard entry: **Connection Surface**, **Impressed Current Surface**, **Passive Metal Surface**, **Protected Metal Surface**, **Thin Passive Metal Surface**, **Thin Protected Metal Surface**, **Connection Point** (under **Edge Electrode**), and **Impressed Current Point**.

- Stabilization sections are added to the Current Distribution, Boundary Elements interface for dissolving-depositing species, which are enabled when Deformed Geometry is active.

GENERAL NEW FUNCTIONALITY IN THE CHEMICAL TRANSPORT INTERFACES

- The **Porous Material** node, used for defining and sharing properties for modeling porous media processes, has been significantly improved.

  - The **Phase-Specific Properties** section lists all properties requested by physics features with support for a porous material.

  - For each phase-specific property, the phase is indicated in the right-most column.

  - When requested, multiple entries of the same quantity, one for each phase, can be listed.

  - Tables listing properties can be sorted using any of the columns.

  - By right-clicking any property, it is now possible to select **Go to Requesting Mode** to navigate to where it is used. In the same manner, it is also possible to select **Go to Source** to inspect how the property is defined.

  - The **Homogenized Properties** section lists all properties that are not phase specific, like the permeability. It also lists properties requested by interfaces without support for a porous material. The homogenized properties can be entered directly by selecting **Locally defined** in the **Material** list.

  - When defining properties in a **Fluid** or **Solid** subnode to a **Porous Material** node, **Locally defined** can be selected, in the **Material** list, to enter the properties directly.

- A new **Reacting Flow** multiphysics coupling feature has been introduced. In addition to isothermal flow, it now also supports nonisothermal flow, provided that both a **Chemistry** interface and a **Heat Transfer** interface are included in the coupling. Opening a model created in a previous version, the old coupling feature is migrated into a new one.

- Postprocessing variables for the resulting flow rates (mass or molar) on boundaries are now available. These are available in all mass transport interfaces and are listed under the respective boundary conditions used. These can, for example, be used to evaluate the mass conservation.

- The out-of-plane geometry extension can now be specified in all mass transport interfaces. In 2D the out-of-plane thickness is available, while in 1D it is the cross-sectional area. The out-of-plane dimension is accounted for in the governing equations and in postprocessing variables for the flow rate.

- A new **Out-of-Plane Flux** feature is available in the Transport of Diluted Species and Transport of Concentrated Species interfaces. It can be used to model flux out of slender geometries in 2D and 1D.

- The **Partition Condition** feature now includes the possibility to specify a user-defined relation, possibly nonlinear, between the concentration on the two sides.

- A new **Porous Medium** feature is available in the Transport of Concentrated Species interface. This feature replaces the Porous Media Transport Properties feature. Opening a model created in an older version, the previous features are kept in the model.

- A new **Transport of Concentrated Species in Porous Media** interface is available in the Chemical Species Transport branch when adding a physics interface.

REACTING FLOW, DILUTED SPECIES NOW SUPPORTS TURBULENT FLOW

- The **Reacting Flow, Diluted Species** coupling feature can now be used to model transport of dilute solutes in a fluid subjected to turbulent flow. The coupling feature applies turbulence modeling for the species transport. Turbulent mixing is accounted for, and mass transfer close to walls is modeled using wall functions.

- When licensed to use models for turbulent flow, multiphysics interfaces are available in the **Turbulent Flow, Diluted Species** group under **Reacting Flow** in the **Chemical Species Transport** branch.

- The **Turbulent Mixing** node is now obsolete. It was previously available under the **Transport Properties** feature in the Transport of Diluted Species and Transport of Concentrated Species interfaces.

CHEMISTRY INTERFACE IMPROVEMENTS

- It is no longer needed to explicitly type in variable names to specify the mixture composition in the **Species Matching** section. Instead, use the new **Species solved for** list to first select a mass transport interface solving for concentrations. Then, for

each of the bulk species in the Chemistry interface, select which of the concentrations to use.

• Using the Chemistry interface, it is no longer possible to define a species as solvent when modeling a concentrated mixture (when **Type** is set to **Concentrated Species** in the **Mixture Properties** section).

New and Updated Models in Version 6.0

• New Oxide Jacking of Reinforced Concrete tutorial

• New Pitting Corrosion tutorial

• The Stress Corrosion model was extended to account for deformations at the corrosion defect also in the stress calculation

• The *oil_platform*, *ship_hull*, and *monopile* tutorials were updated to use the new Cathodic Protection interface and related features.

• Updated *localized_corrosion_ls* and *localized_corrosion_pf* models using the Time Dependent with Phase Initialization study node.

• Resistor-Modulated Pipeline Cathodic Protection (changed name from prior version).

• The following models are updated to use the new Deformed Geometry node: *crevice_corrosion_with_deformation*, *galvanic_corrosion_with_deformation*, *localized_corrosion*, and *under_deposit_corrosion*.

• The *oil_platform*, *anode_film_resistance*, and *monopile* tutorials were updated to use Seawater material for the electrolyte conductivity.

ECAD Import Module

New Functionality in Version 6.0

The ECAD Import Module includes significant improvements for generating the 2D geometries from the layouts imported from IPC-2581, ODB++, and GDS files. The geometry for complex layouts is now generated in a more robust way, which eliminates the need for slower fallback methods and thereby improves the performance for the import.

Electrochemistry Module

New Functionality in Version 6.0

GENERAL NEW FUNCTIONALITY IN THE ELECTROCHEMISTRY INTERFACES

- The Highly Conductive Porous Electrode domain node is no longer available when using a Primary Current Distribution type in the Primary Current Distribution and Secondary Current Distribution interfaces.

- The **Thermodynamic Equilibrium** electrode kinetics option is no longer available in the **Highly Conductive Porous Electrode** domain node in the Primary Current Distribution, Secondary Current Distribution, and Tertiary Current Distribution interfaces.

- A new **Adsorbing-Desorbing Species** section has been added to the **Electrode Surface**, **Porous Electrode**, and related nodes in the electrochemistry interfaces. The section can be used to model adsorption-desorption and the resulting surface coverage.

- The implementation of the **Concentration Dependent Kinetics** kinetics type in the **Electrode Reaction**, **Porous Electrode Reaction**, and related nodes has changed in order to improve accuracy and numerical convergence for well-posed problems. Prior to version 6.0, the `max()` operator was used on the CO and CR parameter expressions before applying them internally in the definition expression for the `iloc` variable. The usage of the `max` operator has been removed in version 6.0. To get old models to behave exactly as before, rewrite the CO and CR parameter expressions as `max(CO, eps^2)` and `max(CR, eps^2)`, respectively.

- New Adsorption-Desorption Voltammetry tutorial added to the Electrochemistry Module Application Library.

- The **Porous Material** node, used for defining and sharing properties for modeling porous media processes, has been significantly improved.

  - The **Phase-Specific Properties** section lists all properties requested by physics features with support for a porous material.

  - For each phase-specific property, the phase is indicated in the right-most column.

  - When requested, multiple entries of the same quantity, one for each phase, can be listed.

  - Tables listing properties can be sorted using any of the columns.

  - By right-clicking any property, it is now possible to select **Go to Requesting Mode** to navigate to where it is used. In the same manner, it is also possible to select **Go to Source** to inspect how the property is defined.

  - The **Homogenized Properties** section lists all properties that are not phase specific, like the permeability. It also lists properties requested by interfaces without support for a porous material. The homogenized properties can be entered directly by selecting **Locally defined** in the **Material** list.

  - When defining properties in a **Fluid** or **Solid** subnode to a **Porous Material** node, **Locally defined** can be selected, in the **Material** list, to enter the properties directly.

- A new **Reacting Flow** multiphysics coupling feature has been introduced. In addition to isothermal flow, it now also supports nonisothermal flow, provided that both a **Chemistry** interface and a **Heat Transfer** interface are included in the coupling. Opening a model created in a previous version, the old coupling feature is migrated into a new one.

- Postprocessing variables for the resulting flow rates (mass or molar) on boundaries are now available in all mass transport interfaces and are listed under the respective boundary conditions used. They can, for example, be used to evaluate the mass conservation.

- The out-of-plane geometry extension can now be specified in all mass transport interfaces. In 2D the out-of-plane thickness is available, while in 1D it is the cross-sectional area. The out-of-plane dimension is accounted for in the governing equations and in postprocessing variables for the flow rate.

- A new **Out-of-Plane Flux** feature is available in the Transport of Diluted Species and the Transport of Concentrated Species interfaces. It can be used to model flux out of slender geometries in 2D and 1D.

- The **Partition Condition** feature now includes the possibility to specify a user-defined relation, possibly nonlinear, between the concentration on the two sides.

- A new **Porous Medium** feature is available in the Transport of Concentrated Species interface. The new feature replaces the Porous Media Transport Properties feature. If you open a model created in an older version, the previous features are kept in the model.

- A new **Transport of Concentrated Species in Porous Media** interface is available in the Chemical Species Transport branch when adding a physics interface.

REACTING FLOW, DILUTED SPECIES NOW SUPPORTS TURBULENT FLOW

- The **Reacting Flow, Diluted Species** coupling feature can now be used to model transport of dilute solutes in a fluid subjected to turbulent flow. The coupling feature applies turbulence modeling for the species transport. Turbulent mixing is accounted for, and mass transfer close to walls is modeled using wall functions.

- When licensed to use models for turbulent flow, multiphysics interfaces are available in the **Turbulent Flow, Diluted Species** group under **Reacting Flow** in the **Chemical Species Transport** branch.

- The **Turbulent Mixing** node is now obsolete. It was previously available under the **Transport Properties** feature in the Transport of Diluted Species and Transport of Concentrated Species interfaces.

CHEMISTRY INTERFACE IMPROVEMENTS

- It is no longer needed to explicitly type in variable names to specify the mixture composition in the **Species Matching** section. Instead, use the new **Species solved for** list to first select a mass transport interface solving for concentrations. Then, for each of the bulk species in the Chemistry interface, select which of the concentrations to use.

- Using the Chemistry interface, it is no longer possible to define a species as solvent when modeling a concentrated mixture (when **Type** is set to **Concentrated Species** in the **Mixture Properties** section).

New and Updated Models in Version 6.0

A New Adsorption-Desorption Voltammetry tutorial has been added to the Electrochemistry Module Application Library.

Electrodeposition Module

New Functionality in Version 6.0

GENERAL NEW FUNCTIONALITY IN THE ELECTROCHEMISTRY INTERFACES

- The Highly Conductive Porous Electrode domain node is no longer available when using a Primary Current Distribution type in the Primary Current Distribution and Secondary Current Distribution interfaces.

- The **Thermodynamic Equilibrium** electrode kinetics option is no longer available in the **Highly Conductive Porous Electrode** domain node in the Primary Current Distribution, Secondary Current Distribution, and Tertiary Current Distribution interfaces.

- A new **Adsorbing-Desorbing Species** section has been added to the **Electrode Surface**, **Porous Electrode**, and related nodes in the electrochemistry interfaces. The section can be used to model adsorption-desorption, and the resulting surface coverage.

- The implementation of the **Concentration Dependent Kinetics** kinetics type in the **Electrode Reaction**, **Porous Electrode Reaction**, and related nodes has changed in order to improve accuracy and numerical convergence for well-posed problems. Prior to version 6.0, the `max()` operator was used on the CO and CR parameter expressions before applying them internally in the definition expression for the `iloc` variable. The usage of the `max` operator has been removed in version 6.0. To get old models to behave exactly as before, rewrite the CO and CR parameter expressions as `max(CO,eps^2)` and `max(CR,eps^2)`, respectively.

- A new Adsorption-Desorption Voltammetry tutorial has been added to the Electrochemistry Module Application Library.

NEW ELECTRODEPOSITION MODULE-SPECIFIC FUNCTIONALITY

- The **Electrodeposition, Deformed Geometry** model wizard entries now make use of the new Deformed Geometry formulation. The new multiphysics interfaces also use new **Nondeforming Boundary** and **Deforming Electrode Surface** multiphysics features.

- **Stabilization** sections are added to the Current Distribution, Boundary Elements interface for dissolving-depositing species, which are enabled when Deformed Geometry is active.

- The **Porous Material** node, used for defining and sharing properties for modeling porous media processes, has been significantly improved.

    - The **Phase-Specific Properties** section lists all properties requested by physics features with support for a porous material.

    - For each phase-specific property, the phase is indicated in the right-most column.

    - When requested, multiple entries of the same quantity, one for each phase, can be listed.

    - Tables listing properties can be sorted using any of the columns.

    - By right-clicking any property, it is now possible to select **Go to Requesting Node** to navigate to where it is used. In the same manner, it is also possible to select **Go to Source** to inspect how the property is defined.

    - The **Homogenized Properties** section lists all properties that are not phase specific, like the permeability. It also lists properties requested by interfaces without support for a porous material. The homogenized properties can be entered directly by selecting **Locally defined** in the **Material** list.

    - When defining properties in a **Fluid** or **Solid** subnode to a **Porous Material** node, **Locally defined** can be selected, in the **Material** list, to enter the properties directly.

- A new **Reacting Flow** multiphysics coupling feature has been introduced. In addition to isothermal flow, it now also supports nonisothermal flow, provided that both a **Chemistry** interface and a **Heat Transfer** interface are included in the coupling. Opening a model created in a previous version, the old coupling feature is migrated into a new one.

- Postprocessing variables for the resulting flow rates (mass or molar) on boundaries are now available in all mass transport interfaces and are listed under the respective boundary conditions used. They can, for example, be used to evaluate the mass conservation.

- The out-of-plane geometry extension can now be specified in all mass transport interfaces. In 2D the out-of-plane thickness is available, while in 1D it is the cross-sectional area. The out-of-plane dimension is accounted for in the governing equations and in postprocessing variables for the flow rate.

- A new **Out-of-Plane Flux** feature is available in the Transport of Diluted Species and Transport of Concentrated Species interfaces. It can be used to model flux out of slender geometries in 2D and 1D.

- The **Partition Condition** feature now includes the possibility to specify a user-defined relation, possibly nonlinear, between the concentration on the two sides.

- A new **Porous Medium** feature is available in the Transport of Concentrated Species interface. The new feature replaces the Porous Media Transport Properties feature. If you open a model created in an older version, the previous features are kept in the model.

- A new **Transport of Concentrated Species in Porous Media** interface is available in the Chemical Species Transport branch when adding a physics interface.

REACTING FLOW, DILUTED SPECIES NOW SUPPORTS TURBULENT FLOW

- The **Reacting Flow, Diluted Species** coupling feature can now be used to model transport of dilute solutes in a fluid subjected to turbulent flow. The coupling feature applies turbulence modeling for the species transport. Turbulent mixing is accounted for, and mass transfer close to walls is modeled using wall functions.

- When licensed to use models for turbulent flow, multiphysics interfaces are available in the **Turbulent Flow, Diluted Species** group under **Reacting Flow** in the **Chemical Species Transport** branch.

- The Turbulent Mixing node is now obsolete. It was previously available under the **Transport Properties** feature in the Transport of Diluted Species and Transport of Concentrated Species interfaces.

CHEMISTRY INTERFACE IMPROVEMENTS

- It is no longer needed to explicitly type in variable names to specify the mixture composition in the **Species Matching** section. Instead, use the new **Species solved for** list to first select a mass transport interface solving for concentrations. Then, for each of the bulk species in the Chemistry interface, select which of the concentrations to use.

- Using the Chemistry interface, it is no longer possible to define a species as solvent when modeling a concentrated mixture (when **Type** is set to **Concentrated Species** in the **Mixture Properties** section).

New and Updated Models in Version 6.0

- A new model in the Electrodeposition Application Library: Copper Deposition in a Through-Hole Via, which demonstrates the effect of additives during electrodeposition. The new model uses the Adsorbing-Desorbing Species feature.

- The following models were updated to use the new **Deformed Geometry** node: *cu_trench_deposition*, *ecm_microbore*, *inductor_coil*, *microconnector_bump_3d*, *pulse_reverse_plating*, and *electrode_insulator_growth*.

Fatigue Module

New Functionality in Version 6.0

RANDOM VIBRATION FATIGUE

A new **Random Vibration** feature is provided. It is used to compute the fatigue life of a structure subjected to random vibrations. The response of the structure is defined by spectral moments of stress, computed from the results of a preceding random vibration analysis. Two cycle counting models are available — the **Dirlik** and **Bendat** models.

As an effect of this addition, the old Vibration Fatigue feature has been renamed to **Harmonic Vibration** in order to improve clarity.

IMPROVEMENTS TO CUMULATIVE DAMAGE

Additional S-N curve types have been added to the **Cumulative Damage** feature. You can now use an **S-N curve with R-value dependence**, an **S-N curve with mean stress dependence**, or an **S-N curve for amplitude stress**. For the **S-N curve for amplitude stress** option, you have the option of using a **Gerber**, **Goodman**, or **Soderberg** mean stress correction.

IMPROVEMENTS TO THE EVALUATION OF FATIGUE

The various features in the Fatigue interface now base the fatigue computations on stresses, strains, and energies evaluated at the integration points of the elements.

New Models in Version 6.0

BRACKET — RANDOM VIBRATION FATIGUE

This model uses a solution from a random vibration analysis of a bracket, and computes the fatigue life using different cycle counting models. The fatigue properties of the material are represented by the Basquin fatigue model.

Fuel Cell & Electrolyzer Module

New Functionality in Version 6.0

GENERAL NEW FUNCTIONALITY IN THE ELECTROCHEMISTRY INTERFACES

- The Highly Conductive Porous Electrode domain node is no longer available when using a Primary Current Distribution type in the Primary Current Distribution and Secondary Current Distribution interfaces.

- The **Thermodynamic Equilibrium** electrode kinetics option is no longer available in the **Highly Conductive Porous Electrode** domain node in the Primary Current Distribution, Secondary Current Distribution, and Tertiary Current Distribution interfaces as well as for the Lithium-Ion Battery and Battery with Binary Electrolyte interfaces.

- A new **Adsorbing-Desorbing Species** section has been added to the **Electrode Surface**, **Porous Electrode**, and related nodes in the electrochemistry interfaces. The section can be used to model adsorption-desorption and the resulting surface coverage.

- The implementation of the **Concentration Dependent Kinetics** kinetics type in the **Electrode Reaction** and **Porous Electrode Reaction** (and related) nodes has changed in order to improve accuracy and numerical convergence for well-posed problems. Prior to version 6.0, the `max()` operator was used on the CO and CR parameter expressions before applying them internally in the definition expression for the `iloc` variable. The usage of the `max` operator has been removed in version 6.0. To get old models to behave exactly as before, rewrite the CO and CR parameter expressions as `max(CO,eps^2)` and `max(CR,eps^2)`, respectively.

- New Adsorption-Desorption Voltammetry tutorial added to the Electrochemistry Module Application Library.

NEW FUEL CELL AND ELECTROLYZER-SPECIFIC FUNCTIONALITY

- A new Fuel Cell and Electrolyzer Material Library was added.

New features in the Hydrogen Fuel Cell and Water Electrolyzer interfaces:

• The "No potential constraint", "There are domains selected on interface level that are not selected by any phase node", and "Zero convective velocity" model warning messages have been added.

• A new **Water Condensation-Evaporation** domain subnode has been added to the **H2 Gas Phase** and **O2 Gas Phase** domain nodes.

• A new **Water Gas Shift Reaction** domain subnode has been added to the **H2 Gas Phase** domain node

• A new **Flux** boundary subnode has been added to the **H2 Gas Phase** and **O2 Gas Phase** domain nodes.

• A new **Separator** domain node has been added.

• Updated description of H2 Flow Channel and O2 Flow Channel domain nodes to **H2 Gas Flow Channel** and **O2 Gas Flow Channel**, respectively.

• New functionality added for gas crossover transport of H2, O2, and N2 species across the membrane domain for proton exchange and hydroxide exchange electrolyte types.

• New functionality has been added for electroosmotic water drag in the membrane domain for proton exchange and hydroxide exchange electrolyte types.

• New liquid water properties (density and dynamic viscosity) have been added to the **H2 Gas Phase** and **O2 Gas Phase** domain nodes for announcement to other interfaces, if H2O selected in the corresponding H2 or O2 gas mixture. Additionally, the

descriptions of the announced gas phase properties have been changed to **Density of gas phase** and **Dynamic viscosity of gas phase**, for clarity.

- New **H2 Gas-Electrolyte Compartment** and **O2 Gas-Electrolyte Compartment** domain nodes have been added.

- New **H2 Electrode Surface**, **O2 Electrode Surface**, **Internal H2 Electrode Surface**, and **Internal O2 Electrode Surface** boundary nodes have been added.

- A new composition option of **Humidified mixture** has been added to **H2 Inlet**, **O2 Inlet**, and **Initial Values** nodes, and to the **Composition** section of **H2 Gas Phase** and **O2 Gas Phase** nodes, if an H2O species is present in the corresponding gas mixture.

- A new composition option of **Humidified air** has been added to **O2 Inlet** and **Initial Values** nodes, and to the **Composition** section of **O2 Gas Phase** nodes, if H2O and N2 species are present in the O2 gas mixture.

- A new option of **From initial values** has been added to the **H2 Inlet** and **O2 Inlet** nodes. The default has been changed from **Mole fractions** to **From initial values**. Note that COMSOL API backward compatibility for this change is not supported.

- Corrected the built-in reference equilibrium potential calculation to account for reference pressures in the Hydrogen Fuel Cell and Water Electrolyzer interfaces.

This bug fix will affect models that have reference pressures set to values other than 1 atm.

- Built-in calculation of gas phase thermal conductivities has been added in the **H2 Gas Phase** and **O2 Gas Phase** domain nodes. The variables (named `fc.kgas_mix` or `we.kgas_mix`) can be used in, for instance, heat transfer simulations.

- The description of sigmas has changed to "Effective electrical conductivity" in nodes for gas diffusion electrodes and gas diffusion layers. Description remains as before ("Electrical Conductivity") in **Current Collector** nodes.

- The species CH4 (methane) is now supported in the hydrogen gas mixture.

- The default value of the **Electrolyte conductivity** material input in the **Electrolyte Phase** node has been changed to **From material** (from **User defined**). Note that COMSOL API backward compatibility for this change is not supported.

- A **Domain material** option has been added to the **Electrolyte material** list in the **Electrolyte Phase** node and is set as the default value. The previously available None option has been removed.

- The new permeability option for Straight Channels in H2/O2 Gas Flow Channel is now available when Darcy's law is enabled.

- The **Continuity** node has been updated to support mass fraction and pressure continuities on internal pair gas phase boundaries. The updated continuity node now supports continuity of all domain-dependent variables solved for.

- The **Porous Material** node, used for defining and sharing properties for modeling
  porous media processes, has been significantly improved.

  - The **Phase-Specific Properties** section lists all properties requested by physics
    features with support for a porous material.

  - For each phase-specific property, the phase is indicated in the right-most column.

  - When requested, multiple entries of the same quantity, one for each phase, can
    be listed.

  - Tables listing properties can be sorted using any of the columns.

  - By right-clicking any property, it is now possible to select **Go to Requesting Node**
    to navigate to where it is used. In the same manner, it is also possible to select **Go
    to Source** to inspect how the property is defined.

  - The **Homogenized Properties** section lists all properties that are not phase specific,
    like the permeability. It also lists properties requested by interfaces without
    support for a porous material. The homogenized properties can be entered
    directly by selecting **Locally defined** in the **Material** list.

  - When defining properties in a **Fluid** or **Solid**, subnode to **Porous Material**, **Locally
    defined** can be selected, in the **Material** list, to enter the properties directly.

- A new **Reacting Flow** multiphysics coupling feature has been introduced. In addition
  to isothermal flow, it also supports nonisothermal flow, provided that both a
  Chemistry interface and a **Heat Transfer** interface are included in the coupling.
  Opening a model created in a previous version, the old coupling feature is migrated
  into a new one.

- Postprocessing variables for the resulting flow rates (mass or molar) on boundaries
  are now available in all mass transport interfaces and are listed under the respective
  boundary conditions used. They can, for example, be used to evaluate the mass
  conservation.

- The out-of-plane geometry extension can now be specified in all mass transport
  interfaces. In 2D the out-of-plane thickness is available, while in 1D it is the cross-
  sectional area. The out-of-plane dimension is accounted for in the governing
  equations and in postprocessing variables for the flow rate.

- A new **Out-of-Plane Flux** feature is available in the Transport of Diluted Species and Transport of Concentrated Species interfaces. It can be used to model flux out of slender geometries in 2D and 1D.

- The **Partition Condition** feature now includes the possibility to specify a user-defined relation, possibly nonlinear, between the concentration on the two sides.

- A new **Porous Medium** feature is available in the Transport of Concentrated Species interface. The new feature replaces the Porous Media Transport Properties feature. When opening a model created in an older version, the previous features are kept in the model.

- A new **Transport of Concentrated Species in Porous Media** interface is available in the **Chemical Species Transport** branch when adding a physics interface.

REACTING FLOW, DILUTED SPECIES NOW SUPPORTS TURBULENT FLOW

- The **Reacting Flow, Diluted Species** coupling feature can now be used to model transport of dilute solutes in a fluid subjected to turbulent flow. The coupling feature applies turbulence modeling for the species transport. Turbulent mixing is accounted for, and mass transfer close to walls is modeled using wall functions.

- When licensed to use models for turbulent flow, multiphysics interfaces are available in the **Turbulent Flow, Diluted Species** group under **Reacting Flow** in the Chemical Species Transport branch.

- The **Turbulent Mixing** node is now obsolete. It was previously available under the **Transport Properties** feature in the Transport of Diluted Species and Transport of Concentrated Species interfaces.

CHEMISTRY INTERFACE IMPROVEMENTS

- It is no longer needed to explicitly type in variable names to specify the mixture composition in the **Species Matching** section. Instead, use the new **Species solved for** list to first select a mass transport interface solving for concentrations. Then, for each of the bulk species in the Chemistry interface, select which of the concentrations to use.

- Using the Chemistry interface, it is no longer possible to define a species as solvent when modeling a concentrated mixture (when **Type** is set to **Concentrated Species** in the **Mixture Properties** section).

THERMODYNAMICS

- The **Thermodynamics** wizards can now be started from the **Physics** toolbar.

- The filter search used when including species in a thermodynamic system has been improved. Species whose molecular formula includes any combination of the given atoms are now shown. When searching for ammonia, NH3 or H3N can hence be used. It is also possible to filter species using aliases defined in the database. For example, "dihydrogen oxide" can be used to find water.

- The Davidson model (including pressure correction) can now be used for the dynamic viscosity.

- Second derivatives, with respect to temperature and pressure, are now available for the density and the specific volume.

New and Updated Models in Version 6.0

- A new *alkaline_electrolyzer* model, coupled to Euler–Euler two phase laminar flow. The model also makes use of the new Separator, H2 Electrolyte-Gas Compartment, O2 Electrolyte-Gas Compartment, H2 Electrode Surface, and O2 Electrode Surface nodes.

- A new *pemfc_serpentine_flow_field* model. The model makes use of the new electroosmotic drag feature in Membrane Transport.

- A new *nonisothermal_pem_fuel_cell* model. This tutorial models the intercoupled electrochemical reactions, charge and species transport, and heat transfer in a polymer electrolyte membrane (PEM) fuel cell.

- The *pem_mea_1d* model was updated to use the Hydrogen Fuel Cell interface, including the new Membrane Transport features.

Geomechanics Module

New Functionality in Version 6.0

NONLOCAL PLASTICITY

Nonlocal plasticity has been added to **Plasticity**, **Soil Plasticity**, and **Elastoplastic Soil**. You can enable and control it in the new **Nonlocal Plasticity Model** section in the settings for these material models.

With nonlocal plasticity, the equivalent plastic strain is a distributed field, rather than defined at a point. This makes it possible to solve problems that would otherwise fail due to high strain localization.

NEW INPUT PROPERTY FOR HARDENING SOIL MODEL

In the **Hardening Soil** material model, the **Ellipse aspect ratio** has been added as a new material parameter. It has the options **From material**, **From coefficient of earth pressure at rest**, and **User defined**.

NEW INPUT PROPERTY FOR EXTENDED BARCELONA BASIC MODEL

In the **Extended Barcelona Basic** material model, the **Slope of critical state line** has been added as a new material parameter. It has the options **From material**, **Match to Mohr-Coulomb criterion**, **Match to Matsuoka-Nakai criterion**, and **User defined**.

Also, the **Match to Matsuoka-Nakai criterion** option has been added to the corresponding material parameter in the **Modified Cam-Clay** and **Modified Structured Cam-Clay** material models.

CREEP IMPROVEMENTS

The functionality in the **Creep** node has been subject to major changes:

- A new general framework gives large improvements in computational speed and memory usage. For larger models, a speedup by a factor of 10 or more can be expected.
- You can select the type of scheme that is used for the time integration of the creep equations. In the **Time Stepping** section, you can set **Method** to **Automatic, Backward Euler**, **Forward Euler**, or **Domain ODEs**. The last method provides backward compatibility with versions prior to 6.0 and is usually significantly slower for large models.

- It is no longer possible to add several **Creep** nodes acting in parallel on the same selection. When more than one creep mechanism is acting, you can instead add one or more **Additional Creep** nodes under a **Creep** node. As an effect, only one set of creep strain degrees of freedom is needed, even if several creep models are used. This gives additional performance gains for models of this type.

- The Norton–Bailey creep model has been removed. It is replaced by a general isotropic "Bailey-type" hardening formulation, which can be combined with all creep models, not only the Norton model.

- The type of equivalent stress used to determine the creep rate is now a user input. You can select **von Mises**, **Hill orthotropic**, **Pressure**, or **User defined**. Thus, nonisotropic creep can now be modeled.

- The old creep models **Potential**, **Volumetric**, and **Deviatoric** have been removed. The same functionality is now achieved either through the **User defined** creep model, or by using the new **Inelastic Strain Rate** node.

PRESCRIBED INELASTIC STRAIN RATE

A new **Inelastic Strain Rate** subnode has been added to **Linear Elastic Material**, **Layered Linear Elastic Material**, and **Nonlinear Elastic Material**. The effect of this node is similar to the existing **External Strain** node, in that it injects an inelastic strain in the material model. The difference is that here you instead specify a rate, which is then integrated into an inelastic strain contribution. By using this feature, you do not need to add an extra ODE interface when you have a rate equation describing an extra strain contribution.

The strain rate can be specified in terms of a strain tensor, a deformation gradient, an inverse deformation gradient, or three orthogonal stretches.

REDUCED INTEGRATION

The new framework for reduced integration will in many cases make the solution of models with nonlinear materials, such as plasticity and creep, significantly faster. Reduced integration is controlled from the **Quadrature Settings** section in material models such as **Linear Elastic Material**, **Nonlinear Elastic Material**, and **Elastoplastic Soil Material**.

TRANSIENT RESPONSE OF A SHALLOW FOUNDATION ON UNSATURATED SOIL

In this example, the transient response of a shallow foundation resting on an unsaturated soil stratum is investigated. The changes in pore suction due to water infiltration and evaporation is investigated using the Extended Barcelona Basic soil model. The results show the deformation of the clay stratum due to footing pressure and changes in pore suction.

Heat Transfer Module

New and Improved Functionality in Version 6.0

IMPROVED COMPUTATIONAL PERFORMANCE FOR SURFACE-TO-SURFACE RADIATION

The Surface-to-Surface Radiation interface has a new **Jacobian Contribution** section where you can control the contribution of the radiosity terms to the Jacobian matrix. The default option, **Only local contributions to radiosity**, computes an approximate of the Jacobian matrix. Combined with improved solver settings, it provides a dramatic decrease of the CPU time and memory requirements for surface-to-surface radiation computations, in particular with the hemicube method. The previously used approach, computing the full Jacobian matrix, remains available and can be activated by selecting the **Include contributions from total irradiation** option.

In addition, the Surface-to-Surface Radiation interface has a new **Store view factors on disk** option in the **View Factor** section. The view factors can be stored in the model when they have been computed once, provided that the mesh, the selection of the radiating boundaries, the radiation direction, and the external radiation sources or symmetry conditions are unchanged. This may significantly reduce the computational time for cases where the view factor calculation is demanding, typically with specular or semitransparent surfaces.

EXTENDED FEATURES FOR SURFACE-TO-SURFACE RADIATION

The **Ray shooting** option is now available as a **Surface-to-surface radiation method**, as well as in 2D axisymmetric models, for the computation of the view factors.

The user interface of the surface-to-surface radiation features has been simplified as follows:

- The new **View factor update threshold** list in the **View Factor** section makes it possible to easily control the frequency of view factor computations in time-dependent studies, when using moving symmetry planes or time-dependent surface properties.

- In the **Opaque Surface** and **Semitransparent Surface** features, a single **Define properties on each side** check box controls the display of side-specific user inputs for the

definition of the emissivity, reflectivity, transmissivity, and critical angle on two sides of a shell.

- The **Wavelength unit** for the spectral band endpoints can now be defined once for all at the interface level in the **Radiation Settings** section.

In the **View Factor** section, the new **High order mesh elements** check box enables the use of mesh elements defined by the geometry shape functions to compute the view factors with high accuracy, or the use of linearized mesh elements to reduce computational cost. In addition, the **Mesh element self-irradiation** check box has been added to account for self-irradiation in concave elements.

The **Prescribed Radiosity** feature now has a **Polar** option to specify the **Directional dependence** of emissivity. In addition, this feature has a new user input for the external temperature, T_{ext}, when the external radiation is defined from a blackbody or graybody model. Either a user-defined temperature can be entered or an ambient temperature can be selected.

In the **Prescribed Radiosity** and **Semitransparent Surface** features, there is a new **Coordinate System Selection** section for the specification of the reference vector of the azimuthal angle. The choice must be made among boundary coordinate systems.

The radiosity is now assumed to be discontinuous by default on edges, which corresponds to the definition of a slit on the radiosity variables. This benefits cases where only one of two faces connected by an edge is exposed to solar radiation, and this improves the robustness of the computation.

Finally the adaption algorithm available for the **Ray shooting** method has been improved to more efficiently detect radiosity or topology changes.

EXTENDED FEATURES FOR HEAT TRANSFER IN POROUS MEDIA

The **Porous Medium** domain feature now includes three different models, which correspond to the thermal equilibrium and thermal nonequilibrium assumptions and to the particular configuration of packed beds of pellets. These models can be selected from the new **Porous medium type** list in the **Porous Medium** section:

- When the **Porous Medium** feature is added by the Heat Transfer in Porous Media predefined interface, the **Porous medium type** is set by default to **Local thermal equilibrium**. This corresponds to the model previously available within the **Porous Medium** feature, solving for a single equation for the temperature with effective properties, with some extra functionality. New averaging models are available for the **Effective thermal conductivity**: **Solid spherical inclusions**, **Fluid spherical inclusions**,

Wrapped screen, and **Sintered metal fibers**. In addition, the **Volume average** and **Reciprocal average** models have been renamed **Plane layers parallel to heat flow** and **Plane layers perpendicular to heat flow**, respectively, for clarity. Finally, using the new **Equivalent thermal conductivity** option, you can specify a user-defined expression.

- When the **Porous Medium** feature is added by the new Local Thermal Nonequilibrium predefined interface, the **Porous medium type** is set by default to **Local thermal nonequilibrium**. This replaces the model previously available within the Local Thermal Nonequilibrium multiphysics interface (including the Heat Transfer in Solids interface, the Heat Transfer in Fluids interface, and the Local Thermal Nonequilibrium multiphysics coupling). The settings for the fluid and solid temperature equations are available in the **Fluid** and **Solid** subnodes of the **Porous Medium** feature, which provides the setting of the I**nterstitial convective heat transfer coefficient** (previously available in the multiphysics coupling node). In this configuration, the **Local Thermal Nonequilibrium Boundary** feature is automatically added to delimit the domain in which a two-temperature model is used.

- When the **Porous Medium** feature is added by the new Heat Transfer in Packed Beds predefined interface, the **Porous medium type** is set by default to **Packed bed**. This option corresponds to a new multiscale modeling approach specifically tailored for pellet beds, as described in the section below.

For the three **Porous medium type** options mentioned above, the postprocessing variables (thermal properties, heat fluxes, heat sources, heat and energy balance) are available in a unified way for homogenized quantities.

MULTISCALE HEAT TRANSFER IN PELLETS BEDS

The new Heat Transfer in Packed Beds predefined interface is available under the **Porous Media** group in the **Heat Transfer** branch in the model wizard. It adds the **Porous Medium** feature, with **Porous medium type** set to **Packed bed**, to model the pellets of a packed bed as spherical homogenized porous particles in which the temperature varies radially. The temperature distribution in the pellets is computed for every position in the packed bed and is coupled to the temperature in the surrounding fluid through an interstitial heat flux or a temperature continuity condition between the pellets' surface and the fluid.

In this configuration, the **Porous Medium** feature has three subnodes (**Fluid**, **Pellets**, and **Pellet-Fluid Interface**), and the **Local Thermal Nonequilibrium Boundary** feature is automatically added to delimit the domain in which a model solving for multiple temperatures is used. Note that the new **Pellet** subnode, available under the **Porous Material** node, can be used to define the properties of the pellet phase.

EXTENDED FEATURES FOR HEAT TRANSFER IN THIN STRUCTURES

In the Heat Transfer in Shells interface, the formulation of heat sources and fluxes has been improved to account for the effect of the surface curvature on the layers' area and length. Because of the use of an extra dimension through the shell's thickness instead of representing it explicitly in the geometry, the dimensions of the selected boundary or edge in the geometry have to be scaled when computing fluxes and sources through the thickness. The application of this scaling factor to variables and equations makes it possible to accurately handle thick and curved layered shells.

The new **Lumped System Connector** and **Lumped System Connector, Interface** features are now available in the Heat Transfer in Shells interface to connect a lumped thermal system to a shell through a layer's edge or interface, respectively. When a Lumped Thermal System interface with an **External Terminal** node is present in the model, this terminal can be selected as a **Source** in the **Terminal Inputs** section of any of the new features. The use of the **Lumped System Connector, Interface** feature is illustrated in the *Lumped Composite Thermal Barrier with Shells* tutorial.

The new **Symmetry** subnode is available under the **Thin Layer**, **Thin Film**, and **Fracture** boundary features of the Heat Transfer interface.

The new **Thermal Insulation, Interface** feature is added by default in the Heat Transfer in Shells interface and specifies thermal insulation on the exterior interfaces of the shell. When added manually, the thermal insulation condition can be applied on any interface of the layers.

A new **Layerwise constant properties** check box is available and active by default when the **Layer type** is **Thermally thin approximation** in the **Solid** and **Fluid** features of the Heat Transfer in Shells interface and in the **Thin Layer** and **Thin Film** features of the Heat Transfer interface. A significant speedup can be obtained with this option, which assumes that the material properties are constant per layer but still changing with layers or the layers' temperature.

NONISOTHERMAL REACTING FLOW

The new Nonisothermal Reacting Flow multiphysics interface is available in the **Laminar Flow** and **Turbulent Flow** versions, under the **Nonisothermal Reacting Flow** group in the **Chemical Species Transport** branch. It adds the **Reacting Flow** multiphysics coupling and the Chemistry, Transport of Concentrated Species, and Heat Transfer in Fluids interfaces, together with a laminar or turbulent version of the Single Phase Flow interface. It provides the heat transfer functionality to reacting flow models, by accounting for the cross contributions between the heat and species equations, in particular for the enthalpy of phase change and the enthalpy diffusion term. It handles

the temperature, pressure, and concentration dependence of the material properties; and accurate heat and energy balance is therefore expected using the predefined variables.

EXTENDED FEATURES FOR MOISTURE TRANSPORT

The **Moisture Flow** multiphysics coupling has a new option to account for the leakage velocity at walls where evaporation and condensation take place. The **Account for Stefan velocity at walls** check box is available in the **Moisture Transport at Walls** section, when the **Mixture type for moist air** is set to **Concentrated species** in the Moisture Transport interface. This functionality is available for free and porous media and automatically adds the corresponding contribution to the velocity of the fluid flow interface.

In the Moisture Transport interface, the **Hygroscopic Porous Medium** feature has been updated to match the classic design for features using porous materials. It now contains the **Liquid Water**, **Moist Air**, and **Porous Matrix** subnodes for the settings of each phase.

In addition, the Moisture Transport interface has extended functionality for several boundary conditions:

- The **Periodic Condition** feature is now available. Discontinuous Galerkin constraints or classical constraints are applied on the relative humidity, vapor concentration, or mass fraction, depending on the adjacent domain features.

- In the **Inflow**, **Open Boundary**, and **Moisture Content** features, it is now possible to specify the vapor mass fraction.

- In the **Open Boundary** feature, the formulation of the Discontinuous Galerkin constraints has been improved.

- In the **Moisture Content** and **Continuity** features, the Discontinuous Galerkin constraints have been added.

Finally, several postprocessing variables are now available to check the mass balance. In particular, the `mt.massBalance`, `mt.dwcInt`, `mt.ntfluxInt`, and `mt.GInt` predefined variables make it possible to control the mass balance, the total accumulated moisture rate, the total net moisture rate, and the total mass source, respectively.

EXTENDED FEATURES FOR PHASE CHANGE

The **Phase Change Interface** feature now accounts for the solid translation velocity in the velocity of the fluid phase at the interface. This can be applied to continuous casting processes, for example.

The new **Phase Change Interface, Exterior** feature is available to define the temperature and position of a phase change interface with the Stefan condition, in the same way as the **Phase Change Interface** feature, but applicable on exterior boundaries adjacent to **Fluid**, **Solid**, or **Porous Medium** domains. The properties of the phase not represented in the computational domain have to be set in the user interface instead of being gathered from an adjacent domain.

THERMAL CONDUCTIVITY MODEL FOR LARGE STRAINS

When the material and spatial frames differ, a new **Deformation model for thermal conductivity** list is available in the features of the heat transfer interfaces representing solid domains (**Solid**, **Biological Tissue**, and **Shape Memory Alloy**).

The **Standard** (previously existing) option assumes that thermal conduction is not affected by the deformation of the solid. In other words, a temperature difference between two faces of a solid induces the same heat flux as in the undeformed configuration, whatever the deformation is. The new option, **Large strain**, defines the effective thermal conductivity in deformed configurations accounting for the rotation between the undeformed and the deformed configuration but not for the strain. In the case of isotropic thermal conductivity, this option ensures that the thermal conductivity in the deformed solid is the same as in the undeformed solid.

NUCLEATE BOILING HEAT FLUX

The Rohsenow correlation, describing the heat flux due to nucleate boiling at a solid surface immersed in a fluid, is now available as a predefined option in the following heat flux features: **Heat Flux**, **Out-of-Plane Heat Flux**, **Point Heat Flux**, and **Isothermal Domain Interface** in the Heat Transfer interface; **Heat Flux** and **Heat Flux, Interface** in the Heat Transfer in Shells interface; and **Nucleate Boiling Thermal Resistor** and **Heat Rate** in the Lumped Thermal System interface. The **Nucleate boiling heat flux** option is available from the **Flux type** list, and the coefficients needed to define the correlation are predefined for some liquids and solid surfaces. Alternatively, user-defined coefficients can be specified. The critical heat flux that can be expected with nucleate boiling is available as a predefined variable for postprocessing.

MARANGONI EFFECT

The **Marangoni Effect** multiphysics coupling has been updated to account for both the tangential and normal effects of surface tension. In addition, the contact angle can now be set in the user interface, either directly or through Young's equation. The new formulation results in improved computational performance.

- A new set of predefined variables is available for the efficient postprocessing of heat and energy balances. These variables are exact in cases of constant material properties and can be a good approximate of the default variables in cases of moderate material property variation in the temperature and pressure ranges present in the model. These variables are much faster to evaluate than the default variables for heat and energy balances. The `ht.dEiInt_cst`, `ht.dEi_cst`, `ht.Ei_cst`, `ht.H_cst`, and `ht.DeltaH_cst` variables are available for heat balance. The `ht.dEiOInt_cst`, `ht.dEiO_cst`, `ht.EiO_cst`, `ht.H_cst`, and `ht.DeltaH_cst` variables are available for energy balance.

- The default color table for the temperature plots has been changed from **ThermalLight** to **HeatCameraLight**. The default color table for the relative humidity has been changed from **Rainbow** to **JupiterAuroraBorealis** with a **Reverse** color map transformation.

- An efficient formulation of the sensible enthalpy predefined variable, `ht.DeltaH`, is now available for free and porous media containing moist air. This analytical formulation, taking advantage of the polynomial definition of the heat capacity for moist air, drastically reduces the evaluation time of the accumulated heat and energy rates in postprocessing.

- In fluid domains, when **Fluid type** is **From material** or **Gas/Liquid**, the ratio of specific heats is now computed by default from the density and the heat capacity, using Mayer's relation.

- The **Semitransparent Surface** feature is now available in the Radiation in Absorbing and Scattering Media interface. The settings are similar to those of the **Semitransparent Surface** feature available in the Radiation in Participating Media interface.

- The **Semitransparent Surface** and **Incident Intensity** boundary features in the Radiation in Participating Media and Radiation in Absorbing-Scattering Media physics interfaces have an additional user input for the external temperature, T_{ext}, when the external radiation is defined from a blackbody or graybody model. Either a user-defined temperature can be entered or an ambient temperature can be selected.

- In the **Incident Intensity** feature of the Radiative Beam in Absorbing Media interface, the definition of the radiative intensity has been fixed when using built-in beam profiles in configurations where the beam is not perpendicular to the surface.

- In the **Deposited Beam Power** feature of the Heat Transfer interface and in the **Incident Intensity** feature of the Radiative Beam in Absorbing Media interface, the definition of the distribution function has been fixed for 2D models.

- The contributions of Heat Transfer with Surface-to-Surface Radiation and Heat Transfer with Radiative Beam in Absorbing Media interfaces to the radiative heat flux of the Heat Transfer interface, `ht.rflux`, have been added. Contributions of the Heat Transfer with Radiation in Participating Media and Heat Transfer with Radiation in Absorbing-Scattering Media interfaces have been fixed.

- In the Surface-to-Surface Radiation interface, the operator `rad.radGpEval(`*`expr`*`)`, used to evaluate expressions at Gauss points, has been replaced by the built-in operator `gpeval(`*`order, expr`*`)`.

- In the **Moist Porous Medium** feature of the Heat Transfer interface, the definition of the effective properties has been fixed to include the liquid water contribution.

- The **Stationary, One-way with Initialization, Moisture Flow** and **Time Dependent, One-way with Initialization, Moisture Flow** preset studies are now available for turbulent moisture flow models with wall functions.

- The Heat Transfer in Shells interface now supports the Eigenfrequency study.

- In the **Thin Layer**, **Thin Film**, and **Fracture** features of the Heat Transfer interface, the handling of the initial condition has been fixed when **Layer type** is **General**. The **Initial Values** subnode is now also available when the **Layer type** is set to **Thermally thick approximation**. For the **Thermally thick approximation** layer type, it is applicable on exterior boundaries only to define the temperature on the thin structure's outer side, the temperature on inner sides being defined from the adjacent domain's initial temperature. In addition, for the **General** and **Thermally thick** layer types, there is a new **From adjacent domains** option in the **Initial Values** subnode, to get the average value of the initial temperature in the adjacent domains.

- The default solver suggestions have been improved when solving for the thermal damage indicator with the **Irreversible Transformation** feature.

- A new **Coordinate System Selection** section is available in the Thermoelectric Effect multiphysics coupling.

- The **Longitudinal dispersivity** and **Transverse dispersivity** settings in the **Thermal Dispersion** feature are now available with all COMSOL products that include the feature.

- The definition of the isothermal Joule–Thomson coefficient variable, `ht.mujtT`, has been improved.
- Pair features have been updated following the new framework and are now also applicable on interior boundaries.

New Applications in Version 6.0

LUMPED COMPOSITE THERMAL BARRIER WITH SHELLS

This example is a variant of the *Composite Thermal Barrier* tutorial and shows how to set up multiple sandwiched thin layers with different thermal conductivities in two different ways. First, the composite is modeled as a 3D object. In the second approach, the Lumped Thermal System physics interface is used to avoid resolving the thin domains by use of thermal circuit modeling. In addition, the top and bottom parts of the steel column are represented with two shells.

THE CONTINUOUS CASTING — ARBITRARY LAGRANGIAN-EULERIAN METHOD

This example describes the cooling and solidification, from melt to solid metal, in a continuous casting process. The model includes nonisothermal properties, temperature distribution, a flow field, and phase change. This example uses the arbitrary Lagrangian–Eulerian (ALE) method for modeling the phase change, and the casting velocity is handled automatically.

Updated Applications in Version 6.0

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 *Evaporation in Porous Media with Large Evaporation Rates* tutorial has been replaced with the *Drying of a Potato Sample* tutorial, with simplified mesh and solver settings.
- The use of the new predefined variables for mass balance is demonstrated in the *Evaporative Cooling of Water*, *Condensation Detection in an Electronic Device with Transport and Diffusion*, *Condensation Risk in a Wood Frame Wall*, and *Drying of a Potato Sample* tutorials.

- The *Heat Sink* tutorial has been updated with the control of energy balance using the new predefined variable `ht.energyBalance`.

- In the *Electronic Chip Cooling* tutorial, Study 1 is not used anymore to get the initial conditions for Study 2.

- The *Vacuum Flask* tutorial has been updated with the use of a **Fluid** feature with 0 velocity in the air domain for clarity.

- In the *Thermal Performances of Windows* and *Glazing Influence on Thermal Performances of a Window* tutorials, the **Open Boundary** features have been replaced by **Heat Flux** features.

Backward Compatibility with Version 5.6

- In the **Prescribed Radiosity** feature in the **Surface-to-Surface Radiation** interface, if the graybody model is selected to define the radiative intensity, the emissivity is temperature dependent, and the temperature model input is automatically defined from a multiphysics coupling in an API script, then you must update the script to define T_{ext}. In other cases, T_{ext} is set to the value that was set to the model input in version 5.6 or earlier so that the model behavior is unchanged.

- In **Heat Transfer** interfaces, in the **Deposited Beam Power** boundary condition, for **User defined** beam profiles, the beam intensity is now defined from its value in the beam cross section, which corresponds to the deposited beam power divided by the cosine of the incidence angle. Nevertheless, for **User defined** beam profiles, the models and API scripts created in version 5.6 or earlier behave identically in version 6.0.

- In the **Opaque Surface** feature of the Surface-to-Surface Radiation interface, the coordinate system choice is restricted to the boundary system. When opening a model created in version 5.6 or earlier with another coordinate system selected, this input should be checked for consistency.

- In the Surface-to-Surface Radiation interface, when the **Expression** or **Tolerance** settings had been modified in the **View Factor Update** section of a model created in version 5.6 or earlier, these settings are reported in the new **View Factor** section.

- A modification of the numerical results within the solver tolerance range may be observed when computing surface-to-surface radiation models created in version 5.6 or earlier, due to the introduction in the Surface-to-Surface interface of the **Jacobian contribution** option set by default to **Only local contributions to radiosity**. When opening a model created in version 5.6 or earlier, a warning is displayed under

the Surface-to-Surface Radiation interface. Note that the accuracy level remains the same because the convergence criteria and the solver tolerance are unchanged.

- The Local Thermal Nonequilibrium predefined multiphysics interface and the **Local Thermal Nonequilibrium** multiphysics coupling are now obsolete. When opening a model created in version 5.6 or earlier, this feature is not migrated. It should be manually replaced by the **Porous Medium** feature with the **Porous medium type** set to **Local thermal nonequilibrium**. 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.

- In the Moisture Transport interface, the **Hygroscopic Porous Medium** feature not supporting the **Porous Material** functionality is now obsolete. When opening a model created in version 5.6 or earlier, this feature is not migrated. It should be manually replaced by the new **Hygroscopic Porous Medium** feature. 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.

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.

Liquid & Gas Properties Module

New Functionality in Version 6.0

- The **Thermodynamics** wizards can now be started from the **Physics** toolbar.

- The filter search used when including species in a thermodynamic system has been improved. Species whose molecular formula includes any combination of the given atoms are now shown. When searching for ammonia, NH3 or H3N can hence be used. It is also possible to filter species using aliases defined in the database. For example, "dihydrogen oxide" can be used to find water.

- The Davidson model (including pressure correction) can now be used for the dynamic viscosity.

- Second derivatives, with respect to temperature and pressure, are now available for density and specific volume.

MEMS Module

New Functionality in Version 6.0

MAGNETOMECHANICS MULTIPHYSICS INTERFACE

Two new physics interfaces for the analysis of coupled magnetic and mechanical effects have been added: **Magnetomechanics** and **Magnetomechanics, No Currents**. When you add such an interface, two physics interfaces are added to the model: Solid Mechanics and either Magnetic Fields or Magnetic Fields, No Currents. The new **Magnetomechanical Forces** multiphysics coupling is also added.

You can find these interfaces under the **Electromagnetics-Structure Interaction** branch in the **Structural Mechanics** folder in the **Add Physics** tree.

This interface also requires the AC/DC Module.

MAGNETOSTRICTION COUPLED TO ROTATING MACHINERY, MAGNETIC INTERFACE

The **Magnetostriction** multiphysics coupling can now also be used to connect the Solid Mechanics and Rotating Machinery, Magnetic physics interfaces.

COMPONENT MODE SYNTHESIS

Linear components built using the Solid Mechanics and Multibody Dynamics interfaces can be reduced to computationally efficient reduced-order models using the Craig–Bampton method. A reduced component can then be used in dynamic or stationary analyses, either in a model consisting of only reduced components, or together with ordinary full FE models. The latter can then be nonlinear.

To create such components, you add a **Reduced Flexible Components** node in the physics interface.

RIGID JOINT IN SOLID MECHANICS INTERFACE

A **Rigid Joint** node has been added to the Solid Mechanics interface. Such a joint can be used to connect two attachments rigidly. The feature is similar to the same joint in the Multibody Dynamics interface. The main purpose is for joining CMS components.

ATTACHMENTS NO LONGER REQUIRE MULTIBODY DYNAMICS LICENSE

You can now add the **Attachment** node with a Structural Mechanics Module or MEMS Module license. The main usage is in the context of creating and joining CMS components.

POINT LOADS AT ARBITRARY LOCATIONS IN SOLID MECHANICS

With the new **Point Load, Free** and **Ring Load, Free** features, you can apply point loads at arbitrary locations that do not coincide with a geometrical point or a mesh node. This is particularly useful when working with moving loads.

NEW DEFAULT BEHAVIOR FOR CONTACT AND CONTINUITY

If any contact pairs are defined under **Definitions**, a default **Contact** node will be added in the Solid Mechanics interface. Similarly, if there are any identity pairs, a default **Continuity** node will be added. In many cases, the pair handling can be considered automatic because pairs can be generated during the finalization of the geometry sequence.

As an effect of the general reformulation of the pair functionality, the **Source external to current physics** check box is no longer needed and has been removed.

TWISTING DEGREE OF FREEDOM IN 2D AXISYMMETRY

In the Solid Mechanics interface in 2D axisymmetry, it is now possible to include circumferential deformation. To do so, you select the **Include circumferential displacement** check box in the **Axial Symmetry Approximation** section in the settings for the physics interface.

INITIAL IMPERFECTION IN BUCKLING ANALYSIS

You can now use the buckling modes from a linear buckling analysis as initial imperfections to the geometry when performing a full nonlinear buckling analysis. This functionality is controlled from the new **Buckling Imperfection** node, which can be added from **Physics Utilities** under **Definitions**.

VISCOELASTICITY IMPROVEMENTS

There are several important additions to the viscoelastic material models:

- For frequency-domain and time-dependent analyses, all of the viscoelasticity models have been augmented with the possibility to include viscoelasticity in the volumetric deformation.
- The Generalized Maxwell model now includes the possibility to prune branches representing frequency ranges outside the bandwidth of prescribed loads, improving

the performance in time-dependent analysis for models with many viscoelastic branches.

- For frequency-domain analyses, a new user-defined viscoelasticity model makes it possible to enter frequency-dependent expressions for the loss and storage moduli or compliances.

- Through a new formulation of the viscoelastic equations, it is now possible to solve for eigenfrequencies in a structure containing viscoelastic materials using a standard procedure for damped eigenfrequency problems. Previously, the eigenvalue problem was nonlinear in the frequency, and only one eigenfrequency at a time could be computed. The new approach is available for all viscoelastic models except user defined, under the assumption that fractional derivatives are not used.

IMPROVED MIXED FORMULATION

In material models that have an option to select a mixed formulation, you can now modify the discretization for the auxiliary dependent variable (pressure or volumetric strain). To do so, **Advanced Physics Options** must be selected. Then, a new section named **Discretization** will appear in the settings for the material model as soon as a mixed formulation is selected. In this section, you can choose between different types of shape functions for the auxiliary dependent variable.

Another change is that the auxiliary dependent variable now has feature scope, rather than physics scope.

NEW DAMPING MODELS

Two new damping models have been added for the mechanical material models:

- The **Wave attenuation** model is essentially a viscous model but with parameters given by measured data for the attenuation of elastic waves in the material. It is available in the **Linear Elastic Material** in Solid Mechanics.

- The **Maximum loss factor** model is mainly intended for time-domain analysis of materials for which a loss factor representation provides a good description in the frequency domain. This damping model is available for all material models that support viscous damping.

The new damping models can be used in both frequency-domain and time-domain studies.

DAMPING FOR PIEZOELECTRIC MATERIALS

Two new options have been added to the **Dielectric Loss** subnode under **Piezoelectric Material** in the Solid Mechanics interface: **Complex Permittivity**, intended for frequency-domain analysis, and **Maximum loss factor**, intended for time-domain analysis.

MULTIPOLE DEBYE DISPERSION FOR PIEZOELECTRIC MATERIALS

New damping models have been added for piezoelectric materials. Under **Charge Conservation, Piezoelectric**, you can now add a **Dispersion** subnode. There, you can choose between the **Debye** and **Multipole Debye** dispersion models.

REDUCED INTEGRATION

A new framework has been added for reduced integration. It is now available in the Solid Mechanics interface. Reduced integration is particularly useful when the computational cost per integration point is high, which is true for many advanced material models.

Reduced integration is controlled from the **Quadrature Settings** section in various material models. It is available in top-level material models like **Linear Elastic Material**. The selected integration rule will then be inherited by any subnodes that may be added.

EASIER GAUSS POINT EVALUATION IN DOMAINS

A set of physics-scoped operators have been added as tools for easier evaluation of Gauss point results in domains. The operators automatically adapt to the Gauss point pattern in each element. These operators are:

- `<phys>.gpeval(expr)`: Map a field of suitable order to Gauss point data.
- `<phys>.elemgpmax(expr)`: Maximum value in any Gauss point in an element.
- `<phys>.elemgpmin(expr)`: Minimum value in any Gauss point in an element.
- `<phys>.elemavg(expr)`: Average value of the Gauss points in an element.
- `<phys>.elemavg(expr)`: Integration over an element using Gauss point data.

INTERCALATION STRAIN

A new subnode, **Intercalation Strain**, has been added to **Linear Elastic Material** and **Nonlinear Elastic Material** in the Solid Mechanics interface. The purpose is to model inelastic strains caused by ion transport.

This feature requires the Battery Design Module.

INPUT OF RESIDUAL STRESS

In the **External Stress** node, there is now an option called **Residual Stress**. A residual stress will not directly affect the displacements. That is, if you just enter a residual stress and no other loads, there will be no displacements. The stress is, however, added to the stress tensor in the sense that it will be part of the stress state that is used in various material models.

EXTERNAL STRESS CAN BE ENTERED AS FIRST PIOLA-KIRCHHOFF STRESS TENSOR

In the **External Stress** node in the Solid Mechanics interfaces, it is now possible to enter the stress state in the form of a 1st Piola–Kirchhoff stress tensor.

SYMMETRIC FORMULATION FOR RIGID CONNECTOR AND RIGID DOMAIN

Under geometric nonlinearity, the consistent stiffness and mass matrices generated by the **Rigid Connector** and **Rigid Domain** features may become nonsymmetric. The solution of a nonsymmetric problem requires significantly more memory and computation time than solving a symmetric problem of the same size. In version 6.0, you can choose to use a less accurate symmetric matrix instead. Doing so may require some more iterations, but each iteration will be significantly faster.

In order to use the symmetric formulation, select **Use symmetric formulation for geometric nonlinearity** in the **Formulation** section in the settings for the feature.

COORDINATE SYSTEM TYPES RESTRICTED FOR RIGID CONNECTOR

For a rigid connector, it is not meaningful to select a coordinate system with position-dependent axis orientations. This is no longer allowed, neither in the **Rigid Connector** node itself nor in its subnodes.

EXPRESSIONS FOR THE ROTATION VECTOR CHANGED IN RIGID CONNECTOR AND RIGID DOMAIN

The expression defining the variables giving the rotation vector in a rigid connector and rigid domain has been changed in such a way that it is more robust with respect to computing Jacobian contributions. This can be important if you want to, for example, control such a variable using other equations.

The affected variables for a rigid connector are named *<phys>*`.th`*<dir>*`_`*<tag>* (for example, `solid.thz_rig1`). For a rigid domain, the names are of the type *<phys>*`.`*<tag>*`.th`*<dir>* (for example, `solid.rd1.thy`).

CONSTRAINT GROUPS FULLY SUPPORTED FOR RIGID CONNECTORS AND RIGID DOMAINS

In earlier versions, constraint groups could not be used for rigid connectors or rigid domains if the option **Evaluate reaction forces** was selected. This limitation has been removed.

STRICTER DEFAULT TOLERANCE FOR TIME-DEPENDENT ANALYSIS

The default tolerance for time-dependent analysis has been decreased from 0.01 to 0.001 for the Solid Mechanics interface. This will affect both new and old models in which the solver sequence has not been manually edited. The effect is that solutions will be more accurate but typically at the cost of longer solution times.

THE TIME-DEPENDENT, PRESTRESSED, MODAL STUDY TYPE REMOVED

Due to problems with obtaining a correct linearization in all situations, the **Time-Dependent, Prestressed, Modal** study type is no longer available.

AZIMUTHAL MODE NUMBER AUTOMATICALLY TRANSFERRED TO REVOLUTION DATASET

When using circumferential mode extension for 2D axisymmetry, the azimuthal mode number (for example, `solid.mk`) is now automatically transfered to the revolution datasets that are created as default plots. This means that you do not need to modify the settings in the dataset when switching between results for different mode numbers.

IMPROVEMENTS IN RANDOM VIBRATION FUNCTIONALITY

There are several improvements within the field of random vibration analysis:

- The number of study steps generated by a Random Vibration (PSD) study sequence is now two instead of three. The "dummy" Frequency Domain study now resides inside the Model Reduction study.

- A new operator, `q2sq`, has been added. It is similar to the existing `q2` operator but has more convenient arguments and output. In particular, it is targeted at computing the RMS of the von Mises stress and vector norms.

- In the **Random Vibration** node, it is possible to enter default parameters for operators like `rms`, which perform an integration. It is then possible to invoke these operators without the last three arguments.

- It is now easy to add the base acceleration to computed acceleration PSDs, thus facilitating direct evaluation of absolute as well as relative spectra.

- There are significant performance improvements in the random vibration computations.

CHANGED NOMENCLATURE FOR FORCE LINEAR STRAIN

The label of the **Force linear strain** check box, which is present in many material models, has been changed to **Geometrically linear formulation** in order to improve clarity.

CHANGES IN DEFAULT PLOTS

There are many improvement in the default plots. Some of the more prominent are:

- The default stress plot in 3D now uses a **Volume Plot**, rather than a **Surface Plot**. There are several advantages, including easier selection of individual domains and more consistent evaluation of some expressions defined at the domain level. However, in larger models, you may find that the plotting is slower. Also, the visualization of curved boundaries with a coarse mesh is smoother in a surface plot.

- A new Prism color table is used for the default stress plots. This color table will emphasize regions with high stresses.

- In the default stress plots, the smoothing is conditional, so that stress jumps larger than 20% between adjacent elements will be plotted without smoothing. This makes it easier to identify a solution computed on a too coarse mesh.

EASIER USE OF MULTIPLE CELL PERIODICITY NODES

The way studies are generated has been changed so that unique studies are created for each **Cell Periodicity** node. This makes it possible to, for example, run nested homogenizations.

A new section, **Advanced**, has been added to the settings for **Cell Periodicity**. Here you can, if needed, select to use the old scheme to keep the old behavior.

SYMMETRY PLANE FOR ELECTROSTATICS AND ELECTRIC CURRENTS

A new **Symmetry Plane** feature is available for the Electrostatics and Electric Currents interfaces. It provides symmetry and antisymmetry conditions for the electric field. For the antisymmetric case, a reference potential can be set to specify around what electric potential the field is antisymmetric (the default is ground).

IMPROVED HANDLING OF LOAD GROUPS FOR THERMAL EXPANSION AND HYGROSCOPIC SWELLING

The handling of load groups in the **Thermal Expansion** and **Hygroscopic Swelling** features has been improved. In particular, using several such features with the same selection now works in a more intuitive way.

REMOVAL OF SEVERAL PAIR FEATURES

As part of a major overhaul of the pair handling, several pair features have been removed. These nodes were seldom or never used, and the interpretation was difficult and error prone. The removed pair features are **Fixed Constraint**, **Prescribed Displacement**, **Roller**, **Symmetry**, **Antisymmetry**, **Rigid Connector**, and **Boundary Load**. In case any such feature was used in a model, it can easily be replaced by standard boundary conditions.

CHANGE IN PRESCRIBED DISPLACEMENT WHEN ELASTIC PREDEFORMATION IS USED

The interpretation of the input in the **Prescribed Displacement** node in connection with **Elastic Predeformation** has been changed. It is now the total displacement, which is given in the same way as if **Elastic Predeformation** is not present.

CHANGED API FOR INITIAL VALUES

Due to changes in the implementation of the **Initial Values** node in Solid Mechanics, there are changes in the API syntax. In version 5.6 and earlier, if the displacement field had a name that was not u, the field name was reflected in the API syntax when setting an initial value. Also, the syntax for the initial velocity has changed. Assume that the field name is u_solid:

Earlier versions:
```
...feature("init1").set("u_solid", new int[]{1, 0, 0});
...feature("init1").set("du_solid/dt", new int[]{2, 0, 0});
```

Version 6.0:
```
...feature("init1").set("u", new int[]{1, 0, 0});
...feature("init1").set("ut", new int[]{2, 0, 0});
```

If you are reading a file containing the old syntax, there is an automatic conversion for some common cases. The cases that are handled include when the field name is u* or u_solid*, where * is a number less than or equal to 10.

For all other cases when a nondefault field name is used, you must manually replace the argument in the call to set() by the new names, u or ut, to set the initial displacements or velocities.

BUG FIXES

The Compliance matrix and Elasticity matrix of the piezoelectric material Polyvinylidene fluoride (PVDF) has been corrected to be consistent with the Voigt notation (as of 5.6 update 1).

The formulation for rarefied (general accommodation) thin-film flow has been corrected (as of 5.6 update 2).

New Models in Version 6.0

SOLIDLY MOUNTED RESONATOR 2D

A solidly mounted resonator (SMR) is a piezoelectric MEMS resonator formed on top of an acoustic mirror stack deposited on a thick substrate. This tutorial shows how to simulate an SMR in 2D. In this example, the eigenmodes were computed and the frequency response from 500 to 1200 MHz was analyzed. The computed impedance curve and displacement profile agree reasonably well with the simulation results found in literature.

SOLIDLY MOUNTED RESONATOR 3D

A solidly mounted resonator (SMR) is a piezoelectric MEMS resonator formed on top of an acoustic mirror stack deposited on a thick substrate. This tutorial shows how to simulate an SMR in 3D. In this example, the eigenmodes were computed with a varying number of particles attached to the sensor surface for computing the sensitivity, and the corresponding change in frequency response was analyzed. The resonant frequency decreases with more attached particles, and the sensitivity depends on the attachment location relative to the mode shape — both observations are as expected.

COMPONENT MODE SYNTHESIS TUTORIAL

In this tutorial example, the concept of component mode synthesis (CMS) is introduced through a simple solid model of a beam. Parts of the beam are reduced into CMS components. Dynamic as well as stationary analyses are performed. Moreover, you will learn how to apply loads directly on a CMS component and work with such components during result evaluation. The results from the full and reduced versions of the model are compared.

MODEL UPDATES

The *biased_resonator_3d_freq* model has been updated to use the same AC drive voltage as in the reference paper.

The *comb_drive_tuning_fork_gyroscope_manufacturing_variation* model has been updated to use **Component>Deformed Geometry** branch features.

Metal Processing Module

New Functionality in Version 6.0

IMPORT OF PHASE TRANSFORMATION DATA

It is now possible to import phase transformation data from JMatPro® for steel quenching purposes. Phase transformation data is imported for both diffusive and diffusionless phase transformations. The appropriate phase transformation models are configured automatically, and parameters for start and finish temperatures, latent heat, and phase transformation strains are also imported.

NEW PHASE TRANSFORMATION MODEL

The **Kirkaldy-Venugopalan** phase transformation model has been added to the **Phase Transformation** node in the Metal Phase Transformation and Austenite Decomposition interfaces. This phase transformation model is mainly suitable for diffusive phase transformations during steel quenching.

MISCELLANEOUS

- The **Leblond-Devaux**, **Johnson-Mehl-Avrami-Kolmogorov (JMAK)**, and **Kirkaldy-Venugopalan** phase transformation models can use **TTT diagram data** directly, as input.

- The **Koistinen-Marburger** phase transformation model can now be defined in terms of the **Martensite start temperature** and **Martensite finish temperature**.

- In 0D, it is now possible to compute the time at which a phase transformation has stopped. This effectively corresponds to the time at which the rate of formation of the destination phase falls below a lower threshold. You can model this by using the **Compute time to completion** option in a **Metallurgical Phase** node, where the numerical lower threshold, the **Completion rate**, is additionally supplied.

Microfluidics Module

New Functionality in Version 6.0

- A Fluid-Solid Interaction, Fixed Geometry interface for Two-Phase Flow, Phase Field has been added under the Fluid-Structure Interaction section in the Fluid Flow branch.

- The Brinkman Equations, Free and Porous Media Flow and Darcy's Law interfaces have a new **Porous Medium** feature, which replaces the old **Fluid and Matrix Properties** feature. The new feature contains two subfeatures, **Porous Matrix** and **Fluid**, in which the characteristic properties can be specified directly or by linking to a Material node.

- The Brinkman Equations and Free and Porous Media Flow interfaces have a new option, **Porous treatment of no slip condition**, with two options, **Standard no slip formulation** and **Porous slip**. The latter is suitable for models with underresolved boundary layers.

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.

Mixer Module

New Functionality in Version 6.0

ROTATING MACHINERY, PHASE TRANSPORT MIXTURE MODEL

The predefined Rotating Machinery, Phase Transport Mixture Model interfaces for laminar and turbulent flow of multiple dispersed phases have been added under the Multiphase Flow section in the Fluid Flow branch. These interfaces can be used to simulate phase separation; fractionation by density, size, or shape; and the mixing of several phases that would otherwise separate due to sedimentation or flotation processes.

Multibody Dynamics Module

New Functionality in Version 6.0

IMPROVED RIGID BODY CONTACT

The **Rigid Body Contact** feature has been extended so that more shapes are available in 3D. The source geometry can be a sphere or cylinder. The destination geometry can be a sphere, cylinder, planar shape, or user-defined arbitrary shape.

In both 2D and 3D, the geometrical parameters of the standard shapes can now be automatically determined from the geometry of a selected rigid domain.

COMPONENT MODE SYNTHESIS

Linear components built using the Solid Mechanics and Multibody Dynamics interfaces can be reduced to computationally efficient reduced-order models using the Craig–Bampton method. A reduced component can then be used in dynamic or stationary analyses, either in a model consisting of only reduced components or together with ordinary full finite-element models. The latter can then be nonlinear.

To create such components, you add a **Reduced Flexible Components** node in the physics interface.

POINT LOADS AT ARBITRARY LOCATIONS

With the new **Point Load, Free** feature, you can apply point loads at arbitrary locations that do not coincide with a geometrical point or a mesh node. This is particularly useful when working with moving loads.

IMPROVED MIXED FORMULATION

When using a mixed formulation in the **Linear Elastic Material**, you can now modify the discretization for the auxiliary dependent variable (pressure or volumetric strain). To do so, **Advanced Physics Options** must be selected. Then, a new section named **Discretization** will appear in the settings for **Linear Elastic Material** as soon as a mixed formulation is selected. In this section, you can choose between different types of shape functions for the auxiliary dependent variable.

Another change is that the auxiliary dependent variable now has feature scope rather than physics scope.

NEW DAMPING MODELS

Two new damping models have been added for the **Linear Elastic Material**:

- The **Wave attenuation** model is essentially a viscous model, but with parameters given by measured data for the attenuation of elastic waves in the material.

- The **Maximum loss factor** model is mainly intended for time-domain analysis of materials for which a loss factor representation provides a good description in the frequency domain.

The new damping models can be used in both frequency-domain and time-domain studies.

REDUCED INTEGRATION

A new framework has been added for reduced integration. It is controlled from the **Quadrature Settings** section in the **Linear Elastic Material** node's settings.

EASIER GAUSS POINT EVALUATION IN DOMAINS

A set of physics-scoped operators have been added as tools for easier evaluation of Gauss point results in domains. The operators automatically adapt to the Gauss point pattern in each element. These operators are:

- `<phys>.gpeval(expr)`: Map a field of suitable order to Gauss point data.
- `<phys>.elemgpmax(expr)`: Maximum value in any Gauss point in an element.
- `<phys>.elemgpmin(expr)`: Minimum value in any Gauss point in an element.
- `<phys>.elemavg(expr)`: Average value of the Gauss points in an element.
- `<phys>.elemavg(expr)`: Integration over an element using Gauss point data.

SYMMETRIC FORMULATION FOR RIGID CONNECTOR AND RIGID DOMAIN

Under geometric nonlinearity, the consistent stiffness and mass matrices generated by the **Rigid Connector** and **Rigid Domain** features may become nonsymmetric. The solution of a nonsymmetric problem requires significantly more memory and computation time than solving a symmetric problem of the same size. In version 6.0, you can choose to use a less accurate symmetric matrix instead. Doing so may require some more iterations, but each iteration will be significantly faster.

In order to use the symmetric formulation, select the **Use symmetric formulation for geometric nonlinearity** check box in the **Formulation** section in the settings for the feature.

COORDINATE SYSTEM TYPES RESTRICTED FOR RIGID CONNECTOR

For a rigid connector, it is not meaningful to select a coordinate system with position-dependent axis orientations. This is no longer allowed, neither in the **Rigid Connector** node itself nor in its subnodes.

EXPRESSIONS FOR THE ROTATION VECTOR CHANGED IN RIGID CONNECTOR AND RIGID DOMAIN

The expression defining the variables giving the rotation vector in a rigid connector and rigid domain has been changed in such a way that it is more robust with respect to computing Jacobian contributions. This can be important if you want to, for example, control such a variable using other equations.

The affected variables for a rigid connector are named `<phys>.th<dir>_<tag>` (for example, `solid.thz_rig1`). For a rigid domain, the names are of the type `<phys>.<tag>.th<dir>` (for example, `solid.rd1.thy`).

CONSTRAINT GROUPS FULLY SUPPORTED FOR RIGID CONNECTOR AND RIGID DOMAIN

In earlier versions, constraint groups could not be used for rigid connectors or rigid domains if the **Evaluate reaction forces** option was selected. This limitation has been removed.

STRICTER DEFAULT TOLERANCE FOR TIME-DEPENDENT ANALYSIS

The default relative tolerance for time-dependent analysis has been decreased from 0.01 to 0.001 for the Multibody Dynamics interface. This will affect both new and old models in which the solver sequence has not been manually edited. The effect is that solutions will be more accurate but typically at the price of longer solution times.

THE TIME-DEPENDENT, PRESTRESSED, MODAL STUDY TYPE REMOVED

Due to problems with obtaining the correct linearization in all situations, the **Time-Dependent, Prestressed, Modal** study type is no longer available.

CHANGED NOMENCLATURE FOR FORCE LINEAR STRAIN

The **Force linear strain** check box, which is present in **Linear Elastic Material**, has been changed to **Geometrically linear formulation** in order to improve clarity.

IMPROVED HANDLING OF LOAD GROUPS FOR THERMAL EXPANSION

The handling of load groups in **Thermal Expansion** has been improved. In particular, using several such features with the same selection now works in a more intuitive way.

REMOVAL OF SEVERAL PAIR FEATURES

As part of a major overhaul of the pair handling, several pair features have been removed. These nodes were seldom or never used, and the interpretation was difficult and error prone. The removed pair features are **Fixed Constraint**, **Prescribed Displacement**, **Roller**, **Rigid Connector**, and **Boundary Load**. In case any such feature was used in a model, it can easily be replaced by standard boundary conditions.

New Models in Version 6.0

DYNAMICS OF A ROLLER CONVEYOR

This model simulates the dynamics of a roller conveyor belt transporting a spherical object or a ball. The ball is placed at one end of a roller conveyor and then transported to the other end, where it is collected in a rectangular tray. The conveyor system consists of cylindrical rollers inserted between two frames. Guideways are provided on both sides of the conveyor to prevent the ball from falling off. All the components of the system are assumed to be rigid. The rollers are assumed to be connected to a power source to rotate at a prescribed speed, whereas the frame, guideways, and tray are fixed.

The contact between the ball and rollers is modeled using rigid body contact with friction. Similar contact conditions are also used between the ball and other components. The connection between the rollers and frame is simplified using hinge joints. A transient study is performed to analyze the system dynamics and also find the contact forces, friction forces, and energy dissipation due to friction.

COMPONENT MODE SYNTHESIS TUTORIAL

In this tutorial example, the concept of component mode synthesis (CMS) is introduced through a simple solid model of a beam. Parts of the beam are reduced into CMS components. Dynamic as well as stationary analyses are performed. Moreover, you will learn how to apply loads directly on a CMS component and work with such components during result evaluation. The results from the full and reduced versions of the model are compared.

MODELING VIBRATION AND NOISE IN A GEARBOX: CMS VERSION

This example illustrates the modeling of vibration and noise in a 5-speed synchromesh gearbox of a manual transmission vehicle. The example is essentially the same as *Modeling Vibration and Noise in a Gearbox* but with some elastic components reduced to computationally efficient lower-order models using the component mode synthesis (CMS) technique.

Nonlinear Structural Materials Module

New Functionality in Version 6.0

NEW HYPERELASTIC MATERIAL MODEL: VAN DER WAALS

A new hyperelastic material model has been added, called *van der Waals*. It is based on thermodynamical principles for polymer chains. This material model is also known as the *Kilian* model.

FIBER-REINFORCED HYPERELASTIC AND NONLINEAR ELASTIC MATERIALS

By adding one or more **Fiber** subnodes under a **Hyperelastic Material** or **Nonlinear Elastic Material** node, you can augment the stiffness by the effect of distributed fibers. For hyperelastic materials, there are three material models for the fibers: **Holzapfel-Gasser-Ogden**, **Linear elastic**, and **User defined**. The fiber content is assumed to be a small fraction of the total material volume. The new **Fiber** node makes it easy to model anisotropic hyperelastic materials in general.

For nonlinear elastic materials, the only material model for the fibers is linear elastic.

You can model thermal expansion of the fibers by adding a **Thermal Expansion** subnode under the **Fiber** node.

ALTERNATIVE INPUT PROPERTIES FOR SOME HYPERELASTIC MODELS

In the Neo-Hookean and St. Venant–Kirchhoff hyperelastic material models, you can now enter the constitutive parameters on several different forms, for example, by Young's modulus and Poisson's ratio.

NEW VARIANT OF MULLINS EFFECT

When using **Ogden-Roxburgh** as a **Damage function** in the **Mullins Effect** node, it is now possible to choose between two different interpolation functions for the damage. In the original version, a *tanh* function is used. By selecting the **Use error function** check box, you can replace it with the *erf* function.

NEW DEFAULT VALUE FOR BULK MODULUS IN HYPERELASTIC MATERIALS

In hyperelastic materials that are of a nearly incompressible type, the bulk modulus κ now has the default value `100*phys.Gequ`. The built-in variable `phys.Gequ` is the shear modulus at zero strain. In earlier versions, the default value was zero; a value that always had to be replaced by something reasonable.

NONLOCAL PLASTICITY

Nonlocal plasticity has been added to the **Plasticity** and **Porous Plasticity** features. You can enable and control it in the new **Nonlocal Plasticity Model** section in the settings for these material models.

With nonlocal plasticity, the equivalent plastic strain is a distributed field, rather than defined at a point. This makes it possible to solve problems that would otherwise fail due to high strain localization.

POROUS PLASTICITY ENHANCEMENTS

There are several enhancements to the porous plasticity material model:

- The hardening variable in the matrix (`<item>.epm`) is constrained to be positive.

- When the effective void volume function reaches its maximum value, a small stiffness now remains, thus avoiding a breakdown of the analysis. You can control the **Maximum damage** from the **Advanced** section in the **Porous Plasticity** node.

- In the Gurson–Tvergaard–Needleman model, the effective void volume fraction function f_e can now be selected and has the options **Bilinear**, **Asymptotic**, and **User defined**.

- It is possible to account for void growth in shear in 2D axisymmetry.

- Gauss point interpolated variables have been added for more accurate result evaluation.

CREEP IMPROVEMENTS

The functionality in the **Creep** node has been subject to major changes:

- A new general framework gives large improvements in computational speed and memory usage. For larger models, a speedup by a factor of 10 or more can be expected.

- You can select the type of scheme that is used for the time integration of the creep equations. In the **Time Stepping** section, you can set **Method** to **Automatic, Backward Euler**, **Forward Euler**, or **Domain ODEs**. The last method provides backward

compatibility with versions prior to 6.0 and is usually significantly slower for large models. The **Backward Euler** method is not available in the Layered Shell interface or in the layered material models in the Membrane and Shell interfaces.

- It is no longer possible to add several **Creep** nodes acting in parallel on the same selection. When more than one creep mechanism is acting, you can instead add one or more **Additional Creep** nodes under a **Creep** node. As an effect, only one set of creep strain degrees of freedom are needed, even if several creep models are used. This gives additional performance gains for this type of model.

- The Norton-Bailey creep model has been removed. It is replaced by a general isotropic "Bailey-type" hardening formulation, which can be combined with all creep models, not only the Norton model.

- The type of equivalent stress used to determine the creep rate is now a user input. You can select **von Mises**, **Hill orthotropic**, **Pressure**, or **User defined**. Thus, nonisotropic creep can now be modeled.

- The old creep models **Potential**, **Volumetric**, and **Deviatoric** have been removed. The same functionality is now achieved either through the **User defined** creep model or by using the new **Inelastic Strain Rate** node.

PRESCRIBED INELASTIC STRAIN RATE

A new subnode, **Inelastic Strain Rate**, has been added to **Linear Elastic Material**, **Layered Linear Elastic Material**, **Nonlinear Elastic Material**, **Hyperelastic Material**, and **Layered Hyperelastic Material**. The effect of this node is similar to the existing **External Strain** node, in that it injects an inelastic strain in the material model. The difference is that here you instead specify a rate, which is then integrated into an inelastic strain contribution. By using this feature, you do not need to add an extra ODE interface when you have a rate equation describing an extra strain contribution.

The strain rate can be specified in terms of a strain tensor, a deformation gradient, an inverse deformation gradient, or three orthogonal stretches.

UPDATE OF ANAND VISCOPLASTIC MODEL

The **Anand** model under **Viscoplasticity** has been updated. The main change is with respect to the time integration. You can now select the type of scheme that is used. In the **Time Stepping** section, you can set **Method** to **Automatic, Backward Euler**, or **Domain ODEs**. The last method provides backward compatibility with versions prior to 6.0 and is usually significantly slower for large models. This section is not present in the Layered Shell interface or in the layered material models in the Membrane and Shell interfaces.

As an effect of these changes, there are some potential backward compatibility issues:

- API code that references the old field names in order to modify solver settings will fail.

- Manual variable scaling of the viscoplastic degrees of freedom will be lost, even when **Domain ODEs** is selected.

DAMPING FOR SHAPE MEMORY ALLOY

In a dynamic analysis, you can now use a **Damping** subnode under **Shape Memory Alloy** in both the Solid Mechanics and Truss interfaces.

ENHANCEMENTS IN PHASE-FIELD-BASED DAMAGE

When using the **Damage** node, there are two new options for the **Crack driving force**: **Total strain energy density** and **User defined**. The old option, **Strain energy density**, has been renamed to **Elastic strain energy density** for the sake of clarity.

IMPROVED PERFORMANCE FOR LARGE-STRAIN PLASTICITY

Large-strain plasticity is up to 20% faster than in earlier versions because a more efficient formulation is used.

REDUCED INTEGRATION

The new framework for reduced integration will, in many cases, make the solution of models with nonlinear materials, such as plasticity and creep, significantly faster. Reduced integration is controlled from the **Quadrature Settings** section in material models such as **Linear Elastic Material**, **Nonlinear Elastic Material**, **Hyperelastic Material**, and **Shape Memory Alloy**.

New Models in Version 6.0

CREEP ANALYSIS OF A TURBINE STATOR BLADE

In this example, the deformations caused by secondary creep in a turbine stator blade are computed. The creep rate is highly influenced by temperature, and the deformation and stress relaxation is thus to a large extent controlled by the temperature field, which is computed.

PHARMACEUTICAL TABLETING PROCESS

Powder compaction is a popular manufacturing process not only in powder metallurgy, but also in the pharmaceutical industry. The capped Drucker–Prager model is commonly used for simulating the compaction processes of pharmaceutical powders,

where the material properties depend on the powder density. In this example, microcrystalline cellulose (MCC) is compacted, and the constitutive material properties are calibrated based on experimental data.

TWO-STAGE POWDER COMPACTION PROCESS

Powder compaction is a key process in powder metallurgy, where it gives the flexibility to produce quality products of complex shapes for sintering. The density of the compact is a key factor to determine the overall quality of the sintered product, as regions with lower density could reduce its mechanical strength.

A multistage compaction process is one way to achieve homogeneous density in the workpiece. In this example, a two-stage compaction process is demonstrated in a simple setup, and the outcome is compared with the results of a single-stage process.

BIVENTRICULAR CARDIAC MODEL

The standard biventricular cardiac model of the heart is used to show how to set up fiber directions in a complex geometry. The fibers are then used to model the large deformation of the myocardium with the Holzapfel–Gasser–Ogden anisotropic material model. The Aliev–Panfilov equations are included to model the electric cardiac excitation.

Optimization Module

New Functionality in Version 6.0

EASIER ACCESS TO SHAPE AND TOPOLOGY OPTIMIZATION

The shape and topology optimization features have been moved up the model tree. In version 5.6, they were available from the **Component** node's **Definitions** branch, but in 6.0, they reside directly under the **Component** (similar to physics interfaces).

It is now also possible to add a **Control Variable Field** directly under the **Component** node, so in many cases, it is no longer necessary to use the Optimization interface.

PARAMETER ESTIMATION IMPROVEMENTS

The user interface for parameter estimation has been improved with the addition of a new **Global Least-Squares Objective** feature (available under the **Component** node) and a new **Parameter Estimation** study step. The old Parameter Estimation study step has been renamed to **Curve Fitting**. For most parameter estimation models, this means that it is no longer necessary to use the Optimization interface.

SENSITIVITY ANALYSIS IMPROVEMENTS

There have been several improvements to the robustness of transient sensitivity analysis, and it is now possible to force solving of the adjoint problem using the same time steps as the forward problem. This setting is available in the **Optimization Solver** node. Oftentimes, robustness can be further improved by the use of fixed time steps.

Additional Improvements
A **Control Function** feature has been added, which can be used with arbitrary arguments. One possibility is to use it for introducing a scalar transient control variable. Regularization using a Bernstein polynomial or a Helmholtz filter is supported by this feature.

The **Transformation** shape optimization feature has been introduced to simplify simple operations such as translation, rotation, and scaling. This feature interacts with the other shape optimization features, and it also supports scaling and translation in custom coordinate systems.

New and Updated Models in Version 6.0

The following new models are now available:

- The *coil_shape_optimization* model demonstrates how to find a coil geometry giving a uniform magnetic field on an axis and minimal field near where the axis ends. This model requires the AC/DC Module.

- The *optimal_heating_control* model provides an example of optimal control, where the input heating power to a rod is optimized to give a certain temperature on the outside of the rod. This problem corresponds to the situation where you want to reach a certain minimum temperature as fast as possible all over, without exceeding some maximum temperature.

- The *rectangular_horn_shape_optimization* model demonstrates how to use the Control Function feature to perform shape optimization on a rectangular horn for an improved on-axis response. The off-axis spatial response is improved using the Free Shape Boundary and Transformation shape optimization features. This model requires the Acoustics Module and either the AC/DC Module, Battery Design Module, MEMS Module, Plasma Module, RF Module, or Semiconductor Module.

- The *tesla_microvalve_transient_optimization* model performs a topological optimization for a Tesla microvalve for an oscillating pressure drop. A Tesla microvalve inhibits backward flow using friction forces rather than moving parts, and therefore the objective is to maximize the average flow rate. The design can be optimized by distributing material within the modeling domain.

- In the *waveguide_filter_optimization_transformation* model, a waveguide filter is designed using shape optimization by moving and scaling rectangles in the geometry. The irises of the initial geometry are optimized to ensure good bandpass response and out-of-band rejection, while maintaining the double mirror symmetry. This model requires the RF Module.

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 6.0

SIMPLIFIED NONLOCAL COUPLING NAMES

The built-in couplings that evaluate the sum, average, maximum, and minimum of expressions over all particles have been renamed. In an instance of the Mathematical Particle Tracing interface with the default name `pt`, the old name for the sum was `pt.ptop1()`; the new name is `pt.sum()`. Similarly, the average, maximum, and minimum are now `pt.ave`, `pt.max`, and `pt.min`, respectively.

Backward Compatibility

The old coupling names, such as `pt.ptop1()` in the example above, will be labeled as deprecated but they are still supported in version 6.0.

TRANSFORMATIONS OF LOADED PARTICLE COORDINATES

When loading initial particle position and velocity from a file using the **Release from Data File** node, the distribution of initial particle positions can now be dilated, rotated, and translated. If a rotation is applied, you can also choose to apply the same rotation to the initial particle velocity vectors if they are also loaded from the same file.

FAST BOUNDARY SEARCH FOR DRAG IN WALL-BOUNDED FLOWS

In the **Drag Force** for Particle Tracing for Fluid Flow interface, when you select the **Include wall corrections** check box to define a more accurate drag force in narrow pipes or channels, you can now specify the **Mesh search method** used to compute the direction and distance from each particle to the closest wall. Similar functionality is used to detect the closest wall when activating anisotropic turbulent dispersion in the **Drag Force** or to find the distance to the closest wall for the wall-induced **Lift Force**.

The default option in the **Mesh search method** list, **Closest point**, is the behavior from versions 5.6 and earlier. It is the slowest but most robust option. The **Use tolerance** and **Walk in connected component** options may be significantly faster in finely meshed 3D geometries.

ACCUMULATED DOSE FOR PARTICLE-MATTER INTERACTIONS

In the **Particle-Matter Interactions** node, you can now select any combination of the **Absorbed dose**, **Absorbed dose from ionization losses**, or **Absorbed dose from nuclear stopping** check boxes. Selecting a check box causes an accumulated variable to be

defined in the selected domains to compute the accumulated dose and dose equivalent from the passage of energetic particles (usually ions) through the medium.

DISSIPATED PARTICLE HEAT

You can use the new **Dissipated Particle Heat** node to accumulate a heat source term based on the heating or cooling of particles by their surroundings. To use this node, the particle temperature must be solved for, and at least one **Convective Heat Losses** or **Radiative Heat Losses** node must be present. A particle that is heated by its surroundings will define a volumetric heat sink, while a particle that is cooled by its surroundings will define a volumetric heat source.

New and Updated Applications in Version 6.0

PINCHED FLOW FRACTIONATION

This tutorial model demonstrates the continuous separation of particles by size via purely inertial means. Particles enter a microchannel through one inlet while pure liquid enters the same microchannel through a second inlet at a much higher velocity. When the two streams are joined, the distribution of particles is pushed against the wall of the microchannel opposite the high-velocity stream. In close proximity to the microchannel wall, the wall-induced lift force separates out the different particle sizes. Then, at the other end of the channel, particles exit through different outlet branches based on their relative size.

THREE-BODY PROBLEM

The classical three-body problem involves the prediction of the trajectories of three point masses under mutual gravitational attraction. This example compares two known periodic solutions to the three-body problem: the stable figure-eight configuration and the unstable Lagrange configuration.

Important Fixes in Version 6.0

ACCURACY IMPROVEMENTS FOR BOUNDARY ACCUMULATORS

Significantly improved the accuracy of accumulated variables defined on curved boundaries.

CHARGE ACCUMULATION WITH MULTIPLE SPECIES

In the Particle Tracing for Fluid Flow interface, using the **Charge Accumulation** node with two or more different particle species no longer fails with an "Undefined value found" error message.

NUMBER OF PARTICLE DIAMETER VALUES PARSING ISSUE

When computing particle diameters in the Particle Tracing for Fluid Flow interface, most particle release features allow you to sample from a normal, lognormal, or uniform distribution of initial diameter values. In version 5.6, entering an expression or a parameter for the **Number of values** field resulted in the error message "Field is not an integer" even if an integer value was given. This has been fixed in version 6.0.

VOLUME FORCE IN THE INERTIALESS FORMULATION

In the Particle Tracing for Fluid Flow interface, using the **Newtonian, ignore inertial terms** formulation together with the **Volume Force Calculation** node caused an "Undefined variable" error in version 5.6. This has been fixed in version 6.0.

WALL CORRECTIONS AFFECTING THE SOLUTION WHILE DISABLED

In the Particle Tracing for Fluid Flow interface, the **Drag Force** node can be used to model drag in wall-bounded flows by selecting the **Include wall corrections** check box. This check box is only supported for the **Stokes** drag law. However, in version 5.6, it was possible to define drag corrections in other drag laws by selecting this check box and then switching to a different drag law. This has been fixed in version 6.0.

WALL CORRECTIONS FROM PAIR CONTINUITY BOUNDARIES

In the Particle Tracing for Fluid Flow interface, the **Drag Force** node can be used to model drag in wall-bounded flows by selecting the **Include wall corrections** check box. In version 5.6, interior boundaries of an assembly where the **Pair Continuity** boundary condition was used would count as walls for the purpose of applying such wall corrections. In version 6.0, **Pair Continuity** boundaries will be ignored when computing wall corrections for the drag force.

THERMAL RE-EMISSION ON INTERIOR MOVING BOUNDARIES

In version 5.6, it was possible for the **Thermal Re-Emission** boundary condition to reflect particles on the wrong side of interior boundaries if those boundaries were moving at speeds comparable to or faster than the thermal velocity of the particles. This has been corrected in version 6.0.

CONFLICT WITH MULTIPLE PARTICLE-MATTER INTERACTIONS NODES

In the Charged Particle Tracing interface, variables defined by multiple copies of the **Particle-Matter Interactions** node will no longer overwrite each other.

ERROR MESSAGES FROM INTERFACES NOT SOLVED FOR

In version 5.6, an error message was given when trying to run any study if a particle tracing interface in the model had no particle release features, or if the particle release features had no selection. In version 6.0, such error messages will be ignored when running a study in which the particle tracing interface is not solved for.

Backward Compatibility with Version 5.5

PARTICLE PROPERTIES FROM MATERIAL

In the Particle Tracing for Fluid Flow interface, the option to take particle properties such as density **From material** was added in version 5.6. When opening a model built in version 5.5 or earlier, **User defined** will be selected from the list instead of **From material** in the settings for the **Particle Properties** node.

SAMPLING OF AUXILIARY DOFS 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 versions 5.5 and earlier, the mean and standard deviation were specified. In versions 5.6 and later, the maximum and minimum are specified.

When opening a model built in version 5.5 or earlier, the text fields for the maximum and minimum values in uniform distributions of auxiliary variables are automatically defined to give the same result as in the older model:

```
New minimum = (Old mean) - (Old standard deviation)*sqrt(3)
New maximum = (Old mean) + (Old standard deviation)*sqrt(3)
```

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.

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.

Pipe Flow Module

New Functionality in Version 6.0

NON-NEWTONIAN FLOW IN PIPES

A new Herschel–Bulkley fluid model is now available for the **Pipe Flow** interface. Using the Herschel–Bulkley model, you can simulate flow of fluids exhibiting viscoplastic behavior.

ADDITIONAL OPTION FOR T-JUNCTIONS

The **Pipe Flow** interface now includes an additional option for the T-junction specification called **Loss coefficient with respect to common branch**. This option automatically calculates the dimensionless loss coefficients accounting for a particular flow situation, for example, if the flow in the junction is joining or separating.

Updated Models in Version 6.0

STRESS IN COOLING PIPELINE NETWORK

This example demonstrates how to model coupled flow, heat transfer, and structural deformation and stress in a pipeline network. Gravity loads from the pipe and fluid are also taken into account.

TOPOLOGY OPTIMIZATION OF A DISTRICT HEATING NETWORK

This example shows how to design the layout of a district heating network using topology optimization. The model requires the Pipe Flow Module and the Optimization Module.

Plasma Module

New Functionality in Version 6.0

CONSISTENT STABILIZATION
Streamline diffusion for electrons and ions is available when the **Formulation** is set to **Finite element (linear shape function)** and **Finite element (quadratic shape function)**. This stabilization technique makes it easier to solve plasma problems with a large drift component like corona discharges and streamers.

INCONSISTENT STABILIZATION
Isotropic diffusion for electrons and ions is available when the **Formulation** is set to **Finite element (linear shape function)** and **Finite element (quadratic shape function)**. This stabilization technique adds a diffusion component to the transport equations in order to dampen the effect of oscillations.

STATIONARY AND FREQUENCY-STATIONARY STUDIES ARE AVAILABLE FOR THE PLASMA INTERFACE
Stationary and frequency-stationary studies are available in the Plasma interface. These types of studies increase computational speed for DC and ICP discharges, allows for fast parameterization, and makes the use of optimization technology more practical.

TIME DEPENDENCY ADDED TO THE BOLTZMANN EQUATION IN THE TWO-TERM APPROXIMATION
Time dependency was added to the isotropic part of the two-term expansion of the Boltzmann equation in the **Boltzmann Equation, Two-Term Approximation** interface and to the Plasma interface when using the **Local Field Approximation**. This new functionality makes it possible to study cases in which the isotropic part of the electron distribution function cannot be assumed to be stationary.

IMPROVEMENTS IN THE STABILITY OF THE EQUILIBRIUM DISCHARGE INTERFACES
Several improvements were made in the equilibrium discharge interfaces to increase their stability. In general, models solve smoother and faster. The improvements include changes in the solver settings and how the fluid properties are interpolated and extrapolated.

DC CORONA DISCHARGE IN AIR IN A POINT-TO-PLANE CONFIGURATION

This tutorial model presents a study of a negative corona discharge in dry air at atmospheric pressure in a point-to-plane configuration. A millimeter-sized elliptical electrode creates a high-intensity electric field where the corona discharge occurs. A ground plane is placed 10 cm away from the corona electrode. The voltage is varied between −5 and −50 kV. The creation and transport of charged species is solved self-consistently coupled with Poisson's equation. The local field approximation is used, which means that transport and source coefficients are assumed to be well parameterized through the reduced electric field.

DOUBLE-HEADED STREAMER IN NITROGEN

This tutorial model presents a study of a double-headed streamer in nitrogen at atmospheric pressure. An initial seed of electrons is placed between two electrodes that apply an initial electric field of 52 kV/cm to the gas. A negative and positive streamer propagate toward the electrodes. The electron density, electric field, and the propagation velocity of the streamer agree well with simulation results published by Bessières and others.

PLASMA DC ARC

This example investigates the electrical and thermal characteristics of a plasma DC arc created in a point-to-plane configuration. The discharge is assumed to be in local thermodynamic equilibrium. The plasma is considered a conductive fluid medium and is modeled using a magnetohydrodynamics approach. This model shows how to use the Combined Inductive/DC Discharge interface to simulate the plasma generated in a DC arc.

PLASMA-PULSED ARC

This example investigates the electrical and thermal characteristics of a plasma-pulsed arc created in a point-to-plane configuration. The discharge is assumed to be in local thermodynamic equilibrium. The plasma is considered a conductive fluid medium and is modeled using a magnetohydrodynamics approach. This model shows how to use the Combined Inductive/DC Discharge interface to simulate the plasma generated in a pulsed arc.

COIL OPTIMIZATION OF AN ICP REACTOR

This model shows how shape optimization can be used to design the coils of an ICP reactor to obtain plasma uniformity. The reactor in study is a planar ICP with the coils distributed along the radial direction. The Optimization study step is used to find the best coil placement so that the gradient of the electron density in the reactor center is minimized. A target electron density in the reactor center is also imposed by varying the coil power.

TRANSIENT NEGATIVE MOBILITY AND NEGATIVE DIFFERENTIAL CONDUCTIVITY EFFECTS IN XENON

This tutorial model presents a study showing the transient negative mobility and the negative differential conductivity effects in xenon. The stationary and time-dependent Boltzmann equation in the two-term approximation is used to compute the electron energy distribution function.

Updated Applications in Version 6.0

INDUCTIVELY COUPLED PLASMA (ICP) TORCH

This model before version 6.0 was solved using a time-dependent study. Now, it is solved with a stationary solver that allows for much faster computation times.

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 6.0

POROELASTICITY, LARGE DEFORMATION, SOLID MULTIPHYSICS INTERFACE

The new **Poroelasticity, Large Deformation, Solid** multiphysics interface is used for modeling poroelasticity under finite structural deformations. It is similar to the existing **Poroelasticity, Solid** multiphysics interface but with an added **Elastic Predeformation** node in the Solid Mechanics interface.

You can find this interface under the **Poroelasticity** branch in the **Structural Mechanics** folder in the **Add Physics** tree. This new interface also requires the Structural Mechanics Module.

IMPROVED HANDLING OF POROUS MATERIALS

Porous materials are now defined using phase-specific properties tabulated in the **Porous Material** node. In addition, subnodes can be added for the solid and fluid features. Several subnodes can be defined for each phase. This functionality makes it possible to use one and the same porous material for fluid flow, chemical species transport, and heat transfer without having to duplicate material properties and settings.

IMPROVEMENTS TO BRINKMAN EQUATIONS INTERFACE

In the Brinkman Equations interface, the new **Porous Medium** feature now supports the **Porous Material** functionality. When opening a model created in version 5.6 or earlier, the old **Fluid and Matrix Properties** feature is not migrated, but a warning message is prompted. It should be manually replaced by the new **Porous Medium** feature (which adds a **Porous Matrix** and **Fluid** subfeatures) to allow an updated definition of the material properties. Support for obsolete features is not ensured in future versions; thus it is strongly recommended to transfer the old feature to the new feature.

POROUS SLIP IN THE BRINKMAN EQUATIONS INTERFACE

The boundary layer between free and porous media flow may be very thin and impractical to resolve. The new porous-slip wall treatment makes it possible to account for porous walls without resolving the full flow profile in the boundary layer. Instead, a stress condition is applied at the porous surfaces by utilizing an asymptotic solution,

yielding a good accuracy in the bulk flow. The functionality is activated in the new **Porous treatment of no slip condition** settings, and it is then used as the default wall condition. This new feature can be used in most problems where the model domain is large.

TWO-PHASE FLOW IN POROUS MEDIA

The new Brinkman Equations multiphysics interface combines the **Brinkman Equations** and **Level Set** interfaces and automatically adds a **Two-Phase Flow, Level Set** coupling node. It solves the conservation of momentum and the continuity of mass with the Brinkman equations. The front between two immiscible fluids in porous media is tracked with the level-set function.

You can find this interface under the **Multiphase Flow>Two-Phase Flow, Level Set** branch in the **Fluid Flow** folder in the **Add Physics** tree.

NONISOTHERMAL FLOW IN POROUS MEDIA

The new **Nonisothermal Flow, Brinkman Equations** multiphysics interface automatically adds the coupling between heat transfer and fluid flow in porous media. It combines the **Heat Transfer in Porous Media** and **Brinkman Equations** interfaces and automatically adds a **Nonisothermal Flow** coupling node.

You can find this interface under the **Nonisothermal Flow** branch in the **Fluid Flow** folder in the **Add Physics** tree.

EXTENDED FEATURES FOR HEAT TRANSFER IN POROUS MEDIA

The **Porous Medium** domain feature now includes three different models, which correspond to the thermal equilibrium, thermal nonequilibrium assumptions, and the configuration of packed beds. These models can be selected from the new **Porous medium type** list in the **Porous Medium** section:

- The default **Porous Medium** feature in the Heat Transfer in Porous Media interface by default selects the **Local thermal equilibrium** option under the **Porous medium type** list. This corresponds to the previously available averaging techniques within the **Porous Medium** feature, which solved a single equation for the average temperature field with effective properties. New averaging techniques are available for computing the **Effective thermal conductivity** in porous media: **Solid spherical inclusions**, **Fluid spherical inclusions**, **Wrapped screen**, and **Sintered metal fibers**. In addition, the **Volume average** and **Reciprocal average** techniques have been renamed to **Plane layers parallel to heat flow** and **Plane layers perpendicular to heat flow**,

respectively, for clarity. Finally, using the new **Equivalent thermal conductivity** option, you can enter user-defined expressions.

- When the new Local Thermal Nonequilibrium interface is added, the **Porous medium type** list is set to **Local thermal nonequilibrium**. This setting replaces the previously available Local Thermal Nonequilibrium multiphysics interface (which included the Heat Transfer in Solids interface, Heat Transfer in Fluids interface, and Local Thermal Nonequilibrium multiphysics coupling node). The settings for the fluid and solid materials are available in the **Fluid** and **Solid** subnodes of the **Porous Medium** feature, which provides the setting of the **Interstitial convective heat transfer coefficient** (previously available in the multiphysics coupling node). The **Local Thermal Nonequilibrium Boundary** feature is automatically added to surrounding domains where the fluid and solid temperatures are not in equilibrium.

- When the new Heat Transfer in Packed Beds interface is added, the **Porous medium type** list is set to **Packed bed**. This option corresponds to a new multiscale modeling approach tailored for packed beds.

For these three **Porous medium type** options, the postprocessing variables (thermal properties, heat fluxes, heat sources, and heat and energy balance) are available in a unified way for homogenized quantities.

MULTISCALE HEAT TRANSFER IN PACKED BEDS

The new Heat Transfer in Packed Beds interface is useful to model packed beds made of spherical homogenized porous particles in which the temperature varies radially. The temperature distribution in the pellets is computed for every position in the bed, and it is coupled to the temperature in the surrounding fluid through an interstitial heat flux or a temperature continuity condition between the pellets' surface and the fluid.

In this configuration, the **Porous Medium** feature has three subnodes (**Fluid**, **Pellets**, and **Pellet-Fluid Interface**), and a **Local Thermal Nonequilibrium Boundary** feature is automatically added to surrounding domains where the fluid and solid temperatures are not in equilibrium. The new **Pellet** subnode, available under the **Porous Material** node, can be used to define the properties of the pellet and fluid.

You can find this interface under the **Porous Media** branch in the **Heat Transfer** folder in the **Add Physics** tree.

EXTENDED FEATURES FOR MOISTURE TRANSPORT

The **Moisture Flow** multiphysics coupling has a new option to account for the leakage velocity at walls where evaporation and condensation take place. When the **Mixture**

type for moist air is set to **Concentrated species** in the Moisture Transport interface, the **Account for Stefan velocity at walls** check box is available in the **Moisture Transport at Walls** section. This functionality is available for free and porous media, and it automatically adds the corresponding contribution to the fluid flow velocity.

In the Moisture Transport interface, the **Hygroscopic Porous Medium** feature has been updated to match the new design for features using porous materials. It now contains the **Liquid Water**, **Moist Air**, and **Porous Matrix** subnodes for the settings of each phase.

In addition, the Moisture Transport interface has extended functionality for several boundary conditions.

Several postprocessing variables are now available to check mass balance. In particular, the `mt.massBalance`, `mt.dwcInt`, `mt.ntfluxInt`, and `mt.GInt` predefined variables make it possible to check the mass balance, the total accumulated moisture rate, the total net moisture rate, and the total mass source, respectively.

MISCELLANEOUS

- An efficient formulation of the sensible enthalpy predefined variable, `ht.DeltaH`, is now available for free and porous media containing moist air. This analytical formulation, taking advantage of the polynomial definition of the heat capacity for moist air, drastically reduces the evaluation time of the accumulated heat and energy rates in postprocessing.

- In the **Moist Porous Medium** feature of the Heat Transfer interface, the definition of the effective properties has been fixed to include the liquid water contribution.

- The **Longitudinal dispersivity** and **Transverse dispersivity** settings in the **Thermal Dispersion** feature are now available with all the products that include the feature.

Backward Compatibility with Version 5.6

- The Local Thermal Nonequilibrium predefined multiphysics interface and the **Local Thermal Nonequilibrium** multiphysics coupling are obsolete. These features are not migrated when opening a model created in version 5.6 or earlier. They should be manually replaced by the new **Porous Medium** feature, with the **Porous medium type** set to **Local thermal nonequilibrium**. Support for features marked as obsolete is not ensured in future versions, so it is strongly recommended to transfer the old features to the new feature.

- In the Moisture Transport interface, the old **Hygroscopic Porous Medium** feature not supporting the **Porous Material** functionality is now obsolete. This feature is not

migrated when opening a model created in version 5.6 or earlier. It should be manually replaced with the new **Hygroscopic Porous Medium** feature. Support for features marked as obsolete is not ensured in future versions, so it is strongly recommended to transfer the old feature to the new feature.

Polymer Flow Module

New Functionality in Version 6.0

NEW VISCOELASTIC MODEL

The exponential Phan–Thien Tanner viscoelastic model (**EPTT**) has been added in the Viscoelastic Flow interface.

SISKO MODEL FOR INELASTIC NON-NEWTONIAN MODEL

The new **Sisko** inelastic model has been added to model the flow of non-Newtonian fluids at moderate and high shear rates.

CURING REACTION INTERFACE

The new Curing Reaction interface has been added to model the crosslinking of thermosetting polymers. The interface includes predefined models for the curing reaction rate and models for correlating the viscosity and degree of cure.

FLUID-STRUCTURE INTERACTION FOR VISCOELASTIC FLUIDS

New multiphysics interfaces — Fluid-Solid Interaction, Viscoelastic Flow; Fluid-Solid Interaction, Viscoelastic Flow; and Fixed Geometry — have been added to facilitate a convenient setup of models combining flow of viscoelastic fluids and deformations of solids. The Fluid-Solid Interaction, Viscoelastic Flow interface also requires a license for either the Structural Mechanics Module, MEMS Module, or Multibody Dynamics Module.

LICENSE UPDATE FOR BRINKMAN EQUATIONS AND FREE AND POROUS MEDIA FLOW INTERFACES

The Brinkman Equations and Free and Porous Media Flow interfaces are now available in the Polymer Flow Module.

TWO-PHASE FLOW IN POROUS MEDIA

A new multiphysics interface combines the Brinkman Equations and the Level Set interfaces and automatically adds a Two-Phase Flow, Level Set coupling node. It solves for conservation of momentum and mass with the Brinkman equations. The front between two immiscible fluids in porous media is tracked with the level set function.

PASTA EXTRUSION

This example shows how to simulate the nonisothermal flow of the dough in the metering zone of the pasta extruder. To account for shear thinning behavior of the dough, the power law non-Newtonian model with temperature-dependent parameters is used.

VISCOELASTIC FLOW THROUGH A CHANNEL WITH A FLEXIBLE WALL

This model demonstrates how to use the new Fluid-Solid Interaction, Viscoelastic Flow interface. This example models the flow of a viscoelastic fluid in a channel in which part of the upper wall is replaced by an elastic plate subjected to an external pressure.

Ray Optics Module

OPTICAL LIBRARY IMPROVEMENTS

The Optical Material Library has been significantly improved in version 6.0. Glasses from CDGM Glass Co., Ltd.; Corning Incorporated; Ohara Inc.; and Schott AG now include a more complete set of material properties. In addition to optical dispersion coefficients or tabulated values of the refractive index, many of these materials now include the internal transmittance, thermal conductivity, specific heat capacity, Young's modulus, Poisson's ratio, and coefficient of thermal expansion. This extension of the Optical material library greatly facilitates coupled structural-thermal-optical performance (STOP) analysis.

SIMPLIFIED NONLOCAL COUPLING NAMES

The built-in couplings that evaluate the sum, average, maximum, and minimum of expressions over all rays have been renamed. In an instance of the Geometrical Optics interface with the default name gop, the old name for the sum was `gop.gopop1()`; the new name is `gop.sum()`. Similarly, the average, maximum, and minimum are now `gop.ave`, `gop.max`, and `gop.min`, respectively.

Backward Compatibility

The old coupling names, such as `gop.gopop1()` in the example above, will be labeled as deprecated, but they are still supported in version 6.0.

GAUSSIAN BEAM RAY RELEASE FEATURE

In a Geometrical Optics model that solves for ray intensity or power, you can add the **Gaussian Beam** node to release rays with a Gaussian intensity or power distribution.

Because geometric ray tracing does not consider diffraction in the vicinity of the beam waist, the **Gaussian Beam** node is appropriate only when the wavefront normals can be approximated as following straight lines. This is a reasonable assumption when the model geometry is much larger than the Rayleigh range of the beam, in which case the rays can be seen to emanate out from a point with angle-dependent intensity or power; or occasionally when the Rayleigh range is much larger than the geometry, in which case the released beam is collimated and the intensity or power is a Gaussian function of radial distance from the beam axis.

BLACKBODY RADIATION RAY RELEASE FEATURE

You can use the **Blackbody Radiation** node to release rays diffusely from a surface, with initial ray intensity or power based on the surface temperature. If tracing of polychromatic light has been enabled in the Geometrical Optics interface settings, then the wavelength or frequency of the released rays is sampled from the Planck distribution.

The more general-purpose ray release features, such as **Release from Grid** and **Release from Boundary**, now also support sampling the initial wavelength or frequency from the Planck distribution, but for these release features, the power or intensity must be explicitly given; it is not automatically derived from a surface temperature.

NEW WAYS TO ASSIGN A WEIGHTED INTENSITY DISTRIBUTION

When releasing a distribution of rays from a point, using a node such as **Release from Grid**, you can now choose to assign a **Weighted distribution** of initial intensity or power. The sum of the initial power over all released rays will still add up correctly to the specified total source power, but the intensity and power of individual rays can be a function of initial position or direction.

TRANSFORMATIONS OF LOADED RAY COORDINATES

When loading initial ray position and direction from a file using the **Release from Data File** node, the distribution of initial ray positions can now be dilated, rotated, and translated. If a rotation is applied, you can also choose to apply the same rotation to the initial ray directions if they are also loaded from the same file.

EASIER POSTPROCESSING OF REFRACTIVE INDEX

Built-in postprocessing variables allow you to easily visualize the refractive index of materials at the d-line, F-line, and C-line; as well as the Abbe number. Usually, the refractive indices of dispersive materials can only be plotted on discrete rays because they are expressed as functions of the ray frequency or vacuum wavelength.

NEW WAYS TO DEFINE ABSORBING MEDIA

There are several new ways to define the material properties of absorbing media in the Geometrical Optics interface. The default behavior is to specify the imaginary part of the refractive index directly. Alternatively, you can specify the intensity attenuation coefficient or the internal transmittance for a material sample of a certain thickness.

RELEASE FROM ELECTRIC FIELD IMPROVEMENT

When releasing rays from a boundary with the **Release from Electric Field** node, which initializes the ray Stokes parameters based on the electric field in an adjacent domain, you can now choose to initialize the ray direction using the Poynting vector of the adjacent field. In previous versions, rays could only be released normal to the boundary.

IMPROVED HANDLING OF CURVED GRATINGS

The interpretation of diffraction gratings on curved surfaces has been fine-tuned in version 6.0. You can now control whether the grating constant should be interpreted as the distance between unit cells of the grating along the curved surface, or as the projected distance between unit cells on a tangent plane.

New Applications in Version 6.0

PETZVAL LENS OPTIMIZATION

This example extends the basic Petzval Lens tutorial by demonstrating how an optical prescription can be modified when introducing a replacement glass. To maintain the same level of image quality using the replacement glass, small adjustments to the radii of curvature of the lenses are required. The changes in the lens radii of curvature were computed using the BOBYQA gradient-free optimization method. The perturbations in the lens radii of curvature were used as control parameters, while the global objective function was the sum of the root mean square spot size at three different field angles.

MICROLITHOGRAPHY LENS

In this example, geometrical optics simulation is performed on a system of 21 fused silica spherical lens elements for use in deep ultraviolet (DUV) photolithography. The spot diagram shows excellent image quality over a 23.4 mm image circle.

ROWLAND CIRCLE SPECTROMETER

The Rowland circle is a circle of radius R that lies tangent to a concave curved diffraction grating of radius $2R$. When light enters through an arrow slit located anywhere along the Rowland circle and then hits the grating, the reflected rays of each diffraction order will be focused at different points along the same circle. For nonzero diffraction orders, the rays are also sorted by wavelength.

Important Fixes in Version 6.0

WAVEFRONT CURVATURE REINITIALIZATION BY GRATINGS

When solving for ray intensity, the reinitialization of the wavefront radii of curvature has been improved when releasing rays of nonzero diffraction order at the grating surface.

SCATTERING BOUNDARY CONDITION IMPROVEMENTS

When solving for ray intensity or power, the **Scattering Boundary** node shows significantly improved energy conservation in version 6.0 compared to version 5.6. This is because the ray splitting algorithm that produces reflected and refracted rays at the rough surface now uses the same value of the random perturbation to the surface normal to compute the Fresnel coefficients of the reflected ray as it does for the refracted ray.

ACCURACY IMPROVEMENTS FOR BOUNDARY ACCUMULATORS

The accuracy of accumulated variables defined on curved boundaries has been significantly improved.

COUNT REFLECTIONS CHECK BOX NO LONGER CAUSES AN ERROR

Previously, an error message was given if the intensity formulation for graded media was selected in a 3D model and the **Count reflections** check box was selected. In version 6.0, these features can be used together without causing an error message to appear.

UNIT SYNTAX IN PART LIBRARIES

Some of the parts in the Part Libraries for the Ray Optics Module have been updated to use proper unit syntax in their definitions. This makes the parts more robust when toggling between different unit systems.

ERROR MESSAGES FROM INTERFACES NOT SOLVED FOR

In version 5.6, an error message was given when trying to run any study if an instance of the Geometrical Optics interface had no ray release features, or if the ray release features had no selection. In version 6.0, such error messages will be ignored when running a study in which the Geometrical Optics interface is not solved for.

IMPROVED PHASE REINITIALIZATION FOR POLARIZED RAYS

A stability improvement has been made to the reinitialization of phase when rays are reflected or refracted at boundaries. The phase variable is solved for when ray intensity

is solved for and the **Compute phase** check box is selected in the Settings window for the Geometrical Optics interface.

Backward Compatibility with Version 5.5

NEW OPTIONS FOR SPECIFYING REFRACTIVE INDEX

Beginning in version 5.6, in the settings for the **Medium Properties** node you can either specify the absolute refractive index or the refractive index relative to air at a given temperature and pressure. If you selected **Specify refractive index** from the **Optical Dispersion Model** list in version 5.5 or earlier, then in a later version **Specify absolute refractive index** will be selected from the **Refractive index of domains** list.

SAMPLING OF AUXILIARY DOFS FROM UNIFORM DISTRIBUTIONS

When a ray optics model includes an **Auxiliary Dependent Variable**, or when polychromatic light is released with a distribution of frequency or vacuum wavelength, you have the option of sampling the initial values of these extra variables from a **Uniform** distribution. In versions 5.5 and earlier, the mean and standard deviation were specified. In versions 5.6 and later, the maximum and minimum are specified.

When opening a model built in version 5.5 or earlier, the text fields for the maximum and minimum values in uniform distributions of auxiliary variables are automatically defined to give the same result as in the older model:

```
New minimum = (Old mean) - (Old standard deviation)*sqrt(3)
New maximum = (Old mean) + (Old standard deviation)*sqrt(3)
```

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>`.rmrms.
- `<phys>`.`<feat>`.rmaxrel has been renamed to `<phys>`.`<feat>`.rmax.
- `<phys>`.`<feat>`.rmidrms has been renamed to `<phys>`.`<feat>`.rmrms.

RF Module

New Functionality in Version 6.0

ELECTROMAGNETIC WAVES, BOUNDARY ELEMENTS

This new physics interface is for modeling the scattering properties of objects, evaluating electric fields far from the scatterer or the far fields of an antenna placed on an electrically large platform.

LAYERED TRANSITION BOUNDARY CONDITION

Multiple thin layers, such as the gold-plated copper of a circuit board trace or close-to-normal incidence on an antireflection coating to an optical lens, can be described by the Layered Transition Boundary Condition feature. It requires combining this physics feature, Layered Material in the global Materials, and Layered Material Link in the Materials node.

THREE-PORT NETWORK FEATURE

This feature characterizes the response of a three-port network component using S-parameters. Import a Touchstone file to describe the physical behavior and response of a three-port device or system through three-port boundaries without addressing a complicated geometry.

B-H CURVE MAGNETIC FIELD CONSTITUTIVE RELATION

The B-H curve option is available in the magnetic field constitutive relation for modeling nonlinear magnetic phenomena. The material properties from the Nonlinear Magnetic material library, available with the RF Module, can be used to relate the magnetic field and magnetic flux density.

REFINE CONDUCTIVE EDGES

This new physics-controlled mesh option quickly identifies the exterior edges of signal path boundaries, configured by perfect electric conductors or transition boundary conditions, and applies a user-specified mesh size.

SHIFTED LAPLACE CONTRIBUTION ON MULTIGRID LEVELS

If no geometrical feature sizes are smaller than half a wavelength, when the operating frequency is high, modeling with a higher-order element such as cubic element

discretization is beneficial for faster computation. The computation efficiency can be further boosted by imposing the Shifted Laplace contribution on multigrid levels.

SUGGESTED ITERATIVE SOLVER FOR PERIODIC STRUCTURES

Typical periodic problems are solved with a direct solver. However, the direct solver consumes a lot of memory, when the periodic unit cell size is not subwavelength. In this case, switch to the Suggested Iterative Solver to finish the computation faster with less memory usage.

CONSTRAINT-FREE PORT OPTION

The constraint-free port option is available to calculate the expansion coefficients as an overlap integral, while in the default port formulation, the expansion coefficients (or S-parameters) are calculated by adding a scalar dependent variable for each coefficient and then adding a constraint to enforce the series expansion.

FREQUENCY DOMAIN, RF ADAPTIVE MESH STUDY

The dedicated study step for mesh adaptation makes the workflow much easier when setting up mesh adaptation for modeling microwave and millimeter-wave antennas and circuits. This dedicated study step automatically provides a physics-controlled functional, goal-oriented termination expressions, and other settings. It requires the use of linear element discretization in the physics interface. The adapted mesh can be used for further frequency-domain simulations to characterize the device under test.

REFLECTION COEFFICIENT WITH MULTIPLE EXCITATION

When exciting all ports as in a phased antenna array, it is possible to compute the reflection coefficient at each excited port that includes the impedance mismatching as well as coupling by adjacent active ports.

SYMMETRY AXIS REFERENCE POINT

The **Symmetry Axis Reference Point** subnode is available in 2D axisymmetry, when there is an incident field defined in the parent node. Then this subnode is available as a default subnode to the **Scattering Boundary Condition** parent node. The **Symmetry Axis Reference Point** subnode defines a reference position at the intersection point between the parent node's boundary selection and the symmetry axis.

DEFAULT PLOTS FOR NUMERIC PORT MODE FIELDS

Plots are automatically created of the port mode fields for numeric ports.

MODELING OF A CPW USING NUMERIC TEM PORTS

This model shows how to set up a coplanar waveguide (CPW) using numeric ports analyzed as a TEM option, under the assumption that the ports mode fields are close to transverse electromagnetic (TEM).

MODELING OF A GROUNDED CPW USING NUMERIC TEM PORTS

This model shows how to set up a grounded coplanar waveguide (GCPW), using TEM-like numeric ports. The edges of metalized vias are included on the numeric port boundaries to capture a proper mode propagating through the GCPW circuit.

FEM-BEM COUPLING OF A MICROSTRIP PATCH ANTENNA

This example shows how to couple the finite element method (FEM), analyzing a microstrip patch antenna, to the boundary element method (BEM), for evaluating the field outside the FEM computational domain. The model computes the S-parameter, near-field distribution, and far-field radiation pattern through the FEM and the electric fields outside the near-field air domain sphere with the BEM.

CPW RESONATOR FOR CIRCUIT QUANTUM ELECTRODYNAMICS

This model demonstrates a transmission line resonator, one of the main components of CPWs. The resonator can be built from CPW transmission lines, terminated with a combination of open and short ends. These ends create a resonator out of a CPW, with the open and short ends functioning as zero current and zero voltage boundary conditions, respectively.

WI-FI BOOSTER YAGI-UDA ANTENNA

This models demonstrates how to design a conventional Yagi–Uda antenna for the 2.4–2.5 GHz band. The antenna can direct the signal in a particular direction and prevent intruders from eavesdropping on personal Wi-Fi signals.

RAT-RACE COUPLER

The rat-race coupler model has been updated to show how to use the layered transition boundary condition — a new physics feature — to model two thin metal layers as a boundary condition.

Backward Compatibility with Version 5.6 and Earlier

The port mode fields are now defined as the outgoing waves, whereas in version 5.6 and earlier the port mode fields were defined as the incoming (exciting) waves. This change only affects User defined and Periodic port types because input fields can be provided for these types. When loading MPH-files from version 5.6 or earlier, the input fields for User defined and Periodic ports are migrated to function with the new mode field definition. However, for User defined ports, the migration assumes that the material adjacent to the port is homogeneous.

Rotordynamics Module

New Functionality in Version 6.0

NEW INTERFACE: SOLID ROTOR, FIXED FRAME

A new physics interface, Solid Rotor, Fixed Frame, has been added. Its functionality is essentially the same as in the existing Solid Rotor interface, but the rotor must be axially symmetric. The difference is that it considers the rotor from a fixed frame rather than from a corotating frame, just like the Beam Rotor interface.

In long rotors, there can be mixed whirling modes that are difficult to interpret in a corotational frame. This is one example where you would prefer the new interface over the existing Solid Rotor interface.

LIQUID ANNULAR SEAL

A new feature, **Liquid Annular Seal**, has been added to all rotordynamics interfaces. It can be used to model the effects of the pressure drop across the seal, as well as swirling of the fluid at the entrance of the seals. Both these effects affect the stability characteristics of the rotor.

TURBULENCE AND SURFACE ROUGHNESS IN HYDRODYNAMIC THRUST BEARING

The **Hydrodynamic Thrust Bearing** feature in the Hydrodynamic Bearing interface has been enhanced by the possibility to take turbulence and surface roughness into account. You can control this from the new **Contact Surface Property** section in the settings for this node.

REMOVAL OF NONLINEAR MATERIAL MODELS FROM SOLID ROTOR

The possibility to model plasticity, creep, and viscoelasticity in the Solid Rotor interface has been discontinued. This capability seems never to have been used and would block other important developments.

REDUCED INTEGRATION

A new framework has been added for reduced integration. It is now available in the Solid Rotor and Solid Rotor, Fixed Frame interfaces. Reduced integration is controlled from the **Quadrature Settings** section in **Linear Elastic Material**.

COMPONENT MODE SYNTHESIS

Linear components built using the Solid Mechanics and Multibody Dynamics interfaces can be reduced to computationally efficient reduced-order models using the Craig–Bampton method. A reduced component can then be used in dynamic or stationary analyses (for example, as foundations for a rotor).

New Models in Version 6.0

COMPARISON OF CAMPBELL PLOTS USING DIFFERENT ROTOR INTERFACES

Different types of elements can be used for modeling a rotor, depending on the level of complexity and the type of the system being modeled. The modeling steps and representation of the results will vary with the type of idealization. In this tutorial model, an eigenfrequency analysis is performed on a stepped rotor, using three different physics interfaces for rotordynamics: Solid Rotor; Solid Rotor, Fixed Frame; and Beam Rotor. The resulting Campbell plots from these interfaces are compared with each other. The model also helps in understanding the different steps involved when using each interface.

RESPONSE OF A ROTOR UNDER INFLUENCE OF SEAL FORCES

The fluid flow in seals causes a strong restoring radial force, opposing the shaft motion. Also, there is a cross-coupled force due to the axial variation of the circumferential flow. The first type of force always has a stabilizing effect on the rotor, whereas the second one in certain situations can cause an instability.

In this example, an axial compressor is modeled using the Beam Rotor interface. The compressor has 10 stages of impellers, with a seal near each impeller to avoid flow leakage. In addition, a balance piston seal is present at the end of the impeller stages. The time-dependent response of the system is studied for a gradually increasing rotor speed. Two cases are considered for the simulation: excluding all seals and including the seals. The responses of the system for the two cases are compared. The result shows a better stability in the presence of seals.

Semiconductor Module

New Functionality in Version 6.0

TRANSITION BETWEEN DISCRETE LEVELS

A new feature of transition between discrete trap levels has been added to the three trap-assisted recombination features (domain, boundary, and heterointerface) when the explicit trap option is selected and more than one discrete trap level is created. This makes it possible to simulate the transition between the trap levels and to simulate the transition between the quantized levels of a quantum well and a quantum dot.

You can view this new feature in the A Solar Cell with InAs Quantum Dots Embedded in AlGaAs/GaAs Quantum Wells tutorial model.

CONTACT RESISTANCE

A new **Contact resistance** option has been added to the Metal Contact boundary condition for both ohmic and Schottky contact types and all five driving modes: voltage, current, power, circuit current, and circuit voltage.

This feature can be enabled by selecting the **Contact resistance** check box (disabled by default) in the Settings window.

The A Cross-Bridge Kelvin Resistor Model for the Extraction of Specific Contact Resistivity tutorial model shows this new option.

KLAASSEN UNIFIED MOBILITY MODEL (LIC)

The Semiconductor interface now includes a **Klaassen Unified Mobility Model (LIC)** feature (sometimes referred to as Philips Unified Mobility Model) available under the Semiconductor Material Model node. In this model, the total carrier mobility is given by combining the lattice (L), donor (I), acceptor (I), and carrier-carrier (C) scattering effects. Screening of the impurities by charge carriers and clustering of impurities at high doping levels are also included. Both the Trench-Gate IGBT 3D and Trench-Gate IGBT 2D tutorial models use this new mobility model.

HETEROJUNCTION HEAT SOURCE

The Joule heating contribution to the boundary heat sources at heterojunctions has been included in the built-in variables of the Semiconductor interface. This makes it easier to perform coupled thermal analysis of heterostructures.

DEFAULT SOLVER

A new physics suggested solver sequence has been added to the Semiconductor interface to streamline the study setup for models with field-dependent mobility or impact ionization generation. These existing Application Library models are now more efficient with the new solver sequence: *mosfet_mobility*, *lombardi_surface_mobility*, c*aughey_thomas_mobility*, and *insb_pfet_density_gradient*.

TRAPPING FEATURES

A new option of user-defined additional electron and hole capture rates has been added to the three discrete trap level features (domain, boundary, and heterointerface) when the explicit trap option is selected.

The continuous trap distribution feature has been enhanced by expanding the range of the trap energy levels outside of the band gap and allowing the capture probability to depend on the trap energy level.

The trapping feature for the density-gradient formulation has been enhanced with the new option of solving for the trap occupancy (as opposed to the trap Fermi level).

MOBILITY MODELS

The Caughey–Thomas Mobility Model has been enhanced with several new options for the formulation of the driving forces. This adds versatility to the feature for users interested in different types of driving forces.

METAL CONTACTS

A new built-in global variable for the average terminal current density has been added for the Metal Contact boundary condition. This makes it convenient to obtain scaling independent output quantities for comparison purposes.

CORRECTIONS AND BACKWARD COMPATIBILITY

- The default expression, description, and equation display for the user-defined energy distribution of continuous trap levels have been updated.
- The selection of the Schrödinger–Poisson multiphysics coupling has been extended to include domains of the Second Order Hamiltonian domain condition (in 5.6 update 1).
- The Caughey–Thomas Mobility Model and Lombardi Surface Mobility Model have been improved for the finite volume formulation by applying the circumcenter operator on the driving forces.
- A formulation issue of WKB tunneling feature for Schottky contacts has been fixed.

- The trapping **Formulation** list under the discrete and continuous trap energy level subnodes has been moved to the parent trap-assisted recombination node (three kinds: domain, boundary, and heterointerface) to simplify model setup.

- The unit of the default value for the **Hole thermal velocity** under the **Carrier Capture** section for explicit traps has been corrected from m/s to cm/s.

New Models in Version 6.0

SURFACE-TRAP-INDUCED HYSTERESIS IN AN INAS NANOWIRE FET, A DENSITY-GRADIENT ANALYSIS

This tutorial analyzes the hysteresis of the conductance-gate-voltage (G-Vg) curves of an InAs nanowire FET, using the density-gradient theory to add the effect of quantum confinement to the conventional drift-diffusion formulation, without a large increase of computational costs. The hysteresis is caused by the dynamic charging effects of fast and slow semiconductor-oxide interface traps of continuous energy distributions and of both donor and acceptor types. The capture probability is modeled as thermally activated with a barrier height that varies with the trap energy level. The qualitative behavior and the order of magnitude of the computed G-Vg curves under various voltage ramping conditions agree well with simulation and experiment results found in the literature.

A CROSS-BRIDGE KELVIN RESISTOR MODEL FOR THE EXTRACTION OF SPECIFIC CONTACT RESISTIVITY

This benchmark example builds two models of a cross-bridge Kelvin resistor used for extracting the specific contact resistivity. The first model simulates the system in 3D, using the contact resistance feature built in the Semiconductor interface. The other model is a 2D approximation of the system developed in the reference paper, implemented using a boundary PDE math interface. The computed Kelvin contact resistance as a function of the contact window size and the diffusion tap width correlates well between the two models and agrees well with the result shown in the reference paper.

A SOLAR CELL WITH INAS QUANTUM DOTS EMBEDDED IN ALGAAS/GAAS QUANTUM WELLS

This example shows an approximate approach to model a dot-in-well solar cell as described by Asahi and others in the reference paper. The quantum wells and layers of quantum dots are each treated as lumped energy levels in the band gap. The authors specify transitions between the dot/well levels and the energy bands. The continuum

part of the current density is otherwise unimpeded by the wells and dots. This description is equivalent to the trapping feature in the Semiconductor interface, so it is used to model the wells and dots in this example. The computed trend of the photocurrent and occupancy of the quantum dot states agrees well with the result shown in the paper.

TRENCH-GATE IGBT 2D

In this first half of a two-part example, a 2D model of a trench-gate IGBT is built, which will be extended to 3D in the second half. In general, it is the most efficient to start with a 2D model to make sure everything works as expected before extending it to 3D. The Caughey–Thomas mobility model is combined with the Klaassen unified mobility model to account for velocity saturation and phonon, impurity, and carrier-carrier scattering. The contact resistance option of metal contact boundary conditions is used to implement the mixed-mode simulation with parasitic resistance at the collector and emitter as mentioned in the reference paper. The computed collector current density as a function of the collector voltage agrees reasonably well with the published result.

TRENCH-GATE IGBT 3D

In this second half of a two-part example, a 3D model of a trench-gate IGBT is built by extruding the 2D model from the first half. Unlike the 2D model, now it is possible to arrange the alternating n+ and p+ emitters along the direction of extrusion as in the real device. This more realistic arrangement leads to better quantitative agreement with experimental data. The computed collector current density as a function of the collector voltage agrees reasonably well with the published result.

IMPROVED MODELS

The two tutorial models MOSCAP 1D and MOSCAP 1D Small Signal have been expanded to demonstrate how to compute the differential capacitance using metal contacts where the terminal charge is not readily available.

The meshes for the two tutorial models Interface Trapping Effects of a MOSCAP and MOSCAP 1D Small Signal have been improved for a better compromise between discretization error and round-off error.

The *mosfet_mobility*, *lombardi_surface_mobility*, *caughey_thomas_mobility*, and *insb_pfet_density_gradient* library models have been updated to take advantage of a new autogenerated solver sequence.

The Thermal Analysis of a Bipolar Transistor model has been updated to use the available temperature and heat source items in the lists for the temperature and heat source couplings, as opposed to entering the corresponding variables manually.

Structural Mechanics Module

New Functionality in Version 6.0

MAGNETOMECHANICS MULTIPHYSICS INTERFACE

Two new physics interfaces for the analysis of coupled magnetic and mechanical effects have been added: **Magnetomechanics** and **Magnetomechanics, No Currents**. When you add such an interface, two physics interfaces are added to the model: Solid Mechanics and either Magnetic Fields or Magnetic Fields, No Currents. The new **Magnetomechanical Forces** multiphysics coupling is also added.

You can find these interfaces under the **Electromagnetics-Structure Interaction** branch in the **Structural Mechanics** folder in the **Add Physics** tree.

This interface also requires the AC/DC Module.

POROELASTICITY, LARGE DEFORMATION, SOLID MULTIPHYSICS INTERFACE

The new **Poroelasticity, Large Deformation, Solid** multiphysics interface is used for modeling poroelasticity under finite structural deformations. It is similar to the existing **Poroelasticity, Solid** multiphysics interface, but with an added **Elastic Predeformation** node in the Solid Mechanics interface.

You can find this interface under the **Poroelasticity** branch in the **Structural Mechanics** folder in the **Add Physics** tree.

This interface also requires the Porous Media Flow Module.

BEAM CROSS SECTION INTERFACE IN 3D

The Beam Cross Section interface is now available in 3D. In the 3D version, you have the possibility to extrude the cross section and then show a full 3D representation of the stresses in a straight beam.

As an effect of this addition, the 2D version of the interface has also been significantly updated. One major change is that the interface can handle more than one cross section. The selection of the cross section is thus no longer made at the interface level but in a node named **Homogeneous Cross Section**. It is possible to have any number of such nodes. The **Hole** node is thus no longer a top-level node but a subnode under **Homogeneous Cross Section**. Because there now can be results for several sections in the

same interface, the computed cross-section property variables now have feature scope rather than physics scope. For example, a cross-section area that in previous versions was named `bcs.A` will now be named `bcs.hcs1.A`. When opening an old model, it is automatically converted to the new structure.

STRESS FROM INTERNAL PRESSURE IN BEAM CROSS SECTION

For use with the Pipe Mechanics interface, it is now possible to enter a pressure in the **Hole** node in the Beam Cross Section interface. This will be used to compute the stress contribution from an internal gauge pressure.

TRANSFER OF DATA BETWEEN BEAM CROSS SECTION AND BEAM TYPE INTERFACES

Two new multiphysics couplings handling data transfer between the Beam Cross Section and the Beam or Pipe Mechanics interfaces have been added.

- The **Beam Cross Section - Beam Coupling** multiphysics coupling transfers cross-section data computed in the Beam Cross Section interface to the Beam or Pipe Mechanics interface. It is available in 2D and 3D.

- The **Beam - Beam Cross Section Coupling** multiphysics coupling transfers section forces computed in a Beam or Pipe Mechanics interface to a Beam Cross Section interface for visualization. This coupling is only available in 3D.

MAGNETOSTRICTION COUPLED TO ROTATING MACHINERY, MAGNETIC INTERFACE

The **Magnetostriction** multiphysics coupling can now also be used to connect the Solid Mechanics and Rotating Machinery, Magnetic physics interfaces.

COMPONENT MODE SYNTHESIS

Linear components built using the Solid Mechanics and Multibody Dynamics interfaces can be reduced to computationally efficient reduced-order models using the Craig–Bampton method. A reduced component can then be used in dynamic or stationary analyses, either in a model consisting of only reduced components, or together with ordinary full FE models. The latter can then be nonlinear.

To create such components, you add a **Reduced Flexible Components** node in the physics interface.

RIGID JOINT IN SOLID MECHANICS INTERFACE

A **Rigid Joint** node has been added to the Solid Mechanics interface. Such a joint can be used to connect two attachments rigidly. The feature is similar to the same joint in the Multibody Dynamics interface. The main purpose is for joining CMS components.

ATTACHMENTS NO LONGER REQUIRE MULTIBODY DYNAMICS LICENSE

You can now add the **Attachment** node with a Structural Mechanics Module or MEMS Module license. The main usage in the context of creating and joining CMS components.

LOADS ON SHELLS AT TOP OR BOTTOM SURFACE

It is now possible to apply loads in the Shell and Plate interfaces not only on the midsurface but also on the top and bottom surfaces. Using the actual location in the thickness direction can be important in the case of shells with significant thickness, in particular when curved.

The thickness direction location is selected in the new **Through-Thickness Location** section, available in the **Face Load**, **Edge Load**, **Point Load**, **Point Load, Free**, **Ring Load** and **Ring Load, Free** nodes' settings.

COUPLINGS BETWEEN DISJOINT SHELLS

Three new nodes have been added in the Shell interface, facilitating easier coupling between parts of shells that are located so that there are gaps in the geometry. These are: **Edge to Edge**, **Edge to Boundary**, and **Boundary to Boundary**. The couplings can be rigid or elastic. Some applications are:

- Imported geometries where there are slits between parts.
- Midsurface generation having left slits between parts.
- Avoiding the extra flexibility caused by using a common edge in, for example, a T-joint. The shell thickness can be taken into account in a more accurate way.
- Weld modeling, in which case you can use the flexible version of the coupling and evaluate the forces in it.

POINT LOADS AT ARBITRARY LOCATIONS

With the new **Point Load, Free** and **Ring Load, Free** features, you can apply point loads at arbitrary locations that do not coincide with a geometrical point or a mesh node. This is particularly useful when working with moving loads.

This functionality is available in the Solid Mechanics, Shell, Plate, Membrane, Beam, and Truss interfaces.

NEW DEFAULT BEHAVIOR FOR CONTACT AND CONTINUITY

If any contact pairs are defined under **Definitions**, a default **Contact** node will be added in the Solid Mechanics, Shell, and Membrane interfaces. Similarly, if there are any identity pairs, a default **Continuity** node will be added. In many cases, the pair handling can be considered as automatic because pairs can be generated during the finalization of the geometry sequence.

As an effect of the general reformulation of the pair functionality, the **Source external to current physics** check box is no longer needed and has been removed except in the Layered Shell interface.

TWISTING DEGREE OF FREEDOM IN 2D AXISYMMETRY

In the Solid Mechanics interface in 2D axisymmetry, it is now possible to include circumferential deformation. To do so, you select the **Include circumferential displacement** check box in the **Axial Symmetry Approximation** section in the settings for the physics interface.

INITIAL IMPERFECTION IN BUCKLING ANALYSIS

You can now use the buckling modes from a linear buckling analysis as initial imperfections to the geometry when performing a full nonlinear buckling analysis. This functionality is controlled from the new **Buckling Imperfection** node, which can be added from **Physics Utilities** under **Definitions**.

COMPUTATION OF SECTION FORCES IN SOLIDS

By adding the new **Section Forces** node in Solid Mechanics, you can compute section forces (axial force, shear forces, bending moments, and twisting moment) on a cross section in a solid structure.

FIBER-REINFORCED LINEAR ELASTIC MATERIAL

By adding one or more **Fiber** subnodes under a **Linear Elastic Material** node, you can augment the stiffness by the effect of distributed fibers. The fiber content is assumed to be a small fraction of the total material volume.

You can model thermal expansion of the fibers by adding a **Thermal Expansion** subnode under the **Fiber** node.

WRINKLING IN MEMBRANES

When a principal stress in a membrane is less than or equal to zero, it loses its stability. Physically, this means that wrinkling will occur. This situation can now be handled by

adding the new **Wrinkling** subnode under **Linear Elastic Material** in the Membrane interface.

VISCOELASTICITY IMPROVEMENTS

There are several important additions to the viscoelastic material models:

- For frequency-domain and time-dependent analyses, all of the viscoelasticity models have been augmented with the possibility to include viscoelasticity in the volumetric deformation.

- The Generalized Maxwell model now includes the possibility to prune branches representing frequency ranges outside the bandwidth of prescribed loads, improving the performance in time-dependent analysis for models with many viscoelastic branches.

- For frequency-domain analyses, a new user-defined viscoelasticity model makes it possible to enter frequency-dependent expressions for the loss and storage moduli or compliances.

- Through a new formulation of the viscoelastic equations, it is now possible to solve for eigenfrequencies in a structure containing viscoelastic materials using a standard procedure for damped eigenfrequency problems. Previously, the eigenvalue problem was nonlinear in the frequency, and only one eigenfrequency at a time could be computed. The new approach is available for the all viscoelastic models except user defined, under the assumption that fractional derivatives are not used.

IMPROVED MIXED FORMULATION

In material models that have an option to select a mixed formulation, you can now modify the discretization for the auxiliary dependent variable (pressure or volumetric strain). To do so, **Advanced Physics Options** must be selected. Then, a new section named **Discretization** will appear in the settings for the material model as soon as a mixed formulation is selected. In this section, you can choose between different types of shape functions for the auxiliary dependent variable.

Another change is that the auxiliary dependent variable now has feature scope rather than physics scope.

NEW DAMPING MODELS

Two new damping models have been added for the mechanical material models:

- The **Wave attenuation** model is essentially a viscous model, but with parameters given by measured data for the attenuation of elastic waves in the material. It is available in the **Linear Elastic Material** in Solid Mechanics.

- The **Maximum loss factor** model is mainly intended for time-domain analysis of materials for which a loss factor representation provides a good description in the frequency domain. This damping model is available for all material models that support viscous damping.

The new damping models can be used in both frequency-domain and time-domain studies.

DAMPING FOR PIEZOELECTRIC MATERIALS IN TIME DOMAIN

Two new options have been added to the **Dielectric Loss** subnode under **Piezoelectric Material** in the Solid Mechanics interface: **Complex Permittivity** intended for frequency-domain analysis and **Maximum loss factor** intended for time-domain analysis.

MULTIPOLE DEBYE DISPERSION FOR PIEZOELECTRIC MATERIALS

New damping models have been added for piezoelectric materials. Under **Charge Conservation, Piezoelectric**, you can now add a **Dispersion** subnode. There, you can choose between the **Debye** and **Multipole Debye** dispersion models.

This feature requires the MEMS Module or AC/DC Module.

REDUCED INTEGRATION

A new framework has been added for reduced integration. It is now available in the Solid Mechanics and Membrane interfaces. Reduced integration is particularly useful when the computational cost per integration point is high, which is true for many advanced material models.

Reduced integration is controlled from the **Quadrature Settings** section in various material models. It is available in top-level material models like **Linear Elastic Material**. The selected integration rule will then be inherited by any subnodes that may be added.

EASIER GAUSS POINT EVALUATION IN DOMAINS

A set of physics-scoped operators have been added as tools for easier evaluation of Gauss point results in domains. The operators automatically adapt to the Gauss point pattern in each element. These operators are:

- `<phys>.gpeval(expr)`: Map a field of suitable order to Gauss point data.

- `<phys>.elemgpmax(expr)`: Maximum value in any Gauss point in an element.

- `<phys>.elemgpmin(expr)`: Minimum value in any Gauss point in an element.

- `<phys>.elemavg(expr)`: Average value of the Gauss points in an element.

- `<phys>.elemavg(expr)`: Integration over an element using Gauss point data.

INTERCALATION STRAIN

A new subnode, **Intercalation Strain**, has been added to the **Linear Elastic Material** and **Nonlinear Elastic Material** nodes in the Solid Mechanics interface. The purpose is to model inelastic strains caused by ion transport.

This feature requires the Battery Design Module.

BOLT PRETENSION FOR BEAMS

The **Bolt Pretension** feature has been added to the Beam interface, thus facilitating a simplified modeling of prestressed bolts. The functionality is analogous to that in the Solid Mechanics interface, with the exception that the bolt cross section is represented by a point selection rather than by a boundary selection.

SEQUENTIAL TIGHTENING OF BOLTS

In the **Bolt Selection** node, a new check box, **Sequential tightening**, has been added. When selected, it is possible to enter expressions for time or parameter values when the prestress force in the individual bolt is changed. Using this new approach, several bolts can be sequentially tightened within a single study step.

BOLT PRESTRESS BY TIGHTENING TORQUE

In the **Bolt Pretension** node, it is now possible to specify the bolt force also in terms of the tightening torque. In addition, when using one of the other options (bolt force or bolt stress), the corresponding torque can be computed.

DEFAULT EVALUATION GROUP FOR BOLT FORCES

When a **Bolt Pretension** node is active in a study, a default evaluation group containing a list of forces in all bolts selected in **Bolt Selection** subnodes is automatically generated.

AUTOMATIC GENERATION OF BOLT LABELS

When adding a new **Bolt Selection** subnode, the bolt label automatically gets a sequentially numbered default name. The base name that is used for the name generation is given in the new **Bolt Label** section in the parent **Bolt Pretension** node.

CHANGES IN BOLT THREAD CONTACT

As a consequence of the general contact improvements, there are significant changes in how bolt thread contact is handled. The **Bolt Thread Contact** node, which used to be a global feature, is now located under **Pairs** at the boundary level. The old **Thread Boundary Selection** subnode has been removed. The selection of pairs is now done directly in the **Bolt Thread Contact** node. This means that you need to add one **Bolt Thread Contact** node per bolt because the pair selection should represent one single bolt.

Also, the old **Bolt Label** input field has been removed because it is not used.

When opening a file created in a version prior to 6.0, the old representation is automatically converted to the new one. When using the API, there is no such conversion. Any file that references the old `BoltThreadContact` and `ThreadBoundarySelection` must be updated so that it uses the new `PairBoltThreadContact`.

COUPLING FROM BEAM POINTS TO SOLID EDGES

The **Solid-Beam Connection** multiphysics coupling has been extended with an option for connecting a point on a beam to edges in a solid. One use case is when combining bolts idealized by beams with components modeled with solid elements. Then the edge of the bolt hole can be connected to the end of the beam to form an abstract bolt head.

CIRCUMFERENTIAL MODE EXTENSION FOR AXISYMMETRIC SHELL

The capability to compute different harmonics in the azimuthal direction for eigenfrequency and frequency-domain analysis has been added to the Shell interface in 2D axisymmetry. You enable it by selecting **Circumferential mode extension** in the settings for the Shell interface.

WEAK CONSTRAINTS FOR AXISYMMETRIC SHELL

The foldline constraints, as well as the shell normal displacement constraints, can now be of the weak form also in the 2D axisymmetric version of the Shell interface. You can control the constraint formulations in the **Foldline Settings** and **Advanced Settings** sections in the settings for the Shell interface.

IMPROVED SHELL DATASET

The Shell dataset has received an important upgrade. Now, shells will be visualized as solids, with thickness and offset taken into account. Any variable that has a dependency on the through-thickness location will be displayed accordingly. This is, for example, used in the default stress plots in the Shell interface. In order to make use of the new functionality, the definitions of some variables have been changed. For example:

- New variables `shell.u`, `shell.v`, and `shell.w` have been added. They contain the displacement, including the effect of rotation and distance from the midsurface. These variables can be used in **Deformation** nodes, for example.

- The displacement magnitude `shell.disp` is now based on these new displacement variables, so that it represents a field also in the thickness direction.

UPDATED INPUT OF THICKNESS AND OFFSET IN SHELL

The settings in the **Thickens and Offset** node in the Shell interface have been changed, so that it is easier to enter the more common offset values. **Top surface on boundary**, **Midsurface on boundary**, and **Bottom surface on boundary** are now predefined choices.

FLUID-STRUCTURE INTERACTION WITH SCREEN

The **Screen** boundary condition on interior boundaries in fluid-flow interfaces will now automatically work together with the Shell and Membrane interfaces when using the **Fluid-Structure Interaction** multiphysics coupling.

COUPLINGS BETWEEN RIGID CONNECTORS

It is now possible to couple a rigid connector in the Pipe Mechanics interface to a rigid connector in Solid Mechanics, Shell, or Multibody Dynamics interfaces. To do so, select a rigid connector defined in the other physics interface from the **Connect to** list in the **Advanced** section in the settings for **Rigid Connector**.

INPUT OF RESIDUAL STRESS

In the **External Stress** node, there is now an option called **Residual Stress**. A residual stress will not directly affect the displacements. That is, if you just enter a residual stress, and no other loads, there will be no displacements. The stress is, however, added to the stress tensor in the sense that it will be part of the stress state that is used in various material models.

EXTERNAL STRESS CAN BE ENTERED AS FIRST PIOLA-KIRCHHOFF STRESS TENSOR

In the **External Stress** node in the Solid Mechanics, Layered Shell, and Membrane interfaces, it is now possible to enter the stress state in form of a 1st Piola–Kirchhoff stress tensor.

SYMMETRIC FORMULATION FOR RIGID CONNECTOR AND RIGID DOMAIN

Under geometric nonlinearity, the consistent stiffness and mass matrices generated by the **Rigid Connector** and **Rigid Domain** features may become nonsymmetric. The solution of a nonsymmetric problem requires significantly more memory and computation time than solving a symmetric problem of the same size. In version 6.0, you can choose to use a less accurate symmetric matrix instead. Doing so may require some more iterations, but each iteration will be significantly faster.

In order to use the symmetric formulation, select the **Use symmetric formulation for geometric nonlinearity** check box in the **Formulation** section in the settings for the feature.

IMPROVED SHEAR CORRECTION FACTORS FOR BEAMS

The shear correction factors generated by the standard beam cross sections have been updated using more accurate expressions that depend on the aspect ratio of the cross section as well as the material properties.

COORDINATE SYSTEM TYPES RESTRICTED FOR RIGID CONNECTOR

For a rigid connector, it is not meaningful to select a coordinate system with position-dependent axis orientations. This is no longer allowed, neither in the **Rigid Connector** node itself nor in its subnodes.

EXPRESSIONS FOR THE ROTATION VECTOR CHANGED IN RIGID CONNECTOR AND RIGID DOMAIN

The expression defining the variables giving the rotation vector in a rigid connector and rigid domain has been changed in such a way that it is more robust with respect to computing Jacobian contributions. This can be important if you want to, for example, control such a variable using other equations.

The affected variables for a rigid connector are named *<phys>*.th*<dir>_<tag>* (for example, `solid.thz_rig1`). For a rigid domain, the names are of the type *<phys>.<tag>*.th*<dir>* (for example, `solid.rd1.thy`).

CONSTRAINT GROUPS FULLY SUPPORTED FOR RIGID CONNECTOR AND RIGID DOMAIN

In earlier versions, constraint groups could not be used for rigid connectors or rigid domains if the option **Evaluate reaction forces** was selected. This limitation has been removed.

SLIP VELOCITY ADDED TO SHELL AND MEMBRANE INTERFACES

The **Slip Velocity** subnode under **Contact**, in which you can describe friction with a given slip direction, has been added also to the Shell and Membrane interfaces.

IMPROVEMENTS TO SAFETY FACTOR COMPUTATIONS

Some of the models for safety factor computation in the **Safety** node have been updated.

- In the Tsai–Hill criterion, there is now a separate input for the compressive strengths.
- The **Modified Tsai-Hill** criterion, intended for 2D analysis, has been removed from the list of failure models. To obtain the same effect, use the **Tsai-Hill** criterion and select the new **Use plane stress formulation** check box. When you open an old model that contained the **Modified Tsai-Hill** criterion, it will automatically be updated to the new settings.
- 2D formulations of the Tsai–Wu and Hashin criteria have been added. To use them, choose **Tsai-Wu orthotropic** or **Hashin** as the failure criterion, and then select the new **Use plane stress formulation** check box.

STRICTER DEFAULT TOLERANCE FOR TIME-DEPENDENT ANALYSIS

The default tolerance for time-dependent analysis has been decreased from 0.01 to 0.001 for all structural mechanics interfaces. This will affect new and old models in which the solver sequence has not been manually edited. The effect is that solutions will be more accurate but typically at the cost of longer solution times.

THE TIME-DEPENDENT, PRESTRESSED, MODAL STUDY TYPE REMOVED

Due to problems with obtaining a correct linearization in all situations, the **Time-Dependent, Prestressed, Modal** study type is no longer available.

TORQUE ARROWS FOR LOADS IN 2D

In the default load plots, out-of-plane moments in 2D are now represented by a new type of arrow, called **Torque** in the **Point** plot.

AZIMUTHAL MODE NUMBER AUTOMATICALLY TRANSFERRED TO REVOLUTION DATASET

When using circumferential mode extension for 2D axisymmetry, the azimuthal mode number (for example, `solid.mk`) is now automatically transferred to the revolution datasets that are created as default plots. This means that you do not need to modify the settings in the dataset when switching between results for different mode numbers.

MODELING OF CRACK CLOSURE

When modeling cracks using the **Crack** feature, you can now inhibit closure of the crack by adding a **Crack Closure** subnode. This subnode will add a contact condition, in which friction in the crack can also be taken into account.

CRACK FACE LOAD CAN BE ASSIGNED TO LOAD GROUPS

When using a **Face Load** under **Crack**, you can now assign a load group, in order to limit and scale the load for certain load cases.

IMPROVEMENTS IN STRESS INTENSITY FACTOR COMPUTATION

The stress intensity factors K_I, K_{II}, and K_{III} are now computed with a sign. It is thus possible to determine the range of, for example, K_{II} for a set of load cases. A negative value of K_I indicates crack closure.

In order to control the definition of the signs of the stress intensity factors in 3D, a new subnode, **Reverse Crack Front**, has been added under **Crack**.

The ratios between K_I, K_{II}, and K_{III} are determined through an analysis of the displacements on the crack surface. In the new **Stress Intensity Factor** section in the setting for the **J-integral** node, you can fine-tune the part of the crack that is used for these evaluations.

STRESS LINEARIZATION IMPROVEMENTS

The **Stress Linearization** feature has two important improvements:

- It is no longer necessary to use a geometrical line to define a stress classification line through the thickness. You can now use a line between two arbitrary points. The points can be geometrical points or just locations specified by coordinates.
- Stress linearization values can be presented as a field over a boundary. In this case, the evaluation is performed using a large number of automatically generated lines, extending orthogonally to the boundaries. By using this method, you can find the worst location for a stress classification line.

CHANGES IN RANDOM VIBRATION FUNCTIONALITY

There are several news within the field of random vibration analysis:

- The number of study steps generated by a Random Vibration (PSD) study sequence is now two instead of three. The "dummy" Frequency Domain study now resides inside the Model Reduction study.

- A new operator, q2sq, has been added. It is similar to the existing q2 operator but has more convenient arguments and output. In particular, it is targeted at computing the RMS of the von Mises stress and vector norms.

- In the **Random Vibration** node, it is possible to enter default parameters for operators like rms, which perform an integration. It is then possible to invoke these operators without the last three arguments.

- It is now easy to add the base acceleration to computed acceleration PSDs, thus facilitating direct evaluation of absolute as well as relative spectra.

- There are significant performance improvements in the random vibration computations.

CHANGED NOMENCLATURE FOR FORCE LINEAR STRAIN

The label of the **Force linear strain** check box, which is present in many material models, has been changed to **Geometrically linear formulation** in order to improve clarity.

OVERRIDING OF GEOMETRIC NONLINEARITY IN THE BEAM INTERFACE

In studies that are geometrically nonlinear, it is now possible to, per material selection, enforce geometrical linearity also in the Beam and Pipe Mechanics interfaces. You do this by selecting the **Geometrically linear formulation** check box. This can be useful for improving efficiency when there are parts where effects of geometric nonlinearity are negligible. One such case is when bolts are modeled using beams in a contact analysis.

CHANGES IN DEFAULT PLOTS

There are many improvement in the default plots. Some of the more prominent are:

- The default stress plot in 3D now uses a **Volume Plot** rather than a **Surface Plot**. There are several advantages with that change, including easier selection of individual domains and more consistent evaluation of some expressions defined at the domain level. However, you may find that the plotting is slower in larger models. Also, the visualization of curved boundaries with a coarse mesh is smoother in a surface plot.

- A new Prism color table is used for the default stress plots. This color table will emphasize regions with high stresses.

- In the default stress plots, the smoothing is conditional, so that stress jumps larger than 20% between adjacent elements will be plotted without smoothing. This makes it easer to identify a solution computed on a too coarse mesh.

EASIER USE OF MULTIPLE CELL PERIODICITY NODES

The way studies are generated has been changed so that unique studies are created for each **Cell Periodicity** node. This makes it possible to, for example, run nested homogenizations.

A new section, **Advanced**, has been added to the settings for **Cell Periodicity**. Here you can, if needed, select to use the old scheme to keep the old behavior.

IMPROVED GUI IN SPRING-DAMPER NODE

In the **Spring-Damper** node, options that are related to rotational degrees of freedom are now only active if such degrees of freedom are actually present at the attachment points.

THERMAL EXPANSION IMPROVEMENTS

The input options **Tangent coefficient of thermal expansion** and **Thermal strain** have been added to the **Thermal Expansion** node in the Shell interface as well as to the **Layered Thermal Expansion** multiphysics coupling. These options have also been added to the **Thermal Expansion** subnodes under all constraints.

IMPROVED HANDLING OF LOAD GROUPS FOR THERMAL EXPANSION AND HYGROSCOPIC SWELLING

The handling of load groups in **Thermal Expansion** and **Hygroscopic Swelling** has been improved. In particular, using several such features with the same selection now works in a more intuitive way.

REMOVAL OF SEVERAL PAIR FEATURES

As part of a major overhaul of the pair handling, several pair features have been removed. These nodes were seldom or never used, and the interpretation was difficult and error prone. The removed pair features are **Fixed Constraint**, **Prescribed Displacement**, **Roller**, **Symmetry**, **Antisymmetry**, **Pinned**, **No Rotation, Rigid Connector**, and **Boundary Load**. In case any such feature was used in a model, it can easily be replaced by standard boundary conditions.

CHANGE IN PRESCRIBED DISPLACEMENT WHEN ELASTIC PREDEFORMATION IS USED

The interpretation of the input in the **Prescribed Displacement** node in connection with **Elastic Predeformation** has been changed. It is now the total displacement, which is given in the same way as if **Elastic Predeformation** is not present.

CHANGED DEFINITION OF CENTROID OF RIGID CONNECTOR IN SHELL INTERFACE

When the centroid of a **Rigid Connector** in the Shell interface is defined by **Centroid of selected entities**, and the entity type is points, the computation of the centroid has been changed. In version 6.0, the average of the coordinates of the points is used without modification. In earlier versions, a possible offset from the reference surface was included, so that the actual midsurface of the shell was used. The reason for the change is that since it is now possible to select points that are not part of the shell interface, an offset cannot be defined in all points.

CHANGED API FOR INITIAL VALUES

Due to changes in the implementation of the **Initial Values** node in Solid Mechanics, there are changes in the API syntax. In version 5.6 and earlier, if the displacement field had a name that was not u, the field name was reflected in the API syntax when setting an initial value. Also, the syntax for the initial velocity has changed. Assume that the field name is u_solid:

Earlier versions:
```
...feature("init1").set("u_solid", new int[]{1, 0, 0});
...feature("init1").set("du_solid/dt", new int[]{2, 0, 0});
```

Version 6.0:
```
...feature("init1").set("u", new int[]{1, 0, 0});
...feature("init1").set("ut", new int[]{2, 0, 0});
```

If you are reading a file containing the old syntax, there is an automatic conversion for some common cases. The cases that are handled include when the field name is u* or u_solid*, where * is a number less than or equal to 10.

For all other cases when a nondefault field name is used, you must manually replace the argument in the call to set() by the new names, u or ut, to set the initial displacements or velocities.

HANGING CABLE

A cable is a structural member that has stiffness only in its tangential direction but virtually no bending stiffness. When supported only at its two ends, it deflects under gravitational load forming a curve known as a catenary. This example shows how to model components such as cables, wires, or strings using the Truss interface.

INFLATION OF A SQUARE AIRBAG

A square airbag is inflated using a constant air pressure. In some regions of the airbag the material wrinkles during inflation, as a negative principal stress develops. The wrinkling membrane model within the tension field theory gives a correct description of the stress distribution and wrinkling pattern in inflated membranes.

TORSION OF A CIRCULAR MEMBRANE

In this example, a torque is applied to the inner edge of a circular annulus-shaped membrane while the outer edge is fixed, resulting in membrane wrinkling. The wrinkling membrane model avoids the equilibrium instability that would be produced by the compressive stresses.

UNIAXIAL STRETCHING OF A RECTANGULAR MEMBRANE

This example demonstrates the wrinkling phenomenon in a thin sheet stretched uniaxially. The modified membrane theory, which incorporates the wrinkling model, ensures noncompressive principal stresses in the wrinkled region. The numerical results from the simulation are compared to analytical results.

AXISYMMETRIC TWIST AND BENDING

Including circumferential displacements in a 2D axisymmetric Solid Mechanics interface makes it possible to compute twist and bending deformations. This model determines stress concentration factors for a hollow shaft for load cases of axial extension, torsion, and bending, using a computationally lean 2D axisymmetric formulation. To demonstrate the equivalence, the model setup and results are compared with a full 3D analysis.

DEFORMATION OF AN IRON PLATE BY MAGNETIC FORCE

A strong permanent magnet is placed close to a clamped thin plate made of iron. The magnetic force causes the plate to be deflected. This example studies the plate elastic deformation and stress. The deformation of the plate has an influence on the

distribution of the magnetic field. This effect is accounted for using a moving mesh in the air surrounding the plate. The model is set up using the Magnetomechanics, No Currents multiphysics interface.

STRESS IN COOLING PIPELINE NETWORK

This example demonstrates how to model coupled flow, heat transfer, and structural deformation and stress in a pipeline network. Gravity loads from the pipe and fluid are also taken into account.

MODELING OF PRETENSIONED BOLTS

This is a tutorial example, showing how to work with pretensioned bolts. Various aspects of bolt modeling are explored:

- Modeling bolts by solids or by beams
- Connections of beams to solid components
- Bolts ended by nuts or by internal threads in the attached component
- Import of bolt and nut geometries from the Part Libraries
- Connection of bolts using full contact, or by continuity
- Sequential tightening of a set of bolts

COMPONENT MODE SYNTHESIS TUTORIAL

In this tutorial example, the concept of component mode synthesis (CMS) is introduced through a simple solid model of a beam. Parts of the beam are reduced into CMS components. Dynamic as well as stationary analyses are performed. Moreover, you will learn how to apply loads directly on a CMS component and work with such components during result evaluation. The results from the full and reduced versions of the model are compared.

Subsurface Flow Module

New Functionality in Version 6.0

IMPROVED HANDLING OF POROUS MATERIALS

Porous materials are now defined using phase-specific properties tabulated in the **Porous Material** node. In addition, subnodes can be added for the solid and fluid features. Several subnodes can be defined for each phase. This functionality makes it possible to use one and the same porous material for fluid flow, chemical species transport, and heat transfer without having to duplicate material properties and settings.

IMPROVEMENTS TO BRINKMAN EQUATIONS INTERFACE

In the Brinkman Equations interface, the new **Porous Medium** feature now supports the **Porous Material** functionality. When opening a model created in version 5.6 or earlier, the old **Fluid and Matrix Properties** feature is not migrated, but a warning message is displayed. It should be manually replaced by the new **Porous Medium** feature (which adds **Porous Matrix** and **Fluid** subfeatures) to allow an updated definition of the material properties. Support for obsolete features is not ensured in future versions; thus it is strongly recommended to transfer the old feature to the new feature.

POROUS SLIP IN THE BRINKMAN EQUATIONS INTERFACE

The boundary layer between free and porous media flow may be very thin and impractical to resolve. Using the new porous-slip wall treatment, you can account for porous walls without resolving the full flow profile in the boundary layer. Instead, a stress condition is applied at the porous surfaces by utilizing an asymptotic solution, yielding a good accuracy in the bulk flow. The functionality is activated in the new **Porous treatment of no slip condition** settings, and it is then used as the default wall condition. This new feature can be used in most problems where the model domain is large.

NONISOTHERMAL FLOW IN POROUS MEDIA

The new **Nonisothermal Flow, Brinkman Equations** multiphysics interface automatically adds the coupling between heat transfer and fluid flow in porous media. It combines the **Heat Transfer in Porous Media** and **Brinkman Equations** interfaces and automatically adds a **Nonisothermal Flow** coupling node.

You can find this interface under the **Nonisothermal Flow** branch in the **Fluid Flow** folder in the **Add Physics** tree.

EXTENDED FEATURES FOR HEAT TRANSFER IN POROUS MEDIA

The default **Porous Medium** feature in the Heat Transfer in Porous Media interface by default selects the **Local thermal equilibrium** option under the **Porous medium type** list. This corresponds to the previously available averaging techniques within the **Porous Medium** feature, which solved a single equation for the average temperature field with effective properties. New averaging techniques are available for computing the **Effective thermal conductivity** in porous media: **Solid spherical inclusions**, **Fluid spherical inclusions**, **Wrapped screen**, and **Sintered metal fibers**. In addition, the **Volume average** and **Reciprocal average** techniques have been renamed to **Plane layers parallel to heat flow** and **Plane layers perpendicular to heat flow** for clarity. Finally, the new **Equivalent thermal conductivity** option makes it possible to enter user-defined expressions.

For these new options, the postprocessing variables (thermal properties, heat fluxes, heat sources, and heat and energy balance) are available in a unified way for homogenized quantities.

SOURCE TERMS FOR THE SHALLOW WATER EQUATIONS INTERFACE

The shallow water equations give a 1D or 2D approximation of shallow flows by averaging the water column along the depth. Rain, local upwelling, pumping devices, or boundary stresses enter as source terms in the model equations. Introducing source terms was previously possible through the equation view, but it is now possible to add momentum and mass sources as predefined quantities directly in the interface.

MISCELLANEOUS

- The **Longitudinal dispersivity** and **Transverse dispersivity** settings in the **Thermal Dispersion** feature are now available with all COMSOL products that include the feature.

- A new **Discharge** boundary condition in the Darcy's Law and Richards' Equation interfaces makes it possible to specify the discharge in outlets in term of cubic meters of water per second (in general, volume of liquid per unit of time).

GLACIER FLOW: A 2D STUDY OF COLD AND TEMPERATE GLACIERS

This example shows how to set up a glacier flow model containing several important aspects of glacier modeling: the creation of the 2D geometry, modeling of non-Newtonian flow, and implementation of basal sliding. The example uses two different glacier types: a cold glacier and a temperate glacier.

Wave Optics Module

New Functionality in Version 6.0

ELECTROMAGNETIC WAVES, BOUNDARY ELEMENTS

This new physics interface is useful for modeling the scattering properties of objects, evaluating electric fields far from the scatterer.

PART LIBRARY FOR SLAB AND RECTANGULAR WAVEGUIDE ELEMENTS

The new Wave Optics Module Part Library contains slab waveguide shapes for 2D geometries and rectangular waveguide shapes for 3D geometries, including the following waveguide elements:

- Straight waveguides
- Tapered waveguides
- Bent (ring) waveguides
- S-bend waveguides
- Couplers

The parts are built from fully parameterized sequences of geometry instructions.

LAYERED TRANSITION BOUNDARY CONDITION

Multiple thin layers, such as an antireflection coating of an optical lens, can, for close to normal incidence, be described by the Layered Transition Boundary Condition feature. It requires combining this physics feature, Layered Material in the global Materials, and Layered Material Link in the Materials node.

SHIFTED LAPLACE CONTRIBUTION ON MULTIGRID LEVELS

If no geometrical feature sizes are smaller than half a wavelength, when the operating frequency is high, modeling with a higher-order element such as cubic element discretization is beneficial for faster computations. The computation efficiency can be further boosted by imposing the Shifted Laplace contribution on multigrid levels.

SUGGESTED ITERATIVE SOLVER FOR PERIODIC STRUCTURES

Typical periodic problems are solved with a direct solver. However, the direct solver consumes a lot of memory, when the periodic unit cell size is not subwavelength. If

this is the case, switch to the Suggested Iterative Solver to finish the computation faster with less memory usage.

CONSTRAINT-FREE PORT OPTION

The constraint-free port option is available to compute the expansion coefficients as an overlap integral, while in the default port formulation, the expansion coefficients (or S-parameters) are calculated by adding a scalar dependent variable for each coefficient and then adding a constraint to enforce the series expansion.

SMOOTHED HEAT SOURCE CALCULATION

Use an averaged loss calculation option in the bidirectional formulation for the Electromagnetic Waves, Beam Envelopes interface to remove cross terms between the two waves that are not resolved by the mesh. If this spatially, fast-varying heat source distribution anyhow is "washed out" by the heat transfer, it can be advantageous to not include the cross terms when calculating the electromagnetic loss (and heat source).

SYMMETRY AXIS REFERENCE POINT

The **Symmetry Axis Reference Point** subnode is available in 2D axisymmetry, when there is an incident field defined in the parent node. Then this subnode is available as a default subnode to the **Scattering Boundary Condition** or **Matched Boundary Condition** parent node. The **Symmetry Axis Reference Point** subnode defines a reference position at the intersection point between the parent node's boundary selection and the symmetry axis.

DEFAULT PLOTS FOR NUMERIC PORT MODE FIELDS

Plots are automatically created of the port mode fields for numeric ports.

New and Updated Models in Version 6.0

OPTIMIZATION OF A PHOTONIC CRYSTAL FOR DEMULTIPLEXING

This model demonstrates how to apply shape optimization to a photonic crystal. The objective function is to maximize the output power ratio between two narrow frequency bands, while constraining the loss from below. This effect is achieved by letting GaAs pillars change position but not shape. The implementation makes use of the Free Shape Domain and Transformation features, so that gradient-based optimization can be applied.

WHISPERING GALLERY MODE RESONATOR

Dielectric microspheres can support whispering gallery modes with high optical quality factors. This model illustrates how to compute the different eigenmodes and resonance frequencies. The resonance frequencies are either filtered by their spatial localization in the resonator or by comparing the losses of the bound modes and the, for this application undesired, air modes.

HEXAGONAL PLASMONIC COLOR FILTER

Robust and easy-to-fabricate color filters have many applications. This model demonstrates how to build a model of an absorbing color filter, based on a hexagonal array of holes in a thin aluminum layer. The structure is hexagonally periodic, but in the model, it is demonstrated how to define the problem both as a hexagonally and a rectangularly periodic problem.

MACH-ZEHNDER MODULATOR

The Mach–Zehnder Modulator model has been updated to use parts from the Wave Optics Module part library.

OPTICAL RING RESONATOR NOTCH FILTER

This optical ring resonator model has been updated to use parts from the Wave Optics Module part library.

Backward Compatibility with Version 5.6 and Earlier

The port mode fields are now defined as the outgoing waves, whereas in version 5.6 and earlier the port mode fields were defined as the incoming (exciting) waves. This change only affects User defined and Periodic port types, as input fields can be provided for these types. When loading MPH-files from version 5.6 or earlier, the input fields for User defined and Periodic ports are migrated to function with the new mode field definition. However, for User defined ports, the migration assumes that the material adjacent to the port is homogeneous.

Material Library

In version 6.0, the Material Library contains 10,328 materials and 84,573 material property datasets. The following improvements and additions have been made:

- Added several new precious metal alloys: CuPd, Cu-Pt, Au-Pd, Au-Ag, Ag-Cu, Au-Cu, Au-Pt, Ag-Cd, Au-Ag, Ag-Pd, Ag-Mg, Ag-Mn, Ag-Pt, Pt-Fe, Pd-Ag, Pt-Au, and Pt-Cu-Co.

- Added several new refractory metal alloys: Cr-based, Ti-Ge, Ti-Cu, Ti-Nb, Ti-Sn, Ti-Zn, Zr-0.14Hf, Zr-1Hf, and Zr-1Hf-1Ta.

- Added thermal expansion data for TRIP 700, TRIP 1000, ALCHROME alloy DK, Stellite 23, Stellite 30, W – 20 Cu, W – 40 Cu, W – 7 Ni – 2.5 Cu, Al-7Si, Al-13Si, Al-17Si, AZ81, EK30, HK31, HZ32, CuAu3, Cu3Au, 44 Pd – 33 Ni – 23 Cr (GE-76 brazing alloy), and Alloy 19-2.

- Added new materials: TD-NiCr, Fe-Si alloys, Nimonic 100, 30 Co - 23 Fe - 21 Cr - 20 Ni,Mg – 7 Al – 1 Si, Sn-Mg binary alloys, and Zn-Cu binary alloys.

- Added several foods: honey, orange juice, grape juice, apple puree, pear puree, and peach puree. Foods were also put into their own group.

- Added data for several oxides, silicates, and electroceramics.

- Added data for Controlled expansion 45, Temp compensator 30 (types 2, 3, and 4), 1065, 1095, Kovar, 7068, several Cu alloys, AZ81, HM31, ZK60, and Au2Mn.

- The heat capacity data for diamond was incorrect and has been updated. The specific heat data is not affected by this error.

- Refit the thermal expansion data for Alloy 39; there was a significant change in the values.

- The thermal expansion data for: Hastalloy B, Hastalloy C, several Monels, Inconel X-750 Inconel HX, Alloy 72 and Alloy 22-3 has been changed, the new values differ by 5–10%.

- The data for Ta2O3 is actually for Ta2O5. The name of the material has been updated.

- The expansion data for HfTiO4 and ZrTiO4 was changed to a new reference with the expansion in different crystallographic directions.

- Added new materials: TaN, Schott Borofloat 33, spider silk, wheat, Inconel 718SPF, UNS N06333, and several cast irons.
- Added specific heat data for many salts, borides, carbides, nitrides, and other nonmetallic materials.
- Added data for Inconel X-750, AerMet 100, AM-355, FMDP pipeline steel, API-X70 pipeline steel, Incoloy A-286, 304, A9, 17-7 PH, 10B46, several Ti-based alloys, Pb – 55 Bi (eutectic), Zr50Cu40Ag10, rare-earth trifluorides, Dy2WO6, Gd2WO6, Er6WO12, Ho6WO12, Y2W3O12, and Yb6WO12.
- Data for N08330 was removed and combined with Incoloy 330.
- Added apparent viscosity for several polymers.
- The expansion of aluminum was extended down to 10K with a new reference. The CTE value near 20K changed substantially, the others (dL/L and MTE) were only slightly affected.
- Added HgCr2Se4.
- Added data for the following oxides: Dy2TiO5, Er2TiO5, Gd2TiO5, Ho2TiO5, Lu2TiO5, Y2TiO5, La2TiO5, and Nd2TiO5.
- Added data for Lexan 101 and refit the thermal expansion; dL/L increased by 5%.
- Two new magnetic properties were added: magnetic core loss versus flux density and magnetic exciting power versus flux density.
- Several electrical steels were added.
- Several iron-base alloys, nickel-base alloys, titanium-base alloys, magnesium-base alloys, and aluminum-base alloys were added.
- The compressive stress-strain curves for Ta, Ta-2.5W, Ta-5W, and Ta-10W were incorrect and have been fixed.
- Data was added for Zr-based alloys: Zr-1Nb and Zr-2.5Nb; for Mg-based alloys: ZK10, ZN10, and ZX10; and for Al-based alloys: 1085, 6016, 6110, and 7075.
- Data was added for: NILOMAG alloy 77, Ni-SPAN C alloy 902, Inconel 601, Inconel 625LCF, Incoloy 25-6HN, 316L, Inconel X-750, Super 304H (304HCu), Fe-2Mn-1.3Si-0.7Cr-0.4C, X100 pipeline steel, NiCrMo 2.5-IG filler wire, Ductile Ni-Resist D-5S, SiMo51 ductile iron (3.2 C – 4.5 Si – 1 Mo), and SiMo1000 ductile iron.
- Data for Haynes 25 and L-605 were combined as Haynes 25.
- Data has been added for several Cu-based alloys and several Nitronic alloys.

- The thermal conductivity for Kapton HN was incorrect; it was actually for the MT grade. This material has been updated with the correct data.

- Data was added for: UNS S31266, LSHR disk alloy, Rene-41, Haynes 25 (L-605), S350, S420, and G500.

LiveLink™ for Excel®

New Functionality in Version 6.0

IMPROVED VBA SUPPORT

Support for Visual Basic for Applications (VBA) has been improved. For all buttons on the COMSOL ribbon tab, a VBA command has been introduced. This functionality makes it possible to use toolbar button commands functionality directly from VBA code, which makes it easier to automate steps that were easy to perform using the ribbon but perhaps difficult to perform using the COMSOL API.

ADDITIONAL IMPROVEMENTS

- The LiveLink™ interface now supports Excel® 2021.

- When saving Excel data from apps, it is now possible to include expressions in the exported data.

- The new **Break All Links** button on the COMSOL ribbon tab removes all the comments with links to the COMSOL models from the current worksheet. Doing so breaks the links between all cell ranges in the worksheet and the model.

- Models can be loaded from a database that is stored using Model Manager. Models cannot be saved in a database using Excel.

- The stability of connections has been improved as well as improved handling when Excel is disconnected from a COMSOL Multiphysics Server.

LiveLink™ for MATLAB®

New Functionality in Version 6.0

SYSTEM REQUIREMENTS

LiveLink™ for MATLAB® is supported using MATLAB 2021a or 2021b. Older versions may work, but a Java VM version 1.8 or newer is required. It is available in MATLAB 2017b or newer.

UPDATED USER INTERFACES

All user interfaces have been remade using the App Designer in MATLAB (`mphnavigator`, `mphopen`, `mphsearch`, and `mphmodellibrary`). This update provides more modern and responsive user interfaces that now permit resizing. The new user interfaces are available for MATLAB versions from 2020a and newer. The old user interfaces are still available, but they will not be updated and they will be removed in a future version of COMSOL Multiphysics. Note that some differences may be observed depending on the exact version of MATLAB used.

AUTOCOMPLETION

Autocompletion for wrapper functions is supported in the Command Window, the regular Editor, and the Live Editor for Live Scripts. This functionality makes it much easier to use the wrapper functions.

NEW GEOMETRY AND MESH FUNCTIONS

COMSOL 6.0 permits more advanced uses of geometries and multiple meshes in a model. In order to make it easier to follow such models from MATLAB, the new `mphgeominfo` and `mphcomponentinfo` functions have been introduced. The function `mphmeshstats` has been improved as well.

MODEL MANAGER SUPPORT

Models can be loaded from a database that is stored with Model Manager using `mphload` and `mphopen`. Models cannot be saved in a database using MATLAB. Instead, use the command `mphlaunch` to start the COMSOL Desktop and access the Model Manager there.

- The `mphmin` and `mphmax` functions are now able to return the position of the minimum or maximum value when using the position property.

- The `mphinterp` and `mpheval` functions have been updates with improved error messages in case the expression used is outside the geometry or the domain where the expression is used.

- The `mphreport` function now supports generating reports as Microsoft® PowerPoint® presentations in addition to creating reports in HTML or Microsoft® Word® formats.

Backward Compatibility with Version 5.6

COLOR TABLES

All functions that have to do with color tables (for example, `cividis`, `cyclic`, and `disco`) except for `colortable` are deprecated. When using the `colortable` function, you can use autocompletion to get a list of available color tables. As an alternative to using, for example, `cividis` you can use

```
table = colortable('cividis');
```

instead.

REPLACED AND DEPRECATED FUNCTIONS

The functions `mphint`, `mphgetp`, and `mphgetmodelnode` are deprecated. `mphgetmodelnode` has been replaced by `mphcomponentinfo` where new features have been added. Th deprecated functions still exist but will result in a warning when used and may be removed in a future release of COMSOL Multiphysics. Such warnings can be disabled using this command

```
warning('off','COMSOL:DEPRECATED')
```

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.

LiveLink™ for Simulink®

New Functionality in Version 6.0

A significant performance increase has been obtained for cosimulation when simulating time-dependent COMSOL models together with Simulink.®

Initialization and stability of simulations have also been improved on all platforms.

The COMSOL API for Use with Java®

COMSOL 6.0 API Changes

PAIR FEATURES

The removal of the fallback features under pair features can break API code that you have written that access those features. The presence of a default pair feature may also create a different configuration compared to what previous API runs did.

Java or MATLAB program that accessed fallback features under a pair feature will not work anymore. This is a necessary limitation because any attempt to support such API backward compatibility will be both unintuitive and unsafe. For most physics created from the API, the default feature from the physics will act as fallback to the pair features. As long as the default fallback feature in 5.6 was of the same type, the behavior in 6.0 will be equivalent. If you used a special fallback feature for a pair feature in 5.6, you now place it either before or after the pair feature and use the same selection as the pair feature. Placing the fallback feature after makes it possible to use the pair feature's selection directly. The example below show such code for 5.6 and how the equivalent code in 6.0 can be:

```
// Creating a pair feature in both 5.6 and 6.0
model.component("comp1").physics("es").create("cont1",
"Continuity", 2);
// Assign some pairs to the pair feature
model.component("comp1").physics("es").feature("cont1").
set("pairs", new String[]{"ap1", "ap2"});

// Creating a fallback feature under a pair in 5.6 (this will not
work in 6.0)
model.component("comp1").physics("es").feature("cont1").
create("sfcd1", "SurfaceChargeDensity", 2);
// Accessing a settings in 5.6 (this will not work in 6.0)
model.component("comp1").physics("es").feature("cont1").
feature("sfcd1").set("rhoqs", "1e-9");

// Creating an equivalent feature in 6.0 that act as fallback to
the pair feature
model.component("comp1").physics("es").create("sfcd1",
"SurfaceChargeDensity", 2);
// Use the same selection as the pair feature
int[] ent = model.component("comp1").physics("es").
feature("cont1").selection().entities();
```

```
model.component("comp1").physics("es").feature("sfcd1").
selection().set(ent);
// Accessing a settings in 6.0
model.component("comp1").physics("es").feature("sfcd1").
set("rhoqs", "1e-9");
```

When a physics interface is created from the API in 6.0, it will also add default pair features that did not exist in 5.6. As long as the final model use all created pairs the behavior will be the same in 6.0; otherwise, it may be necessary to disconnect the default pair feature with an extra command. Below is an example for the Electric Currents interface, but the actual set operation is identical for all interfaces:

```
model.component("comp1").physics("ec").feature("dcont1").
set("pairDisconnect", true);
```

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.

This concludes the release notes for COMSOL Multiphysics version 6.0.

Index