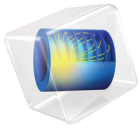

COMSOL Multiphysics

Release Notes

6.3

COMSOL

COMSOL Multiphysics Release Notes

Version: COMSOL 6.3

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 on the COMSOL Access page at www.comsol.com/support/case. Useful links:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/product-update
- 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
- Learning Center: https://www.comsol.com/support/learning-center

Part number: CM010001

Contents

Chapter 1: Release Notes

1

Release Notes

COMSOL Multiphysics® version 6.3 includes a new product: the Electric Discharge Module. It also provides new functionality and improvements across the COMSOL products. These *Release Notes* provide information regarding new functionality in version 6.3 for the entire COMSOL product suite.

COMSOL Multiphysics

New Product

ELECTRIC DISCHARGE MODULE

The Electric Discharge Module provides comprehensive simulation capabilities spanning a wide spectrum of electric discharge scenarios, encompassing atmospheric pressure gas discharges, breakdown phenomena in liquids such as transformer oil, and solid materials like insulating polymers. It provides specialized interfaces for simulating breakdown thresholds, surface charge dynamics, and discharge processes in various environments, from atmospheric gases to microscale events in MEMS devices.

The Electric Discharge Module enables you to predict the impact of electric discharges on system performance, ensuring that designs are optimized for reliability and compliance with regulatory standards, particularly in EMC/EMI design for electronic devices. Engineers and scientists can simulate and analyze lightning-induced electromagnetic pulses, electrostatic discharges, and other discharge events, making it a valuable tool for advancing product development and reducing the costs associated with experimental testing and prototyping.

The module's built-in transport models and customizable electric discharge chemistries provide the flexibility needed to conduct accurate and innovative analyses. You can easily implement these customizations within the COMSOL Desktop® without having extensive scripting or programming knowledge. The Electric Discharge Module can be used with products throughout the COMSOL product suite, including those for electromagnetics, structural mechanics, and fluid dynamics, to explore the multiphysics effects that often accompany electric discharges.

New General Functionality

LMADMIN LICENSE SERVER MANAGER

The `lmadmin` license server manager for version 11.19.5 is now included in the COMSOL installation. It is available in `licenses/<platform>/lmadmin`. Note that it is not supported on ARM64 Linux. Further, note that `lmgrd` is the preferred license server manager for COMSOL Multiphysics.

Also see the *Model Manager Reference Manual* for more information.

DATABASE INTEGRATION WITH BATCH MODE

The command for solving COMSOL models in batch mode from a command-line interface now supports using model versions stored in a Model Manager database — both as input and output. This, together with the Model Manager API for use with Java®, enables you to set up powerful simulation pipelines in which input and output can be managed via a database. You can also combine a version of a model in a database as the input with an MPH-file on the file system as the output, and vice versa.

COPY AND PASTE OF MODEL CONTENTS

Support has been added for copying model nodes in the **Contents** tree of a model version and pasting those nodes into the model tree in the COMSOL Desktop. This enables you to quickly reuse modeling and results evaluation setups from your existing models when building new models. Not all nodes in the **Contents** tree can be copied and not all nodes in the COMSOL Desktop can be the target of a paste. The model tree being copied from and the model tree being pasted to can, however, belong to different COMSOL Multiphysics program sessions.

SEARCH AUTOCOMPLETION

Autocompletion assistance has been added for the Model Manager search syntax. You can easily find and complete the names of search filters, the available values for a specific search filter, and the various symbols used to express and combine such filters. The new autocompletion functionality is available in all windows in the COMSOL Desktop where version-controlled models and data files can be searched.

IMPROVED INTEGRATION BETWEEN WORKSPACE WINDOWS

The overall integration between the various windows of the Model Manager workspace has been improved. Selected results in the **Commits**, **Versions**, and **References** windows will now be automatically displayed in the **Settings** window, thereby making it simpler to browse the database. You will also find it easier to perform various database tasks, as buttons in the workspace toolbars now target these windows when the windows are in focus.

REDUCED MEMORY USAGE WHEN SAVING LARGE MODELS

Earlier versions of COMSOL Multiphysics could sometimes run out of available heap space memory when trying to save large models to a Model Manager server database. This issue has now been addressed.

MODEL MANAGER API IMPROVEMENTS

The COMSOL Model Manager API for use with the Java® programming language now supports Java 11 in the Application Builder's Method Editor and in the new **Java Shell** window.

The following new methods and types have been added to the API:

Preset Item and Version Keys when Saving

The identifying keys of items and their versions can now be preset when saving to the database. Knowing these keys beforehand enables you, for example, to more easily reference your saved items and versions in other parts of your code.

You can generate unique keys via the following new methods:

```
ItemKey.generate(<item-type>);
ItemVersionKey.generate(<item-type>);
```

and then apply the generated keys as input parameters when saving. For example,

```
SaveModelItemParam param = DatabaseApiUtil.param()
  .forSaveModel()
  .withItemKey(<generated-item-key>)
  .withItemVersionKey(<generated-item-version-key>)
  // Set additional input parameters...

DatabaseApiUtil.api()
  .databaseByAlias(<database-alias>)
  .defaultRepository()
  .defaultBranch()
  .saveModel(param, "Saved a new model.");
```

Write Options when Exporting Versions

You can now specify what should happen when an existing file is found on the file system when exporting a model or file version from a database. To replace an existing MPH file, for example, write

```
ExportModelItemVersionParam param = DatabaseApiUtil.param()
  .forExportModelVersion()
  .withTargetWriteOptions(
    ExportModelItemVersionTargetWriteOption.REPLACE_EXISTING)
  // Set additional input parameters...
```

```
DatabaseApiUtil.api()
  .modelVersionByLocationUri(<model-version-location>)
  .export(param);
```

GENERAL IMPROVEMENTS

- A **Refresh Connection** button has been added to the **Settings** windows of local and server databases. Clicking the button refreshes the connection to the database using the window's current configuration settings.
- The **Settings** window for a server database now displays the version number of the Model Manager server.
- The **Import to Database** button in the **Auxiliary Data** window now supports output files. Previously, only input files were supported.
- When attempting to open the most recently saved version available in the **Versions** window, Model Manager will perform a preliminary check if there is an even later version found in the database — possibly saved by another user. If one is found, you will be given the option to open that version instead.

COMPATIBILITY BETWEEN VERSIONS

Local Databases

Local databases can be shared between COMSOL Multiphysics versions 6.0–6.3, as the database format is designed for both backward and forward compatibility. Models saved to a database from a specific COMSOL Multiphysics version can, however, only be opened in that and later versions.

Older Versions of Model Manager Server

COMSOL Multiphysics version 6.3 supports connecting to all previous versions of a Model Manager server, although some new Model Manager functionality is disabled in the COMSOL Desktop when connected to an older server version.

New Functionality in the Model Manager Server

Also see the *Model Manager Server Manual* for more information.

IMPROVED CONTROL OVER DATABASE SETUP

In previous versions of the Model Manager server, the server automatically created a new database using managed server components with default configuration settings. Administrators may now skip this automatic creation, instead manually performing the database setup from the server's web interface on first login. This is especially useful

when the desired setup involves placing data directories and backup directories in locations with more available disk space than those of the default locations.

STREAMLINED CUSTOMIZATION OF THE ASSET MANAGEMENT SYSTEM

Customizing the asset management system has become a lot easier by removing the need for otherwise repetitive steps. You can create new attributes when adding or updating asset types, as well as preview the **Asset** page for a corresponding asset. It is also possible to start out from existing asset types when creating asset types via a new duplication shortcut.

NEW PAGE FOR VIEWING ASSET CHANGES

A new **Asset Comparison** page that enables you to compare two versions of the same asset has been added to the asset management system. This makes it easier, for example, to see all changes made to an asset when going from one version to the next. The comparison includes both built-in fields and any attributes added via customization.

ATTRIBUTE FILTER IMPROVEMENTS

Finding assets by applying filters to attribute values from the **Filters** menu has been improved with the following updates:

- It is now possible to apply filters on table-valued attributes by combining subfilters on cell values found in table rows. Such composite attribute filters were previously only available via the Model Manager search syntax.
- New fields have been added to filter assets on their uploaded attachments, linked model and file versions, and web links.
- The list of available attribute filters in the menu is now based on the current selection of asset type filters, thereby hiding attributes already known to not be present in the search result.

ACCOUNT LANGUAGE

You can now set the language to be used in the Model Manager server's web interface. The languages to choose from are the same as those that are available for the server's default language.

GENERAL IMPROVEMENTS TO THE ASSET MANAGEMENT SYSTEM

- Search results are now automatically updated when you modify a filter in the **Filters** menu on the **Home** page.
- You can now search assets by their identifiers in the search field on the **Home** page.

- Modifying the list of related assets on the **Asset** page has been moved to a separate **Related Assets** page for better overview.
- You can now add links to any model or file version in the database on the **Asset** page, not just the latest version in a branch.
- The description of a composite attribute now shows up as a tooltip to the attribute's label on the **Asset** page.

GENERAL ADMINISTRATION IMPROVEMENTS

- Required fields are now indicated by a red asterisk symbol on edit pages. The **Save** button remains disabled as long all required fields are empty.
- Additional columns have been added to the result tables on various list pages for better data overview.
- A new **Logs** page has been added to the **System** administration area. The page enables administrators to download log files generated by the Model Manager server as a compressed zip archive.
- A **Test Connection** button has been added to the pages for adding and editing database configurations. Previously, such a button was only available on the details page.
- New database configuration settings for Microsoft SQL Server® regarding encryption and server certificates have been added.
- The parent directories to data directories no longer need to exist on the file system beforehand when creating new managed server components set to start automatically.
- Upon pressing Ctrl+C, the Model Manager server will emit a message when it has begun stopping, as long as the server has been launched from a command-line interface.

COMPATIBILITY BETWEEN VERSIONS

Older Versions of COMSOL Multiphysics

COMSOL Model Manager server version 6.3 supports client connections from COMSOL Multiphysics version 6.0 and newer.

Primitive Attribute Filter Syntax

Primitive attribute filter expressions of the form `@<identifier>:<value>` now only applies to attributes appearing as top-level fields on the **Asset** page. To filter primitive attributes inside composite attributes, write `@<composite-identifier>{@<primitive-identifier>:<value>}`.

Terminology Change

The widget types for checkboxes and checkbox lists have been renamed to **Checkbox** and **Checkbox list**.

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 AND CHANGES

Running Compiled Apps Across Minor Versions

Users of compiled apps can now run on an older runtime version than it was compiled for, as long as the major versions are the same. The runtime installer now checks for updates and can update the version directly after installation.

IMPROVEMENTS TO THE FORM EDITOR

New Form Toolbar Object

The **Form Toolbar** is a new form object, which you can add to each form. It is a toolbar that shows up in a manner that depends on how the form is being used, such as the main toolbar of a window, as a section header, or as a view header. The view header option is useful in settings forms, where it prevents the toolbar from scrolling out of view.

IMPROVEMENTS AND CHANGES TO THE METHOD EDITOR AND APPLICATION LANGUAGE

Java Support

Java 11 is now supported in the Method Editor.

Java Shell and Chatbot

You can use the new **Java Shell** and **Chatbot** windows to test, debug, and improve methods in the Application Builder (see below for more information).

Data Viewer

The **Variables** window in previous versions of the Application Builder has been renamed the **Data Viewer** window. The **Data Viewer** window is now also available in the Model Builder and Model Manager workspaces and always show the relevant content in the model. The **Data Viewer** window shows global model parameters, primitive

declarations, and Java Shell variables. When stopped at a breakpoint, it also shows local variables in methods.

Delimiter Improvements

There is a new `readCSVFile(file, delimiters)` syntax that enables users to decide which delimiter to use. It reads a string matrix from a CSV file with the specified delimiter. If there is more than one delimiter provided, this method will try selecting the most probable delimiter from the supplied candidates.

Similar syntax is now also available as `readMatrixFromFile(file, delimiter)`, which reads a double matrix from the given file with the specified delimiter, and as `readStringMatrixFromFile(file, delimiter)`, which reads a string matrix stored in a file with the specified delimiter.

Saving Application Copy

The new `saveApplicationCopyAs` method saves a copy of the application. When running this method, users are presented with a file browser dialog where they can select where to save the copy. The method returns a boolean value where a true value indicates that the save was successful.

```
boolean saveApplicationCopyAs()
```

This command is also available in command sequences under **GUI Commands** > **File Commands**.

New and Improved General Functionality in COMSOL Multiphysics

CHATBOT WINDOW

The **Chatbot** window functionality connects to an external AI system. A subscription agreement with the AI system host is needed to obtain an API key. COMSOL is not a party in this agreement, and use of the external AI system may be subject to additional fees.

It is your responsibility to ensure that your use of the AI system is in accordance with laws and policies that may apply. It is also your responsibility to review the suitability of any suggestions made by the AI system.

In the Windows® version of the COMSOL Desktop, a new **Chatbot** window is available for chatting with the OpenAI GPT-4 model, which can assist with generating and correcting COMSOL API for Java® code from within COMSOL Multiphysics®

and answering modeling questions related to the COMSOL Multiphysics® software. Doing so can help out with programming using the COMSOL API for Java® as well as with general COMSOL modeling using the COMSOL Desktop. The **Chatbot** window appears both in the Model Builder workspace and in the Application Builder workspace. The chatbot functionality requires configuration of an OpenAI API key. It can be turned off in the installer or via a preference setting, if desired. Note that by default, the CHATBOT license feature is not included in the installation; to include it, you need to select to install Chatbot components in the installer.

JAVA SHELL WINDOW

In the Windows® version of the COMSOL Desktop, a new **Java Shell** window is available for interacting with the current model in the Model Builder using the COMSOL API for Java® and in the Application Builder when working with methods. You get completion suggestions when typing commands, and suggested COMSOL API code from the **Chatbot** window can be sent to the **Java Shell** window to run there.

DATA VIEWER WINDOW

The **Data Viewer** window (see also the Application Builder section above) is available in the Model Builder and provides a convenient way to inspect and change values for global parameters, for example. You can also open it from the **Java Shell** window to inspect session variables.

DISPLAYING COMMENTS IN SETTINGS WINDOWS

Comments made in the **Properties** window are now displayed in the corresponding **Settings** window. If you do not want to display comments there, clear the **Comments** > **Display Comments in Settings** checkbox in the **Show More Options** dialog.

UPDATED RIBBON TOOLBARS AND DESKTOP USER INTERFACE

The ribbon component has been updated, as well as the general look and feel of the COMSOL Desktop on Windows®. The new ribbon component supports keytips on all levels. To activate keytips, press the Alt key. The keytips appear next to the buttons. Keytips appear for all buttons, menus, and items (including the Quick Access Toolbar and the **File** menu).

AUXILIARY SLIT FEATURE

For all physics interfaces, a new **Auxiliary Slit** feature is available when you have enabled **Equation-Based Contributions** in the **Show More Options** dialog. You can then add **Auxiliary Slit** features to interior boundaries to create slits that decouple the fields on either sides of the boundaries.

LISTS WITH SEARCH FILTER AND TREE STRUCTURES

Lists in the COMSOL Desktop are now available with a more advanced design that includes a search filter and trees.

Such lists are available in the following cases:

- Named selections in the **Selection List** window
- Color tables in results
- Icon selectors in the Application Builder
- Lists of times in results

LIGHTER COLOR THEMES AND OTHER GRAPHICS COLOR THEME IMPROVEMENTS

The **Default**, **Light**, and **Dark** graphics color themes have been updated with lighter colors. There is also a new **RYBDark** theme that makes it possible to use the RYB color theme together with dark background colors. In addition, the **Classic** and **ClassicDark** themes have been added to enable access the colors in previous versions.

It is now possible to change graphics color theme from the **Graphics** window toolbar's **Colors** menu, and you can now see the theme colors directly in the **Color** list also in the cross-platform version.

RESET LABELS TO DEFAULT

You can now choose **Reset Labels to Default** from the **Model Tree Node Text** menu in the **Model Builder** window. It will reset all Model Builder node labels to their default values in the current language.

FILTER OUT INAPPLICABLE SELECTIONS

All **Settings** windows with an **Override** or **Override and Contribution** section (which includes physics features, multiphysics features, and materials, for example) now includes an **Only Show Applicable** button in the selection section (**Boundary Selection**, for example). When you click this button, all geometric entities that are either overridden or not applicable no longer appears in the selection section's list of geometric entities.

NEW GEOMETRIC VARIABLES

- The arc length parameters `edgparal` and `edgparnal` are available on edges in 2D and 3D. The parameter `edgparnal` is the normalized arc length, that is `edgparal`

divided by the edge length, and goes from 0 to 1. The new variables can be useful for defining distributed loads and constraints such as a parabolic velocity profile.

- A new `concompid` variable evaluates to an integer value that is unique for each connected component of geometric entities, domains, boundaries, edges, and points.

SELECTIONS

Logical Expression supports using Boolean expressions to select all entities belonging to certain components or certain packages from an ECAD import.

ADD-IN LIBRARY UPDATE

Support for adding new add-ins to the COMSOL Add-in Libraries and updating existing ones between product updates is now available, complementing the Application Library Update and Part Library Update services.

FILTERING OF AND FLEXIBLE NUMBER OF RECENT FILES

In the **Open** full-screen window, there is now a text field at the top of the list of recent files where you can type a text to filter the list to find some specific model files.

There is also a new **Maximum number of recent files** setting on the **Files** > **Recent** page in the **Preferences** window. The default value is 150 files.

RESETTING LABELS TO THE DEFAULT LABEL TEXT

On the **Model Tree Node Text** menu in the Model Builder toolbar, you can now choose **Reset Labels to Default** to reset all node labels to their default label text.

MAXIMIZE AND RESTORE BUTTONS

Maximize and **Restore** buttons have been added to the Windows version of the COMSOL Desktop. They provide the same behavior as double-clicking the window header.

IMPORT TO DATABASE IN THE AUXILIARY DATA WINDOW

You can now also use **Import to Database** for output files displayed in the **Auxiliary Data** window.

KEYBOARD SHORTCUT FOR REVERT TO SAVED

You can now use Ctrl+Shift+R (Windows and Linux) or Command+Shift+R (macOS) as a keyboard shortcut for the **Revert to Saved** command on the **File** menu.

GEOMETRY FUNCTIONALITY

- Use the new **Geometry Cleanup** feature to prepare your geometry for meshing. It automatically detects small details and gaps in the geometry. You can either select to run an automatic cleanup, open the **Cleanup Wizard**, or cancel a cleanup.
- There is a new virtual operation, **Merge Faces**, that can be used to merge faces to collapse the gap or region between them.
- **Plane Geometry** and **Geometry Part** sequences now support adding **Form Union** and **Form Assembly** nodes as well as virtual operations and, in 3D, **Remove Details**. **Form Union/Assembly** nodes are added automatically if you select to add a virtual operation or a **Remove Details** node.
- The **Extrude** and **Revolve** operations now support edges and vertices as input. It is also possible to extrude and revolve selections on any 2D objects into 3D.
- You can now create selections for layers in primitives. This simplifies setting up physics and material settings. This concerns the **Block**, **Cone**, **Cylinder**, **Sphere**, **Ellipsoid**, **Circle**, **Ellipse**, **Rectangle**, and **Square** primitives.
- The **Logical Expression Selection** feature supports using Boolean expressions to select all entities belonging to certain components or certain packages from an ECAD import.
- It is now possible to visualize several work planes in the **Graphics** window. For example, it is now possible to see both the **Coordinate System for Objects** and **Coordinate System to Match** work planes in the **Rigid Transform** operation.
- The **Settings** window of the **Import** node has been redesigned with new sections for increased usability.
- When importing a DXF file in 2D, the options **Knit curves** and **Do not knit** now import closed curves as solids.
- The 2D DXF export comes with new settings to specify if the curve representation will be exact or approximated.
- Geometry parts now support **Local Functions** for use inside the parts only. These functions are not accessible in instances of the part.
- Boolean operations built with the COMSOL kernel can now detect surface intersections where no adjacent edge intersects the other face.

MESHING FUNCTIONALITY

- For physics-controlled meshing, a new geometry analysis has been added to let small details and curved faces in the geometry be better resolved while still allowing for the mesh size to grow in other regions. Control this with the **Geometric Analysis, Detail Size** contributor.
- It is now possible to generate a physics-controlled mesh when remeshing imported meshes.
- The **Swept** operation now supports sweeping a mesh through shell-like domains that are bounded by two face components. It also has a new default source face meshing option: **Automatic (prefer hexahedra)**. With this option selected, the algorithm tries to mesh the boundaries with a quad mesh but switches to triangles for the boundaries where the quad mesher fails.
- For the **Create Edges** operation, it is now possible to select to create **Resulting small faces/domains** (3D/2D) and **Complement faces/domains** (3D/2D) selections. There is also a new **Group by continuous tangent** setting that will propagate an edge element selection by including subsequent elements of similar direction.
- Selections including mesh control entities are now handled correctly when meshing.
- The operations **Partition with Ball/Box/Cylinder** and **Partition by Expression** now support selections on resulting entities.
- When importing a mesh from a geometry sequence, the setting **Resolve geometric details** will ensure that the mesh becomes fine enough to close to resolve small details and curved surfaces.
- When exporting meshes on the NASTRAN format, the **Large** field format is now used by default. For export to the **Sectionwise** format, the **Settings** window now has radio buttons for the selection of data to export.

New Functionality in the Physics Interfaces

- The predefined multiphysics handling in the **Model Wizard** can now make use of existing participating physics interfaces.
- It is now possible to copy physics features between COMSOL sessions.
- The new **Explicit Event List** node in the **Events** interface defines multiple explicit events using a list of times and corresponding variable values for a common state variable. The input is in the form of a table, which in turn allows of defining the list of events from a text file.

- In the **Flux/Source** and **Interior Source** boundary conditions, available in the **Wave Form PDE** interface, there is a new load factor setting. The load factor l (default: 1) is multiplied with the flux term g to form a flux term $g\,l$. The use of the load factor is optional (the default of 1 means that it then has no effect), and it can be used for performance reasons.
- Matrix inputs in physics interfaces now include buttons to show the coordinate labels and to show all components when the input normally only shows the in-plane components.

New and Updated Operators, Functions, and Definitions

- For the **Deep Neural Network** function, support for training the DNN using a GPU has been added. Select the **Train on GPU** checkbox to train the DNN on a GPU, if your system includes a supported GPU. On the **Computing** > **GPU Acceleration** page in the **Preferences** window, you can also select the **Train Deep Neural Networks on GPU by default** checkbox. During installation, **CUDA DNN Support** must be selected on the **Products** page, and a NVIDIA® CUDA® Toolkit must also be available (it can be downloaded from the NVIDIA® website).
- A new **Geometry Sampling** method is available by choosing **Surrogate Model Sampling** > **Geometry Sampling** from the **Definition** node's context menu for model components. In its settings, you can select a sample method and set its properties to specify the desired geometry sampling method for sampling quantities of interest for a specific model component. This functionality makes generating spatial-, time-, and frequency-dependent data for use with deep neural network (DNN) surrogate models significantly easier.
- The **Interpolation** function has been updated and improved. When you import data using the **Spreadsheet** format, a new **Data Column Settings** section appears, where you can choose, for each column, if it contains an **Argument** or **Function values**. You can also choose **Ignored column** to ignore its content. There are now also **Include left extrapolation** and **Include right extrapolation** checkboxes in the **Plot Parameters** section so that you can choose whether to include extrapolations in the plot.
- You can now specify the location of the transition zone for smoothing of a step in **Step** functions. You can choose **Beginning of step**, **Middle of step**, or **End of step** from the new **Location definition** list, which can make it easier to use a smoothed step for initialization of time-dependent simulations.
- For **Least-Squares Fit** functions, you can now add lower and upper parameter bounds.

- For **Waveform** functions, you can now use Gaussian amplitude modulation. There is also a new **Equation** section in the **Settings** window that shows that equation for the waveform with the selected settings.
- You can now use `mTorr` to express a pressure in millitorr. Other SI prefixes are also available for the Torr unit.
- °F and °C are now available as Fahrenheit and Celsius units for temperature, respectively.
- There is a new type of pair feature: **Sector Symmetry Pair**. You can use it on a pair between two domains when modeling a sector of a geometry where one of the domains is rotating around the sector center. If a **Sector Symmetry** feature in a physics interface uses sector symmetry pairs instead of identity pairs, the sector settings in the physics feature are no longer needed because they are set in the pair. In this way, you do not have to provide the same settings in several physics features.
- For identity pairs and contact pairs, there are new options to select **All pairs**, **All contact pairs**, or **All identity pairs**.
- The **Partial Fractional Fit** function has been improved with the addition of an iteration-based stop condition and an option to automatically detect and remove Froissart doubles.
- Two new operators, `cubeint(`*r*`,<expr>)` and `cubeavg(`*r*`,<expr>)`, are available for evaluating the integral or average of an expression on the specified shape with radius or side *r*.
- For probes of **Maximum** and **Minimum** type, you can now specify the point type and integration order in the **Integration Settings** section of the settings.
- The new built-in operator `atlintotalpeak(<expr1>,<expr2>)` evaluates `<expr2>` at the phase angle that maximizes `lintotal(<expr1>)`.
- For the **Piecewise** function, a text area with syntax highlighting has been added.

New Functionality in Studies and Solvers

GPU ACCELERATION FOR TIME-DEPENDENT SIMULATIONS USING THE DISCONTINUOUS GALERKIN METHOD

For time-dependent simulations using the discontinuous Galerkin (dG-FEM) method, such as transient acoustics, you can use a GPU to accelerate the time-dependent solver. To do so, right-click the **Time-Dependent Solver** node and choose to add the new **Hardware Acceleration** subnode, where you select the **Accelerated solver for discontinuous Galerkin** checkbox. A CUDA® Toolkit must be available, and its path can

be specified during installation. That path can be changed on the **Computing** > **GPU Acceleration** page in the **Preferences** window, where you can also verify the CUDA installation.

FILTERING AND SORTING OF EIGENVALUE-TYPE SOLUTIONS

A new **Filtering and Sorting** section is available in the **Settings** windows for the **Eigenvalue**, **Eigenfrequency**, and **Boundary Mode Analysis** study steps. You can add filter expressions (such as `real(freq)+1e-6>0`) and sort the eigensolutions. Sorting options include sorting in ascending or descending order and sorting based on the real part, imaginary part, real part magnitude, imaginary part magnitude, or the absolute value.

Sorting of eigensolutions is also available in the **Eigenvalue Solver** node's settings and in the settings for the **Combine Solutions** node when combining solutions of eigenvalue type.

IMPROVED CONCATENATION OF COMBINED SOLUTIONS

In the **Combine Solutions** node's settings, you can now, for concatenation of solutions, choose a concatenation method: to let the first solution have precedence, which excludes solution numbers from the second solution that are also in the first solution, or to include all solutions. There is also an automatic or manual interpolation and sorting of solutions.

PARAMETER ESTIMATION

The **Parameter Estimation** node is now available in the base package, without any additional license requirements.

IMPROVED PHYSICS AND VARIABLES SELECTION

In the **Physics and Variables Selection** section, you can now select physics in a tree table and also select and disable physics to solve for on the component level.

NEW BACKTRACKING GLOBALIZATION METHOD

A new **Backtracking (Newton)** globalization method is now available for the **Fully Coupled** and **Segregated** solvers. This method uses information about the nonlinear residual as well as the system matrix to estimate the damping that minimizes the residual. This way, a faster convergence can be achieved as compared with using a constant damping factor.

NONLINEAR EIGENVALUE SOLVER

A new **ARPACK Nonlinear** solver is now available for eigenvalue-type studies and the **Eigenvalue Solver** node. It is useful for solving nonlinear eigenvalue problems that previously required workarounds and iterative methods.

PROPER ORTHOGONAL DECOMPOSITION (POD) MODEL-REDUCTION METHOD

In the **Model Reduction** study node's **Setting** window, you can now choose **POD** from the **Method** list. The POD (proper orthogonal decomposition) method is a projection-based method that projects an original full-order problem to a low-dimensional space spanned by POD modes obtained through performing singular value decomposition on training solutions and constraint modes. This method is suitable for model reduction of parametric problems without hyperreduction.

CONSTANT STEP BACKTRACKING

A new **Constant step** method can be selected from the **Backtracking** list in the **Fully Coupled** and **Segregated** solvers when the new **Backtracking (Newton)** method is used. This algorithm applies constant damping factors (which you can specify in the **Damping in one step** field) whenever the step taken by the solver does not improve the residual. The advantage of this method is that it is computationally efficient, and for certain models it comes at almost no extra cost. For example, models that use a fully coupled solver and that only need mild damping are well suited for this new algorithm.

STABILITY IMPROVEMENTS FOR BACKTRACKING AND ADAPTIVE TOLERANCE

For backtracking, a **Maximum damping factor** can be specified for the **Full estimate** and **Constant step** options. For adaptive tolerance, you can specify a value in the **Threshold for safeguard** field for the tolerance above which the safeguards are applied. You can also apply CFL weighting for models using pseudo time stepping. For models not using pseudo time stepping, you can apply a nonlinear error weighting. With these new options as well as internal stability improvements, a more stable convergence can be obtained in many cases, especially if backtracking and adaptive tolerance are used together.

CHECKING CONVERGENCE BEFORE SOLVING LINEAR SYSTEMS

The way convergence is controlled has been changed for constant damping and backtracking solvers. If the termination criterion is set to **Residual** or **Solution or residual**, residual-based convergence is checked before solving the linear system. This has the advantage that if the initial solution is already good enough, no resources are

wasted solving the linear system. Iterative linear solvers are also known to potentially stagnate if the initial guess is "too good". To activate this new convergence check, set the termination criterion to either **Residual** or **Solution or residual**.

IMPROVED CONTROL OF JAVA HISTORY LOGGING

A new preference setting, the **Store complete solver history** checkbox, on the **Computing** > **Study Defaults** page in the **Preferences** window, is cleared by default to not include all solver settings. Select it to log all solver settings.

TOLERANCE SETTING ON STUDY LEVEL

You can now specify the tolerance for **Stationary** and **Frequency Domain** study steps from the study settings. This setting provides easier access to the relative tolerance without generating the default solver and then opening the solver settings.

USE OUTPUT SOLUTION FOR TIME OPERATORS

You can now use the output solution together with time operators such as `at()` and `timeint()` by first selecting the new **Use output solution for the time operators** checkbox in the **Settings** window for the **Time-Dependent Solver** node. The access is limited to what is stored in the output solution object. This functionality can be used to implement certain time-delay differential equations (DDEs) or to access a moving time-average.

New and Improved Results and Visualization Functionality

ANIMATED CAMERA TRANSITIONS

In the **Graphics** window, camera transitions are now animated by default to get a smooth transition from, say, a default view in 3D to a view using the *xy*-plane. You can control this behavior using the settings on the **Graphics** > **Interaction** > **Camera** page in the **Preferences** window.

GENERAL NEW RESULTS AND VISUALIZATION FUNCTIONALITY

Natural Color Tables

There is a new **Natural** category of color tables, with 29 color tables resembling naturally occurring color variations, indicated with their names, such as **Algae**, **Amber**, and **Passiflora**. These color tables have a linear lightness gradient, which makes them accurate in representing the underlying data as well as intuitive to read. A linear lightness gradient can also improve readability for users with color deficiency.

Preferred Units Configuration Feature

A new feature is available under **Configurations**. Use the settings in the **Preferred Units** node to define preferred units for physical quantities and other physical expressions under results, which can be useful if you want to use other units than the ones provided by the model's unit systems in plots and other results features.

Result Templates

The **Add Predefined Plot** window in previous versions of the COMSOL Desktop has been renamed **Result Templates** to better reflect that you can set up such templates that include not just plots but also evaluation groups and derived values.

Refreshing Zoom to Extents

If you select the **Refresh zoom to extents** checkbox on the **Graphics** > **Interaction** > **Camera** page in the **Preferences** window, zoom extents will be refreshed when building the geometry if the bounding box for the entire geometry is in view at that moment.

RUNNING EXPORT FEATURES AFTER SOLVING

You can now run all **Export** features automatically after solving the model by first selecting the **Run all exports after solving** checkbox in the **Settings** window for the main **Results** node.

NEW AND IMPROVED DATASET AND EVALUATION FUNCTIONALITY

- A new **Beam** dataset is available for visualizing beams in a model by providing a 3D representation of the beams with a correct visualization of their shape.
- A new **Pipe** dataset is available for visualizing pipes in a model by providing a 3D representation of the pipes with a correct visualization of their shape.
- You can now compute values for mass properties from 3D Solution and Mesh datasets such as center of mass using a **Mass Properties** node. It is available on the **Measure** submenu under **Derived Values** and **Evaluation Group** nodes.
- A new **Radiation Pattern** node is available under **Export** for exporting radiation patterns for models where such radiation patterns are of interest.
- You can now include row numbers in tables by clicking the new **Show Row Numbers** button in the toolbar for **Table** windows.
- Parameters defined under **Results** can now be used in more places than before, such as in interpolated times, data, and color ranges in plots.
- Data export features now support evaluating in Gauss points using the following datasets: **Particle**, **Edge 2D**, **Edge 3D**, **Surface**, **Transformation 2D**, and **Transformation**

3D, in addition to **Solution** datasets, which were the only datasets to support such data evaluations in earlier versions.

- The **Grid** datasets now include an **Advanced** option, with which it is possible to define auxiliary variables and their units. The auxiliary variables have the same value ranges as the parameters in the dataset, but their units can be controlled explicitly. Doing so makes it easier to change the x-axis unit in function plots, for example.
- You can now omit the space coordinate values from data export by clearing the new **Include coordinates** checkbox in the **Advanced** section of the settings for **Export** nodes.

NEW AND IMPROVED GRAPHICS AND PLOT FUNCTIONALITY

- You can now show titles for the color legends. The titles can include math mode LaTeX commands enclosed with `\$` and `\$`.
- It is now possible to choose the default geometry and plot materials as visualization materials from the **Material type** list in the **Appearance** sections in **Material** nodes under components and **Material Appearance** plot subnodes.
- The **Translation** subnode has been replaced with a **Transform** subnode, where you can choose a structured transformation including scaling, rotating, and moving the plot, or choose a general transformation with base vectors and a translation.
- Annotations can now be added by clicking in the **Graphics** window. You can do so by first clicking **Marker Point** in the **Graphics Interaction** section of the plot group's ribbon toolbar. A **Marker** subnode will be created if needed, and additional points will be added to the point evaluation in the **Marker** subnode's settings.
- For **Marker** subnodes, you can now choose to display point evaluations. They can be entered as coordinates in a table or added interactively in the plot, when the **Marker Point** button has been selected.
- The **Marker** subnode is now also available for **Contour Series**, **Isosurface Series**, **Table Surface**, **Table Contours**, and **Table Point** plots as well as for **Surface Data**, **Line Data**, **Tube Data**, and **Point Data** plots.
- The number formatting has been improved for **Annotation** plot, **Max/Min** plots, and **Marker** subnodes. You can now choose from automatic, engineering, scientific, stopwatch, and scientific stopwatch formats.
- A new option, **Last**, is available for time selection of transient solutions in 2D and 3D.
- 3D streamlines can now start from selected edges.

- A new **Unwrap phase** checkbox is available in 1D **Global** and **Point Graph** plots for data with an angle as the unit. When the checkbox is selected, your plot will show the phase values without jumps.
- You can now add multiple **Graph Marker** subnodes to the same plot for applicable plot types.
- The `eval` operator in **Annotation** plots has been improved. It now supports formats of the `%[flags][width][.precision]` type. Allowed types are `d`, `f`, `e`, `E`, `g`, and `G`. For example, `eval(pi*1e10,,%+.3e)` gives the output `+3.142e+10`, showing the number with scientific format with 3 decimal values and with the sign of the number shown.
- For **Point** plots, you can now display values as text. Evaluation can also be done not only at points but also for other geometry levels: line, surface, and volume (depending on the space dimension of the model). The evaluation can now also be done at mesh nodes or Gauss points.
- For models that include a phase, the **Solution at angle (phase)** list in plot groups now includes a **Custom per plot** option, which adds **Solution at angle (phase)** lists for all plots in the plot group. You can then specify the phase individually for each plot.
- For a disabled plot, clicking **Plot** or pressing F8 no longer enables or plots the plot. Clicking **Plot** plots all active and enabled plots in the plot group.

NEW AND IMPROVED REPORT AND PRESENTATION FUNCTIONALITY

- The report level **Brief** now includes settings for **Function** nodes.
- **Physics Interface** report nodes now include variables, shape functions, weak expressions, and constraints by default only for reports of level **Complete**.

New and Improved Java® API Methods

HASTAG METHOD

To check whether a feature with a given tag exists in a list, use the new `hasTag` method. It will return true if the list contains a model entity with the given tag.

The following methods have been added or improved in version 6.3:

ABSOLUTE TOLERANCE IN THE GENERAL EXTRUSION OPERATOR

For `model.cpl()` and a general extrusion operator, if `manualsearchdist` is set to `on` and the source selection is a boundary, edge, or point selection, `searchdist` gives the

search distance in directions orthogonal to the source selection. If `manualsearchdist` is `off`, the search distance is equal to `exttol` times the mesh element size.

IMPROVED SUPPORT FOR EDGE EVALUATIONS AND PROJECTIONS

The methods `edgeX`, `edgeDX`, `edgeDDX`, `edgeDDDX`, `edgeNormal`, `edgeCurvature`, and `edgeTorsion` have been updated to support evaluation on a smooth interpolated edge curve for meshes that define their own geometric model.

REMOVING ROWS IN RESULTS TABLES

Use the new `removeRows(int[])` method to delete rows from a results table. You can delete several rows, provided as an input, given as an integer array of row numbers.

RETRIEVING THE NUMBER OF PARAMETER VALUES THAT ARE AVAILABLE FOR STEPPING OVER

Use the new `getStepCount(int level)` method to retrieve the number of parameter values that are available for stepping over on a given level, given as an integer input value. This functionality can be useful for managing sliders in apps, for example.

General Backward Compatibility Considerations

COMSOL Multiphysics version 6.3 can open MPH-files saved from COMSOL Multiphysics versions 4.0–6.2.

COMSOL Multiphysics version 6.3 can run Java® files saved from COMSOL Multiphysics versions 4.0–6.2. 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 6.3.

Backward Compatibility with Version 6.2

REMOVED DEVICE FEATURE FROM THE PHYSICS BUILDER

In version 6.2 and earlier versions, under the **Device Model Feature** node, a node called a **Device Feature** that created an instance of the **Device Model Feature** directly was available. This functionality was inconsistent with the basic framework, both how **Device Model Feature** nodes should be used and how **Device Model** features generally behave. There is a migration that converts the configuration to an equivalent configuration with a **Device Model** and a **Device** node under a **Global Feature.** So the migration changes the type of the **Device Model Feature** to a global feature, move all device-related nodes to the new **Device Model** node, and finally change the obsolete

Device Feature node to a normal **Device** node. There are some complicated cases when the migration fails to do a completely valid migration. It then adds a warning giving more information about the problem.

CHANGED NAMES FOR PARAMETERIZED CURVE AND PARAMETERIZED SURFACE DATASETS

The datasets called Parameterized Curve 2D, Parameterized Curve 3D, and Parameterized Surface in earlier versions of COMSOL Multiphysics are now called Parametric Curve 2D, Parametric Curve 3D, and Parametric Surface, respectively.

GEOMETRY IMPORT IN WORK PLANES

For **Import**, there is a new setting **Include result form virtual operations** when importing MPHBIN and MPHTXT in a work plane. In 6.2, this setting did not exist in this case, which meant that only the real part of the geometry was imported. In 6.3, the setting is on by default, which means that the whole virtual geometry is imported. This concerns the property `includevirtual` in the Java API.

GEOMETRIC MODEL UPDATE FOR MESH

The method `geometricModel(boolean)` has been replaced with `geometricModel(String)` to specify if a meshing sequence operates on a geometry or on a mesh.

Backward Compatibility with Version 6.1

NEW WEBVIEW2 BROWSER

The CefSharp web browser control used by the .NET client in version 6.1 has been replaced with WebView2.

Backward Compatibility with Version 6.0

NEW CEFSHARP BROWSER

The Chromium™-based CefSharp browser component replaces the Internet Explorer-based .NET WebBrowser component in version 6.1.

In the **Video** and **Web Page** form objects, there is now a **Native browser on Windows** option in the settings to determine if the Internet Explorer®-based .NET WebBrowser component from earlier versions should be used instead. The default value for the native browser is **Chromium™ (CEF)** but all existing models have the setting set to

Internet Explorer® by default to maintain the functionality from earlier versions. Use the launcher flag `cs.legacywebbrowser=true` as a way to keep the previous browser component instead of CefSharp.

The CefSharp browser component stores user data such as cookies, browser history, and local storage in a folder. This `cscefcache` folder is placed among the temporary files of the client and persist between COMSOL and browser sessions. You can delete the `cscefcache` folder to clear this data. The CEF log output is written to `Users\[user]\.comsol\6.2\logs\cef.log`. The log level can be controlled using the `cs.cefloglevel` launcher argument, which accepts the values `info`, `warning`, `error`, and `fatal`.

EQUATION FORM FOR PDE AND ODE INTERFACES

When adding a PDE or ODE interface in the user interface, the equation form will by default be set to **Study controlled**. This will lead to a different interpretation of time derivatives in frequency-domain studies compared to previous versions. When creating a new PDE or ODE interface from the API or opening a model saved in a previous version, the equation form will be set to **Time domain**, which corresponds to the behavior in previous versions.

THE FLASH MOVIE FORMAT

For movie export, the Flash format is no longer available as of version 6.1.

TIME-EXPLICIT SOLVER

The **Time-Dependent Solver** node settings now includes the time-explicit methods from the **Time-Explicit Solver** node that was available in the **Model Builder** in earlier versions. The **Time-Explicit Solver** node is still available in models created in earlier version of COMSOL Multiphysics. It is also available in the COMSOL API.

MODEL REDUCTION STUDY STEP

The **Study step for eigenmodes** and **Study step for constraint modes** settings in version 6.1 correspond to **Defined by study step** in previous versions because it is not possible to use one study step for two different solution types.

PERIODIC CONDITIONS — DESTINATION SELECTION

The **Destination Selection** subnode under **Periodic Condition** nodes has been removed in version 6.1 and has been replaced by an optional **Destination Selection** section in the

Settings windows for **Periodic Condition** nodes. It can still be added from the API for backward compatibility. The **Destination Selection** subnode was used for two purposes:

- To provide a manual destination selection in cases where the automatic detection failed for some reason (for example, in models with curved boundaries). The selection of the feature was used as destination domains. When the **All boundaries** option was selected for the selection list, the automatic destination was used from the parent periodic condition.
- To add a manual orientation for vector transforms of the destination boundaries. The **Orientation of Destination** section was used for this.

For more information about the API compatibility, see COMSOL 6.1 API Changes.

FLASH MOVIE FORMAT

The Flash movie format is no longer available for export of movie files using the **Animation** feature. Animation features in models created using version 6.0 or earlier where the format is set to Flash now use GIF as the format from version 6.1.

STREAMLINE PLOT CHANGES

For 2D **Streamline** and 3D **Streamline Surface** and **Streamline Multislice** plots, the algorithm for finding and plotting streamlines has been improved. The **Density** setting in version 6.0 and earlier versions has been replaced with the **Maximum distance** setting, which is the inverse of the **Density**. A corresponding **Minimum distance** setting has also been added. When opening models from version 6.0 and earlier versions, the **Minimum distance** is assigned a value that is 0.05 times the **Maximum distance**. This is a reasonable heuristic, but it is not guaranteed to give a good result in all cases.

PHYSICS SYMBOLS IN 1D AND 0D GEOMETRIES

The **Physics Symbols** section in the settings for physics interfaces, features, and boundary conditions has been removed for all components with dimension less than 2D. This change might cause issues with old public API code that accesses any physics symbol properties in 1D or 0D, even though they did not have any effect in earlier versions.

AC/DC Module

New and Improved Functionality in Version 6.3

NEW FORMULATION FOR ELECTROSTATICS

The **Electrostatics** interface now supports the **D–V Formulation** in 2D and 3D, allowing for a more accurate electrostatic force evaluation. This is of particular interest for *3D MEMS device modeling*.

As a counterpoint to solving for scalar potential, as is done in the V Formulation, electrostatic problems can be reformulated to solve for flux. The D–V formulation is a mixed formulation technique that solves for the normal components of the electric displacement field, D. It does this by solving a system of two equations: one for the electric displacement field and a second equation for the electric potential, V. The second equation acts as a constraint on the first equation.

Computing the flux directly produces a high accuracy for the electric displacement field and is particularly advantageous for calculating electromechanical forces in geometries that contain sharp corners.

The **D–V Formulation** can be activated by going to the **Discretization** options in the **Settings** window for **Electrostatics** and selecting any of the **Mixed finite element** options.

NEW ELECTROMECHANICS MULTIPHYSICS INTERFACES

Under **AC/DC** > **Electromagnetics and Mechanics** > **Electromechanics**, two new multiphysics interfaces have been added to the **Model Wizard** tree: **Electromechanics, Shell** and **Electromechanics, Membrane**. Both use the new **Electromechanics, Boundary** multiphysics coupling, together with the **Electrostatics** interface and either the **Shell** or **Membrane** interface. Coupling **Electrostatics** with **Shell** or **Membrane** was possible before, but the new interfaces streamline the process.

For the new multiphysics interfaces as well as the existing **Electromechanics, Solid** interface, the contained **Electrostatics** interface is now configured to use the new **D–V Formulation** by default — although, this can be reverted back to the regular formulation if desired. Note that the new **Electromechanics, Boundary** multiphysics coupling requires a license for the Structural Mechanics Module, together with either the MEMS Module or the AC/DC Module.

SUPPORT FOR HIGH-FREQUENCY LOSS IN MULTITURN COILS

The **Homogenized multiturn** conductor model in the **Coil** domain feature — as used in the **Magnetic Fields**, the **Magnetic and Electric Fields**, and the **Rotating Machinery, Magnetic** interfaces — has received multiple updates, most notably for modeling intrinsic loss caused by skin and proximity effects in the coil.

In the **Frequency Domain** study when the **Wire properties** setting is using the **From conductivity** option, a new **High-frequency effective loss** model is available (enabled by default). This loss model introduces an effective wire conductivity and an effective complex permeability in the coil such that the field distribution and the AC resistance resemble that of a fully resolved coil consisting of hexagonally packed wires with skin and proximity effects. For **Stationary** and **Time Dependent** studies, the loss model reverts back to the DC approximation, as used in previous COMSOL® versions.

For both the **Time Dependent** and **Frequency Domain** studies, a new **Wire properties** option is available, called **From resistance and mutually coupled circuit**. It couples the coil to a lumped circuit. When the circuit parameters are properly tuned, the resistive and inductive effects modeled by the circuit resemble those coming from a fully resolved skin and proximity effect.

The advantage of this approach is that it supports both the **Frequency Domain** and **Time Domain** study and is therefore more versatile. The disadvantage is that the circuit needs to be tuned. Oftentimes, an educated guess is possible, but fitting gives more accurate results. Fitting is typically done by matching the average loss to that of an equivalent fully resolved model, or to measurements.

NEW HOMOGENIZED LITZ COIL CONDUCTOR MODEL

For the **Coil** domain feature — as used in the **Magnetic Fields** and the **Rotating Machinery, Magnetic** interfaces — there is now a new **Homogenized litz coil** conductor model. This conductor model shares many similarities with the **Homogenized multiturn** option, but it is more suitable for litz coils.

When the **Litz wire properties** setting uses the **From conductivity** option, the **Number of strands** setting can be specified, along with the **Litz wire DC resistance per unit length**. The latter is used to compensate for the increase in DC resistance caused by the twisting or weaving pattern. (Note that the homogenized multiturn conductor model assumes untwisted wires.) Like the **Homogenized multiturn** conductor model, the **Homogenized litz coil** conductor model supports **High-frequency effective loss**.

When the **Litz wire properties** setting is using the **From resistivity** option, the wire resistance per unit length can be set explicitly. The wire resistance can be taken from a

specification sheet or a measurement, and it can be a constant value or taken from a lookup table or a frequency-dependent curve. Analytical expressions as given by literature are supported too — they are typically based on known parameters such as strand size, conductivity, number of strands, and frequency.

Note that at higher frequencies, the total net AC resistance will depend on both skin and proximity effects within the litz coil (intrinsic effects) and the inductive coupling between the coil and its environment (external effects).

MISCELLANEOUS COIL IMPROVEMENTS

Smaller usability improvements for the **Coil** domain feature — as used in the **Magnetic Fields**, the **Magnetic and Electric Fields**, and the **Rotating Machinery, Magnetic** interfaces — include support for a **Symmetry specification** in 2D and the **Filling factor** variable. The **Filling factor** is an important parameter to check when the wire (or strand) cross-section area is specified explicitly. This is done to avoid coil configurations where the copper mass is larger than would physically fit inside the coil. Typically, the filling factor should not surpass 0.9 (the densest packing for circles), but higher values are allowed. This functionality is useful for experimentation, sweeping, or optimization purposes.

The **Symmetry specification** feature can be used when *sector symmetry* or *mirror symmetry* is used, and the model represents only a fraction of the actual device. It can be useful for setting up stator coil configurations for motor models (for cases where the **Multiphase Winding** feature is not applicable). Another application of interest is the 2D modeling of fully resolved litz coils: The symmetry factors can be used in combination with the **Coil group** option in 2D to model the strands fully resolved with skin and proximity effects included. These 2D models can then be used to verify the **High-frequency effective loss** model in 3D or to fine-tune the circuit parameters used in the **From resistance and mutually coupled circuit** option.

NEW COIL FEATURE FOR THE MAGNETIC MACHINERY, ROTATING, TIME PERIODIC INTERFACE

A new **Coil** feature has been added to the **Magnetic Machinery, Rotating, Time Periodic** interface. It can be seen as a combination of the existing **Passive Conductor** feature and **External Current Density** feature. The main advantage of this new feature is that it combines a simple form of excitation with the support for skin and proximity effects. For more advanced excitation forms, consider using the **Multiphase Winding** feature instead.

NEW MULTI-TERMINAL CONDUCTOR FEATURE

The **Magnetic Fields, Currents Only** interface has a new **Multi-Terminal Conductor** feature. This feature is similar to the existing **Conductor** feature and can be used as an easy way to apply current sources, especially in connection with the **Stationary Source Sweep with Initialization** study step and for conductors containing multiple current inputs to sweep onto. The **Multi-Terminal Conductor** feature has several subnodes (**Terminal**, **Ground**, and **Electric Insulation**) that can be used to define the boundary conditions for the current distribution.

NEW LAMINATED CORE FEATURE

A new domain feature, **Laminated Core**, has been added to the **Magnetic Fields** interface, the **Magnetic Fields, No Currents** interface, and the **Rotating Machinery, Magnetic** interface. For the **Rotating Machinery, Magnetic** interface two flavors are available: the **Laminated Core, Ampère's Law** feature and the **Laminated Core, Magnetic Flux Conservation** feature. These new features are suitable for modeling laminated cores as used in transformers, electromechanical actuators, and electric motors. They are available in 2D and 3D for **Stationary**, **Time Dependent**, and **Frequency Domain** studies.

In the **Laminated Core** feature, laminae are approximated by an anisotropic effective medium. Using the **Stacking factor** option, the ratio between magnetic and nonmagnetic material can be specified. The **Stacking direction** option can be specified as well. For the lamina, both linear and nonlinear magnetic properties are supported: **Relative permeability**, **B–H curve**, and **Effective B–H curve**. The resistive and magnetic loss can be included through empirical loss models, such as **Steinmetz** or **Bertotti**.

EASY EVALUATION OF THE TOTAL MAGNETIC COENERGY

A predefined variable for the total magnetic coenergy is now available for the **Magnetic Fields**; **Magnetic and Electric Fields**; **Magnetic Fields, No Currents**; and **Rotating Machinery, Magnetic** interfaces. Contributions from both domains and boundaries are taken into account. In particular, the magnetic energy now also includes the energy stored in the **Thin Low Permeability Gap** boundary feature.

The magnetic coenergy constitutive relation of a magnetic material now supports sensitivity analysis. This functionality provides easier extraction of magnetic forces by the principle of *virtual work* and easier extraction of the *differential inductance*. Among other things, the new functionality is relevant for extracting a lumped representation of electromechanical devices.

NEW DQ EXCITATION SUPPORT FOR THE MULTIPHASE WINDING

Specifically introduced for motor modeling, the direct quadrature (DQ) transformation support allows for the simulation of electrical machines more in line with common control strategies and for easier calculation of key machine parameters. Essentially, the DQ transformation converts the AC signals from stator coils producing a rotating field to an equivalent set of DC signals for a virtually rotating coil assembly producing a constant field.

In order to use the DQ transformation as a means of excitation, the **DQ currents** option has been added in 2D as an **Excitation type** to the **Multiphase Winding** feature in the **Magnetic Machinery, Rotating, Time Periodic** interface, and the **Rotating Machinery, Magnetic** interface. Furthermore, a set of predefined DQ variables is now available in those interfaces to simplify result evaluation and data extraction.

NEW CONTACT IMPEDANCE AND ELECTRICAL CONTACT BOUNDARY CONDITIONS FOR THE MAGNETIC AND ELECTRIC FIELDS INTERFACE

The resistive boundary conditions in the **Magnetic and Electric Fields** interface have been restructured, and new ones have been added. The **Electric Insulation** boundary feature, the **Contact Impedance** boundary feature, and the **Electrical Contact** boundary feature are now available as subnodes to the **Magnetic Shielding**, the **Magnetic Continuity**, and the **Thin Low Permeability Gap** boundary conditions. This allows for the modeling of electrical contact together with inductive phenomena.

IMPROVED HANDLING OF SOLIDS AND LIQUIDS (FOR SEVERAL PHYSICS INTERFACES)

For several physics interfaces, the handling of solids and liquids has been improved, making it easier to build models with moving materials. In particular, the **Electrostatics** interface now has a **Charge Conservation in Solids** and a **Charge Conservation in Fluids** feature; the **Magnetic Fields, No Currents** interface now has a **Magnetic Flux Conservation in Solids** and a **Magnetic Flux Conservation in Fluids** feature; and the **Magnetic and Electric Fields** interface has an **Ampère's Law and Current Conservation in Solids** and an **Ampère's Law and Current Conservation in Fluids** feature. Note that the previous versions of these features have been deprecated.

The new features provide a more user-friendly implementation that helps to ensure that the appropriate material models and the appropriate coordinate frames of reference are used for liquids, gases, and vacuum as well as for solid objects. Furthermore, the features allow for a more intuitive setup of multiphysics couplings. Multiphysics couplings designed for solids, such as the **Electromechanics** coupling, are only applicable in domains where the corresponding solid feature is active, whereas

multiphysics couplings designed for liquids, such as the **Magnetohydrodynamics** coupling, are only applicable in domains where the corresponding fluid feature is active.

NEW DEFAULT FREE SPACE FEATURE (FOR SEVERAL INTERFACES)

A new default feature, **Free Space**, has been added to the **Electrostatics** interface, the **Magnetic Fields, No Currents** interface, and the **Magnetic and Electric Fields** interface. It is used to specify the physical conditions in close proximity to the modeled device (typically in air or vacuum). The feature provides a starting point to which other features — such as the **Charge Conservation in Solids** feature or the **Ampère's Law and Current Conservation in Fluids** feature (see above) — can be added to locally specify material properties and excitation forms.

For the **Magnetic and Electric Fields** interface, the **Free Space** feature also now comes with a built-in **Stabilization** option. The **Automatic** setting adds an artificial conductivity term that keeps the model from becoming singular (1 S/m for **Stationary** and **Time Dependent** studies, and 0 S/m for **Frequency Domain** studies). A user defined option is available, too, for manual tuning of the stabilization.

Note that replacing the default feature has an impact on backward compatibility: Model methods, Java® code, and MATLAB® code that have been saved using previous COMSOL® versions may need to be revised in order to function properly.

MORE FEATURES NOW SUPPORT THE SPLIT BY CONNECTIVITY OPTION

When a model of a physical device requires hundreds of coils (or terminals), setting up every one of them individually can be time consuming. For such cases, a single selection can be used that contains the union of all coils. This selection is typically provided by the geometry sequence, a geometry import node, or some other form of automation. The **Split By Connectivity** option analyzes the combined selection and automatically identifies all coils within it. With a single click, a physics feature is created for every coil.

This functionality has now been extended to most instances of **Coils** in domains, **Terminals** in domains and on boundaries, and **Lumped Ports** in the **Electrostatics**, **Electrostatics, Boundary Elements**, **Electric Currents**, **Magnetic Fields**, **Magnetic and Electric Fields**, and **Rotating Machinery, Magnetic** features. Furthermore, it is now supported by the public API, meaning that the **Split By Connectivity** action can be included in apps, plug-ins, or model methods.

IMPROVED USABILITY FOR CIRCUIT CONNECTIONS

The usability of circuit connections has improved greatly. A domain physics interface such as the **Electric Currents** interface or the **Magnetic Fields** interface can be connected to the lumped **Electrical Circuit** interface by means of a **Coil** or **Port** feature. When such a domain or boundary feature is set to be connected to a lumped circuit, automated functionality is now available to complete the circuit connection. This includes options to create a new **Electrical Circuit** interface or to connect to an existing one. The lumped elements in the circuit that provide the connection to the finite element model are created automatically, and useful feedback is provided when the connection is incomplete.

PUBLIC API SUPPORT FOR CIRCUIT IMPORT AND EXPORT

The **Electrical Circuit** interface supports the **Import SPICE Netlist** and **Export SPICE Netlist** options. In addition to this, circuit import and export are now supported through the public API. This means that a model method, an app, or a plug-in can automatically retrieve or export a circuit. This is especially useful for *lumped circuit extraction*, where circuits are generated in an automated fashion based on lumped resistance, capacitance, and inductance matrices as retrieved from the finite element model.

BETTER SUPPORT FOR MODELING BIOLOGICAL TISSUES (INCLUDING NEW BIOLOGICAL TISSUE MATERIAL LIBRARY)

For both the **Electrostatics** interface and the **Electric Currents** interface, three new dispersive dielectric material models are available: **Cole–Cole**, **Havriliak–Negami**, and **User defined**. All three are available for **Frequency Domain** studies as well as for **Time Dependent** studies — using the new support for partial fraction fitting (note that this implies that the dispersive materials can also be fitted directly on measured data). This is of particular interest for medical applications.

Furthermore, the existing **Multipole Debye** dispersive material model can now be used in combination with the new **Biological Tissues** material data. The **Biological Tissues** folder in the **AC/DC** material library branch contains values for the electric conductivity, relative permittivity, and reference temperature, together with the relaxation times and relative permittivity contributions of several Debye poles, for 54 biological tissue types.

The dispersion models are available in the **Dispersion** subnode. This subnode is located under the **Charge Conservation in Solids** feature (for **Electrostatics**) and the **Current Conservation** feature (for **Electric Currents**, assuming that the feature's **Material type** is set to **Solid**). The subnode will become visible when the parent feature's **Dielectric model** is set to **Dispersion**.

Note that the **Dispersion** subnode is now *contributing*. This implies that the effect of several dispersion models can be combined by adding multiple **Dispersion** subnodes.

Note that the **Cole–Cole** and **Havriliak–Negami** material models require the AC/DC Module, while the **Multipole Debye** and **User defined** models are also available for the MEMS Module.

BETTER DEFAULT SOLVERS AND DEFAULT SETTINGS FOR MOTOR MODELING

The default solver settings for the **Stationary** and **Time Dependent** studies — as used for the **Magnetic Machinery, Rotating, Time Periodic** and the **Rotating Machinery, Magnetic** interfaces — have been redesigned, with a general focus on performance and robustness. The default discretization is now **Linear** rather than **Quadratic**, allowing for more robustness and better performance in the case of *nonlinear magnetic materials*.

Furthermore, the default **Constraint** type used in the **Continuity** pair feature — as used in the **Rotating Machinery, Magnetic** interface — has now changed from **Pointwise constraints** to **Weak constraints**, allowing for coarser meshes on the pair boundaries. (To see the **Constraint Settings** section, enable **Show More Options** > **Advanced Physics Options**.)

Note that changing the default discretization and the default constraint type has an impact on backward compatibility: Model methods, Java® code, and MATLAB® code that have been saved using previous COMSOL® versions may need to be revised in order to function properly.

NEW ITERATIVE SOLVER FOR THE MAGNETIC FIELDS INTERFACE

The **Auxiliary-Space Algebraic Multigrid** (Auxiliary-Space AMG, or ASAMG) solver has been introduced as the new default iterative solver for the **Magnetic Fields** interface, in particular for stationary studies. For most models, this is expected to reduce memory consumption and improve both performance and robustness.

NEW NAMING CONVENTION FOR VARIABLE NAMES IN THE ELECTRICAL CIRCUIT INTERFACE

The variable names used in the **Electrical Circuit** interface have been updated from using an underscore convention to using a dot-separated convention; for example, "`cir.R1_p_v`" is now "`cir.R1.p.v`". Furthermore, the new format reduces the level of ambiguity for variables where the underscore is used to separate the subscript from the main variable name — as used in "`R_dc`", for example.

Note that changing the naming convention has an impact on backward compatibility: Custom expressions containing these names — including those used in model methods, Java® code, and MATLAB® code — may need to be revised in order to function properly.

MISCELLANEOUS IMPROVEMENTS

- The **Stationary Source Sweep** study now supports cluster distribution.
- The **Lumped Port** feature — as used in the **Magnetic Fields** interface, the **Magnetic and Electric Fields** interface, and the **Rotating Machinery, Magnetic** interface — now supports specifying the excitation direction by means of a **Reference edge**.

New and Updated Models, Apps, and Add-Ins in Version 6.3

PERMANENT MAGNET MOTOR IN STEADY STATE (NEW MODEL)

Obtaining the steady-state performance is an essential task for the electric motor designer. As with many electromagnetic devices, the steady-state condition of an electric motor occurs when the electric and magnetic field variations have stabilized to a time-periodic pattern. The **Magnetic Machinery, Rotating, Time Periodic** interface solves directly for the steady-state operation while fully including the effects of nonlinear materials and induced currents. In this tutorial model, a distributed wound interior permanent magnet (IPM) machine is modeled to demonstrate the fundamentals of the time-periodic interface.

PIEZOMAGNETIC CELL ROVER (NEW MODEL)

This tutorial shows how to model a miniaturized magnetostrictive antenna developed for use inside living cells. The stress in the antenna, the magnetic flux density, the current density, and the displacement of the tip of the device are investigated at the resonance frequency.

B–H CURVE CHECKER (UPDATED APP)

The B–H Curve Checker app has been improved. In particular, it now supports input data that does not reach all the way to full saturation. The data is augmented assuming that the differential permeability will decrease, eventually settling at the vacuum permeability. This gives more realistic saturation effects in models with high-flux concentrations.

CIRCUIT EXTRACTOR (UPDATED ADD-IN)

The **Circuit Extractor** add-in allows for automated extraction of lumped circuits by analyzing the lumped resistance, capacitance, and inductance matrix retrieved from the finite element model. The resistance networks extracted by the add-in have been updated to better follow the SPICE standard.

Backward Compatibility with Version 6.2 and Earlier

- For the **Electromechanics, Solid** multiphysics interface, the contained **Electrostatics** interface is now configured to use the new **D–V Formulation** by default. This version of the **Electromechanics, Solid** multiphysics interface is therefore not compatible with previous versions of COMSOL, considering the support of model methods, Java® code, and stored MATLAB® scripts. This is because these scripts typically do not mention default settings explicitly. To manually migrate scripts that rely on the previous default settings, add an instruction that applies the previous default settings explicitly.
- The **Coil** feature in the **Magnetic Fields** interface, the **Magnetic and Electric Fields** interface and the **Rotating Machinery, Magnetic** interface now has new default settings for the **Coil type** and the **Wire cross-section area**. The **Coil type** is applicable in 3D for the **Homogenized multiturn** conductor model, and the new default is **Numeric**. The **Wire cross-section area** is used for the **Homogenized multiturn** conductor model when the **Wire properties** are set to **From conductivity** (both in 2D, and 3D), and the new default is **Filling factor**. This version of the **Coil** feature is therefore not compatible with previous versions of COMSOL, considering the support of model methods, Java® code, and stored MATLAB® scripts. This is because these scripts typically do not mention default settings explicitly. To manually migrate scripts that rely on the previous default settings (**Linear**, and **User defined**), add an instruction that applies the previous default settings explicitly.
- The **Homogenized multiturn** conductor model in the **Coil** feature — as used in the **Magnetic Fields**, the **Magnetic and Electric Fields**, and the **Rotating Machinery, Magnetic** interfaces — does not support advanced magnetic and dielectric material models any longer. This is because the correct functioning of the **Homogenized multiturn** conductor model combined with these material models could not be guaranteed. This version of the **Coil** feature is therefore not compatible with previous versions of COMSOL. Existing **Coil** features saved in COMSOL 6.2 and before will keep the full set of magnetic and dielectric material models.
- The switch from the old **Charge Conservation** feature in the **Electrostatics** interface, the old **Magnetic Flux Conservation** feature in the **Magnetic Fields, No Currents**

interface, and the old **Ampère's Law and Current Conservation** feature in the **Magnetic and Electric Fields** interface, to the new default **Free Space** feature in all these interfaces, means that this version of these interfaces is not compatible with previous versions of COMSOL, considering the support of model methods, Java® code, and stored MATLAB® scripts. This is because these scripts typically do not mention the creation of the default feature explicitly. To manually migrate scripts that rely on the previous default features, add an instruction that adds the previous default features explicitly.

- The **Charge Conservation** feature in the **Electrostatics** interface has been deprecated and is replaced by a **Charge Conservation in Solids** feature and a **Charge Conservation in Fluids** feature.
- The **Magnetic Flux Conservation** feature in the **Magnetic Fields, No Currents** interface has been deprecated and is replaced by a **Magnetic Flux Conservation in Solids** feature and a **Magnetic Flux Conservation in Fluids** feature.
- The **Ampère's Law and Current Conservation** feature in the **Magnetic and Electric Fields** interface has been deprecated and is replaced with an **Ampère's Law and Current Conservation in Solids** feature and an **Ampère's Law and Current Conservation in Fluids** feature.
- The default discretization for the **Rotating Machinery, Magnetic** interface has changed from **Quadratic** to **Linear**, both for the **Magnetic scalar potential** and the **Magnetic vector potential**. In addition to this, the default **Constraint** type used in the **Continuity** pair feature has changed from **Pointwise constraints** to **Weak constraints**. This version of the **Rotating Machinery, Magnetic** interface is therefore not compatible with previous versions of COMSOL, considering the support of model methods, Java® code, and stored MATLAB® scripts. This is because these scripts typically do not mention default settings explicitly. To manually migrate scripts that rely on the previous default settings, add an instruction that applies these previous default settings explicitly.
- The default solver used for the **Magnetic Fields** interface has changed for the **Stationary** study step. Existing models with a manually configured solver saved in COMSOL 6.2 and earlier may need to have their solver configurations reset, to benefit from the new defaults.
- The default solver used for the **Rotating Machinery, Magnetic** interface has changed for the **Stationary** study step and for the **Time Dependent** study step. Existing models

with a manually configured solver saved in COMSOL 6.2 and earlier may need to have their solver configurations reset, to benefit from the new defaults.

- The variable names used in the **Electrical Circuit** interface have been updated from using an underscore convention to using a dot-separated convention — for example "`cir.R1_p_v`" is now "`cir.R1.p.v`". Custom expressions containing these names — including those used in model methods, Java® code, and MATLAB® code — may need to be revised in order to function properly.

Acoustics Module

New Functionality in Version 6.3

GPU SUPPORT FOR ACCELERATED ACOUSTICS SIMULATIONS WITH PRESSURE ACOUSTICS, TIME EXPLICIT

Support for an accelerated solver that can use GPU has been added for the **Pressure Acoustics, Time Explicit** interface. A new option where the system data is extracted and cashed for efficient evaluation on either GPU or CPU has been added. Important features like the general **Impedance** condition (to include real impedance data of absorbing surfaces) and the **Absorbing Layer** (to model open problems) are available with the GPU. If the solved problem can fit in the memory of the GPU, this results in the most significant speedup.

POROACOUSTICS IN THE TIME DOMAIN FOR PRESSURE ACOUSTICS, TIME EXPLICIT

A new **Poroacoustics** feature has been added to the **Pressure Acoustics, Time Explicit** interface. This allows for modeling porous materials in the time domain using an equivalent fluid model. The method relies on transforming frequency-dependent data for the equivalent density and compliance to the time domain using a rational function approximation. The data can be fitted with the built-in **Partial Fraction Fit** function.

FAST FORMULATION OF THERMOVISCOUS ACOUSTICS

The new **Thermoviscous Acoustics, SLNS Approximation** interface solves for the propagation of acoustic waves using the so-called *sequential linearized Navier–Stokes*, or *SLNS*, approximation, to include the thermoviscous boundary layer losses in a computationally efficient manner. The interface solves the governing equations in the frequency domain. This interface is particularly suited for large system simulations. The main assumption for the decomposition is that the acoustic wavelength is much larger than the thickness of the viscous and thermal boundary layers.

ANALYTICAL PORT MODES FOR LINEARIZED POTENTIAL FLOW

Standard analytical port mode options have been added to the **Port** condition in the **Linearized Potential Flow, Frequency Domain** interface. Both **Annular** and **Circular** port modes can be selected. The modes assume a uniform background flow in the port surface. These built-in options simplify model setup in many cases; in particular, they

are essential when performing a modal expansion of measured sources, as well as when computing modal transmission loss of flow ducts.

ANISOTROPIC POROACOUSTICS

A new **Anisotropic Poroacoustics** material model has been added to the **Pressure Acoustics, Frequency Domain** interface. The new model defines a porous domain (equivalent fluid model) with a porous material that has anisotropic properties. Anisotropic properties can be defined for the relevant poroacoustic properties, namely the flow resistivity, the tortuosity factor, and the viscous characteristic length.

IMPROVEMENTS TO THE PARTIAL FRACTION FIT FUNCTIONALITY

The **Partial Fraction Fit** function is available in the COMSOL Multiphysics base package. The function is essential in acoustics for modeling impedance conditions as well as poroacoustic properties in the time domain. The **Partial Fraction Fit** function has several improvements:

- Explicit handling of data at zero frequency (DC), if included in the input data
- Exact computation of residues
- New advanced stop conditions based on **Iterations** and **Tolerance or iterations**.
- New advanced option to automatically detect and remove Froissart doublets (unstable poles)

These new options allow for more robust fitting of noisy data as well as better control of the fitting.

INTERIOR IMPEDANCE MODELS IN PRESSURE ACOUSTICS AND FEM–BEM MULTIPHYSICS COUPLING

A series of new interior impedance models (transfer impedance models) have been added to pressure acoustics in the frequency domain. This is available for both the **Interior Impedance** condition and the **Pair Impedance** condition in the **Pressure Acoustics, Frequency Domain** interface as well as for the **Impedance** condition that can be added between domains modeled with FEM and BEM as a subfeature to the **Acoustic BEM-FEM Boundary** multiphysics coupling. The new options include **Thin plate**, **Membrane**, **Porous mass layer**, and **Perforated plate** models through a transfer impedance.

IMPERFECT BONDING OPTION FOR POROELASTIC-SOLID BOUNDARY MULTIPHYSICS COUPLING

A new **Imperfect bonding** option exists when coupling poroelastic and solid domains. The existing **Perfect bonding** option represents a scenario where the solid is glued or bonded to porous material. The second new **Imperfect bonding** option represents a case with only coupling in the normal direction; this can represent a case where the porous material is placed on or next to the solid.

ELECTROMECHANICS MULTIPHYSICS COUPLING FOR MEMBRANES AND SHELLS

A new **Electromechanics, Boundary** multiphysics coupling has been added for modeling electromechanics when coupling the **Electrostatics** interface to the **Membrane** or the **Shell** interfaces. The new multiphysics coupling simplifies setting up models of, for example, microphones. When the new multiphysics coupling is used in combination with acoustics and structural mechanics for frequency-domain analysis, an efficient iterative solver suggestion is now also generated.

PHYSICS-CONTROLLED AWE EXPRESSIONS FOR PRESSURE ACOUSTICS, FREQUENCY DOMAIN

A new **AWE Expression** feature is used to define the asymptotic waveform expansion (AWE) expression or metric used for the **Adaptive Frequency Sweep** study type. The feature can define expressions based on a user-defined global expression, values defined at points, or the scattering coefficients evaluated at ports.

EXTENDED PORT OPTIONS FOR PRESSURE ACOUSTICS

The **Port** condition in the **Pressure Acoustics, Frequency Domain** interface has a new built-in **Annular** port option for annular port geometries. Moreover, for both the **Circular** and **Annular** port options, **Sine** and **Cosine** options are available to define the setting in the **Azimuthal angle dependency** list.This option is used to add the two orthogonal modes that exist for a given azimuthal mode number.

POWER VARIABLES FOR PML AND EXTERIOR FIELD

Two built-in variables exist for evaluating the total radiated power through a perfectly matched layer (PML) and the radiated power associated with the **Exterior Field Calculation** feature.

- For the PML, the variable `acpr.P_pml` gives the total radiated power from the physical domain to the PML. The variable is useful for identifying the performance of the PML.

- The variable `acpr.efc1.P_rad` gives the radiated power at the **Exterior Field Calculation** boundary, including symmetries used in the feature.

SOUND SOFT BOUNDARY (ISENTROPIC) FOR LINEARIZED EULER

A sound soft boundary condition has been added to the **Linearized Euler** interfaces. This isentropic condition sets the pressure and the density to zero on the selected boundaries.

RAY ACOUSTICS

Several performance and usability improvements have been made to the **Ray Acoustics** interface. They include:

- To simplify modeling of most room acoustic problems, the default value of the **Count reflections** setting has changed to true, and the default value for the **Intensity computation** has changed to **Compute power**. Both options are necessary for impulse response analysis.
- The **Receiver** feature in the **Ray Acoustics** interface now automatically creates an associated receiver dataset when solving the model. The dataset can then be used for an **Impulse Response** plot.
- The performance of the **Mixed diffuse and specular reflection** condition has been improved. This is the most commonly used condition for room acoustics simulations. As an example, the computation time of a large 50,000 ray model is reduced by about 20 to 25%.
- It is now possible to only store data on the receiver. This option can highly reduce the size of solved ray acoustics models when stored after solving.
- The radiation balloon (or 3D radiation pattern) of a transducer simulation now has a dedicated export functionality for radiation patterns (under the **Export** node.). The data can now easily be used as input to a **Ray Acoustics** interface simulation in the **Source with Directivity** feature.

STABILIZATION IMPROVED FOR LINEARIZED NAVIER–STOKES

The stabilization for the **Linearized Navier–Stokes** interfaces has been improved to ensure that it is not overly diffusive. This effect could happen in structures with small dimensions and for certain flow conditions. This updated reflects recent research in stabilization methods for LES flow modeling.

PERFORMANCE IMPROVEMENTS

- Improved performance during linear iterations for coupled FEM-BEM models.

- The performance when using the frequency-dependent **Impedance** boundary condition in the **Pressure Acoustics, Time Explicit** interface has been improved.
- Models based on the discontinuous Galerkin (dG-FEM) method have improved memory balance when solved on a cluster architecture.

IMPORTANT ENHANCEMENTS AND IMPROVEMENTS

- A new **Unwrap phase** option exists in plots where a phase expression is plotted.
- A predefined **Gaussian** modulated option exists when defining the **Waveform** analytic function. This signal type is useful in many acoustical scenarios.
- A new nonlinear eigenvalue solver is available for eigenfrequency studies. The solver simplifies solving problems that are nonlinear in the eigenvalue, that is, the frequency. This is relevant for many impedance conditions and porous materials with damping.
- It is now possible to simplify results analysis using filtering and ordering of eigenvalues, eigenfrequencies, and boundary modes.
- Weak constraints on interior boundaries have been improved for the **Linearized Navier–Stokes** and **Thermoviscous Acoustics** interfaces.
- New results variables `phys.v_real` and `phys.v_imag` for the **Linearized Navier–Stokes** and **Thermoviscous Acoustics** interfaces. You can now plot the real and imaginary (out-of-phase) part of the velocity magnitude.
- An option to use a Nitsche formulation has been added for constraints for the **Thermoviscous Acoustic-Structure Multiphysics** coupling, when coupling it to a solid mechanics interface.
- Impedance boundary conditions in the acoustics interfaces that use the boundary area now include the capability to use symmetries to set up that area in 3D. When possible, the area multiplication factor can be calculated automatically.

ENHANCED PIEZOELECTRIC MATERIAL OPTIONS FOR ACOUSTICS APPLICATION

The **Piezoelectric** material library has been expanded to include materials from PI Ceramic GmbH.

New and Updated Models in 6.3

NEW MODELS

The following models are new in version 6.3:

- Flow Duct
- Flow Duct — With Boundary Mode Analysis
- Flow Duct — Modes with Impedance Condition
- Sound Radiation from a Circular Duct with Flow
- Full Ear Hearing Aid
- Dome Tweeter with Composite Diaphragm — Eigenfrequency Analysis
- Dome Tweeter with Composite Diaphragm — Frequency-Domain Response
- Anisotropic Porous Absorber
- Porous Absorber with Local and Extended Reacting Approximations for Time-Domain Modeling
- Diesel Particulate Filter Analysis Using an Acoustic Transfer Matrix

UPDATED MODELS

The following models have been updated in version 6.3:

- Small Concert Hall Acoustics
- Chamber Music Hall
- Loudspeaker Driver in a Vented Enclosure
- Open Pipe
- Absorptive Muffler
- Generic 711 Coupler — An Occluded Ear-Canal Simulator

NEW MODELS IN THE APPLICATION GALLERY ON COMSOL.COM

In addition, several models have been added to the Application Gallery found at https://www.comsol.com/models/acoustics-module. The new models are:

- Acoustic Streaming Induced by a Focused Ultrasound Beam
- Conference Speaker Virtual Test Setup
- Conference Speaker System
- Input Impedance of a Tube and Coupler Measurement Setup: Time Domain MOR using Partial Fraction Fit
- Whistling Potential of an In-Duct Orifice in the Presence of Flow
- Acoustic Liner with a Grazing Background Flow
- Car Cabin Acoustics — Iterative Solver Suggestion for Cubic Elements
- Midwoofer Harmonic Distortion Analysis with KLIPPEL Measurements

Backward Compatibility with Version 6.1

- In the Acoustic Diffusion Equation interface, the **Long room** option has been removed from the **Mean free path model** in the **Room** feature. To retrieve the old behavior for the feature `room1` in the interface `ade`, component `comp1`, add the following line after selecting the **Long room** option:

  ```
  model.component("comp1").physics("ade").feature("room1").
  set("lambda", "sqrt(ade.room1.S/(4*pi))");
  ```

- The Linearized Potential Flow interfaces have a new default tag for the default material model. If the default feature has been edited in a Java file, the following line should be added for the physics `lpff` in the component `comp1` to obtain the old behavior:

  ```
  model.component("comp1").physics("lpff").feature().
  create("aem1","LinearizedPotentialFlowModel").selection().all();
  ```

  This line should be added directly after the line where the physics interface is created.

- In the **Prescribed Displacement** feature in the Poroelastic Waves interface, notice that the **General notation** option has been removed.
- The Poroelastic Waves interface has been updated to improve the overview of the features and their functionality. A new feature, **Impervious Layer**, the homogeneous

Neumann condition for the pressure, is now added by default. This feature corresponds to a sound hard wall or a no-flow condition.

The following backward compatibility issues can be seen in certain, less common models:

- The override rules have been improved for the **Porous**, **Free** and **Septum Boundary Load** boundary conditions, which only appear if you open a model created in an earlier version of COMSOL Multiphysics. The **Porous Free** node has been renamed **Free, Sound Soft**.
- Most predefined pair conditions have been removed from the Poroelastic Waves interface. Add the corresponding nonpair boundary conditions instead if needed.
- The **Symmetry** condition is now only available on exterior boundaries. To get a roller condition for the porous matrix, add the **Roller** condition.
- The symmetry settings have been removed from the **Roller** condition. Use a **Symmetry** condition instead.

- Java backward compatibility for the Poroelastic Waves interface:
  - The default features have changed. To retrieve the old behavior, add the following line after creating the physics interface:

    ```
    model.component("comp1").physics("pelw").feature().
    create("pfree", "PorousFree", 2);
    ```

    Then, set the selection to the boundaries where the Porous, free condition should be active, for example, for boundaries 1 and 2:

    ```
    model.component("comp1").physics("pelw").feature("pfree").
    selection().set(new int[]{1, 2});
    ```
  - The default material model in the Poroelastic Waves interface has changed. To retrieve the old behavior, add the following line when the interface has been created and replace all references to `"pelm1"` by `"pelm_def"`:

    ```
    model.component("comp1").physics("pelw").
    create("pelm_def", "PoroelasticWavesMaterial", 3);
    ```

Battery Design Module

New Functionality in Version 6.3

GENERAL

- The new **Two Electrodes** model option in the **Lumped Battery** interface defines the degrees of conversion of the electrodes (which equals the state of lithiation for the case of a lithium-ion battery) and the corresponding half-cell potentials independently for the positive and negative electrode. This new option also allows for defining the individual kinetic and diffusion voltage losses in the positive and negative electrodes. Using the **Particle diffusion** option for the concentration overpotentials, the Two Electrodes model is mathematically identical to a single-particle model (SPM).
- A new **Model Wizard** entry **Lumped Battery, Two Electrodes** has been added, using the **Two Electrodes** option as default.
- In the **Lithium-Ion Battery** and **Battery with Binary Electrolyte** interfaces, the new **Thin Porous Electrode** node can be used to define electrodes using a single-particle approach, where the electrode transport processes in the thickness dimension of the electrode are assumed to be negligible. The new node provides increased flexibility when defining models — for instance, allowing for combining concentrated solution theory in the separator with single-particle electrodes, also known as an SPMe (single particle model with electrolyte) model in literature. The **SOC and Initial Charge Distribution** global node has been updated to support selections of **Thin Porous Electrode** nodes.
- The above new features replace the functionality of the old Single Particle Battery interface, which is now made obsolete, albeit old models can still be run when opened in version 6.3.
- Under **Advanced Settings** in the **Lithium-Ion Battery** and **Battery with Binary Electrolyte** interfaces, it is now possible to enable the **Use logarithmic formulation for the electrolyte salt concentration** dependent variable. The new setting inherently avoids issues related to negative values for the salt concentration in the electrolyte phase. The new setting can be used to improve convergence of models operating at high charge or discharge rates resulting in local depletion of electrolyte salt in the electrodes. A common use case is to enable the feature when running parameter estimation or surrogate model training in order to improve convergence for the whole parameter space.

- Under **Advanced Settings** in the **Lithium-Ion Battery** and **Battery with Binary Electrolyte** interfaces, it is now possible to choose between the conservative and nonconservative form for the electrolyte material balance formulation. This setting impacts how convective contributions and time derivatives of the electrolyte volume fraction are treated in the material balance. The default value is **Nonconservative**, but old models will be migrated to use the **Conservative** option in order to preserve the handling of the time derivative of the electrolyte volume fraction. The difference between the two equation forms are further discussed in the theory chapters for the electrochemistry interfaces in the documentation.
- In the **Lumped Battery** interface and whenever the **SOC and Initial Charge Distribution** node is active in the **Lithium-Ion** or **Battery with Binary Electrolyte** interfaces, it is now possible to specify the total current using a **C-rate multiple** option. This new option, which applies a total current equal to the 1C current times the C-rate multiple, is available in all electrode features where a total current condition can be specified.
- The integration frame has been changed from **Spatial** to **Material** for inventory variables used by **SOC and Initial Charge Distribution** node.
- **Electric Conductivities** values have been updated for most electrode materials in the **Battery Design Material Library**. The present conductivities now refer to the bulk conductivities of the pure active electrode material (that is, not the effective conductivities of a porous conductive binder or electrode material mixture).
- A new **Periodic Condition** boundary node has been added to the **Lithium-Ion Battery** and **Battery with Binary Electrolyte** interfaces.
- A new option to specify a **Thermoneutral voltage** value in the **Lumped Battery** and **Battery Pack** interfaces has been added.
- The new **Explicit Event List** node in the **Events** interface defines multiple explicit events using a list of times and corresponding variable values for a common state variable. The input is in the form of a table, which in turn allows for defining the list of events from a text file. An explicit event is generally used in the **Events** interface to momentarily stop the time-dependent solver, redefine the value of one or several state variables at a given, explicit, time, and then restart the solver. Defining load changes in a battery model using events instead of time-dependent continuous functions often brings considerable performance gains with respect to simulation times, since the continuous transition between the current load steps does not have to be resolved in time.
- A new **Concentrated Electrolyte Transport** interface is now available from the **Model Wizard**. This interface can be used to model transport in any electrolyte solution with an arbitrary number of charged and uncharged species. The interface is based

on concentrated electrolyte theory, where the transport equations are defined using binary Maxwell–Stefan diffusivity coefficients and assume local electroneutrality. In contrast to the Nernst–Planck equations, for instance, the concentrated solution theory does not assume all modeled species to be diluted in a solvent of constant concentration. Typical electrolyte systems that can be modeled using the new interface include ionic liquids, salt melts, and highly concentrated solutions, featuring nonnegligible concentration gradients of the charge-carrying species.

- A **Model Wizard** entry has been added for the **Tertiary Current Distribution, Poisson** charge conservation option.
- The default solver setting suggestions have been updated to set the **Pivoting perturbation** to 1e-13 for the PARDISO solver in the Electrochemistry interfaces.
- The frame of integration has been changed from the **Material** to the **Spatial** frame in most domain transport equations in the electrochemistry interfaces defining electrolyte transport, in conjunction with a general overview of conversion factors used during frame splitting. The changes improve mass conservation in domains subject to material strains and only have an impact on the results when used in combination with **Moving Mesh** or when **Include Geometric Nonlinearity** is enabled in a study step.
- The **Nonconservative** form of the material balance equation in the **Tertiary Current Distribution** interfaces in porous domains has been corrected. The porosity factor is now placed outside of the time derivative operator.

THE CHEMICAL SPECIES TRANSPORT INTERFACES

The plot defaults have been changed in the following respects:

- The **Prism** color table is now used by default instead of the **Rainbow** color table.
- Mass fluxes are visualized using arrows instead of streamlines in 2D and 2D axially symmetric geometries.
- Mass flux streamlines use ribbons instead of lines in 3D geometries.

Two new results templates types are now available: arrayed concentration plots and mole balances.

- In order to visualize multiple concentrations at a time, the arrayed concentration plots include up to four different concentrations in the same plot group. One or several plot groups, as needed to include all species, are available.
- Mole balance evaluation groups are available for all species solved for. The evaluation groups include integration of fluxes over inlets, outlets, and all external boundaries, as well as integration of domain source terms.

New and Updated Models in Version 6.3

- *Lithium Plating and Stripping* (li_plating)
- *Copper Current Collector Dissolution* (cu_current_collector_dissolution)
- *LMO Decomposition* (lmo_decomposition)
- *Surrogate Model of a Battery Test Cycle* (lib_test_cycle_surrogate)
- *Heterogeneous Lithium-Ion Battery* (heterogeneous_lib)
- *Single-Particle Modeling of Lithium-Ion Batteries* (lib_single_particle)
- *Surrogate Model Training of a Battery Rate Capability Model* (lib_rate_capability_surrogate)
- Changed the name of the capacity_fade model to sei_formation
- Updated the following models to use the new C-rate multiple option: li_battery_multiple_materials_1d, li_battery_solid_electrolyte, li_battery_thermal_2d_axi, lumped_li_battery_capacity_loss, and pouch_cell_utilization, sei_formation, sei_formation_seed

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

New Functionality in Version 6.3

- 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 6.3 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/).
- When importing CAD assemblies saved in the file formats of the CATIA® V5, Inventor®, NX®, PTC Creo Parametric™, PTC Pro/ENGINEER®, and SOLIDWORKS® software, it is now possible to preview the assembly tree where you can choose the components to import from the assembly.
- The settings for the Detect Interferences tool now include a **Gap tolerance** that makes it easier to detect gaps between objects. The following new checkboxes are available to select the type of interferences that are visualized in the Graphics: **Show intersections**, **Show touches**, **Show gaps**, **Show containments**.
- The CAD file import functionality now also supports the import of AutoCAD®, CATIA® V5, and Inventor® files on a supported Linux® operating system with an Intel® 64-bit processor.

New Functionality in the Design Module in Version 6.3

- The Chamfer and Fillet operations (in 3D) now have a **Group by continuous tangent** option that automatically extends the selection to tangent edges.

- The capabilities of the Fillet operation in 3D have been extended to generate **Constant width** and **Variable radius** fillets in addition to the previously available **Constant radius** option.
- The new feature **Project to Faces** is now available to create the projection of edges onto faces. Several projection methods are available: perpendicular to the target faces, in the direction of a vector or an edge, and perspective projection when choosing a point that represents the eye position.

Backward Compatibility with Version 6.2 and Earlier

- For the Fillet operation in 3D, the **Propagate to tangent edges** option now propagates the fillet to fewer edges than before, in certain cases. To get the old behavior, you can add more edges to the input edge selection.
- The preference setting **Geometry** > **3D Geometry Representation** > **Use Design Module Boolean operations in new geometries** is now selected by default.

New Functionality in LiveLink™ for AutoCAD® in Version 6.3

The LiveLink™ interface now supports AutoCAD® 2025.

New Functionality in LiveLink™ for Inventor® in Version 6.3

The LiveLink™ interface now supports Inventor® 2025 for synchronizing the geometry when running the Inventor® and COMSOL Multiphysics® side by side.

New Functionality in LiveLink™ for PTC Creo Parametric™ in Version 6.3

- The LiveLink™ interface now supports PTC Creo Parametric™ 11.0.
- The LiveLink™ interface now keeps track of the revision and version of synchronized CAD designs that are under version control in the Windchill PLM software. The revision and version of the synchronized geometry are displayed in the settings for the LiveLink feature and are saved in the model.

New Functionality in LiveLink™ for Revit® in Version 6.3

The LiveLink™ interface now supports Revit® 2025.

CFD Module

New Functionality in Version 6.3

- The **Turbulence model type** RANS in the **Turbulence** section of the **Settings** window for the **Single-Phase Flow** and **High Mach Number Flow** interfaces has been split into **RANS-EVM** (eddy-viscosity model) and **RANS-RSM** (Reynolds-stress model). All previously available **RANS** models are now available when **RANS-EVM** is selected.
- Two new **Turbulent Flow** interfaces have been added under the **Single-Phase Flow** section in the **Fluid Flow** branch: **Turbulent Flow, Wilcox R-w** and **Turbulent Flow, SSG-LRR**.
- A new **Mass flow** option has been added in the **Boundary Condition** section of the **Settings** window for the **Outlet** feature in the **Single-Phase Flow** interfaces.
- The **Vacuum Pump** feature in the **Single-Phase Flow** interfaces has been updated to facilitate the use of vacuum pump performance curves.
- New variables for the traction at walls and interior walls are available when **Wall treatment** is set to **Wall functions** or **Automatic** in the **Turbulence** section of the **Settings** window for the **RANS-EVM**, **LES**, and **DES** interfaces. The traction variables are used by default in the **Fluid–Structure Interaction** multiphysics coupling nodes.
- A new **Limit small time steps effect on stabilization time scale** option is available in the **Consistent Stabilization** section in the **Settings** window for the **Bubbly Flow**, **Euler-Euler**, and **Phase Transport** interfaces. It can be used to avoid the loss of stabilization during initialization and for small time steps.
- Two new domain features, the **Fluid** feature and the **Porous Medium** feature, are available for the **Phase Transport** interfaces. The new **Porous Medium** feature has subnodes to define the phase properties and the properties of the porous matrix. The new domain features replace the previously available **Phase and Transport Properties** and **Phase and Porous Media Transport Properties** features. When opening a model created in a previous version, the previous features are kept in the model.
- A new **Boundary Mass Source** boundary condition feature has been added to the Phase Transport interfaces. This feature accounts for the consumption or production of the different phases due to reactions or other physical processes at the boundary. When the **Phase Transport in Porous Media** interface is coupled to a **Darcy's Law** interface using a **Multiphase Flow in Porous Media** multiphysics coupling node, the net mass transfer is automatically accounted for in the flow field.

- A new **Power law** option has been added to specify predefined relative permeabilities for the different phases in the **Porous Medium** feature for the **Phase Transport in Porous Media** interface. This option allows for the easier implementation of relative permeabilities based on power law expressions.
- A new **Include shear-induce migration** option has been added to the Mixture Model multiphysics coupling. This option allows, for example, for the simulation of the phenomenon that particles in a pressure-driven pipe or channel mixture flow will migrate toward the center of the flow channel.
- The **Krieger type** mixture viscosity model in the Mixture Model multiphysics coupling has been extended and is now also available for mixtures with more than one dispersed phase.
- The **Rotating Machinery, Nonisothermal Flow** multiphysics interface has been added under the **Nonisothermal Flow** section in the **Fluid Flow** branch.
- Two new **Turbulent Flow** interfaces have been added under the **High Mach Number Flow** section in the **Fluid Flow** branch: **High Mach Number Flow, Wilcox R-w** and **High Mach Number Flow, SSG-LRR**.
- A new option for **Include kinetic energy** has been added in the **Physical Model** section of the **Settings** window for the **High Mach Number Flow** interfaces.
- A new **Subsonic** option has been added in the **Flow Condition** section of the **Settings** window for the **Inlet** condition in the **High Mach Number Flow** interfaces.
- The **Suppress backflow** option is now available when **Boundary condition** is set to **Normal stress** in the **Flow Properties** section and **Flow condition** is set to **Subsonic** in the **Flow Condition** section of the **Settings** window for the **High Mach Number Flow** interfaces.

New Models in Version 6.3

SHOCK DIAMONDS FROM A RECTANGULAR NOZZLE

This model uses the High Mach Number Flow, k-ε interface to simulate turbulent compressible flow through a rectangular nozzle and analyzes the attenuation of the asymmetric shock-diamond structure in the emerging supersonic jet.

TWO-PHASE FLOW MODELING OF A DENSE SUSPENSION

This model has been reformulated to use the new shear-induced migration functionality in the **Mixture Model** multiphysics coupling node.

Backward Compatibility with Version 6.1 and Earlier

NEW DEFAULT MAXIMUM STEP CONSTRAINT AND MAXIMUM STEP SETTINGS IN THE TIME-DEPENDENT SOLVER NODE FOR THE LARGE EDDY SIMULATION AND DETACHED EDDY SIMULATION INTERFACES

LES and DES models using default solver settings will get the **Maximum step constraint** set to **Expression** with a **Maximum step** set to `spf.dt_CFL` in the **Time-Dependent Solver** node. This constraint may be more restrictive than the **Automatic** option for the **Maximum step constraint** obtained prior to version 6.2, and as a result, some models will take a longer time to solve. The **Maximum step constraint** option can be reset to **Automatic** to get the old behavior back.

NEW DEFINITION OF THE MEAN FREE PATH IN THE THIN-FILM FLOW INTERFACES

The definition of the mean free path when the **Fluid type** is set to **Gas (modified Reynolds equation)** in the Thin-Film Flow interfaces has been changed to obtain consistency between different expressions for the **Film flow** model. The results using the **Rarefied-total accommodation** option in the **Film flow** model are more accurate than in previous versions.

Backward Compatibility with Version 6.0 and Earlier

RENAMED THIN-FILM FLOW AND PHASE FIELD, THIN-FILM FLOW INTERFACES IN COMPILED JAVA FILES

Compiled Java files using the Thin-Film Flow, Shell (tffs) and Thin-Film Flow, Edge (tffs) interfaces need to be updated manually to run in version 6.1. The identifier for these interfaces has been changed to tff. In order to run Java files, you need to replace all instances of `tffs` with `tff`.

Similarly, the identifier for the Phase Field, Thin-Film Flow, Shell (pftffs), and Phase Field, Thin-Film Flow, Edge (pftffs) needs to be manually changed from `pftffs` to `pftff`.

NON-NEWTONIAN INELASTIC MATERIAL MODELS

Non-Newtonian inelastic material models that were added in older versions will not be able to define the dynamic viscosity in the **Basic (def)** material model.

THE LES SMAGORINSKY INTERFACE

The modeled subgrid stress term in the LES Smagorinsky interface has been recast on symmetric form, and the filtering is now instead included in the expression for the turbulent viscosity. The **Smagorinsky parameter** now enters the turbulent viscosity expression as squared, and its default value has therefore been changed to 0.1. Slight differences in the results may occur for some models.

Chemical Reaction Engineering Module

New Functionality in Version 6.3

INTERFACES FOR PRECIPITATION AND CRYSTALLIZATION

The new **Size-Based Population Balance** interface can be used to model the population balance of particles in a fluid. Examples include crystals in a liquid, gas bubbles in a liquid, and droplets in a gas. The interface solves for the discretized size distribution of particles, with support for nucleation and growth of the particles.

The new **Precipitation and Crystallization in Fluid Flow** multiphysics interface can be used to simulate precipitation of a chemical species in a fluid solution and solve for the evolution of the size distribution of the precipitate. The interface includes the **Size-Based Population Balance** interface and can be used to model precipitation in ideal reactors and space-dependent models.

The new interfaces are available in the **Precipitation and Crystallization** branch under **Chemical Species Transport** when selecting physics interfaces.

The new interfaces superseded the Population Balance Equation Discretization add-in available in the previous release. The add-in is no longer included.

THE CHEMISTRY INTERFACE SUPPORTS TURBULENT FLOW

The **Eddy-Dissipation** turbulent-reaction model for fluid flow reactions in the turbulent regime is now available in the **Reaction** node of the **Chemistry** interface. The functionality requires that the user is also licensed to use turbulence models for fluid flow.

IMPROVED FUNCTIONALITY FOR SPACE-DEPENDENT REACTION SYSTEMS

The **Generate Space-Dependent Model** node in the **Reaction Engineering** interface has been improved in the following manner:

- Multiphysics coupling nodes are now automatically added when creating a space-dependent model. When choosing, for example, to create a **Transport of Diluted Species** and a **Laminar Flow** interface, a **Reacting Flow, Diluted Species** coupling node is also added.

- Predefined multiphysics interfaces can now also be created. The same multiphysics interfaces as available in the **Model Wizard** and the **Add Physics** window can be created from the **Generate Space-Dependent Model** node. You can, for example, select to create a **Nonisothermal Reacting Flow** interface. As in previous versions, it is also possible to manually combine interfaces for fluid flow, mass, and heat transport.
- Space-dependent models with turbulent flow can now be created, provided that an additional license for turbulent fluid flow is available. This has been made possible by the inclusion of the **Eddy-Dissipation** model in the **Chemistry** interface.
- It is now possible to select to only create a **Chemistry** interface and no interfaces for fluid flow, heat transfer, or mass transfer. This functionality can, for example, be used to export an updated reaction mechanism to an existing model.

UPDATED PLOT DEFAULTS

The plot defaults for mass transport interfaces have been changed in the following respects:

- The **Prism** color table is now used by default instead of the **Rainbow** color table.
- Mass fluxes are visualized using arrows instead of streamlines in 2D and 2D axially symmetric geometries.
- Mass flux streamlines use ribbons instead of lines in 3D geometries.

NEW RESULTS TEMPLATES

Two new results templates types are now available for mass transport interfaces, arrayed concentration plots, and mole balances.

- In order to visualize multiple concentrations at a time, the arrayed concentration plots include up to four different concentrations in the same plot group. One or several plot groups, as needed to include all species, are available.
- Mole balance evaluation groups are available for all species solved for. The evaluation groups include integration of fluxes over inlets, outlets, and all external boundaries, as well as integration of domain source terms.

IMPROVED SOLVER DEFAULTS FOR REACTING FLOW

For reacting flow models using a segregated solver, one iteration instead of three is now used for the group with concentration (or mass fraction) variables. If the **Eddy-dissipation** turbulent-reaction model is used, three iterations is still used.

When using a turbulence model with wall functions, the wall functions are now only solved for on walls with non-zero mass flux, that is on boundaries where the concentration or mass flux is prescribed.

NEW DEFAULT SOLVER FOR PACKED BED MODELS

The default iterative solver for large models, including **Packed Bed** features, has been changed. A geometric multigrid solver is now used for the concentration variables. This significantly reduces the solution time for these types of models.

NEW NONISOTHERMAL REACTING FLOW, BRINKMAN EQUATIONS MULTIPHYSICS INTERFACE

A new **Nonisothermal Reacting Flow, Brinkman Equations** multiphysics interface is now available in the **Nonisothermal Reacting Flow** branch under **Chemical Species Transport** when selecting physics interfaces. The new interface can be used to simulate single-phase flow coupled to heat transfer, species transport, and reactions in porous media.

VAPOR–LIQUID EQUILIBRIUM MULTIPHYSICS INTERFACES

The **Vapor Flow** multiphysics interfaces introduced in the previous version are now renamed **Vapor-Liquid Equilibrium**, and are available in a corresponding branch under **Chemical Species Transport** when selecting physics interfaces.

TRANSPORT PROPERTIES NODES RENAMED FLUID

The **Transport Properties** domain feature in physics interfaces for mass transport is now named **Fluid**. The node is applicable in domains with free flow and can be combined with **Porous Medium** or **Solid** nodes to mass transport in corresponding media.

IMPROVEMENTS TO THE GENERATE MATERIAL WIZARD

It is now possible to specify a logarithmic range of interpolation points for the pressure when using the **Generate Material Wizard** to create a **Material** from a **Thermodynamic System**. Regularization of function arguments has been added to prevent evaluation of zero or negative values for the temperature and absolute pressure.

IMPROVED SETTINGS FOR THE PELLET SUBNODE OF POROUS MATERIAL

When giving the pellet size, defined parameters and variables can be accessed by pressing Ctrl+Space. The input is also checked to avoid an empty or a zero value. The definition of the effective diameter is also displayed in the setting window.

INDICATION OF SOLVENT SPECIES AND SPECIES WITH CONSTANT CONCENTRATION

It is now easier to identify solvent species and species at constant concentration in the **Chemistry** and **Reaction Engineering** interfaces. When the **Type** is set to **Solvent** in a species node, the string "(Solvent)" is appended. The label of a water solvent species now typically looks like: **Species: H2O (Solvent)**.

When **Keep concentration/activity constant** is selected in a species node, the string "(Constant)" is appended to the species label. The label of a carbon dioxide species at constant concentration now typically looks like: **Species: CO2 (Constant)**.

New Models in 6.3

ANALYSIS OF NOX AND AMMONIA CONVERSION KINETICS IN A DUAL-BED PLUG-FLOW REACTOR

This example investigates the catalytic NOx and ammonia conversion kinetics in the exhaust system of a heavy-duty diesel truck. NOx is selectively reduced by ammonia in the first selective catalytic reduction (SCR) bed, and downstream of this monolith, the remaining ammonia is oxidized in an ammonia slip catalyst (ASC). The reactions take place as the exhaust gases pass through the channels of the monolithic catalysts placed in series. The study is focused on how temperature, gas composition, and flow rate affect the conversion. The model is set up using the plug flow reactor in the Reaction Engineering interface.

NOX AND AMMONIA CONVERSION IN A DUAL-BED MONOLITHIC REACTOR

NO reduction occurs as flue gases pass through the channels of a monolithic reactor in the exhaust system of a heavy-duty diesel truck. This example investigates a 2D axisymmetric model geometry of the reactor, and the model includes mass transport, heat transfer, and fluid flow.

LIQUID–LIQUID EXTRACTION

Liquid–liquid extraction is a process used to separate or transfer species between two immiscible liquids. Transfer of species from one phase to the other is driven by a difference in relative solubility. In this model, a water-filled extraction column is studied. Oil droplets containing a solute species are injected at the bottom of the column and rise up due to buoyancy. As the oil droplets rise, the solute species is transferred into the aqueous phase. Water is injected at the top of the column. The

column is fitted with a number of alternating horizontal discs in order to increase the residence time of the oil droplets.

BOAT REACTOR FOR LOW-PRESSURE CHEMICAL VAPOR DEPOSITION

Chemical vapor deposition (CVD) is an important step in the process of manufacturing microchips. One common application is the deposition of silicon on wafers in low-pressure reactors to obtain uniform deposition thicknesses. This example models the coupled reaction kinetics, fluid flow, and mass transport in a low-pressure boat reactor. The simulation investigates how the silicon deposition rate and thickness varies in the reactor with operating conditions such as temperature and pressure.

NEW MODELS IN THE APPLICATION GALLERY ON COMSOL.COM

In addition, several models have been added to the Application Gallery found at https://www.comsol.com/models/chemical-reaction-engineering-module. The new models are:

- Chemical Vapor Deposition of TiN on Machining Tool Inserts
- Convective Evaporation of a Water–Acetone Droplet
- Metal Hydride Hydrogen Tank During Charging
- Reduction of Iron Ore Pellets in a Packed Bed Reactor

Updated Models in 6.3

PRECIPITATION OF BARIUM SULPHATE

The model has been rebuilt using the new **Precipitation** and **Crystallization in Fluid Flow** multiphysics interfaces.

PYROLYSIS OF WOOD WITH TIME-INTEGRATED OBJECTIVE

The model has been updated to use a time integrated objective together with the SNOPT solver. The time to solve the model has been reduced by a factor of six compared to the previous version. A license for the Optimization Module is now required to solve the model.

A MULTISCALE 3D PACKED BED REACTOR

The model now uses a geometric multigrid solver which reduces the solution time by a factor of four compared to the previous version.

CROSS-FLOW MASS TRANSFER IN A THIN DOMAIN

The model has been updated to use the Transport of Concentrated Species interface. It now also includes an analysis of the mass conservation.

BEER FERMENTATION

Kinetics of gaseous CO2 updated. Space-dependent geometry has been removed.

MODELS WITH IMPROVED RESULTS PRESENTATION

Physically realistic results presentations (for example, using material appearance nodes on solid structures) have been added to the following models:

- A Multiscale 3D Packed Bed Reactor
- Carbon Deposition Heterogeneous Catalysis
- Dissociation in a Tubular Reactor
- Engine Coolant Properties
- Fine Chemical Production in a Plate Reactor
- Polymerization in a Multijet Tubular Reactor

Composite Materials Module

New Functionality in Version 6.3

PARTS FOR HOMOGENIZATION OF MICROSTRUCTURES

In the part libraries, a new microstructure geometry part for bidirectional spread-tow fiber composite is added under the **Unit Cells and RVEs** folder of the **COMSOL Multiphysics** branch.

A new add-in to generate geometry parts for random particulate composites has been added under the **COMSOL Multiphysics** branch of the Add-in Libraries.

COMPOSITE MATERIAL LIBRARY

A new **Composites** built-in folder is now available in the Material Library. The new material library contains three groups of materials: fiber constituents, matrix constituents, and laminae.

LAYERED SHELL IMPROVEMENTS

The following improvements have been made:

- The kinematics is improved to account for large deformations and rotations.
- The **Creep**, **Viscoplasticity**, and **Polymer Viscoplasticity** subnodes are now available in the **Hyperelastic Material** node.
- A new coordinate system called **Layer Local System** has been added to several subnodes.
- A new option **Resultant** has been added to the **Point Load** node.

IMPROVEMENTS TO THE SHELL INTERFACE

The following improvements have been made:

- The **Creep**, **Viscoplasticity**, and **Polymer Viscoplasticity** subnodes are available in the **Hyperelastic Material, Layered** node. If you have a license for the Composite Materials Module, the material model can be used in multilayered shells, and the individual layers can have different material properties.
- The stiffness and inertia matrices are corrected in the **Linear Elastic Material, Layered** node when the **Merge middle layers** checkbox is selected in the **Layered Material Link** or **Layered Material Stack** features.

MISCELLANEOUS

- A new **Advanced** section has been added to the **Layered Shell–Structure Cladding** multiphysics coupling. The constraint can be set on the layered shell boundaries or on the structural boundaries. A manual search method is added as an alternative to the existing closest point method for the mapping operator.
- The connection and automatic tolerance definitions has been improved for the **Layered Shell–Structure Transition** multiphysics coupling.
- Solver suggestions for multiphysics problems have been improved.

New Models in Version 6.3

DOME TWEETER WITH COMPOSITE DIAPHRAGM — EIGENFREQUENCY ANALYSIS

In this example, a composite material is used to control the diaphragm resonance frequencies in a dome tweeter. First, the homogenized material properties of the composite material are computed. Secondly, an eigenfrequency analysis compares the resonant frequencies for conventional materials, such as titanium, with the eigenfrequencies obtained for the composite diaphragm.

DOME TWEETER WITH COMPOSITE DIAPHRAGM — FREQUENCY DOMAIN RESPONSE

In this example, the frequency response of a dome tweeter is studied for different diaphragm materials. The frequency response of the diaphragm made of a composite material is smoother than the response obtained for a diaphragm made of conventional materials like titanium.

PIEZORESISTIVE PRESSURE SENSOR — LAYERED SHELL VERSION

This tutorial uses the **Electric Currents in Layered Shells** interface in combination with the **Layered Shell** interface to model a piezoresistive pressure sensor.

ALUMINUM NITRIDE LAMB WAVE RESONATOR — LAYERED SHELL VERSION

This example shows how to model an aluminum nitride lamb wave resonator and how to perform eigenfrequency and frequency-response analyses to characterize the device. The tutorial uses the **Electric Currents in Layered Shells** interface in combination with the **Layered Shell** interface to model the piezoelectric effect.

Corrosion Module

New Functionality in Version 6.3

GENERAL

- The new **External Current Source** node may now be added to an **Edge Electrode** node in the **Secondary Current Distribution** and **Cathodic Protection** interfaces. The node can be used to define a current source on long slender metallic objects such as rails or pipelines.
- The **Edge Electrode** and **Sacrificial Edge Anode** charge balance equations have been updated to correctly handle edge electrodes placed on external boundaries.
- The new **Concentrated Electrolyte Transport** interface is now available from the Model Wizard. The interface can be used to model transport in any electrolyte solution with an arbitrary number of charged and uncharged species. The interface is based on concentrated electrolyte theory, where the transport equations are defined using binary Maxwell–Stefan diffusivity coefficients and assume local electroneutrality. In contrast to the Nernst–Planck equations, for instance, the concentrated solution theory does not assume all modeled species to be diluted in a solvent of constant concentration. Typical electrolyte systems that can be modeled using the new interface include ionic liquids, salt melts, and highly concentrated solutions, featuring nonnegligible concentration gradients of the charge-carrying species.
- The **Parameter Estimation** study step and BOBYQA, Levenberg–Marquardt, and IPOPT optimization solvers are now available with a Corrosion Module license. Parameter estimation is commonly used for determining suitable parameter values for models based on experimental data.
- A **Model Wizard** entry has been added for the **Tertiary Current Distribution, Poisson** charge conservation option.
- The default solver setting suggestions have been updated to set the **Pivoting perturbation** to 1e-13 for the PARDISO solver in the Electrochemistry interfaces
- The frame of integration has been changed from the **Material** to the **Spatial** frame in most domain transport equations in the electrochemistry interfaces defining electrolyte transport, in conjunction with a general overview of conversion factors used during frame splitting. The changes improve mass conservation in domains subject to material strains and only have an impact on the results when used in

combination with **Moving Mesh** or when **Include Geometric Nonlinearity** is enabled in a study step.

- The **Nonconservative** form of the material balance equation in the **Tertiary Current Distribution** interfaces in porous domains has been corrected. The porosity factor is now placed outside of the time derivative operator.

THE CHEMICAL SPECIES TRANSPORT INTERFACES

The plot defaults have been changed in the following respects:

- The **Prism** color table is now used by default instead of the **Rainbow** color table.
- Mass fluxes are visualized using arrows instead of streamlines in 2D and 2D axially symmetric geometries.
- Mass flux streamlines use ribbons instead of lines in 3D geometries.

Two new results templates types are now available: arrayed concentration plots and mole balances.

- In order to visualize multiple concentrations at a time, the arrayed concentration plots include up to four different concentrations in the same plot group. One or several plot groups, as needed to include all species, are available.
- Mole balance evaluation groups are available for all species solved for. The evaluation groups include integration of fluxes over inlets, outlets, and all external boundaries, as well as integration of domain source terms.

New and Updated Models in Version 6.3

- *Stray Currents from a Train in a Light Rail Transit System* (stray_current_train)
- *Biodegradation of a Magnesium Stent* (biodegradation_mg_stent)
- *Accelerated Corrosion Test of a Scratched Galvanized Steel Sample* (act_scratched_galvanized_steel)
- The evans_droplet model has been updated to include more species and reactions

ECAD Import Module

New Functionality in Version 6.3

IMPORT OF IPC-2581 AND ODB++ FILES

- The import can now generate simplified component geometries from the component outlines in the files when the new **Import component outlines** checkbox is selected. The following new options are available for creating the component geometries: **Height of components**, **Elevation offset for top components**, **Elevation offset for bottom components**, **Component attribute tag**. In 3D, the components can be generated as 2D outline surfaces or as extruded outlines. The imported components are assigned attributes that make it easy to identify and select the components based on their names and package types when used in Logical Expression Selection features.
- A **Plating thickness** can now be specified to generate a cylindrical layer of a specified thickness for the copper around the vias and plated holes.
- In 3D, the PCB geometry is now generated by default for a more efficient model setup when using Form Assembly for finalizing the geometry for physics together with swept meshing. The new import options **Unite metal layer and in-layer dielectric** and **Modify layer objects** for drill layer handling are selected by default to achieve this. Previously, Boolean geometry operations were needed to create a similar geometry.
- Several new options have been added to control how vias and holes are represented in the PCB geometry when using the new **Modify layer objects** option for **Drill layer handling**: **Create domains for the cores of vias**, **Via cores through metal layers**, **Create domains for the cores of plated holes**, **Plated hole cores through metal layers**, and **Create domains for non-plated holes**. The options **Via cores through metal layers** and **Plated hole cores through metal layers** are also available when the **Drill layer handling** is set to **Create separate objects**.
- The new automatically generated `Metal` and `Dielectric` selections collect all copper and dielectric domains and make it easier to assign material properties. Previously, these selections had to be generated using union selection operations.

EXPORT OF 2D GEOMETRY

2D geometry and geometry from work planes in 3D can now be exported to the OASIS format, a widely used standard format for exchanging photomask designs.

IMPORT OF IPC-2581 AND ODB++ FILES

- The default for **Elevations** has been changed to **Metal layer between dielectric layers**. In the API, the default for the corresponding property `elevationtype` is now `metalbetween`.
- The new **Unite metal layer and in-layer dielectric** checkbox is selected by default. In the API the new property `unitemetalanddiel` is `on` by default.
- The new **Drill layer handling** list has the default option **Modify other objects**. In the API, the new property `drillhandling` is `modify` by default.
- Vertical faces are now included in all selections that are generated by the import.

Electrochemistry Module

New Functionality in Version 6.3

GENERAL

- The new **Concentrated Electrolyte Transport** interface is now available from the **Model Wizard**. The interface can be used to model transport in any electrolyte solution with an arbitrary number of charged and uncharged species. The interface is based on concentrated electrolyte theory, where the transport equations are defined using binary Maxwell–Stefan diffusivity coefficients and assume local electroneutrality. In contrast to the Nernst–Planck equations, for instance, the concentrated solution theory does not assume all modeled species to be diluted in a solvent of constant concentration. Typical electrolyte systems that can be modeled using the new interface include ionic liquids, salt melts, and highly concentrated solutions, featuring nonnegligible concentration gradients of the charge-carrying species.
- The **Parameter Estimation** study step and BOBYQA, Levenberg–Marquardt, and IPOPT optimization solvers are now available with an Electrochemistry Module license. Parameter estimation is commonly used for determining suitable parameter values for models based on experimental data.
- A **Model Wizard** entry has been added for the **Tertiary Current Distribution, Poisson** charge conservation option.
- The default solver setting suggestions have been updated to set the **Pivoting perturbation** to 1e-13 for the PARDISO solver in the Electrochemistry interfaces
- The frame of integration has been changed from the **Material** to the **Spatial** frame in most domain transport equations in the Electrochemistry interfaces defining electrolyte transport, in conjunction with a general overview of conversion factors used during frame splitting. The changes improve mass conservation in domains subject to material strains and only have an impact on the results when used in combination with **Moving Mesh** or when **Include Geometric Nonlinearity** is enabled in a study step.
- The **Nonconservative** form of the material balance equation in the **Tertiary Current Distribution** interfaces in porous domains has been corrected. The porosity factor is now placed outside of the time derivative operator.

THE CHEMICAL SPECIES TRANSPORT INTERFACES

The plot defaults have been changed in the following respects:

- The **Prism** color table is now used by default instead of the **Rainbow** color table.
- Mass fluxes are visualized using arrows instead of streamlines in 2D and 2D axially symmetric geometries.
- Mass flux streamlines use ribbons instead of lines in 3D geometries.

Two new results templates types are now available; arrayed concentration plots and mole balances.

- In order to visualize multiple concentrations at a time, the arrayed concentration plots include up to four different concentrations in the same plot group. One or several plot groups, as needed to include all species, are available.
- Mole balance evaluation groups are available for all species solved for. The evaluation groups include integration of fluxes over inlets, outlets, and all external boundaries, as well as integration of domain source terms.

New and Updated Models in Version 6.3

- *Molten Carbonate Transport* (molten_carbonate_transport)
- *Capacitive Deionization of Saline Water* (capacitive_deionization)

Electrodeposition Module

New Functionality in Version 6.3

GENERAL

- The new **Concentrated Electrolyte Transport** interface is now available from the **Model Wizard**. The interface can be used to model transport in any electrolyte solution with an arbitrary number of charged and uncharged species. The interface is based on concentrated electrolyte theory, where the transport equations are defined using binary Maxwell–Stefan diffusivity coefficients and assume local electroneutrality. In contrast to the Nernst–Planck equations, for instance, the concentrated solution theory does not assume all modeled species to be diluted in a solvent of constant concentration. Typical electrolyte systems that can be modeled using the new interface include ionic liquids, salt melts, and highly concentrated solutions, featuring nonnegligible concentration gradients of the charge-carrying species.
- The **Parameter Estimation** study step and BOBYQA, Levenberg–Marquardt, and IPOPT optimization solvers are now available with an Electrodeposition Module license. Parameter estimation is commonly used for determining suitable parameter values for models, based on experimental data.
- A **Model Wizard** entry has been added for the **Tertiary Current Distribution, Poisson** charge conservation option.
- The default solver setting suggestions have been updated to set the **Pivoting perturbation** to 1e-13 for the PARDISO solver in the Electrochemistry interfaces.
- The frame of integration has been changed from the **Material** to the **Spatial** frame in most domain transport equations in the Electrochemistry interfaces defining electrolyte transport, in conjunction with a general overview of conversion factors used during frame splitting. The changes improve mass conservation in domains subject to material strains and only have an impact on the results when used in combination with **Moving Mesh** or when **Include Geometric Nonlinearity** is enabled in a study step.
- The **Nonconservative** form of the material balance equation in the **Tertiary Current Distribution** interfaces in porous domains has been corrected. The porosity factor is now placed outside of the time derivative operator.

THE CHEMICAL SPECIES TRANSPORT INTERFACES

The plot defaults have been changed in the following respects:

- The **Prism** color table is now used by default instead of the **Rainbow** color table.
- Mass fluxes are visualized using arrows instead of streamlines in 2D and 2D axially symmetric geometries.
- Mass flux streamlines use ribbons instead of lines in 3D geometries.

Two new results templates types are now available: arrayed concentration plots and mole balances.

- In order to visualize multiple concentrations at a time, the arrayed concentration plots include up to four different concentrations in the same plot group. One or several plot groups, as needed to include all species, are available.
- Mole balance evaluation groups are available for all species solved for. The evaluation groups include integration of fluxes over inlets, outlets, and all external boundaries, as well as integration of domain source terms.

Fatigue Module

New Functionality in Version 6.3

FINITE LIFE COMPUTATIONS IN STRESS-BASED FATIGUE

It is now possible to use the **Findley** and **Matake** criteria in Stress-Based fatigue for finite life computations.

Fuel Cell & Electrolyzer Module

New Functionality in Version 6.3

GENERAL

- The new **Concentrated Electrolyte Transport** interface is now available from the **Model Wizard**. The interface can be used to model transport in any electrolyte solution with an arbitrary number of charged and uncharged species. The interface is based on concentrated electrolyte theory, where the transport equations are defined using binary Maxwell–Stefan diffusivity coefficients and assume local electroneutrality. In contrast to the Nernst–Planck equations, for instance, the concentrated solution theory does not assume all modeled species to be diluted in a solvent of constant concentration. Typical electrolyte systems that can be modeled using the new interface include ionic liquids, salt melts, and highly concentrated solutions, featuring nonnegligible concentration gradients of the charge-carrying species.
- The **Parameter Estimation** study step and BOBYQA, Levenberg–Marquardt, and IPOPT optimization solvers are now available with a Fuel Cell & Electrolyzer Module license. Parameter estimation is commonly used for determining suitable parameter values for models based on experimental data.
- A **Model Wizard** entry has been added for the **Tertiary Current Distribution, Poisson** charge conservation option.
- The default solver setting suggestions have been updated to set the **Pivoting perturbation** to 1e-13 for the PARDISO solver in the Electrochemistry interfaces.
- The frame of integration has been changed from the **Material** to the **Spatial** frame in most domain transport equations in the Electrochemistry interfaces defining electrolyte transport, in conjunction with a general overview of conversion factors used during frame splitting. The changes improve mass conservation in domains subject to material strains and only have an impact on the results when used in combination with **Moving Mesh** or when **Include Geometric Nonlinearity** is enabled in a study step.
- The **Nonconservative** form of the material balance equation in the **Tertiary Current Distribution** interfaces in porous domains has been corrected. The porosity factor is now placed outside of the time derivative operator.

FUEL CELL & ELECTROLYZER MATERIAL LIBRARY

- Heat capacity and thermal conductivity have been added to KOH.
- Heat of vaporization and vapor pressure have been added to KOH and NaOH

HYDROGEN FUEL CELL AND WATER ELECTROLYZER INTERFACES

- A new **Periodic Condition** boundary node has been added.
- A new **Electric Ground** point node has been added.
- The **Reaction Sources** node under H2/O2 Gas Phase nodes now allows to set the **Molar sources**, in addition to mass sources, and now supports automatic **Heat source** calculation.
- Mass source variables for both the liquid and gas phase on both boundaries (`N_g`/`N_l`) and domains (`Q_g`/`Q_l`) are now defined. These new variables can be used, for instance, in the Phase Transport interfaces by the new **Boundary Mass Source** node to define phase mass fluxes on a boundary or by the **Mass Source** node to define phase sources in a domain.
- A new **Liquid/Solid Electrolyte** option has been added under the **Electrode Reaction Settings** section. The setting defines if the charge-carrying ion contributes to the liquid mass flux/source variables defined by the interface.
- New **Alkaline** entries have been added to the **Model Wizard**. They use the new **Liquid** electrolyte option by default.
- New material properties for heat of vaporization and vapor pressure at H2/O2 Gas Phase nodes have been added.
- A bug related to the **Fuller Diffusion Volume** for **Auxiliary Species** has been fixed. The value entered in the user interface is now interpreted in the unit of m^3.
- A bug related to the initial values of the global degrees of freedoms `wbnd` and `pbnd` in the H2/O2 Inlet nodes for the Mass Flow Rates and Total Mass Flow Rate options has been fixed. This fix was also added to version 6.2 update 3.
- The **Electrolyte type** list under the **Electrode Reaction Settings** section has been renamed to **Charge-carrying ion** with the following options: **Proton**, **Hydroxide**, **Carbonate**, **Oxide**, and **Generic**.

FLUID FLOW INTERFACES

- Two new domain features, the **Fluid** feature and the **Porous Medium** feature, are available for the Phase Transport interfaces. The new **Porous Medium** feature has subnodes to define the phase properties and the properties of the porous matrix. The new domain features replace the previously available **Phase and Transport**

Properties and **Phase and Porous Media Transport Properties** features. When opening a model created in a previous version, the previous feature are kept in the model.

- A new **Boundary Mass Source** boundary condition feature has been added to the Phase Transport interfaces. This feature accounts for the consumption or production of the different phases due to reactions or other physical processes at the boundary. When the **Phase Transport in Porous Media** interface is coupled to a **Darcy's Law** interface using a **Multiphase Flow in Porous Media** multiphysics coupling node, the net mass transfer is automatically accounted for in the flow field.
- A new **Power law** option has been added to specify predefined relative permeabilities for the different phases in the **Porous Medium** feature for the **Phase Transport in Porous Media** interface. This option allows for the easier implementation of relative permeabilities based on power law expressions.
- A new **Include shear-induce migration** option has been added to the **Mixture Model** multiphysics coupling. This option allows, for example, for the simulation of the phenomenon that particles in a pressure-driven pipe or channel mixture flow will migrate toward the center of the flow channel.

THE CHEMICAL SPECIES TRANSPORT INTERFACES

The plot defaults has been changed in the following respects:

- The **Prism** color table is now used by default instead of the **Rainbow** color table.
- Mass fluxes are visualized using arrows instead of streamlines in 2D and 2D axially symmetric geometries.
- Mass flux streamlines use ribbons instead of lines in 3D geometries.

Two new results templates types are now available: arrayed concentration plots and mole balances.

- In order to visualize multiple concentrations at a time, the arrayed concentration plots include up to four different concentrations in the same plot group. One or several plot groups, as needed to include all species, are available.
- Mole balance evaluation groups are available for all species solved for. The evaluation groups include integration of fluxes over inlets, outlets, and all external boundaries, as well as integration of domain source terms.

New and Updated Models in Version 6.3

- *Two-Phase Nonisothermal Zero-Gap Alkaline Water Electrolyzer* (zero_gap_aec)

- *Water and Carbon Dioxide Co-Electrolysis in a Solid Oxide Electrolyzer Cell* (soec_co2)
- *Ammonia-Fed Solid Oxide Fuel Cell* (sofc_nh3)
- *Two-phase Polymer Electrolyte Membrane Fuel Cell* (two_phase_pemfc)
- *Parameter Estimation of a Polymer Membrane Fuel Cell Model* (pemfc_parameter_estimation)
- *Molten Carbonate Transport* (molten_carbonate_transport)
- The alkaline_electrolyzer model has been updated to use new material properties for heat of vaporization and vapor pressure and the new Alkaline model wizard entry
- The aec_shunt_currents model has been updated to use new Alkaline Model Wizard entry

Geomechanics Module

New Functionality in Version 6.3

PERFORMANCE IMPROVEMENTS

The computational cost of assembly operations including creep, plasticity, and soil plasticity has been reduced. This is accomplished by several enhancements and optimizations of the used solution algorithms.

ENHANCEMENTS TO SOIL PLASTICITY

The functionality in the **Soil Plasticity** node has been subject to the following major changes:

- All the material models available in the **Soil Plasticity** node can be extended with a compression cap surface and a tension cutoff criterion. For this purpose, a new subnode called **Cap and Cutoff** is now available. Two different cap models are added: the elliptic cap and the planar cap. Also, two different tension cutoff criteria are available: the Rankine criterion and the mean stress criterion
- All the material models available in the **Soil Plasticity** node can be extended with an isotropic hardening function.
- The user interface and some API names for the **Soil Plasticity** node have been updated.
- A new **Backward Euler, damped** option is available as a local method in **Soil Plasticity**.
- The definitions of the Cam-clay yield function and plastic potential have been improved.

HYPERELASTIC MATERIAL MODELS

The **Hyperelastic Material** node with the **St Venant–Kirchhoff** and **Neo-Hookean** models is now available under the Geomechanics Module license. This makes it possible to model soils under large compressive strains using a hyperelastic formulation.

IMPROVEMENTS TO COUPLED DAMAGE–PLASTICITY MODEL IN CONCRETE

There are several improvements to the coupled damage–plasticity model in the **Concrete** node.

- The implementation of the plasticity algorithm is more robust and now supports a multiplicative flow rule.
- A new **Backward Euler, damped** option is available as a local method.
- The overall computational cost of the model is also significantly reduced.

PHASE FIELD DAMAGE IMPROVEMENTS

The following improvements have been made:

- There are new and improved result variables for **Phase field damage**.
- The default solver suggestions for **Phase field damage** have been improved.

New and Updated Models in Version 6.3

PHASE FIELD MODELING OF DYNAMIC CRACK BRANCHING

In this example, a benchmark problem in dynamic fracture of brittle materials is analyzed using the AT1 phase-field damage model. An instantaneous tensile load is applied to a planar tension specimen with a preexisting crack. Initially, the crack propagates perpendicular to the loading direction, after which the crack branches off symmetrically until catastrophic failure occurs.

Heat Transfer Module

New and Improved Functionality in Version 6.3

NONEQUILIBRIUM FORMULATION FOR MOISTURE TRANSPORT IN POROUS MEDIA

In the **Moisture Transport** physics interfaces, a new option has been added for the **Hygroscopic Porous Medium** feature. It is now possible to use a nonequilibrium formulation where both the relative humidity and the liquid water saturation are solved for. With this formulation, there is no underlying equilibrium hypothesis between the vapor and the liquid phase, which extends the range of applicability of this feature.

As a consequence, two new moisture transport physics interfaces are available from the Model Wizard. The first one, **Nonequilibrium Moisture Transport in Porous Media**, can be used to compute both the relative humidity and the liquid water saturation in a porous media where vapor-liquid equilibrium is not assumed. The second one, **Moisture Transport in Free and Porous Media**, is parameterized to facilitate the setup of moisture transport models where different media coexist.

HOMOGENIZATION OF MATERIALS PROPERTIES FOR HEAT TRANSFER

The new **Cell Periodicity** feature in the **Heat Transfer** physics interface provides a solution to estimate the effective material properties for heat transfer of a heterogeneous material. Based on the idea of a repeating unit cell (RUC) or representative volume element (RVE), this new feature can compute the equivalent thermal conductivity tensor, the heat capacity at constant pressure, and the density of periodic or any other structures.

USER-DEFINED ORBIT FOR ORBITAL THERMAL LOADS

In the **Orbital Thermal Loads** physics interface, a new **User defined** option for the **Orbit type** in the **Orbital Parameters** feature has been added. It makes it possible to freely define the position of the spacecraft as a function of time and can be used to model, for example, heat transfer during transfer orbit.

PERFORMANCE IMPROVEMENTS FOR THERMAL RADIATION INTERFACES

In the **Surface-to-Surface Radiation** and **Orbital Thermal Loads** physics interfaces, a new **Use adaptive resolution coarsening** option is available with the **Hemicube** algorithm. When selected, groups of four adjacent pixels can be merged as one if the difference of

irradiation between these four pixels is below a certain tolerance. By reducing the resolution, this option reduces the memory required to solve the model.

In the **Surface-to-Surface Radiation** and **Orbital Thermal Loads** physics interfaces with the **Ray shooting** radiation method, boundaries with pure specular reflection can be modeled with the new **Specular Mirror** feature. It is an optimized variant of the existing **Opaque Surface** feature, which can significantly improve performance.

In the **Radiation in Participating Media** and **Radiation in Absorbing-Scattering Media** physics interfaces, an alternative solver with a dedicated stabilization has been introduced. This solver can be chosen in the physics interface under the **Stabilization** section. It has proven to be efficient in a variety of cases with similar memory requirements as the already existing default solver. Since there is no general rule regarding the best solver, it is not used by default.

FORWARD RAY-SHOOTING METHOD FOR EXTERNAL RADIATION SOURCE

In the **Surface-to-Surface Radiation** and **Orbital Thermal Loads** physics interfaces, a new **Use forward ray shooting for external radiation source** option is available with the **Ray shooting** algorithm in 3D and 2D. When selected, the contribution of **External Radiation Source** features to the irradiation is calculated using a forward ray-tracing approach. This option is expected to improve accuracy when rays from **External Radiation Source** are reflected several times.

SYMMETRY FEATURE FOR DISCRETE ORDINATES METHOD

In the **Radiation in Participating Media** and **Radiation in Absorbing-Scattering Media** physics interfaces, it is now possible to use the **Symmetry** feature when the **Radiation discretization method** is set to **Discrete ordinates method**. With this method, the **Symmetry** feature selection is restricted to flat boundaries aligned with the axes of the component.

ANISOTROPIC SCATTERING FOR DISCRETE ORDINATES METHOD IN 2D AXISYMMETRY

In the **Radiation in Participating Media** and **Radiation in Absorbing-Scattering Media** physics interfaces, anisotropic scattering phase functions are now available for the discrete ordinates method (DOM) in 2D axisymmetric configurations.

THERMAL CONNECTION BETWEEN INTERFACES

The **Thermal Connection** feature, available in the **Heat Transfer** physics interfaces, has been extended for the **Heat Transfer in Thin Structures** physics interfaces with the new **Thermal Connection, Interface-Interface** feature. It is designed to connect the interfaces

of two boundary selections by a thermal resistor, a thermal capacitor, or a **Lumped Thermal System** interface. It aims to reduce the complexity of a model by replacing parts of it with equivalent circuit elements to represent, for example, the thermal interaction between parts in large assemblies.

PERFORMANCE IMPROVEMENTS FOR THERMAL DAMAGE AND IRREVERSIBLE TRANSFORMATION

For time-domain analysis using the **Temperature threshold** and **Arrhenius kinetics** transformation models, a new **Use local time integration** option is available. When selected, the transformation indicator is no longer represented by ordinary shape functions, but with state variables in the Gauss points, which can result in a significant performance improvement.

IMPROVED HANDLING OF MULTIPLE PELLETS

In the **Heat Transfer** physics interfaces, the handling of porous media with multiple pellet materials has been improved. Each pellet can now get a dedicated initial temperature, heat source, or coupling specification with the fluid phase. The result template for plotting the temperature along the pellet radius is also available for each of them.

POROUS INTERFACE FOR MOISTURE TRANSPORT INTERFACES

In the **Moisture Transport** physics interfaces, a new **Porous Interface** feature has been added. This feature makes it possible to choose the modeling assumptions between different media, such as evaporating the liquid flux at the interface between a porous medium and moist air.

NONCONSERVATIVE FORM OF HYGROSCOPIC POROUS MEDIUM

In the **Moisture Transport in Air**, **Moisture Transport in Building Materials**, and **Moisture Transport in Porous Media** interfaces, the advanced setting to solve either the conservative or nonconservative form of equations now only has an impact on the **Moist Air** feature. The equations of the **Hygroscopic Porous Medium** features are now always solved in conservative form, independently of the value of the setting.

MISCELLANEOUS IMPROVEMENTS

- In the **Thermoelectric Module** feature available in the **Lumped Thermal System** physics interface, the **Thermal and geometric properties** option has been reworked so that the Seebeck coefficient of p-type semiconductors should be positive and the one of n-type semiconductors should be negative. Before COMSOL Multiphysics® version 6.3, the expected signs were the opposite.

- In the **Moisture Transport** physics interfaces, the capillary model inputs have been moved from the **Hygroscopic Porous Medium** feature, in the **Moisture Transport Properties** section, to its **Liquid Water** subfeature, in the **Liquid Water Properties** section.
- Due to the introduction of a setting to use either an equilibrium or a nonequilibrium formulation for **Hygroscopic Porous Media** features in the **Moisture Transport** physics interfaces, the **Moisture Transport in Porous Media** interface has been renamed **Equilibrium Moisture Transport in Porous Media**.
- When a **Moisture Flow** coupling is applied on a porous domain, the porosity and permeability in the **Fluid Flow** interface are now automatically set using the variables from the **Moisture Transport** interface. These properties take the fraction of fluid in the porous medium and the relative permeability of moist air into account. The latter can be set in the **Moist Air** subfeature of the **Hygroscopic Porous Medium** feature.
- In the **Radiation in Participating Media** and **Radiation in Absorbing-Scattering Media** physics interfaces, a new option for the **Incident Intensity** boundary feature has been added. When the **P1 approximation** is selected, it is now possible to specify the external radiation in terms of incident radiation. This setting is available for the **User defined**, **User-defined for each band**, and **User-defined distribution** options.
- In the **Radiation** physics interfaces except for the **Radiative Beam in Absorbing Media** physics interface, when the **Wavelength dependence of radiative properties** is set to **Solar and ambient**, you can now modify the upper bound of the spectral bound for ambient radiation used for integrations calculation by selecting the new **Specify the upper bound for integrations** checkbox.
- In the **Symmetry for Surface-to-Surface Radiation** feature available in the **Surface-to-Surface Radiation** physics interface, a new selection method **Point selection** is now available when the **Type of symmetry** is set to **Sector symmetry**, **Two perpendicular symmetry planes**, or **Three perpendicular symmetry planes**. When selected, points used to define such symmetry can now be directly selected from the **Graphics** window.
- A new **Filter diurnal variation** checkbox has been added in **Ambient Properties** when computed from any ASHRAE meteorological data. When selected, the temperature as well as the relative humidity conditions are no longer calculated based on date and hour, but on date only. The diurnal fluctuation is then removed.
- In **Ambient Properties**, when a weather station is selected **From reference**, information such as its World Meteorological Organization (WMO) reference, location, and coordinates are displayed in the user interface.

- In the **Ground Pointing** feature available in the **Orbital Thermal Loads** physics interface, the position of the ground station can be defined by the **Weather station** selected in any **Ambient Properties** node that uses ASHRAE meteorological data.
- In the **Ground Pointing** feature available in the **Orbital Thermal Loads** physics interface, a new variable `inTargetSpot` has been added. It indicates if the spacecraft is currently in sight of the target, but is not necessarily pointing at it.
- The color tables in default plots for the **Moisture Transport** physics interfaces and plots from the **Result Templates** have been changed to new ones.

New Models in Version 6.3

SUPERHEATED STEAM DRYING OF A WOOD PARTICLE

This model demonstrates the use of the **Heat and Moisture Flow** features for the simulation of superheated steam drying of a wood particle. A nonequilibrium formulation is used to compute the transport of the liquid water and vapor phases in the wood particle.

MODELING OF THE 3-OMEGA METHOD FOR ESTIMATING THE THERMAL CONDUCTIVITY OF NANOSTRUCTURED MATERIALS

The model demonstrates the basics of the 3-omega method. The coupled electromagnetic heating problem is solved in a solid sample, which is heated by a thin copper strip deposited on the surface of a sample. Analyzing the frequency dependence of the voltage oscillation amplitude, the simulation provides the log-linear region, evaluates the thermal conductivity of a sample, and compares it with the exact value for model validation.

Updated Models in Version 6.3

The following 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 *Drying of a Potato Sample* model is now using the new **Porous Interface** feature.
- The *Lumped Composite Thermal Barrier with Shells* model is now using the new **Thermal Connection, Interface-Interface** feature.

- The *Equivalent Properties of Periodic Microstructures* model is now using the new **Cell Periodicity** feature.
- The *Radiative Heat Transfer in a Utility Boiler* model is now using the new **Symmetry** feature.
- The *Greenhouse Effect* model is now using a user-defined value for the upper bound of the spectral bound for integrations calculation.
- Many models have been updated to use the new **Preferred Units** configuration.

Backward Compatibility with Version 6.2

- In COMSOL Multiphysics® version 6.3, the **Moisture Flow** coupling automatically sets the porosity and permeability of the **Fluid Flow** interface when it is applied on a porous domain. API scripts need to be updated to enable model input editing in the **Porous Matrix** subfeature of **Porous Medium** features to ensure the same values as in version 6.2 are used.
- Several improvements have been made in the **Moisture Transport** physics interfaces. The physics phenomena solved are the same as before, but the numerical treatment has changed to increase stability and accuracy in specific cases. These changes may lead to a slight change in results for models where the mesh was coarse or the solver tolerance too loose.
- In the **Moisture Transport** physics interfaces, the override rules of the **Moisture Flux** feature have been modified. The **Moisture Flux** feature now has a symmetric behavior when it is applied before or after a **Symmetry**, **Inflow**, **Outflow**, or **Open Boundary** feature. In version 6.2, the **Moisture Flux** feature contributed to the **Symmetry**, **Inflow**, **Outflow**, and **Open Boundary** features when it was placed afterward in the model tree, whereas it was overridden when placed before. Now, these features always override each other.
- In the **Moisture Transport** physics interfaces, when a **Moisture Source** feature was applied on a **Hygroscopic Porous Medium** domain, it was formerly contributing as an evaporation source, and therefore accounted as a heat source in the **Heat and Moisture** coupling. The contribution from **Moisture Source** features has now been removed from the evaporation source. This will affect the results only if a **Heat and Moisture** coupling was used.
- In the **Lumped Thermal System** physics interface, all variable names have been reworked to use a dot separator instead of an underscore separator. Backward compatibility is ensured. After a new computation, quantities in the **Results** node

have to be updated. For example, the temperature at the port `p1` of a resistor `R1` was formerly `lts.R1_p1_T` and is now `lts.R1.p1.T`.

Backward Compatibility with Version 6.1

- In COMSOL Multiphysics® version 6.2, the **Isothermal domain** checkbox has been removed from the physics interface settings. The **Isothermal Domain** feature is now always available in the domain features list under the **Specific Media** subfolder. The default **Isothermal Domain Interface** feature is automatically added as soon as one **Isothermal Domain** is present in the physics interface.
- When **Heat Transfer with Surface-to-Surface Radiation**, **Heat Transfer with Radiation in Participating Media**, or **Heat Transfer with Radiation in Absorbing-Scattering Media** couplings are added from the **Multiphysics** node, their selection is set to **All boundaries** by default.
- The **Heat transfer in porous media** checkbox has been removed from the **Heat Transfer in Thin Structures** physics interfaces settings. Backward compatibility is ensured in most cases with two exceptions. API scripts need to be updated to disable **Porous Medium** and **Fracture** features if they were enabled and the checkbox was deselected afterward. In version 6.1, these steps would disable **Porous Medium** and **Fracture** features, while in version 6.2, the features remain active. The second exception is when opening a model from a previous version where **Porous Medium** or **Fracture** features were present but the checkbox was deselected. In that case, the features need to be duplicated to be enabled.
- To be able to handle events internally to the **Orbital Thermal Loads** physics interface while still ensuring backward compatibility, a new **Orbital Thermal Loads** physics interface has been created, and the former one has been deprecated. Backward compatibility for the **Orbital Thermal Loads** interface is ensured for API scripts, but will continue to use the deprecated interface with the **Generate Events Interface** feature as well as the **Events** interface. For simple setups, when opening a model from version 6.1 with one active **Orbital Thermal Loads** interface and exactly one autogenerated **Events** interface, the model is migrated to the new interface with the **Events Timeline** feature. In other cases, no migration is done, but the model will remain fully functional.
- In the **Radiation in Participating Media** and **Radiation in Absorbing-Scattering Media** interfaces, the pair versions of the **Opaque Surface**, **Semitransparent Surface**, and **Incident Intensity** features are now interpreted on the spatial frame as their nonpair versions, instead of the material frame in version 6.1.

- In the **Heat Transfer** interfaces, the default opacity of the **Thin Layer** and **Fracture** features has been changed to opaque. In the **Heat Transfer in Thin Structures** interfaces, the default opacity of the **Solid** and **Porous Medium** features has been changed to opaque. To get the same behavior as in version 6.1, API scripts need to be updated to explicitly change the opacity of these features to **Transparent** if the default opacity of these features were used in a **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Orbital Thermal Loads** study. The same process must be carried when opening a model from version 6.1.
- In the **Surface-to-Surface Radiation** and **Orbital Thermal Loads** physics interfaces, the new **Lower integration order for irradiation** option is available with the **Hemicube** algorithm in 3D and 2D and activated by default. While it should have a negligible effect, this option can be deactivated in the **Jacobian Contribution** section to get the same behavior as in version 6.1.

Backward Compatibility with Version 6.0

- In COMSOL Multiphysics® version 6.0, when the **From material** option was selected in **Shape Memory Alloy** feature for the heat capacity at constant pressure, Cp, the property was taken from the material **Basic** group. In version 6.1, the material properties Cp_{A} and Cp_{M} are taken from the groups **Austenite Phase** and **Martensite Phase**, respectively. Identity rules ensure backward compatibility in most cases. However, if different values were set for Cp and Cp_{A} or Cp_{M}, the results may differ between versions 6.0 and 6.1.
- In the **Symmetry for Surface-to-Surface Radiation** feature of the Surface-to-Surface Radiation interface, the **Apply symmetry to external radiation sources** checkbox is selected by default in version 6.1. API scripts need to be updated by clearing this checkbox to get the same results as in version 6.0 when there are both a **Symmetry for Surface-to-Surface Radiation** and an **External Radiation Source** feature in the model.
- It is now no longer possible to set an **Inflow**, **Outflow**, or **Open Boundary** condition on boundaries of **Shape Memory Alloy** or **Building Material** domains.
- In the **Surface-to-Surface Radiation** interface features, the diffuse irradiance variables (`Idiff`, `Idiffu_band`, `Idiffd_band`, `Idiff_band`) are set to 0 when the **Include diffuse irradiance** checkbox is not selected.
- In the settings of the **Surface-to-Surface Radiation** interface, the **Mesh element self-irradiation** checkbox does not exist in version 6.1. Self-irradiation is always taken into account when **High order mesh elements** is selected.

- In the settings of the **Surface-to-Surface Radiation** interface, the **High order mesh elements** checkbox is available for the following combinations in version 6.1: the ray-shooting method in 3D, 2D axisymmetric, and 2D and the hemicube method in 2D axisymmetric and 2D. In case the model saved in version 6.0 had the **High order mesh elements** checkbox selected, the behavior in version 6.1 is the same as when this checkbox was cleared. View factor results should be checked for consistency.
- The contribution of the shell curvature on gradient definition to account for the surface area change has been corrected for cases where the shell normal is pointing toward the center of curvature.
- The speed of sound variable definition has been corrected in the **Fluid** and **Moist Air** features and subfeatures.
- For turbulent nonisothermal flow, the constraints added to guarantee the continuity of the temperature between multiple wall boundaries has been revised to be much faster to treat in case of a very large number of boundaries, and to improve the readability of the **Equation view**.
- Refactored **Boundary Heat Source** feature to give the control if the continuity is prescribed or not and to control where the source is added: source, destination, or both.
- Updated the heat source contribution in **Heat Transfer with Radiative Beam in Absorbing Media** coupling in case of temperature variation across the boundary to account for the side of the boundary that absorbs the radiation.
- In the **Building Material** feature, the material properties are now interpreted in the material frame. This affects models where a **Building Material** feature is on a domain where the spatial frame is controlled by an interface, such as a **Moving Mesh**. The **Mass Properties** node now handles **Building Material** features and can be used to compute the total mass.

Liquid & Gas Properties Module

New Functionality in Version 6.3

IMPROVEMENTS TO THE GENERATE MATERIAL WIZARD

It is now possible to specify a logarithmic range of interpolation points for the pressure when using the **Generate Material Wizard** to create a **Material** from a **Thermodynamic System**. Regularization of function arguments has been added to prevent evaluation of zero or negative values for the temperature and absolute pressure.

MEMS Module

New Functionality in Version 6.3

NEW FORMULATION FOR ELECTROSTATICS

The Electrostatics interface now supports the D-V Formulation in 3D and 2D, allowing for a more accurate electrostatic force evaluation and is particularly advantageous for calculating electromechanical forces in geometries that contain sharp corners. For details, see New Formulation for Electrostatics in the release notes for the AC/DC Module.

NEW DEFAULT FREE SPACE FEATURE (FOR SEVERAL INTERFACES)

For details, see New Default Free Space Feature (for Several Interfaces) in the release notes for the AC/DC Module.

ELECTROMECHANICS MULTIPHYSICS INTERFACES

For details, see Electromechanics Multiphysics Interfaces in the release notes for the Structural Mechanics Module.

CONTACT FORMULATION FOR INTERIOR BOUNDARIES

For details, see Contact Formulation for Interior Boundaries in the release notes for the Structural Mechanics Module.

FITTING OF VISCOELASTIC DATA

For details, see Fitting of Viscoelastic Data in the release notes for the Structural Mechanics Module.

UPDATED LOAD FEATURES

For details, see Updated Load Features in the release notes for the Structural Mechanics Module.

LINEAR ELASTIC MATERIAL PROPERTIES FOR PIEZOELECTRICITY

For details, see Linear Elastic Material Properties for Piezoelectricity in the release notes for the Structural Mechanics Module.

ENHANCEMENTS TO CELL PERIODICITY

For details, see Enhancements To Cell Periodicity in the release notes for the Structural Mechanics Module.

PRESCRIBED ANGULAR VELOCITY

For details, see Prescribed Angular Velocity in the release notes for the Structural Mechanics Module.

ENERGY CONTRIBUTION FOR SPRING FOUNDATION

For details, see Energy Contribution of Spring Foundation in the release notes for the Structural Mechanics Module.

GENERALIZED POWER LAW FOR DECOHESION

For details, see Generalized Power Law for Decohesion in the release notes for the Structural Mechanics Module.

WALL VELOCITY IN FLUID–STRUCTURE INTERACTION

For details, see Wall Velocity in Fluid–Structure Interaction in the release notes for the Structural Mechanics Module.

LOAD PLOTS FOR MASS AND MOMENT OF INERTIA

For details, see Load Plots for Mass and Moment of Inertia in the release notes for the Structural Mechanics Module.

EQUIVALENT ELASTIC CONSTANTS FOR EXTERNAL STRESS–STRAIN RELATION

For details, see Equivalent Elastic Constants for External Stress–Strain Relation in the release notes for the Structural Mechanics Module.

ENHANCED PIEZOELECTRIC MATERIAL OPTIONS FOR MEMS APPLICATION

The **Piezoelectric** material library has been expanded to include materials from PI Ceramic GmbH.

MICROMACHINED GYROSCOPE WITH MIXED FORMULATION

This model demonstrates the operation of a gyroscope in a rotating frame. The model solves the fully coupled equations of electrostatics and structural mechanics. The model uses the new D-V formulation or **Mixed finite element** formulation in the Electrostatics interface to provide higher accuracy for structures such as comb drives that includes sharp corners.

ALUMINUM NITRIDE LAMB WAVE RESONATOR — LAYERED SHELL VERSION

This example shows how to model an aluminum nitride Lamb wave resonator and how to perform eigenfrequency and frequency-response analyses to characterize the device. The tutorial uses the **Electric Currents in Layered Shells** interface in combination with the **Layered Shell** interface to model the piezoelectric effect.

NORMAL MODES OF A BIASED RESONATOR — 3D GEOMETRY FROM A GDS-FILE

The Normal Modes of a Biased Resonator — 3D Geometry from a GDS-File model has been updated after changes to the **Offset Face** geometry operation.

Metal Processing Module

New Functionality in Version 6.3

PHASE TRANSFORMATION MODELING

By using the new **Microstructure based** phase transformation model in conjunction with the **Steel Composition** node, you gain access to the chemical composition-based **Kirkaldy–Venugopalan** and **Li–Niebuhr–Meekisho–Atteridge** phase transformation model formulations. The former can be modified for the application to boron steels according to Åkerström and Oldenburg. The previously existing **Kirkaldy–Venugopalan** phase transformation model has been renamed to **Kirkaldy–Venugopalan, simplified**.

NEW PHYSICS INTERFACES

Two new physics interfaces have been added for the purpose of simulating phase transformations during steel hardening. The **Austenite Decomposition, Kirkaldy–Venugopalan** interface creates and configures necessary nodes for using the **Kirkaldy–Venugopalan** phase transformation model formulation. Similarly, the **Austenite Decomposition, Li–Niebuhr–Meekisho–Atteridge** interface creates and configures necessary nodes for using the **Li–Niebuhr–Meekisho–Atteridge** phase transformation model formulation.

CALCULATION OF HARDNESS

Functionality for calculating the Rockwell hardness (HRC) after quenching has been added. The hardness calculation is based on the computed Vickers hardness (HV).

IMPORT OF PHASE TRANSFORMATION DATA

The import functionality from JMatPro® has been extended, giving improved agreement across the full range of the imported data.

THERMAL STRAIN MODELING

A new formulation for computing thermal strains has been added. The **Density based** formulation uses the temperature-dependent densities of the individual phases as a starting point for computing the thermal strain of the compound material.

TRANSFORMATION INDUCED PLASTICITY MODELING

The TRIP coefficient can now be computed based on the temperature-dependent volumetric difference between source and destination phases and the yield stress of the

source phase. This **Thermal-strain based** TRIP coefficient is related to the density based thermal strain modeling.

New Models in Version 6.3

DILATOMETRY CURVES FROM CCT

This model computes a CCT diagram and dilatometry curves using imported phase transformation data and phase material properties for a general steel. Both diffusive and displacive phase transformations are used, and the temperature-dependent effective density of the material is used to express an axial thermal strain.

QUENCHING OF A BEVEL GEAR

A 3D model is used to simulate the cooling of a bevel gear from an austenitic state. Phase material properties and phase transformation data are imported into the model. The final phase composition and the residual stress state are computed. Phenomena such as thermal strains, phase plasticity, transformation-induced plasticity (TRIP), and phase transformation latent heat are included in the simulation.

Microfluidics Module

New Functionality in Version 6.3

A new **Mass flow** option has been added in the **Boundary Condition** section of the **Settings** window for the **Outlet** feature in the **Single-Phase Flow** interfaces.

Updated Models in Version 6.3

The tutorials have been updated to take advantage of the latest geometry, mesh, and solver features in order to provide simplified step-by-step instructions for the model construction.

Backward Compatibility with Version 6.0

- An error in the expression for the mean free path when using the **Alternative (Sharipov)** option for the **Mean free path definition** under **User defined** in the **External Slip Wall** and **Interior Slip Wall** features has been corrected. This might lead to slight differences in the results. In addition, the default values for the **Standard** option for the **Mean free path definition** setting have been modified to correspond to those of the **Alternative (Sharipov)** option.
- Non-Newtonian inelastic material models that were added in older versions will not be able to define the dynamic viscosity in the **Basic (def)** material model.

Mixer Module

New Functionality in Version 6.3

- New **Rotating Machinery, High Mach Number Flow** interfaces for all transport-equation **RANS-EVM** (eddy-viscosity models) have been added under the **High Mach Number Flow** section in the **Fluid Flow** branch.
- A new **Mixing Plane** feature, which performs circumferential averaging of flow quantities while accounting for the direction of characteristics, has been added in the **Rotating Machinery, High Mach Number Flow** interfaces. The mixing plane approach is an efficient method for modeling sectors of centrifugal pumps, turbines, and compressors. The predefined study, **Frozen Rotor with Initialization**, automatically adds a **Mixing Plane Initialization** study step, which computes average flow quantities using the initial values.

New Models in Version 6.3

RADIAL PUMP WITH MIXING PLANES

The model computes an averaged flow field in a radial pump model using the mixing plane methodology with a frozen rotor study. The solution approximates an overall flow resulting from various possible positions of the rotor blade. This demonstrates how costly, time-dependent simulations can be circumvented by applying a mixing-plane approach.

Multibody Dynamics Module

New Functionality in Version 6.3

FRICTION IN CAM-FOLLOWER

A new **Friction** subnode is has been added to the **Cam-Follower** node. This subnode can be used to model frictional losses in time-dependent analyses.

MEASURE RELATIVE VALUES

The new **Measure** feature makes it possible to compute the difference in displacement, velocity, acceleration, or any other quantity between two selected points. Built-in result variables are available for postprocessing.

PRESCRIBED VELOCITY AND PRESCRIBED ACCELERATION

Two new features called **Prescribed Velocity** and **Prescribed Acceleration** have been added at the domain and boundary levels. You can use them to prescribe velocity or acceleration in one or more directions for time-dependent and frequency domain studies.

MASS NODE

A new node named **Mass Node** has been added to the Lumped Mechanical System interface. You can use it to model a lumped masses in mechanical systems.

PRESCRIBED ANGULAR VELOCITY

For **Rigid Connector**, **Rigid Material**, and **Gears**, a new option has been added for prescribing the rotation. In addition to explicitly giving the rotation, it is now also possible to prescribe the angular velocity.

RESULT TEMPLATES FOR USER-DEFINED REFERENCE FRAMES

Two new plots are added to the result templates to visualize the relative motion and velocity of objects with respect to a rigid domain, attachment, or gear.

ENHANCEMENTS TO THE LUMPED MECHANICAL SYSTEM INTERFACE

The variable naming convention in the Lumped Mechanical System interface is now changed from an underscore separated format to a dot separated format. For example, the spring force variable `comp1.lms.K1_f` is now named as `comp1.lms.K1.f`. Full

compatibility with the previous version is provided. When building a model using the API, the new format should be used, even for Java files created in older versions.

UPDATED JOINT AXIS SELECTION

The **Joint Axis** subnode can now select only one edge as the edge parallel to the joint axis. Full compatibility is provided when opening old models where multiple edges were selected as the parallel edges.

UPDATED LOAD FEATURES

For details, see Updated Load Features in the release notes for the Structural Mechanics Module.

ENERGY CONTRIBUTION FOR SPRING FOUNDATION

For details, see Energy Contribution of Spring Foundation in the release notes for the Structural Mechanics Module.

WALL VELOCITY IN FLUID–STRUCTURE INTERACTION

For details, see Wall Velocity in Fluid–Structure Interaction in the release notes for the Structural Mechanics Module.

LOAD PLOTS FOR MASS AND MOMENT OF INERTIA

For details, see Load Plots for Mass and Moment of Inertia in the release notes for the Structural Mechanics Module.

New Model in Version 6.3

GEAR SHIFTING IN A SEQUENTIAL GEARBOX

This example illustrates the modeling of a sequential manual transmission where the gears are selected sequentially with the help of a rotating drum. As the drum rotates, the grooves on the drum guide a set of shifting forks, which in turn helps in engaging the required gear. A transient multibody analysis is performed to compute the angular velocities of all the gears for the specified engine speed and external load.

Nonlinear Structural Materials Module

New Functionality in Version 6.3

PERFORMANCE IMPROVEMENTS

The computational cost of assembly operations including creep, plasticity, shape memory alloys, and viscoplasticity has been reduced. This improvement is accomplished by several enhancements and optimizations of the solution algorithms.

PRESSURE-DEPENDENT PLASTICITY

A new **Pressure-Dependent Plasticity** subnode has been added to the **Linear Elastic Material**, **Nonlinear Elastic Material**, and **Hyperelastic Material** nodes in Solid Mechanics.

Three new models are introduced: the **Elliptic** model, the **Foam** model, and the **Parabolic** model, besides the **Drucker-Prager** model that is also available in **Soil Plasticity**. You can also create **User defined** material models.

The new feature can handle large compressive strains, and it is furbished with compression cap and tension cutoff models.

PLASTICITY IMPROVEMENTS

The following improvements have been made:

- A new **Backward Euler, damped** option is available as a local method in **Plasticity**.
- The associative flow rule for **Tresca** criterion has been improved.
- A new **Modified Johnson-Cook** isotropic hardening model is available in **Plasticity**.
- A revamped implementation of the **Capped Drucker-Prager** model is available in the **Porous Plasticity** node.
- The **Void volume fraction** option has been removed from the **Nonlocal Plasticity** section in the **Porous Plasticity** node.

HYPERELASTICITY IMPROVEMENTS

The following improvements have been made to hyperelastic materials:

- The **Porous Plasticity** subnode has been added to the **Hyperelastic Material** node in Solid Mechanics.

- The material properties of the Yeoh model in the **Rubber** material available in the **Built-in** material library have been updated.
- The **Plasticity**, **Creep**, **Viscoplasticity**, and **Polymer Viscoplasticity** subnodes have been added to the **Hyperelastic Material** node in the Membrane interface.
- The **Creep**, **Viscoplasticity**, and **Polymer Viscoplasticity** subnodes have been added to the **Hyperelastic Material, Layered** node in the Shell interface.
- The **Creep**, **Viscoplasticity**, and **Polymer Viscoplasticity** subnodes have been added to the **Hyperelastic Material** node in the Layered Shell interface.
- The **Creep** and **Viscoplasticity** subnodes have been added to the **Hyperelastic Material** node in the **Thin Layer** and **Thin Layer, Pair** subnodes of the Solid Mechanics interface.
- All the predefined material models in the **Hyperelastic Material** node now define the elasticity matrix.
- All the predefined material models in the **Hyperelastic Material** node now define the equivalent elastic moduli.
- The **Hyperelastic Material** node is supported in the **Solid Mechanics** interface for 1D and 1D axisymmetric components.

CREEP IMPROVEMENTS

The user interface for the **Creep** node has been updated. The **Formulation** list has been removed from the **Creep** and **Additional Creep** subnodes. The formulation is now controlled in the parent node.

VISCOPLASTICITY IMPROVEMENTS

The following improvements have been made:

- The default solver settings are improved for the **Viscoplasticity** subnode when added to the **Linear Elastic Material, Layered** node in the Shell and Membrane interfaces.
- The Johnson–Cook material model has been removed from the **Viscoplasticity** subnode. It can be found as an option under **Plasticity**.
- The associative flow rule for **Tresca** criterion has been improved.

PHASE FIELD DAMAGE IMPROVEMENTS

The following improvements have been made:

- There are new and improved result variables for **Phase field damage**.
- The default solver suggestions for **Phase field damage** have been improved.

DIAPHRAGM ACCUMULATOR

This example demonstrates how to model enclosed cavities inside a diaphragm accumulator. A rubber diaphragm separates the hydraulic fluid from an inert, compressible gas. Accumulators can serve various purposes. For example, they can provide temporary energy storage for the hydraulic fluid and compensate for pressure and volume fluctuations, thereby dampening pulsations and shocks.

PHASE FIELD MODELING OF DYNAMIC CRACK BRANCHING

In this example, a benchmark problem in dynamic fracture of brittle materials is analyzed using the AT1 phase-field damage model. An instantaneous tensile load is applied to a planar tension specimen with a preexisting crack. Initially, the crack propagates perpendicular to the loading direction, after which the crack branches off symmetrically until catastrophic failure occurs.

Optimization Module

New Functionality in Version 6.3

NEW EGO GRADIENT-FREE SOLVER

The efficient global optimization (EGO) solver has been added as a new gradient-free optimization solver. This solver shares functionality and settings with the Uncertainty Quantification Module, but it only requires a license for the Optimization Module. It is the first global optimization solver, so it is significantly slower than the existing local methods, but it is less prone to converge to local minima, which is important for difficult optimization problems, typically within the frequency domain. The solver does not require initial values for the control parameters, and it is possible to inspect the Gaussian response afterward. It is recommended to evaluate the output of EGO using one of the local optimization methods.

IMPROVED GRADIENT-BASED OPTIMIZATION

Gradient-based optimization with the time-dependent solver uses a new discrete sensitivity method that is more robust, has better accuracy, and is faster. Transient parameter estimation problems are also significantly faster when solved with the SNOPT or IPOPT solvers because the aggregated sensitivity of the entire objective is computed in a single pass instead of using separate computations for each measurement point. The old continuous sensitivity method is still available, but it is no longer the default method for transient optimization. By default, both methods reduce memory consumption by using checkpointing (that is, recomputation of the forward solution), but for small problems the computational time can be reduced by disabling this.

ADDITIONAL IMPROVEMENTS

- The features in the **Shape Optimization** interface now include support for selecting individual points (or edges in 3D) where the normal continuity should be preserved. Previously, it was only possible to select all or no points.
- The **Density Model** feature in the **Topology Optimization** interface now supports coordinate systems for easier setup of milling directions that are not aligned with the global coordinate system.
- The **Global Least-Squares Objective** in the **Parameter Estimation** interface has been renamed to **Least-Squares Objective** because it now supports coordinate columns.

The **Global Least-Squares Objective** (under **Global Definitions**) is unaffected by this change. Furthermore, there are now more ways of using parameter continuation when doing parameter estimation.

- It is now possible to set up more advanced eigenfrequency optimization problems because the **Eigenfrequency** study step now includes support for filtering and sorting of the eigenvalues.
- The **Control Function** feature now supports physical quantities, different extrapolation options, as well as a nonuniform initial value. Bernstein polynomials with orders larger than 33 were causing numerical problems, so they are no longer supported in the user interface.
- The **Control Variable Field** feature under component definitions no longer has bounds enabled by default. The feature now includes support for physical quantities.
- The **Study step** list in all optimization study steps has been changed when a gradient-based optimization solver is selected. Now, the setting is always visible if (and only if) there is more than one supported study step in the study.
- The Coordinate search solver has been removed from the **Optimization** study step.

Backward Compatibility with Version 6.1

The Monte Carlo solver has been removed from the **Optimization** study step.

Backward Compatibility with Version 6.0

The **Free Shape Boundary** and **Free Shape Shell** behave differently in 3D, but the change is limited to lines where the features intersect with **Symmetry/Roller** features. Specifically, in previous versions, the continuity of the slope was not preserved at internal points on such lines, but as of version 6.1, the continuity of the slope is preserved.

Particle Tracing Module

New and Improved Functionality in Version 6.3

IMPROVEMENTS TO THE ROTATING FRAME FEATURE

The **Rotating Frame** feature now automatically adds variables that evaluate the particle positions and velocities with respect to the inertial (laboratory) reference frame. These variables can be used to specify other variables that depend on the coordinates in the inertial frame, for example, in particle–field interactions. These variables are helpful in visualizing the particle trajectories in the inertial reference frame.

The compatibility of the **Rotating Frame** feature with several other physics features has been significantly expanded. The accuracy of the particle dynamics under the combined influence of external forces and the fictitious forces associated with the **Rotating Frame** feature has also been improved. Physics features with improved compatibility with the **Rotating Frame** feature include:

- **Drag Force**
- **Lift Force**
- **Brownian Force**
- **Gravity Force**
- **Acoustophoretic Radiation Force**
- **Electric Force**
- **Magnetic Force**
- **Dielectrophoretic Force**
- **Magnetophoretic Force**
- **Thermophoretic Force**
- **Convective Heat Losses**
- **Radiative Heat Losses**
- **Droplet Evaporation**
- **Nozzle Domain**
- **Charge Accumulation**
- **Thermal Reemission**
- **Secondary Emission**

- **Force**
- **Particle–Particle Interaction**
- **Inlet**
- **Collisions**
- **Friction Force**
- **Particle–Matter Interactions**
- **Thermionic Emission**

COMPATIBILITY OF FROZEN ROTOR SOLUTIONS WITH PARTICLE TRACING IN ROTATING DOMAINS

When simulating particle trajectories in a moving mesh using the **Rotating Domain** feature, it is now possible to map the fluid velocity field solved on a stationary mesh directly onto the moving mesh in a **Time Dependent** study step. A new **Map velocity from stationary mesh to moving mesh** checkbox that enables this mapping is now available in the **Drag Force**, **Lift Force**, **Kelvin–Helmholtz Breakup Model**, **Collisions**, and **Friction Force** features.

This mapping is useful, for instance, when the **Frozen Rotor** study step is used to solve for the fluid velocity field in a rotating domain with a simple geometry. In such cases, when the velocity field defined on the stationary mesh (**Frozen Rotor** solution) is a good approximation of the velocity in the spatial frame, selecting this checkbox avoids having to solve for the velocity fields on a moving mesh.

IMPROVED USER-DEFINED COLLISIONS

In the **Charged Particle Tracing** interface, the **User Defined** subfeature under the **Collisions** feature now supports a much wider range of collisions. It is now possible to choose whether to include the primary particle in the postcollision products. In earlier versions, only one type of secondary particle species was allowed to be included, but as of version 6.3, it is possible to include multiple distinct secondary species.

To do so, a new **Species Properties** node is added as a subfeature to the **User Defined** node. In addition, the settings to control the release of secondary particles are now included in the new **Species Properties** node. Multiple instances of the **Species Properties** node can be added to account for the distinct secondary species.

CONTROL THE RANDOMNESS IN THE PARTICLE RELEASE FEATURES

All **Particle Tracing** interfaces have the **Arguments for random number generation** option that, in previous versions, controlled the seed value of the random number generators

utilized by certain physics features, such as the **Brownian Force** and **Secondary Emission** features. This option now also controls the seed used by the random number generators that the particle release features utilize. This seed can be fixed to either a default value or to a user-defined value. Both options ensure the repeatability of the initial conditions every time a model is solved. Additionally, the seed can also be determined randomly at runtime, which makes it possible to truly randomize the initial conditions and is ideal for Monte Carlo-type simulations.

The random number generators can be used to control the initial position, velocities, and release times as well as the initial values of the auxiliary dependent variables in all of the **Particle Tracing** interfaces. Additionally, in the **Particle Tracing for Fluid Flow** interface, the distributions of particle masses or diameters can also be affected by the random number generators.

NEW OPTIONS FOR INITIALIZING PARTICLE VELOCITY

The particle velocity can now also be initialized by specifying their momentum or kinetic energy. When specifying the particle momentum, it is possible to specify the vector components or the magnitude of momentum and sample the vector directions from spherical, hemispherical, conical, or Lambertian distributions. All of these sampling distributions are now also available when the velocity is initialized using the kinetic energy.

NEW OPTIONS FOR SELECTING PARTICLE GROUPS

For the **Accumulator (Domain)**, **Accumulator (Boundary)**, **Number Density Calculation**, **Particle Counter**, and **Velocity Reinitialization** features, it is now possible to perform computations for a single group of particles. Particles can be grouped by either release features or particle properties. For example, the **Number Density Calculation** feature can be used to compute the number density of all particles, particles released by a particular release feature, or particles with a specified set of properties.

Similarly, the **Particle Counter** feature can be used to count only particles with a specified set of properties that are located in a set of selected domains or on a set of selected boundaries.

NEW TIME-DEPENDENCY OPTIONS FOR ELECTRIC FIELDS IN THE CHARGE ACCUMULATION FEATURE

It is now possible to add time dependencies to the electric fields specified in the **Charge Accumulation** feature. The available time-dependency options are **Time harmonic** or **Periodic**. These options allow an electric field computed in either a time or frequency

domain to be used directly in a **Time Dependent** study step by augmenting the temporal behavior of the electric field.

New Models in Version 6.3

TUBULAR CENTRIFUGE

This model demonstrates the centrifugal separation of particles based on their densities in a tubular centrifuge. The particle–fluid mixture enters the tube, which rotates at several thousand rpm, inducing their sedimentation along the walls of the tube.

The particle motion is solved in the noninertial reference frame using the **Rotating Frame** feature. The particles are thus subject to both fictitious forces and external forces such as the drag force. This model also demonstrates the process of restarting a particle tracing simulation with the initial conditions taken from a previous study step.

RADIO-FREQUENCY QUADRUPOLE ION TRAP

This model demonstrates the conversion of a continuous ion beam into a pulsed beam. A highly energetic ion beam is confined radially in a radio-frequency quadrupole and is cooled using a process known as buffer-gas cooling. The cooled beam is spatially confined (that is, bunched) using an appropriate trapping potential along the axis of the quadrupole. This bunched beam is then released by altering the axial potential.

Backward Compatibility with Version 6.2 and Earlier

USER-DEFINED COLLISIONS

If a model built in version 6.2 or earlier that utilizes the **User Defined** feature is opened in version 6.3, the new implementation of the feature will automatically be applied. If the **Include secondary emission** checkbox is selected in the old model, one instance of the **Species Properties** subnode will be added to the **User Defined** node. All of the properties corresponding to the secondary emission in the **User Defined** node will be replicated in the **Species Properties** subnode. If the **Include secondary emission** checkbox is cleared in the old model, no instances of the **Species Properties** subnode are added.

When building a model using the COMSOL API for use with Java®, the new implementation will be used. This is true even if, for example, a Java® file created from an older version is used when the **Include secondary emission** checkbox is selected. If you want to achieve full backward compatibility using the API and you have code such as:

```
model.component("comp1").physics("cpt").feature("col1").feature("
udi1").set("Ns", 10);
model.component("comp1").physics("cpt").feature("col1").feature("
udi1").set("OffsetInitialPositions", true);
```

it will need to be changed to:

```
model.component("comp1").physics("cpt").feature("col1").feature("
udi1").feature("spp1").set("Ns", 10);
model.component("comp1").physics("cpt").feature("col1").feature("
udi1").feature("spp1").set("OffsetInitialPositions", true);
```

You can then remove the code corresponding to the **Include secondary emission** checkbox:

```
model.component("comp1").physics("cpt").feature("col1").feature("
udi1").set("IncludeSecondaryEmission", "1");
```

RANDOMNESS IN THE PARTICLE RELEASE FEATURES

If a model built in version 6.2 or earlier that has the **Arguments for random number generation** property set to either **Generate random arguments** or **User Defined** and if any initial conditions of the particles depend on random numbers, the initial conditions may be slightly altered when the model is solved in the current version.

NEW OPTIONS TO INITIALIZE PARTICLE VELOCITIES

If a model built in version 6.2 or earlier is opened in version 6.3, the new implementation using a **Velocity specification** list is applied automatically. Any release feature that used the **Maxwellian** or **Thermal** option in the **Initial velocity** list in the old version will now be set up so that the corresponding option is chosen in the **Velocity specification** list instead. Similarly, any feature that utilized the **Kinetic energy and direction** option in the old version will be set up to use the new equivalent option of **Specify kinetic energy** with the an **Expression** that defined the kinetic energy.

This also applies to any models built using the COMSOL API for use with Java®. Any code such as:

```
model.component("comp1").physics("cpt").feature("relg1").set("Ini
tialVelocity", "KineticEnergyAndDirection");

model.component("comp1").physics("fpt").feature("rpt1").set("Init
ialVelocity", "Maxwellian");

model.component("comp1").physics("pt").feature("rel1").set("Initi
alVelocity", "ConstantSpeedSpherical");
```

will continue to work. However, it is recommended to modify such code to conform with the new properties in the COMSOL API for use with Java®. The above code could be modified as:

```
model.component("comp1").physics("cpt").feature("relg1").set("Vel
ocitySpecification", "SpecifyKineticEnergy");
model.component("comp1").physics("cpt").feature("relg1").set("Ini
tialKineticEnergy", "Expression");

model.component("comp1").physics("fpt").feature("rpt1").set("Velo
citySpecification", "Maxwellian");

model.component("comp1").physics("pt").feature("rel1").set("Veloc
itySpecification", "SpecifyVelocity");
model.component("comp1").physics("pt").feature("rel1").set("Initi
alVelocity", "ConstantSpeedSpherical");
```

Pipe Flow Module

New Functionality in Version 6.3

OPEN BOUNDARY

A new **Open Boundary** feature has been added to the **Heat Transfer in Pipes** interface. At the open boundary, both inflow and outflow conditions are supported.

LICENSE UPDATE FOR PIPE CONNECTION COUPLING

The **Pipe Connection** multiphysics coupling is now available with a Pipe Flow Module license only. The coupling can be used to connect the **Pipe Flow** and **Laminar Flow** interfaces. In combination with licenses for the CFD Module or the Heat Transfer Module, the coupling can be used with **RANS-EVM** (Reynolds-averaged Navier–Stokes eddy-viscosity models).

NONISOTHERMAL PIPE CONNECTION

The **Pipe Connection** multiphysics coupling feature has been updated. In addition to isothermal flow, it also now supports nonisothermal flow, provided that both a **Heat Transfer** interface and a **Heat Transfer in Pipes** interface are included in the coupling.

THIN LAYER IN STRUCTURE–PIPE CONNECTION

The **Structure–Pipe Connection** multiphysics coupling now takes the existence of a **Thin Layer** between the coupled boundaries into account.

3D VISUALIZATION OF PIPES

When using the pipe interfaces, it is now possible to get a full 3D representation of pipes with an correct visualization of their shape. When using the **Pipe** dataset, it is possible to plot, for example, the stress and temperature variation across the pipe walls. Several new plots have been added to the result templates to visualize the geometry, stress, temperature, and pressure using the **Pipe** dataset.

Plasma Module

New Functionality in Version 6.3

NONISOTHERMAL PLASMA FLOW INTERFACES

There are new interfaces that automatically couple plasma interfaces with the **Laminar Flow** and **Heat Transfer in Fluids** interfaces. The coupling is implemented with the new **Nonisothermal Plasma Flow** multiphysics feature. The fluid transport and thermodynamic properties are passed automatically between interfaces. The heating resulting from plasma reactions is automatically added to the **Heat Transfer in Fluids** interface.

There is a new model that solves the heavy species energy-conservation equation: *Model of an SF6/argon Inductively Coupled Plasma Reactor*

ELECTRON IMPACT REACTIONS IN THE REACTION GROUP FEATURE

The **Reaction Group** feature now supports electron impact reactions. Reactions can be defined with electron-impact cross sections or Arrhenius parameters. The reactions can be added manually or imported from a file. It is also possible to add **Reaction Group** features and reactions using the **Plasma Chemistry** add-in.

The following new models demonstrate the use of electron impact reactions in the **Reaction Group** feature:

- *Model of an SF_6/Argon Inductively Coupled Plasma Reactor*
- *Global Model of an SF_6/Argon Plasma*

INFLOW, OUTFLOW, SYMMETRY, AND HEAVY SPECIES FLUX FEATURES

The following features were added to the **Plasma** interface: **Inflow**, **Outflow**, **Symmetry**, and **Heavy Species Flux**. With these new features, it is possible to define flow properties for large sets of heavy species in a single feature. In previous versions, flow properties could only be added as subfeatures of individual species.

NEW CIRCUIT TYPES IN METAL CONTACT

The circuit options in the **Metal Contact** feature have changed in the following way: The **Series RC circuit** option from previous versions (6.2 and earlier) is now called **Parallel C with ballast resistor**. However, the equations being solved are the same. The new **Series RC circuit** option in version 6.3 solves for a circuit where the resistance and the

capacitance are in series with the plasma source. The **Blocking Capacitor** option has been numerically improved, and it is now possible to set an initial capacitor voltage.

New and Updated Models in Version 6.3

MODEL OF AN SF_6/ARGON INDUCTIVELY COUPLED PLASMA REACTOR

This model solves for an inductively coupled plasma reactor in a mixture of sulfur hexafluoride (SF_6) and argon. It solves the plasma transport equations fully coupled with fluid flow, heat transfer, and magnetic fields. The new **Plasma Chemistry** add-in is used to import a complete plasma chemistry from a file. Important aspects and strategies for modeling electronegative discharges are discussed in the related model files.

GLOBAL MODEL OF AN SF_6/ARGON PLASMA

This model studies a plasma chemistry of SF_6 and argon mixtures at moderate pressures using a global model. With this model, it is possible to study the mechanisms of dissociation, attachment, and recombination, which are important in such discharges. The heavy species heat equation is solved because the gas temperature can reach significant values in the high-power region simulated here. Parametric sweeps for input powers from 300 to 2000 W and argon mole fraction from 0.1 to 0.9 are computed.

Backward Compatibility with Version 6.2 and Earlier

The **Inlet**, **Outflow**, **Flux**, and **Symmetry** subfeatures are deprecated. When opening models created with version 6.2 and earlier, a warning message will appear: "This feature is obsolete and will be removed in a future version. Consider using the [Inflow/Outflow/Heavy Species Flux/Symmetry] feature instead".

The circuit options in the **Metal Contact** feature have changed in the following way The **Series RC circuit** option from previous versions (6.2 and earlier) is now called **Parallel C with ballast resistor**. The equations being solved are the same. The new **Series RC circuit** option in version 6.3 solves a different equation for a circuit where the resistance and the capacitance are in series with the plasma source. The option **Series RC circuit** used in models created with version 6.2 and earlier will be automatically changed to **Parallel C with ballast resistor** when opening a model in version 6.3.

Porous Media Flow Module

New Functionality in Version 6.3

NONEQUILIBRIUM FORMULATION FOR MOISTURE TRANSPORT IN POROUS MEDIA

In the **Moisture Transport** physics interfaces, a new option has been added for the **Hygroscopic Porous Medium** feature. It is now possible to use a nonequilibrium formulation where both the relative humidity and the liquid water saturation are solved for. With this formulation, there is no underlying equilibrium hypothesis between the vapor and the liquid phase, which extends the range of applicability of this feature.

As a consequence, two new moisture transport physics interfaces are available from the **Model Wizard**. The first one, **Nonequilibrium Moisture Transport in Porous Media**, can be used to compute both the relative humidity and the liquid water saturation in a porous media where vapor–liquid equilibrium is not assumed. The second one, **Moisture Transport in Free and Porous Media**, is parameterized to facilitate the setup of moisture transport models where different media coexist.

NEW FEATURE: BOUNDARY MASS SOURCE

A new **Boundary Mass Source** boundary condition has been added to the Phase Transport interfaces. The new feature accounts for the consumption or production of different phases due to reactions or other physical processes on boundaries. When **Phase Transport in Porous Media** is coupled to **Darcy's Law**, the net mass transfer is automatically accounted for in the flow field.

POROUS INTERFACE FOR MOISTURE TRANSPORT INTERFACES

In the **Moisture Transport** physics interfaces, a new **Porous Interface** feature has been added. This feature makes it possible to choose the modeling assumptions between different media, such as evaporating the liquid flux at the interface between a porous medium and moist air.

NEW STABILIZATION OPTION: LIMIT SMALL TIME STEPS EFFECT ON STABILIZATION TIME SCALE

A new option in the Phase Transport interfaces, **Limit small time steps effect on stabilization time scale**, can be used to avoid the loss of stabilization during initialization and for small time steps.

NEW OPTION FOR RELATIVE PERMEABILITIES

A new **Power law** option has been added to the **Porous Medium** node in the **Phase Transport in Porous Media** interface. The new option provides easier implementation of relative permeabilities based on a power-law expression.

NEW NODES IN THE PHASE TRANSPORT INTERFACES

Two new nodes, **Fluid** and **Porous Medium**, are available in the Phase Transport interfaces. The **Porous Medium** node has subnodes to define the phase properties and the properties of the porous matrix. The new nodes replace the previously available **Phase and Transport Properties** and **Phase and Porous Media Transport Properties** nodes. When opening a model created in a previous version, the old nodes are kept in the model.

NONCONSERVATIVE FORM OF HYGROSCOPIC POROUS MEDIUM

In the **Moisture Transport in Air**, **Moisture Transport in Building Materials**, and **Moisture Transport in Porous Media** interfaces, the advanced setting to solve either the conservative or nonconservative form of equations now only has an impact on the **Moist Air** feature. The equations of the **Hygroscopic Porous Medium** features are now always solved in conservative form, independently of the value of the setting.

MISCELLANEOUS IMPROVEMENTS

- In the **Moisture Transport** physics interfaces, the capillary model inputs have been moved from the **Hygroscopic Porous Medium** node, in the **Moisture Transport Properties** section, to its **Liquid Water** subnode, in the **Liquid Water Properties** section.
- Due to the introduction of a setting to use either an equilibrium or a nonequilibrium formulation for **Hygroscopic Porous Media** features in the **Moisture Transport** physics interfaces, the **Moisture Transport in Porous Media** interface has been renamed **Equilibrium Moisture Transport in Porous Media**.
- When a **Moisture Flow** coupling is applied in a porous domain, the porosity and permeability in the **Fluid Flow** interface are now automatically set using the variables from the **Moisture Transport** interface. These properties take into account the fraction of fluid in the porous medium and the relative permeability of moist air. The latter can be set in the **Moist Air** subnode of the **Hygroscopic Porous Medium** node.
- The add-in method for creating discrete fracture networks (DFN) in fractured geometries is now available in the Porous Media Flow Module Add-in Library.
- A new default color table for the **Volume Fraction** plot in Phase Transport interfaces enhances clarity and is now consistent with other interface plots.

POROELASTICITY, LARGE DEFORMATION, SOLID MULTIPHYSICS INTERFACE REMOVED

The Poroelasticity, Large Deformation, Solid multiphysics interface is no longer available from the **Add Physics** window. The reason for the removal is that, with other recent changes, it differs only marginally from the **Poroelasticity, Solid** multiphysics interface.

New and Updated Models and Applications in Version 6.3

HOMOGENIZED POROUS MATERIAL PROPERTIES APP

This app allows users to examine porous material properties by providing various microscale structures. Users can adjust the dimensions of these structures and calculate key properties like porosity and permeability. The results can also be exported for further analysis, making it a practical tool for studying porous media flow.

SUPERHEATED STEAM DRYING OF A WOOD PARTICLE

This model demonstrates the use of the **Heat and Moisture Flow** features for the simulation of superheated steam drying of a wood particle. A nonequilibrium formulation is used to compute the transport of the liquid water and vapor phases in the wood particle.

Polymer Flow Module

New Functionality in Version 6.3

ROTATING MACHINERY, NONISOTHERMAL FLOW

The **Rotating Machinery, Nonisothermal Flow** multiphysics interface has been added under the **Nonisothermal Flow** section in the **Fluid Flow** branch.

LICENSE UPDATE FOR DARCY'S LAW AND FREE AND POROUS MEDIA FLOW, DARCY INTERFACES

The **Darcy's Law** and **Free and Porous Media Flow, Darcy** interfaces are now available in the Polymer Flow Module.

NON-NEWTONIAN FLUIDS IN POROUS MEDIA

Flows of non-Newtonian fluids in porous media can be simulated using the **Darcy's Law** interface or the **Brinkman Equations** interface. The following inelastic non-Newtonian models are available in the **Fluid** node under the **Porous Medium** feature: **Power Law**, **Carreau**, **Carreau–Yasuda**, **Cross**, **Cross-Williamson**, and **Sisko**.

UPDATED OUTLET FEATURE

A new **Mass flow** option has been added to the **Outlet** feature in the **Single-Phase Flow** interfaces.

New Models in Version 6.3

HOMOGENIZATION OF NON-NEWTONIAN POROUS MEDIA FLOW

This model shows how to use pore-scale modeling of non-Newtonian fluid flow through a porous structure to find homogenized quantities for modeling non-Newtonian fluid flow at the macroscopic scale.

BLOOD FLOW IN A STENOSED PULMONARY ARTERY

In this model, a Carreau fluid is used to capture the non-Newtonian behavior of blood. The example treats a stenosis as a porous medium and demonstrates how to set up a model that combines free and porous media flow of a non-Newtonian fluid.

FLOW IN AN INTERNAL MIXER

This model studies a nonisothermal flow of a highly viscous fluid in an internal mixer. The Carreau non-Newtonian viscosity model together with an exponential thermal dependency is used to determine the temperature- and shear-dependent viscosity of the fluid.

Ray Optics Module

New and Improved Functionality in Version 6.3

GEOMETRIC MODULATION TRANSFER FUNCTION PLOT

It is now possible to create spot diagrams and geometric modulation transfer function (MTF) plots as default plots. A new section, **Results**, has been added to the **Geometrical Optics** interface where automatic plotting of spot diagrams and geometric MTFs can be enabled for imaging applications. A **Plot MTF in single plot group** checkbox is also now available that, when selected, plots all MTFs in a single plot group.

SPECIFY DISPERSION MODEL USING ABBE NUMBER

New functionality is added to the **Medium Properties** feature to define an optical dispersion model using relative refractive index and Abbe number. When the **Specify relative refractive index and Abbe number** option has been selected from the **Refractive index of domains** list, three **Optical dispersion model** options are available: **Cauchy**, **Conrady**, and **Linear**. By selecting the **Change reference wavelengths** checkbox, the reference wavelengths for the refractive index and the Abbe number definitions can be modified.

CONTROL THE RANDOMNESS IN THE RAY RELEASE FEATURES

The **Arguments for random number generation** property (available in the **Advanced Settings** section of the **Settings** window for the physics interface) now also controls the random number generators used by the ray release features.

- When the **Generate unique arguments** option is selected, the seed for the random number generator is set internally and is fixed. This option ensures that the initial conditions of the rays are identical every time the (unaltered) model is solved.
- When the **Generate random arguments** option is selected, the seed is determined randomly at runtime. This option makes it possible to truly randomize the initial conditions and is ideal for Monte Carlo-type simulations.
- The **User defined** option makes it possible to control the seed directly. When this option is selected, a new **Additional input arguments for random number generator** field appears in the **Advanced Settings** section of the **Settings** window for the release feature. The value that you specify will be used to seed the random number generator associated with the release feature.

The random number generators can also be used to control the distributions of the initial ray vector directions and release times, as well as the initial values of the auxiliary dependent variables.

ABSORPTANCE OF COATINGS

It is now possible to specify the absorptance values of the coating layers in lenses or glasses. New functionality has been added to the **Coatings** section of the **Material Discontinuity** feature's settings to make it possible to specify **Absorptance** values when either the **Specify reflectance** or **Specify transmittance** option has been selected from the **Thin dielectric films on boundary** list. Different values for **Absorptance** can be specified for s- and p-polarization by selecting the **Specify different values for s- and p-polarization** checkbox.

NEW OPTIONS TO SELECT RAYS IN THE ACCUMULATOR FEATURES

A new option has been added to the **Accumulator** features in both domain and boundary selections that you can use to select groups of rays based on ray release features. When the model has multiple release features, you can now choose an option from the **Release feature** list to let the accumulated variables be affected by either all of the rays released or by only the rays released by a specific release feature.

New and Updated Models in Version 6.3

ALVAREZ LENS

This model studies a specific freeform design called the Alvarez lens, where two complementary cubic surfaces are shifted laterally with respect to each other to achieve variable optical power.

PETZVAL LENS GEOMETRIC MODULATION TRANSFER FUNCTION

This model is completely revised and uses the newly added default geometric modulation transfer function (MTF) plot instead of using an application method as in previous versions.

PLANO-CONVEX LENS ORIENTATION

This model demonstrates the simulation of a collimated light beam through a plano-convex lens with two orientations. In the first orientation, the plane side of the lens faces the collimated light source, while in second orientation, the convex side faces the light source. The optimal orientation is analyzed using a spot diagram and a geometric modulation transfer function (MTF) plot.

RANDOMNESS IN THE RAY RELEASE FEATURES

If a model built in version 6.2 or earlier has the **Arguments for random number generation** property set to either **Generate random arguments** or **User defined** and if any of the initial conditions of the rays depend on random numbers, the initial conditions of the rays may be slightly altered when the model is solved in the current version.

RF Module

New Functionality in Version 6.3

NEW PREDEFINED MULTIPHYSICS INTERFACE FOR TRANSMISSION LINE PARAMETER ANALYSIS

A new multiphysics interface, **Transmission Line, RLGC Parameters**, combines an **Electric Currents** interface with a **Magnetic Fields** interface to compute transmission line parameters including series resistance, series inductance, shunt conductance, and shunt capacitance, calculated per unit length for each parameter. This multiphysics interface also computes the characteristic impedance and propagation constant. Frequency-domain modeling is supported in 2D.

NEW PHYSICS INTERFACE: TRANSMISSION LINE, TRANSIENT

A new physics interface, **Transmission Line, Transient**, solves the time-domain transmission line equation for the electric potential. This interface can be used to study wave propagation in the time domain along one-dimensional transmission lines.

INTERFACE TOOLBAR BUTTON FOR QUICK MODELING

A new toolbar button integrated into the **Electromagnetic Waves, Frequency Domain** interface automatically configures lossy physics features, such as the **Transition Boundary Condition** and **Impedance Boundary Condition**, based on the selection characteristics of material definitions.

NEW DEFAULT PLOT FOR MORE INTUITIVE VISUALIZATION

A new default plot has been added to the **Electromagnetic Waves, Frequency Domain** interface. This plot is based on the characteristics of physics features and selections and uses a logarithmic scale to provide a more intuitive visualization of field intensity decay. It also emphasizes conductive surfaces with an enhanced 3D perspective, positioning the camera more inward.

UPDATED PART LIBRARY WITH NEW CONNECTORS

The Part Library offers not only simplified versions of edge launch connectors for faster simulation but also a full range of high-precision edge launch connectors from Signal Microwave, LLC. This update also includes new top and vertical launch connectors, as well as extended-length edge launch connectors.

EXTENDED USAGE OF PHYSICS FEATURES IN THE ELECTROMAGNETIC WAVES, BOUNDARY ELEMENTS INTERFACE

The **Perfect Electric Conductor** and **Lumped Port** features can now be applied to voids as well as the exterior surfaces of domains. This improvement makes it possible to use the **Electromagnetic Waves, Boundary Element** interface for simulating printed circuit boards.

REVISED SOLVER CONFIGURATION FOR THE ELECTROMAGNETIC WAVES, TIME EXPLICIT INTERFACE

The **Runge–Kutta** method in the **Time Stepping** setting now utilizes the **Cash–Karp 5** option, which can provide faster computation.

ENHANCED REFLECTION COEFFICIENT WITH MULTIPLE EXCITATION

The generation of the **Reflection coefficient with multiple excitation** output variables is no longer limited to scenarios where all ports are excited. Variables can also be generated when multiple ports are partially excited, supporting more dynamic scenarios in phased array antenna simulations.

ADDITIONAL VARIABLE WITH THE SCATTERED FIELD FORMULATION

A monostatic radar cross-section output variable is now available in 2D and 3D for the **Electromagnetic Waves, Frequency Domain** interface when using the scattered field formulation.

IMPROVED PHYSICS-CONTROLLED MESH FOR HIGHER-ORDER ELEMENTS

The physics-controlled mesh for the **Electromagnetic Waves, Frequency Domain** interface has been customized for higher-order elements, enabling more efficient computation with higher-order discretization, such as with cubic or quartic elements.

MATCHED BOUNDARY CONDITION ADDED TO THE ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN INTERFACE

The **Matched Boundary Condition** is now available for use with the **Electromagnetic Waves, Frequency Domain** interface. This boundary condition is similar to the **Scattering Boundary Condition**. However, whereas the **Scattering Boundary Condition** mainly absorbs the waves propagating in the normal direction to the boundary, the **Matched Boundary Condition** absorbs waves propagating in the directions specified in its settings.

SIMPLIFIED POLARIZATION SETTINGS FOR PORTS

There are new settings for specifying the polarization for **Port** features of the **Periodic** type. In addition to the existing possibility to specify the electric or magnetic mode

field amplitude, it is now possible to directly specify linear transverse electric (TE) or transverse magnetic (TM) polarization or right-handed or left-handed circular polarization.

When automatically adding **Diffraction Order** and **Orthogonal Polarization** ports by clicking the **Add Diffraction Orders** button, the refractive index can now be selected from the domain adjacent to the **Port**. Instead, if the **User defined** option is selected as the **Refractive index, real part**, the entered value can now only be a scalar parameter expression, as anisotropic refractive index values are no longer supported by the periodic ports.

Furthermore, the number of ports added can be controlled by either adding only the ports necessary for the parameter values present when clicking the **Add Diffraction Orders** button or by adding all **Diffraction Order** ports relevant for covering all angles of incidence.

NEW DEFAULT VALUES FOR PORTS

For **Port** nodes of the **Periodic** type, the mode field amplitude should never be zero. Thus, there are now nonzero default values for the **Electric mode field amplitude** and the **Magnetic mode field amplitude**.

Similarly, for **Port** nodes of the **User defined** type, there are nonzero default values for the **Electric mode field** and **Magnetic mode field** fields. Furthermore, the **Propagation constant** is now given the default value `emw.k`, where `emw` is the name of the physics interface.

NEW GLOBAL PORT MODE FIELD VARIABLES

Port features of the **Periodic** type, **Diffraction Order** ports, and **Orthogonal Polarization** ports now create variables that can be useful for analyzing the periodic problem. Variables representing the following values are now available for evaluation:

- Electric mode field amplitude
- Electric mode field
- Port mode wave vector
- Elevation angle
- Azimuth angle

The elevation and azimuth angle variables are available both for each port name and for each mode number. Since the variables are globally available, they can be evaluated on any type of selection and not just on the port boundary.

ELECTRIC POINT DIPOLE ADDED IN 2D AXISYMMETRY

In 2D axisymmetry, it is now possible to add the **Electric Point Dipole** node to the **Electromagnetic Waves, Frequency Domain** interface. The node is only enabled when **Electric field components solved for** is set to either **Three component** or **In-plane vector**. This point feature should only be added to points located on the symmetry axis. Furthermore, the implementation is restricted to the **Azimuthal mode number** $m = 0$.

NEW VARIABLES AND DEFAULT PLOTS FOR THE SCATTERED FIELD FORMULATION

When solving models using the **Electromagnetic Waves, Frequency Domain** interface in the **Scattered field** formulation, new variables are available for the following quantities:

- Instantaneous background electric field norm
- Background plane wave intensity, when **Background wave type** is set to either **Linearly polarized plane wave** or **Circularly polarized plane wave**
- Background Gaussian beam intensity
- Background Gaussian beam maximum intensity
- Background Gaussian beam total power

All variables are available for global evaluation, except for the domain variables for the instantaneous background electric field norm and background Gaussian beam intensity.

By default, the **Scattered field** formulation simulations now automatically add a plot of the instantaneous background electric field norm.

IN-PLANE WAVELENGTH USED AS THE TYPICAL WAVELENGTH FOR PERFECTLY MATCHED LAYERS

In 2D, it is possible to specify a prescribed out-of-plane wave vector component k_z. When calculating the typical wavelength to use in **Perfectly Matched Layer** nodes, the in-plane wavelength is now used. That is, given the wave number k, the in-plane wave number is:

$$k_T = \sqrt{k^2 - k_z^2}$$

and the typical wavelength is given by:

$$\lambda_{PML} = 2\pi / k_T .$$

ELEVATION AND AZIMUTH ANGLE FAR-FIELD VARIABLE NAME UPDATES WITH USER-DEFINED FAR-FIELD VARIABLE NAME

The far-field variables `ewfd.Efartheta` and `ewfd.Efarphi` now update their names when the **Far-field variable name** has been changed in the settings for the **Far-Field Calculation** node. By default, the **Far-field variable name** is `Efar`. However, if it has been changed to `Efar2`, the new variable names will be `emw.Efar2theta` and `emw.Efar2phi`, respectively.

New and Updated Models in Version 6.3

BOW TIE ANTENNA OPTIMIZATION

A bow tie antenna patterned on a dielectric substrate is optimized by adjusting the length of the arms and the flare angle to reduce the magnitude of S_{11}, the reflection coefficient. The two geometric dimensions used as design variables directly control the antenna's size and shape and also affect the dimensions of the dielectric substrate. The efficient global optimization (EGO) method is used to improve the objective function, S_{11}, and a reliability analysis is conducted to assess the robustness of the optimized design with respect to design parameter variations.

MESH ADAPTATION STUDY FOR A MICROSTRIP PATCH ANTENNA

The model uses a dedicated mesh refinement study, **Frequency Domain, RF Adaptive Mesh**, which dynamically refines the mesh in the region of interest. For the Microstrip Patch Antenna model from the RF Module application library, it increases the mesh resolution around areas with high field variations. The primary goal of mesh adaptation is to improve the accuracy of the simulation by allocating more computational resources to important parts of the simulation domain. This helps in precisely evaluating the resonant frequency for high-Q or narrow-band devices.

UNCERTAINTY QUANTIFICATION STUDY FOR A MICROSTRIP PATCH ANTENNA

In this tutorial model, an uncertainty quantification analysis is performed on the Microstrip Patch Antenna model from the RF Module application library to explore how variations in input parameters, such as material properties or geometric variations, impact the antenna's performance in terms of S-parameters.

ROTATING MICROWAVE OVEN WITH PHASE TRANSITION

A spherical water-rich load (a potato) is placed on the rotating glass tray of a microwave oven some distance from the center of rotation. 1 kW of 2.45-GHz microwave power

is applied via a waveguide feed. The plate rotates at 9 degrees per second. The heat-transfer model involves the transition from a water-rich phase to a dehydrated phase, with the corresponding changes in material properties and microwave losses. In the simulation, the rotation is performed stepwise, with one step every 0.25 s, and the mesh is tailored to assure mesh conformity in each angular step. The model makes use of the **Phase Change Material** feature that is readily available in the **Heat Transfer in Solids** interface. The stepwise motion that, in combination with the tailored structured mesh on the sliding mesh boundaries, assures conforming meshes, is implemented using the **Events** interface in the COMSOL Multiphysics core product.

DIELECTRIC RESONATOR ANTENNA

The existing Dielectric Resonator antenna model has been simplified for a faster modeling process, and its frequency range is now tuned for X-band applications.

Rotordynamics Module

New Functionality in Version 6.3

IMPROVED ESTIMATION OF DYNAMIC COEFFICIENTS IN HYDRODYNAMIC BEARING INTERFACE

The built-in functionality for determining dynamic coefficients in hydrodynamic bearings has been significantly improved for increased accuracy. The functionality is based on an infinitesimal perturbation method, which is used to determine the pressure perturbations due to translations of the runner. These improvements make it possible to use pressure-dependent fluid properties and are available for all three options: **Liquid (Reynolds equation)**, **Liquid with cavitation**, and **Gas (modified Reynolds equation)**.

NEW AND IMPROVED PLOTS IN RESULT TEMPLATES

The **Campbell Diagram** has been improved. The new functionality makes it easier to distinguish between forward and backward modes. The new plot is also created much faster than in the previous version.

In addition, two new plots have been added to the result templates for eigenfrequency study types: the **Stability Diagram** and the **Frequency Variation Diagram**. The stability diagram can be used to determine the onset of instability, while the frequency variation diagram is useful for visualizing how natural frequencies change as a function of a swept parameter.

The unwrapped plots, **Unwrapped Fluid Pressure** and **Unwrapped Velocity**, have been improved for better readability.

IMPROVEMENTS FOR JOURNAL BEARING

The **Cylindrical Hydrodynamic** bearing type in the **Journal Bearing** feature has been improved for increased accuracy. The feature now uses a nonlinear formulation for the fluid forces, which is based on the Reynolds equation with a short bearing approximation.

SOLVER SUGGESTIONS FOR THE HYDRODYNAMIC BEARING INTERFACE

Solver suggestions have been added to the **Hydrodynamic Bearing** interface. The parallel sparse direct solver (PARDISO) is now the default solver for stationary as well as time-dependent studies. As a result, the computation time can be reduced by up to 50%.

RENAMING OF BUILT-IN JOURNAL BEARING TYPES

The **Plain** and **Elliptic** journal bearing types have been renamed to **Cylindrical** and **Elliptical**, respectively.

UPDATED LOAD FEATURES

For details, see Updated Load Features in the release notes for the Structural Mechanics Module.

New Models in Version 6.3

ELASTOHYDRODYNAMIC LUBRICATION IN A CYLINDRICAL JOURNAL BEARING

In this example, the importance of including elastic deformations when analyzing heavily loaded fluid-film bearings is investigated. The structural parts are modeled using the **Solid Mechanics** interface, while the pressure in the lubricant is determined by means of the **Hydrodynamic Bearing** interface.

The journal and the bearing are assumed to be made of aluminum, and the lubricant corresponds to a generic oil with pressure-dependent fluid properties.

The elastohydrodynamic results are compared with a simplified hydrodynamic simulation.

SIMPLY SUPPORTED BEAM ROTOR

In this model, an eigenfrequency analysis of a rotor with multiple disks and hydrodynamic bearings is conducted. This model illustrates how to use Campbell and stability diagrams to identify critical speeds and the stability threshold.

The rotor is modeled using the **Beam Rotor** interface, and the disks and bearings are modeled using the built-in **Disk** and **Journal Bearing** features.

Semiconductor Module

New Functionality in Version 6.3

TRANSPORT OF CHARGE CARRIERS INTERFACE

The new **Transport of Charge Carriers** interface enables the modeling of charge carriers, such as electrons, holes, ions, and neutral species like molecules, and their excited states. It solves for the number density of these carriers, accounting for their transport and reactions. The interface handles drift, convection, and diffusion, driven by electromagnetic fields, flow fields, or concentration gradients. It can be applied in various semiconductor and quantum systems, such as organic semiconductors when coupled with the **Electrostatics** interface, ion-sensitive field-effect transistors (ISFETs) when used with the **Semiconductor** interface, and quantum mechanical models when integrated with the **Schrödinger Equation** interface.

You can explore this new interface in tutorial models such as *isfet* and *egofet*.

MIXED FINITE ELEMENT FORMULATION

The new mixed finite element formulation improves the resolution of small-level dark currents, which are critical in certain semiconductor devices. Traditional methods often struggle with this due to the cancellation effects between drift and diffusion currents. By introducing additional dependent variables for electron and hole currents, and using a divergence element to locally enforce current conservation, the new formulation offers a more accurate and reliable solution for resolving dark currents in semiconductor models.

You can explore this new functionality in the *reverse_bias_leakage_current* model.

SILICON CARBIDE MATERIAL DATA

The Semiconductor material library has been updated to include new data for silicon carbide (SiC). This update covers various material properties, such as impact ionization, the Arora mobility model, the Caughey–Thomas mobility model, direct recombination, and more.

You can explore this new material in the *sic_diode_breakdown* model.

SILICON CARBIDE DIODE BREAKDOWN

This model shows how to model the avalanche breakdown due to the impact ionization in a silicon carbide (SiC) diode. The current–voltage (I–V) characteristics of the device are presented as well as the electric field distribution plot. Furthermore, the carrier generation term has been computed to demonstrate the pathways of the breakdown current.

FIN FIELD-EFFECT TRANSISTOR

This model shows how to model a FinFET in 3D. The I–V characteristics of the device is represented. First, the gate voltage is swept to obtain the drain-current-vs.-gate-voltage plot. Then the drain-current-vs.-drain-voltage characteristics are computed for fixed gate voltages.

MOSFET WITH EXPLICIT METAL AND DIELECTRIC DOMAINS

This example is an adaptation of our DC Characteristics of a MOS Transistor (MOSFET) model where the metal and dielectric domains are modeled explicitly and not via a boundary condition. Therefore, the potential profile inside the metal and the insulator can be observed.

REVERSE BIAS LEAKAGE CURRENT

In a diode or a transistor, when a p–n junction is reverse biased (when the p-side is connected to a lower potential than the n-side), ideally, no current should flow. However, due to minority carriers (electrons in the p-side and holes in the n-side), a small current, known as the reverse-bias leakage current, does flow. This example shows how to use the mixed formulation to calculate the small leakage current accurately.

Structural Mechanics Module

New Functionality in Version 6.3

ELECTROMECHANICS MULTIPHYSICS INTERFACES

New multiphysics interfaces for modeling electromechanics for thin structures have been added:

- The new **Electromechanics, Shell** multiphysics interface consists of a **Shell** interface, an **Electrostatics** interface, and an **Electromechanics, Boundary** multiphysics coupling.
- The new **Electromechanics, Membrane** multiphysics interface consists of a **Membrane** interface, an **Electrostatics** interface, and an **Electromechanics, Boundary** multiphysics coupling.
- As an effect, the old Electromechanics multiphysics interface has been renamed to **Electromechanics, Solid**.
- A new **Electromechanics, Boundary** multiphysics coupling is now available.
- The old Electromechanical Forces multiphysics coupling has been renamed **Electromechanics**.

SHRINKAGE AND SWELLING MULTIPHYSICS COUPLING

A new **Shrinkage and Swelling** multiphysics coupling, which connects the **Solid Mechanics** and **Transport in Solids** interfaces, has been added. You can use it to model the effects of volume changes due to diffusion.

SIMPLIFIED MODELING OF FASTENERS

In the **Shell** interface, a new **Fasteners** feature has been added. You can use it to model a large number of rivets or bolts connecting shell boundaries. Coincident holes on adjacent surfaces can be automatically detected and connected. You can also evaluate, for example, safety factors for each connector.

SPOT WELDS

The new **Spot Welds** feature in the **Shell** interface makes it possible to model structures with a large number of spot welds. The implementation is mesh independent, and the locations of the weld nuggets are given as a list of coordinates.

CONTACT FORMULATION FOR INTERIOR BOUNDARIES

A simplified contact formulation that can be used without contact pairs and assemblies has been introduced. The feature, which is available in the **Solid Mechanics** interface is called **Interior Contact**. It can be applied to internal boundaries between solid domains.

Friction, **Adhesion**, and **Decohesion** subnodes can be added to the **Interior Contact** node.

FITTING OF VISCOELASTIC DATA

User-defined frequency-dependent viscoelastic data in terms of loss and storage moduli can be approximated using functions that can be directly used in a time-domain analysis. The same technique can also be used for materials defined using fractional derivatives. In the **Viscoelasticity** node, this is controlled in the **Time Domain and Eigenfrequency** section. This section is available when the **Material model** is selected as **User defined** or when **Standard linear solid** is selected together with **Use fractional derivatives**.

ELECTRODE SURFACE COUPLING

An **Electrode Surface Coupling** node has been added to the **Transport in Solids** interface. It can be used to specify the mass flux into or out of the model domain based on current densities specified in an **Electrochemistry** interface.

PREDEFINED MOBILITY TYPES IN CHEMICAL POTENTIAL

In the **Chemical Potential** node in the **Transport in Solids** interface, two predefined models for mobility have been added: **Linear** and **Quadratic**. The default is still **User defined**.

Also, a **Mobility** section has been added to the **Solid** node in the **Transport in Solids** interface. Its purpose is to provide the mobility to the **Shrinkage and Swelling** multiphysics coupling to compute the stress-induced flux. The section is visible when **Advanced Physics Options** is selected.

DIFFUSIVITY FROM MATERIAL IN TRANSPORT IN SOLIDS

In the **Solid** node in the **Transport in Solids** interface, the diffusivity can now be taken from a material under **Materials**. This is analogous to how diffusivity is handled in the **Transport of Diluted Species** interface.

3D VISUALIZATION OF BEAMS

When using the **Beam** and **Pipe Mechanics** interfaces, it is now possible to get a full 3D representation of beams. This is accomplished through two new types of datasets, **Beam**

and **Pipe**. When using these datasets, it is possible to plot, for example, the stress variation across beams in a 3D plot.

UPDATED LOAD FEATURES

The features for entering loads have been updated for all structural mechanics interfaces. This change affects nodes like, for example, the **Boundary Load, Edge Load**, and **Point Load** nodes. The most visible change is that you can now choose between a load applied in the initial configuration or applied in the deformed configuration.

In physics interfaces with rotational degrees of freedom, you can now also specify the total moment acting on the selection.

When using the **Resultant** type in a load feature, the effect of finite deformations is now taken into account.

Several details have also been homogenized between different load types and different physics interfaces, and new variable names have been introduced.

In the predefined load plots, it is now always the force per deformed area that is displayed.

When using the COMSOL API for use with Java®, you may encounter some backward compatibility issues with earlier releases, in particular, if this is related to when feature inputs are used to provide forces from other physics interfaces or if load plots have been created manually.

AUTOMATIC SIDE DETECTION IN BOUNDARY-TO-BOUNDARY COUPLING

In the **Boundary to Boundary** coupling node, it is now possible to detect the connected side of the shell (top or bottom) automatically. As an effect, the settings have changed slightly. The new **Connected location** list is set to **Automatic** by default, but you can set it to **Manual** to allow detailed control. When opening an old model, the manual mode is selected to maintain full backward compatibility.

IMPROVED STRAIGHT EDGE CONSTRAINT

The design of the **Straight Edge Constraint** node in the **Truss** interface has been improved to provide better control over when the constraint is active or not.

NEW DEFAULT CONSTRAINT METHOD IN SHELL CONNECTIONS

In the **Boundary to Boundary**, **Edge to Boundary**, and **Edge to Edge** couplings in the Shell interface, the default value for the **Constraint method** has been changed from **Elemental** to **Nodal**. This will generate substantial speedups for some configurations.

IMPROVED PERFORMANCE IN COUPLINGS BETWEEN BOUNDARIES

In the **Solid–Thin Structure Connection** and **Layered Shell–Structure Cladding** multiphysics couplings, the search for corresponding points on connected boundaries can be computationally intensive when many elements are involved in the coupling. You now have the option to use a faster but potentially less robust method. This is done by selecting **Manual** as the **Search method** in the **Advanced** section of these multiphysics couplings.

THIN LAYER IN MULTIPHYSICS COUPLINGS

Multiphysics couplings that connect boundaries now take into account the existence of a **Thin Layer** between the coupled boundaries. It is, for example, possible to add a shell on top of a solid while at the same time taking a thin glue layer into account. The affected couplings are:

- **Solid–Thin Structure Connection**
- **Solid–Beam Connection**
- **Structure–Pipe Connection**
- **Layered Shell–Structure Cladding**
- **Layered Shell–Structure Transition**

LINEAR ELASTIC MATERIAL PROPERTIES FOR PIEZOELECTRICITY

Linear elastic material properties (isotropic, orthotropic, or fully anisotropic) can now be used directly by a **Piezoelectric Material** node. Stress-charge and strain-charge forms are supported.

ENHANCEMENTS TO ENCLOSED CAVITY

The **Enclosed Cavity** functionality for modeling enclosed fluids has been extended:

- The feature is now also available in the **Membrane** interface.
- You can add a **Filter** subnode to specify if only parts of boundaries should contribute to the fluid volume. This is useful, for example, when a piston is traveling through a cylinder.
- In the **Advanced** section of the **Enclosed Cavity** node's **Settings** window, you can enter a value for a fixed external volume. This is a part of a system that is not explicitly modeled but will contribute to the total volume when computing pressure changes, for example.
- You can now select boundaries that are not part of the physics interface itself when defining the cavity boundaries.

ENHANCEMENTS TO CELL PERIODICITY

There are several enhancements for the **Cell Periodicity** feature:

- You can now also use a finite strain formulation.
- For the average strain and average stress boundary conditions, it is now possible to include a scale factor to the applied boundary conditions. This makes it possible to incorporate some types of nonlinear materials.
- There are more choices for effective properties that can be computed, and the choices are independent of each other.

STABILIZATION IN MEMBRANE INTERFACE

It is common that membrane structures are unstable before a load has been applied. In order to remedy this, you can now add a **Stabilization** node in the **Membrane** interface. This feature will add a stiffness in the normal direction, which is then gradually removed.

SAFETY FACTOR USING GRIFFITH FAILURE CRITERION

The Griffith failure criterion has been added to the **Safety** node. This criterion is suitable for fractures in brittle materials.

COMBINED SAFETY FACTORS

When using several **Safety** nodes with the same selection, you now also have access to a set of variables containing the worst-case combination of the individual criteria.

PRESCRIBED ANGULAR VELOCITY

For the **Rigid Connector** and **Rigid Material** features, a new option has been added for prescribing the rotation. In addition to explicitly giving the rotation, it is now also possible to prescribe the angular velocity.

FOLD-LINE CONSTRAINTS

The **Fold-Line Settings** section in the settings for the **Shell** interface has been updated. When using pointwise constraints (the default), it is now possible to choose between a nodal or an elemental application of the constraints.

ROTATION VECTOR IN THE SHELL INTERFACE

The expressions defining the variables for the rotation vector (`shell.th`) in the **Shell** interface have been revised. The rotation vector is primarily considered as an output quantity, computed from the actual rotational degrees of freedom. With the new

formulation, however, it is differentiable in such a way that it can be used also in weak-form expressions.

STRESS LINEARIZATION

When using the **Distributed** type in the **Stress Linearization** node, you can enter the start coordinates of a stress classification line. In previous versions, the given point had to be located on the boundary. This requirement has now been relaxed so that the given point is projected to the nearest location on the boundary.

Also, the **Postprocessing** section in the settings for the **Stress Linearization** node has been renamed **Results**.

ENERGY CONTRIBUTION OF SPRING FOUNDATION

The various **Spring Foundation** features now contribute to the total stored energy variable, `Wh_tot`.

GENERALIZED POWER LAW FOR DECOHESION

A new **Mixed mode criterion** option named **Generalized power law** has been added to the settings for the **Decohesion** subnode. It differs from the ordinary power law criterion in that the exponents for mode I and mode II can be different.

UPDATES TO FIBER MODELING

In the **Fiber** subnode, the following changes have been made:

- In the **Solid Mechanics** interface, the **Volume fraction** input has been moved to the **Distribution and Orientation** section.
- In the **Shell**, **Plate**, and **Membrane** interfaces, as well as in the **Thin Layer** node, you can now select between **Surface distribution** and **Volume distribution**. The way the fiber content is specified depends on this choice.

UPDATED STRAIN FORMULATION FOR POROELASTICITY

The strain formulation (linear or geometrically nonlinear) used in the **Poroelasticity** multiphysics coupling is no longer controlled directly by the corresponding setting in the study step. Instead, it is consistent with the formulation used in the connected solid material model. That is, if the strain formulation from the study step is overridden by, for example, a **Linear Elastic Material** node, the choice is propagated to the **Poroelasticity** multiphysics coupling.

WALL VELOCITY IN FLUID–STRUCTURE INTERACTION

When using a segregated solver for fluid–structure interaction (FSI) problems, the wall velocity degrees of freedom will now be placed in the same segregated group as the displacement field for the participating structural mechanics interface.

LOAD PLOTS FOR MASS AND MOMENT OF INERTIA

When **Mass and Moment of Inertia** subnodes under **Rigid Domain** or **Rigid Connector** contribute to inertial forces, these contributions will now also be present in load plots under **Result Templates**.

EQUIVALENT ELASTIC CONSTANTS FOR EXTERNAL STRESS–STRAIN RELATION

Equivalent elastic constants for material models are now used in several places for proper scaling, for example, when determining default penalty factors in contact analysis.

Such equivalent elastic constants are now also available for user-defined materials given through **External Stress–Strain Relation** node. They can either be automatically computed from the material model, or entered manually in the **Advanced** section of the settings for **External Stress–Strain Relation**.

VIRTUAL CRACK EXTENSION CAN COEXIST WITH CRACK CLOSURE

The correct energy release rate can now be computed using a **Virtual Crack Extension** node also when a crack is closed through the action of a **Crack Closure** node.

TEST MATERIAL WITH ISOTROPIC TEST

The **Isotropic test** option in the **Test Material** node is now available with a Structural Mechanics Module license.

POROELASTICITY, LARGE DEFORMATION, SOLID MULTIPHYSICS INTERFACE REMOVED

The Poroelasticity, Large Deformation, Solid multiphysics interface is no longer available from the **Add Physics** window. The reason for the removal is that, with other recent changes, it differs only marginally from the **Poroelasticity, Solid** multiphysics interface.

COMPUTE BOUNDARY FLUXES REMOVED FROM PHASE FIELD IN SOLIDS

The Compute Boundary Fluxes option has been removed from the settings for the **Phase Field in Solids** interface. The reason is that the fluxes for this interface are not of significant physical interest.

New Models in Version 6.3

RESPONSE SPECTRUM GENERATOR

The purpose of this application is to generate response spectra from a time history that can consist of either measured data on a file or an analytical function. Primarily, it serves to provide input to a response spectrum analysis. However, the graphs can also be used to understand how much an eigenmode at a given natural frequency and damping will be excited by the transient event.

PHONONIC CRYSTAL

This model shows how to use the scattered field formulation to compute the transmission coefficient for P- and S- plane elastic waves impinging on a finite size phononic crystal. The transmission tends to zero in the frequency range corresponding to P- and S-wave band gaps, as predicted by a preliminary study aimed at computing the dispersion relation.

MICROMECHANICAL MODEL OF AN OCTET-TRUSS LATTICE CELL

Lattice materials enable the creation of advanced additive manufacturing materials with customized mechanical properties. At the macroscopic level, these heterogeneous materials can be modeled as homogeneous. Homogenization techniques can accurately determine the material's effective properties based on the characteristics of the lattice structure and its constituents.

PIEZOMAGNETIC CELL ROVER

This tutorial shows how to model a miniaturized magnetostrictive antenna developed for use inside living cells.

The stress in the antenna, the magnetic flux density, the current density, and the displacement of the tip of the device are investigated at the resonance frequency.

Subsurface Flow Module

New Functionality in Version 6.3

NEW FEATURE: BOUNDARY MASS SOURCE

A new **Boundary Mass Source** boundary condition has been added to the Phase Transport interfaces. The new feature accounts for the consumption or production of different phases due to reactions or other physical processes on boundaries. When the **Phase Transport in Porous Media** interface is coupled to a **Darcy's Law** interface, the net mass transfer is automatically accounted for in the flow field.

NEW STABILIZATION OPTION: LIMIT SMALL TIME STEPS EFFECT ON STABILIZATION TIME SCALE

A new option, **Limit small time steps effect on stabilization time scale**, in the Phase Transport interfaces can be used to avoid the loss of stabilization during initialization and for small time steps.

NEW OPTION FOR RELATIVE PERMEABILITIES

A new **Power law** option has been added to the **Porous Medium** node in the **Phase Transport in Porous Media** interface. The new option provides easier implementation of relative permeabilities based on a power-law expression.

NEW NODES IN THE PHASE TRANSPORT INTERFACES

Two new nodes, **Fluid** and **Porous Medium**, are available in the Phase Transport interfaces. The **Porous Medium** node has subnodes to define the phase properties and the properties of the porous matrix. The new nodes replace the previously available **Phase and Transport Properties** and **Phase and Porous Media Transport Properties** nodes. When opening a model created in a previous version, the old nodes are kept in the model.

New App in Version 6.3

HOMOGENIZED POROUS MATERIAL PROPERTIES APP

This app allows users to examine porous material properties by providing various microscale structures. You can adjust the dimensions of these structures and then

calculate key properties like porosity and permeability. The results can also be exported for further analysis, making it a practical tool for studying porous media flow.

Wave Optics Module

New Functionality in Version 6.3

PERIODIC STRUCTURE FEATURE ADDED TO THE ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN INTERFACE

The **Periodic Structure** node is a domain condition that simplifies the modeling of periodic problems. Add the node and select the **Excited Port Selection**. Then, the **Periodic Structure** node will handle the selections for all other ports and periodic conditions.

NEW CROSS-SECTION CALCULATION FEATURE

For the **Electromagnetic Waves, Frequency Domain** interface, in the **Scattered field** formulation, the new **Cross Section Calculation** node is available. This node makes variables for the absorption, scattering, and extinction cross sections available for evaluation.

MATCHED BOUNDARY CONDITION ADDED TO THE ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN INTERFACE

The **Matched Boundary Condition** is now available for use with the **Electromagnetic Waves, Frequency Domain** interface. This boundary condition is similar to the **Scattering Boundary Condition**. However, whereas the **Scattering Boundary Condition** mainly absorbs the waves propagating in the normal direction to the boundary, the **Matched Boundary Condition** absorbs waves propagating in the directions specified in the feature settings.

SIMPLIFIED POLARIZATION SETTINGS FOR PORTS

There are new settings for specifying the polarization for **Port** features of the **Periodic** type. In addition to the existing possibility to specify the electric or magnetic mode field amplitude, it is now possible to directly specify linear s- or p-polarization or right-handed or left-handed circular polarization.

When automatically adding **Diffraction Order** and **Orthogonal Polarization** ports by clicking the **Add Diffraction Orders** button, the refractive index can now be selected from the domain adjacent to the **Port**. Instead, if **User defined** is selected as the **Refractive index, real part**, the entered value can now only be a scalar parameter

expression, as anisotropic refractive index values are not supported by the periodic ports.

Furthermore, the number of ports added can be controlled by either adding only the ports necessary for the parameter values present when clicking the **Add Diffraction Orders** button or by adding all **Diffraction Order** ports relevant for covering all angles of incidence.

NEW DEFAULT VALUES FOR PORTS

For **Port** nodes of the **Periodic** type, the mode field amplitude should never be zero. Thus, there are now nonzero default values for the **Electric mode field amplitude** and the **Magnetic mode field amplitude**.

Similarly, for **Port** nodes of the **User defined** type, there are nonzero default values for the **Electric mode field** and **Magnetic mode field** input fields. Furthermore, the **Propagation constant** is now given the default value `ewfd.k`, where `ewfd` is the name of the physics interface.

NEW GLOBAL PORT MODE FIELD VARIABLES

Port features of the **Periodic** type, **Diffraction Order** ports, and **Orthogonal Polarization** ports now create variables that can be useful for analyzing the periodic problem. The following variables are now available for evaluation:

- Electric mode field amplitude
- Electric mode field
- Port mode wave vector
- Elevation angle
- Azimuth angle

The elevation and azimuth angle variables are available for each port name and for each mode number. Since the variables are globally available, they can be evaluated on any type of selection, not just on the port boundary.

ELECTRIC POINT DIPOLE BACKGROUND FIELD

It is now possible to select **Electric point dipole** as the **Background wave type** when solving problems using the **Scattered field** formulation.

ELECTRIC POINT DIPOLE ADDED IN 2D AXISYMMETRY

In 2D axisymmetry, it is now possible to add the **Electric Point Dipole** node to the **Electromagnetic Waves, Frequency Domain** interface. The node is only enabled when

Electric field components solved for is set to either **Three component** or **In-plane vector**. This point feature should only be added to points located on the symmetry axis. Furthermore, the implementation is restricted to the **Azimuthal mode number** $m = 0$.

NEW VARIABLES AND DEFAULT PLOTS FOR THE SCATTERED FIELD FORMULATION

When solving models using the **Electromagnetic Waves, Frequency Domain** interface, in the **Scattered field** formulation, new variables are available for the following quantities:

- Instantaneous background electric field norm
- Background plane wave intensity, when **Background wave type** is set to either **Linearly polarized plane wave** or **Circularly polarized plane wave**.
- Background Gaussian beam intensity
- Background Gaussian beam maximum intensity
- Background Gaussian beam total power

All variables are available for global evaluation, except for the following domain variables: Instantaneous background electric field norm and the Background Gaussian beam intensity.

By default, **Scattered field** formulation simulations now automatically add a plot of the instantaneous background electric field norm.

NEW DEFAULT PLOTS FOR THE BIDIRECTIONAL ELECTROMAGNETIC WAVES, BEAM ENVELOPES INTERFACE

When using the **Electromagnetic Waves, Beam Envelopes** interface in the **Bidirectional** formulation, by default, plots of the electric field norms of the first and second waves are created, instead of a plot of the norm of the total electric field.

IN-PLANE WAVELENGTH USED AS THE TYPICAL WAVELENGTH FOR PERFECTLY MATCHED LAYERS

In 2D, it is possible to specify a prescribed out-of-plane wave vector component k_z. When calculating the typical wavelength to use in **Perfectly Matched Layer** nodes, the in-plane wavelength is now used. That is, given the wave number k, the in-plane wave number is:

$$k_T = \sqrt{k^2 - k_z^2}$$

and the typical wavelength is given by:

$$\lambda_{PML} = 2\pi/k_T.$$

FROM WAVE VECTOR IS NEW DEFAULT VALUE FOR PROPAGATION DIRECTION

In the **Impedance Boundary Condition**, **Transition Boundary Condition**, and **Layered Transition Boundary Condition** nodes in the **Electromagnetic Waves, Beam Envelopes** interface, the default value for **Propagation Direction** has been changed from **Normal Direction** to **From wave vector**.

ELEVATION AND AZIMUTH ANGLE FAR-FIELD VARIABLE NAME UPDATES WITH USER-DEFINED FAR-FIELD VARIABLE NAME

The far-field variables `ewfd.Efartheta` and `ewfd.Efarphi` now update their names when the **Far-field variable name** has been changed in the settings for the **Far-Field Calculation** node. By default, the **Far-field variable name** is `Efar`. However, if it is changed to `Efar2`, the new variable names will be `ewfd.Efar2theta` and `ewfd.Efar2phi`, respectively.

REVISED SOLVER CONFIGURATION FOR THE ELECTROMAGNETIC WAVES, TIME EXPLICIT INTERFACE

The **Runge–Kutta** method in the **Time Stepping** setting now utilizes the **Cash–Karp 5** option, which can provide faster computations.

ADDITIONAL OUTPUT VARIABLE WITH THE SCATTERED FIELD FORMULATION

A monostatic radar cross-section variable is now available in 2D and 3D for the **Electromagnetic Waves, Frequency Domain** interface when using the scattered field formulation.

IMPROVED PHYSICS-CONTROLLED MESH FOR HIGHER-ORDER ELEMENTS

The physics-controlled mesh for the **Electromagnetic Waves, Frequency Domain** interface has been customized for higher-order elements, enabling more efficient computation with higher-order discretization, such as with cubic or quartic elements.

New and Updated Models in Version 6.3

ACOUSTO-OPTIC MODULATOR

An acousto-optic modulator (AOM) is a device that can be used for controlling the power, frequency, or spatial direction of a laser beam with an electrical drive signal. It

is based on the acousto-optic effect — that is, the modification of the refractive index by a propagating sound wave.

This example shows how an AOM can be modeled as a periodic problem.

DISPERSION OF SURFACE PLASMON POLARITON IN THIN METAL EMBEDDED IN DIELECTRICS

This model demonstrates how to simulate surface plasmon polaritons in a thin metal layer embedded in dielectric layers. It calculates the dispersion and propagation length of surface plasmon polaritons as a function of photon energy.

GAUSSIAN BEAM PROPAGATION THROUGH AN OPTICAL PRISM

This model illustrates the refraction of light while passing through an optical prism using s- and p-polarized incident Gaussian beams. The prism boundaries are coated with antireflection coatings. The model also calculates the reflectance and transmittance to analyze the performance of antireflection coatings.

MODELING A SCATTERER NEAR AN OPTICAL WAVEGUIDE

This model simulates a small lossy scatterer in the proximity of an optical waveguide to show how it interacts with the fields. The model calculates the reflection and transmission along the waveguide as well as losses and scattering.

SIMULATION OF DISPERSION AND HYPERBOLIC WAVES IN A METAL–DIELECTRIC LAYERED METAMATERIAL

This model discusses how to simulate hyperbolic waves propagating in a hyperbolic metamaterial constructed of periodically organized metal–dielectric layers, using an electric point dipole located in the air above the structure. It also demonstrates the procedure to compute the effective relative permittivity tensor components of the metamaterial.

SURFACE PLASMON POLARITON EXCITATION VIA OTTO AND KRETSCHMANN CONFIGURATIONS

This example demonstrates a simulation setup for exciting surface plasmon polaritons using Otto and Kretschmann configurations. The underlying mechanism is enabled by the interplay between total internal reflection and evanescent-wave coupling phenomena. It also calculates the reflectance, transmittance, and absorptance to analyze the surface plasmon resonance condition.

MODELS UPDATED TO USE THE CROSS SECTION CALCULATION NODE

The Optical Scattering off a Gold Nanosphere and the Scatterer on Substrate models have been updated to use the new **Cross Section Calculation** node and associated results plots.

PERIODIC STRUCTURE NODE NOW USED IN MANY PERIODIC MODELS

The following models have been updated to use the new **Periodic Structure** node:

- Frequency-Selective Surface, Periodic Complementary Split Ring Resonator
- Fresnel Equations
- Hexagonal Grating
- Hexagonal Plasmonic Color Filter
- In-Plane Switching of a Liquid Crystal Cell
- Metasurface Beam Deflector
- Plasmonic Wire Grating
- Scatterer on Substrate

OPTICAL YAGI–UDA ANTENNA

This model has been updated to use the new **Electric point dipole** option in the **Background wave type** list.

HEXAGONAL GRATING

This model has been updated to include a plot of the elevation and azimuth angles for the different diffraction orders.

SELF-FOCUSING

A calculation of the spot radius along the propagation direction has been added to this model. The calculation uses linear projection operators to compute the beam variance for each position along the propagation direction.

Material Library

New and Updated Material Data in Version 6.3

In version 6.3, the Material Library contains 17,131 materials and 152,896 material property datasets. The following improvements and additions have been made:

- There is a new reference for thermal expansion for 321H stainless steel; the MTE values are lower by 0% to 3%.
- There are new references for the density and thermal expansion of liquid lead and liquid iron.
- New reference for thermal expansion for titanium aluminide Ti_3Al.
- The thermal expansion data for Nitronic 32® was too high by a factor of 1.8 and have been fixed.
- Added new references for the thermal conductivity for Ti-6Al-4V, 304 stainless steel, 316 stainless steel, and for the thermal diffusivity of Ti-6Al-4V at high temperatures. The thermal diffusivity for Ti-6Al-4V has been increased by approximately 50%.
- Added the UL 94 flammability ratings for some polymers.
- The specific heat for the following materials were incorrect and have been fixed: YAG, Dy_3TaO_7, Er_3TaO_7, Eu_3TaO_7, Gd_3TaO_7, Ho_3TaO_7, La_3TaO_7, Lu_3TaO_7, and Sm_3TaO_7.
- The thermal expansion data for some ferrosilicon (Fe-Si) binary alloys were refitted; the changes were in the range of 1% to 5%.
- The thermal conductivity for tin(IV) oxide (SnO_2) was erroneous and has been corrected.
- The thermal expansion data for liquid tin was changed.
- The elastic data (E, G, υ, and K) for indium nitrate (InN) was erroneous and has been corrected.
- The expansion data for NatureWorks® Ingeo™ polylactic acid (PLA) polymer grade 78000D was too low by a factor of 100. This has been fixed.
- The electrical resistivity and conductivity data for $Bi_{0.5}Sb_{1.5}Te_3$ and dilute synthetic seawater was incorrect and has been corrected.
- Additional data was added for the thermal expansion for Stellite 31.

- The low melting point (LMP) for Inconel® 8091 was refitted.
- Several semiconductors and thermoelectric materials have been moved and combined.

LiveLink™ for Excel®

New Functionality in Version 6.3

- When opening models with the LiveLink™ add-in in the Excel® user interface, the browsing of models saved in a Model Manager database has been improved with search and filtering capabilities.
- It is now possible to save models to a Model Manager database with the LiveLink™ add-in in the Excel® user interface.

LiveLink™ for MATLAB®

New Functionality in Version 6.3

MPHSTARTCOMSOLMPHSERVER WRAPPER

A new wrapper for launching a COMSOL Multiphysics server, `mphstartcomsolmphserver`, has been released, designed to simplify the process of starting a server directly from within MATLAB® when MATLAB® has been started independently of COMSOL Multiphysics.

This wrapper is particularly useful in scenarios where a complex software solution is developed that only requires COMSOL functionality occasionally or for a limited time. Another important use case is when using the Parallel Computing Toolbox™, where multiple servers need to be started and connected to within the same computation. The `mphstartcomsolmphserver` function can return a port number, which can then be used with the `mphstart` function to establish a connection between MATLAB® and the newly launched server.

REGULAR EXPRESSION SUPPORT

Support for searching regular expressions in the Model Library Browser (`mphmodellibrary`) has been added, allowing for more advanced searches and condensing the results more precisely. The `doc regexp` command can be used to obtain a description of the regular expression syntax that is supported by MATLAB®.

MPHWRITEMESH FUNCTION

For writing meshes as MPHTXT-, MPHBIN-, or STL-files, a new `mphwritemesh` function is available. This function makes it possible to select the type of mesh elements to write and to specify which parts of the mesh to export, where the export uses the full precision of the mesh data. If the mesh contains selections, these can be exported as MPHTXT- or MPHBIN-files. The saved files can be imported into COMSOL Multiphysics or another software that supports these formats.

AUTOCOMPLETION

During the last few years some effort has been made to support the autocompletion functionality in MATLAB® that makes it possible for you to write a partial command, press Tab, and then get suitable suggestions for variables and values for the function's arguments. In version 6.3, new syntax has been introduced for the `mphgetpair`,

`mphgetframe`, and `mphgetode` functions that makes autocompletion easier to use. For example, if you write `mphgetpair(` and then press Tab, the model variable will automatically be filled in. If you then write a comma and press Tab, the permitted frame types will appear as a drop-down list in order to easily enter this command: `mphgetframe(model,"geometry1")`. The old syntax `mphgetframe(model.frame('geometry1'))` is still supported.

MPHGLOBAL FUNCTION

The `differential` and `evalmethod` arguments have been added to the `mphglobal` function to add support for linearization similar to the support recently added to the `mphinterp` function.

Backward Compatibility with Version 6.0

The `mphint` and `mphgetp` functions that have been deprecated for some time have been removed.

All functions that were named after the names of color tables (`prism`, `heat`, `spectrum`, and so on) that have been deprecated for some time have been removed. Use the `mphcolortable` function instead to get information and data about color tables that are distributed with the COMSOL Multiphysics software, stored on disk, or embedded inside models. The `colortable` function is deprecated but can still be used.

LiveLink™ for Simulink®

General Backward Compatibility

Models that were built with earlier versions of COMSOL must be reexported from COMSOL again. If simulations with old models are attempted, the following error message will appear in the COMSOL Multiphysics server window as well as the functional mockup unit (FMU) log file that can be opened from Simulink®: *The COMSOL Cosimulation Block and COMSOL Multiphysics server version mismatch. Please reexport the cosimulation FMU-file from COMSOL Multiphysics.*

The COMSOL API for Use with Java®

COMSOL 6.3 API Changes

MATERIAL MODEL API METHODS

The following new methods are available:

- The `setPropertyInfo` method for specifying property information (references, temperature conditions, and so on) for the given material property:

  ```
  MaterialModel setPropertyInfo(String name, String info);
  ```

- The `getPropertyInfo` method for returning property information for the given material property:

  ```
  String getPropertyInfo(String name);
  ```

For the following methods, "parameter" has been changed to "property" in the descriptions for names and group types that are sent to and returned: `getString`, `getStringArray`, `getType`, `getValueType`, `hasParam`, `setMixingRule`, `size`, and `getMixingRule`.

COMSOL 6.2 API Changes

MESH ENTITY NUMBERS

For meshes that define their own geometric models, such as imported meshes, the entity numbering will change if any of the following features are used in the meshing sequence: `Union`, `IntersectPlane`, `IntersectLine`, `MergeEntities`, `CreateDomains`, `CreateEdges`, `CreateFaces`, `CreateVertices`, or `FillHoles`.

COMSOL 6.1 API Changes

PERIODIC CONDITIONS — DESTINATION SELECTION

The **Destination Selection** subnode under **Periodic Condition** nodes was removed in version 6.1 and replaced by an optional **Destination Selection** section in the **Settings** windows for **Periodic Condition** nodes. It can still be added from the API for backward compatibility. The only difference in behavior for old API code when using Destination Selections is what the entities of the feature's selection are when it is set to all boundaries:

A periodic condition:

```
PhysicsFeature pc = model.physics("ec").feature("pc1");
```

Set the selection to all boundaries of the destination domains feature:

```
pc.feature("dd1").selection().all();
```

The result of this statement typically differs between versions 6.0 and 6.1:

```
int[] entities = pc.feature("dd1").selection().entities();
```

In version 6.0, only the automatic destination entities from the parent's automatic destination selection were returned. In version 6.1, the software returns all entities that overlap with the parent selections (source and destination). The interpretation of the all boundaries flag being set is still the same: making the periodic condition use the automatic destination. The only difference is what the API returns for the selection of the destination domains.

FILLETS AND CHAMFERS IN 2D GEOMETRIES

In 2D models, when adding `Fillet` and `Chamfer` features, you typically do not need to set the `selectinsketch` property, as it is set to `on` by default.

GEOMETRY EXPORT

The following method now sets the file format of the geometry export:

```
model.component(<ctag>).geom(<tag>).export().set<Type>(<format>);
```

where *<format>* can take the following values: `nativeascii`, `nativebin`, `parasolidbin`, `parasolidascii`, `acisbin`, `acisascii`, `iges`, `step`, `stlbin`, `stlascii`, or `dxf`.

To get the file format that is set for the geometry export, use:

```
String fileType = model.component(<ctag>).geom(<tag>).export().
                  get<Type>();
```

COMSOL 6.0 API Changes

PAIR FEATURES

The removal of the fallback features under pair features can break any API code that you have written that accesses these features. The presence of a default pair feature may also create a different configuration compared to what previous API runs did.

Java® or MATLAB® programs 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 the 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 afterward makes it possible to use the pair feature's selection directly. The example below shows such code for 5.6 and how the equivalent code in 6.0 can be written:

```
// 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 uses 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);
```

This concludes the release notes for COMSOL Multiphysics version 6.3.

Index