



COMSOL Multiphysics

Release Notes

COMSOL Multiphysics Release Notes

© 1998–2023 COMSOL

Protected by patents listed on www.comsol.com/patents, or see Help>About COMSOL Multiphysics on the File menu in the COMSOL Desktop for less detailed lists of U.S. Patents that may apply. Patents pending.

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

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

Version: COMSOL 6.2

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. Other useful links include:

- 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

Part number: CM010001

C o n t e n t s

Chapter I: Release Notes

COMSOL Multiphysics	10
New General Functionality	10
New Functionality in the Model Manager	10
New Functionality in the Model Manager Server	14
New Functionality in the Application Builder	16
New and Improved General Functionality in COMSOL Multiphysics	18
New Geometry and Mesh Functionality	22
New Functionality in the Physics Interfaces	25
New and Updated Operators, Functions, and Definitions	26
New Functionality in Studies and Solvers	30
New and Improved Results and Visualization Functionality	35
New Java [®] API Methods	39
New COMSOL Server Functionality	40
General Backward Compatibility Considerations	40
Backward Compatibility with Version 6.1.	40
Backward Compatibility with Version 6.0.	41
Backward Compatibility with Version 5.6.	43
AC/DC Module	45
New and Improved Functionality in Version 6.2	45
New and Updated Models in Version 6.2.	50
Backward Compatibility with Version 6.1 and Earlier	52
Acoustics Module	54
New Functionality in Version 6.2.	54
New and Updated Model Examples in 6.2	60
Backward Compatibility with Version 6.1.	61
Battery Design Module	63
New Functionality in Version 6.2.	63
New and Updated Models in Version 6.2.	65

CAD Import Module, Design Module, and LiveLink™ Products for CAD	67
New Functionality in Version 6.2	67
New Functionality in the Design Module in Version 6.2	67
New Functionality in LiveLink™ for AutoCAD® in Version 6.2	68
New Functionality in LiveLink™ for Inventor® in Version 6.2	68
New Functionality in LiveLink™ for PTC Creo Parametric™ in Version 6.2	68
LiveLink™ for PTC Pro/ENGINEER®	69
New Functionality in LiveLink™ for Revit® in Version 6.2	69
New Functionality in LiveLink™ for Solid Edge® in Version 6.2	69
CFD Module	70
New Functionality in Version 6.2	70
New Models in Version 6.2	71
Backward Compatibility with Version 6.1 and Earlier	72
Backward Compatibility with Version 6.0 and Earlier	72
Backward Compatibility with Version 5.6 and Earlier	73
Chemical Reaction Engineering Module	74
New Functionality in Version 6.2	74
New Models in Version 6.2	77
Updated Models in Version 6.2	78
Composite Materials Module	79
New Functionality in Version 6.2	79
New Models in Version 6.2	80
Corrosion Module	81
New Functionality in Version 6.2	81
New and Updated Models in Version 6.2	82
ECAD Import Module	83
New Functionality in Version 6.2	83
Electrochemistry Module	84
New Functionality in Version 6.2	84

Electrodeposition Module	86
New Functionality in Version 6.2	86
New Model in Version 6.2	87
Fatigue Module	88
New Functionality in Version 6.2	88
New Model in Version 6.2	88
Fuel Cell & Electrolyzer Module	89
New Functionality in Version 6.2	89
New Model in Version 6.2	91
Geomechanics Module	92
New Functionality in Version 6.2	92
New Models in Version 6.2	93
Heat Transfer Module	95
New and Improved Functionality in Version 6.2	95
Updated Models in Version 6.2	99
Backward Compatibility with Version 6.1.	99
Backward Compatibility with Version 6.0.	101
Backward Compatibility with Version 5.6.	102
MEMS Module	104
New Functionality in Version 6.2	104
New Models in Version 6.2	107
Metal Processing Module	109
New Functionality in Version 6.2	109
Microfluidics Module	110
New Functionality in Version 6.2	110
Updated Models in Version 6.2	111
Backward Compatibility with Version 6.0.	111
Mixer Module	112
New Functionality in Version 6.2	112

Multibody Dynamics Module	113
New Functionality in Version 6.2	113
New Models in Version 6.2	115
Nonlinear Structural Materials Module	116
New Functionality in Version 6.2	116
New Models in Version 6.2	119
Optimization Module	122
New Functionality in Version 6.2	122
Backward Compatibility with Version 6.1.	123
Backward Compatibility with Version 6.0.	123
Particle Tracing Module	124
New and Improved Functionality in Version 6.2	124
Plasma Module	125
New Functionality in Version 6.2	125
New and Updated Applications in Version 6.2.	125
Backward Compatibility with Version 6.0 and Earlier	126
Porous Media Flow Module	127
New Functionality in Version 6.2	127
New and Updated Models in Version 6.2.	130
Polymer Flow Module	131
New Functionality in Version 6.2	131
Backward Compatibility with Version 6.0.	132
Ray Optics Module	133
New and Improved Functionality in Version 6.2	133
New Model in Version 6.2	134
RF Module	135
New Functionality in Version 6.2	135
New and Updated Models in Version 6.2.	139

Rotordynamics Module	141
New Functionality in Version 6.2	141
Semiconductor Module	143
New Functionality in Version 6.2	143
New Models in Version 6.2	145
Structural Mechanics Module	146
New Functionality in Version 6.2	146
New Models in Version 6.2	156
Subsurface Flow Module	160
New Functionality in Version 6.2	160
New and Updated Models in Version 6.2	163
Uncertainty Quantification Module	164
New Functionality in Version 6.2	164
Wave Optics Module	165
New Functionality in Version 6.2	165
New and Updated Models in Version 6.2	168
Material Library	171
New and Updated Material Data in Version 6.2	171
LiveLink™ for Excel®	173
New Functionality in Version 6.2	173
LiveLink™ for MATLAB®	174
New Functionality in Version 6.2	174
Backward Compatibility with Version 6.0	175
Backward Compatibility with Version 5.6	175
LiveLink™ for Simulink®	176
New Functionality in Version 6.2	176
General Backward Compatibility	176

The COMSOL API for Use with Java[®] 177
COMSOL 6.2 API Changes 177
COMSOL 6.1 API Changes 177
COMSOL 6.0 API Changes 178
COMSOL 5.6 API Changes 179

Index 181



Release Notes

COMSOL Multiphysics® version 6.2 includes new functionality and improvements across the COMSOL® products. These *Release Notes* provide information regarding new functionality in version 6.2 for the entire COMSOL product suite.

COMSOL Multiphysics

New General Functionality

All COMSOL programs running on Windows® now use the native Windows-ROOT keystore as the default trust store for HTTPS connections. This is convenient when the Windows keystore contains custom root certificates for an organization (for example, due to firewalls intercepting encrypted traffic).

New Functionality in the Model Manager

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

MODEL MANAGER APPLICATION PROGRAMMING INTERFACE

You can now access models and data files stored in a Model Manager database via a Model Manager application programming interface (API) for use with the Java® programming language. This access includes, for example, loading and saving of models, using data files as input or output, and searching versions. The Model Manager API can be used in the Application Builder's Method Editor, from a standalone Java® application, and from the LiveLink™ for MATLAB® interface.

MAINTENANCE IMPROVEMENTS

The Model Manager now supports computing the total disk space usage; clearing built, computed, and plotted data; and permanently deleting models and data files via multiple selections. You can also collect various subsets of versions you want to perform such maintenance operations on in the **Maintenance** window, which now supports showing versions belonging to multiple items instead of just a single item. A subset may, for example, be specified as all versions matching the current search in the **Model Manager** window. Maintenance operations that automatically target all versions in the **Maintenance** window are available from a new contextual **Maintenance** toolbar.

GEOMETRY PARTS IN A DATABASE

Geometry parts can now be version controlled in a database by saving them as model versions via, for example, the **Save** window. The Model Manager will automatically detect if a model version contains reusable geometry parts and, if so, extract and store metadata in the database that enables you to easily find such parts using the Model Manager search functionality. Geometry parts stored in a database can be loaded into the model opened in the COMSOL Desktop — for example, by right-clicking a

geometry part node in the **Contents** section of the **Settings** window and selecting **Insert into Model**. If the opened model is subsequently saved to the same database, the geometry part will appear as a referenced version in the **References** window for the saved model.

IMPROVEMENTS TO SEARCHING AND FILTERING

Search All Versions in the Database

You can now search either the latest versions in a branch or all versions in the database by selecting the **Latest Versions for Location** or **All Versions in Database** option in the **Open**, **Select File**, **Select Model**, and **Model Manager** windows. The first search mode was already available in earlier versions of the Model Manager; the second is new for 6.2.

Sorting Search Results

Search results can be sorted when searching versions in the **Open**, **Select File**, **Select Model**, and **Model Manager** windows. Results can be sorted by title, saved date, total size, and size of built, computed, and plotted data — in either ascending or descending order. It is also possible to sort search results based on their relevance ranking, with the most relevant matches appearing first. The sort field and sort order is remembered for each window between COMSOL program sessions.

Search Filters

New filters have been added to the **Filters** menu in the **Open**, **Select File**, **Select Model**, and **Model Manager** windows: The **Part** filter matches versions containing reusable geometry parts; the **Commit Comment** filter matches versions on the optional comment written when they were saved; the **Size** filter matches versions on their size in bytes; and the **Computed Data** filter matches versions based on the size of their built, computed, and plotted data.

The **Last Modified** and **Last Modified By** filters have been replaced by the equivalent filters **Saved** and **Saved By**. They correspond, respectively, to the instant in time when a version was saved and the user that saved the version. The former filters are still supported in the Model Manager search syntax.

Search History

You can reapply previous searches made in a Model Manager database via a new **Search History** menu list. Each list entry contains the search term used in the search field as well as any applied filters.

General Search Improvements

- You can temporarily disable an added search filter. When disabled, the filter is not applied when clicking **Search**.
- When changing to, for example, a search mode in which an already applied filter is not supported, that filter is automatically disabled. A warning icon is also shown for the filter.
- You can set the page size used for results in Model Manager via a new **Result page size** preference. The default page size is 100 — the same as in earlier versions.

INSERT PARAMETERS

In the **Contents** section of the **Settings** window, the new **Insert into Model** button can be used to insert parameters into the model opened in the COMSOL Desktop. You will be asked to either overwrite or rename parameters if there is overlap between the names of new and existing parameters.

UPDATING AND PREVIEWING FILE CONTENTS

The **Settings** window for a data file has been updated to simplify working with filesets, that is, a file version containing multiple file resources.

- An entire folder containing file resources can be added via a new **Add Folder** option.
- The **Replace** button no longer changes the filename of a replaced file resource, only its contents.
- A new **Preview File** button enables you to open a selected file resource with the default application for its file type.

SECURITY IMPROVEMENTS

A new security preference, **Allow access to Model Manager databases**, has been added to the **Security** page of the COMSOL Multiphysics installer. When the **Enforce security restrictions** check box is selected and the **Allow access to Model Manager databases** check box is cleared, methods and Java[®] libraries are disallowed from accessing Model Manager databases. This includes both reading data from and writing data to a Model Manager database. The same security preference has also been added to the **Preferences** dialog box under **Security>Methods and Java Libraries**.

GENERAL IMPROVEMENTS

- You can now connect to a nondefault Model Manager database hosted by a Model Manager server by providing its server database alias via the server address field.

- Built, computed, and plotted data can be optionally excluded when importing and exporting models between the file system and a Model Manager database.
- Geometry parts referenced by an MPH file will be automatically imported into the database when the **Include auxiliary files found in imported models** check box is selected. The opposite is true when exporting and the **Include auxiliary files stored in the database** check box is selected.
- Multiple selections can now be made when setting owners of database objects.
- The Model Manager now uses a temporary resource directory within the resources directory of each of its local databases when saving models, instead of using the temporary directory of COMSOL Multiphysics itself. This reduces the disk space usage in the temporary directory when saving large models to a local database.

COMPATIBILITY BETWEEN VERSIONS

Older Versions of Model Manager Server

COMSOL Multiphysics version 6.2 supports connecting to all previous versions of a Model Manager server. Some of the new functionality in version 6.2 will, however, be disabled in the COMSOL Desktop.

HTTPS by Default

In version 6.2, the Model Manager will always attempt to connect to a Model Manager server using a secure connection via HTTPS. Only if the connection fails and the **Require secure connection** check box is cleared will a nonsecure connection attempt via HTTP be made.

Changed Terminology and Search Syntax

The terminology used to characterize a regular model versus a draft model has changed from *save type* to *item save type*. The corresponding field expression used in the Model Manager search syntax is now `@itemSaveType:<value>` instead of `@saveType:<value>`. Using the latter will result in an error.

The terminology used to characterize a model, application, physics, file, fileset, or tag version has changed from *item type* to *item version type*. The corresponding field expression used in the Model Manager search syntax is now `@itemVersionType:<value>` instead of `@itemType:<value>`. The old terminology and syntax remain but are now meant to match the type of an item itself and not the type of a version of an item. The supported values in this case are model, file, and tag.

New Meaning of Branch Search Option

The **Only text and tags** option found in the **Search** list in the **Settings** window for a branch has been replaced with **Only item fields**. The old option meant that no search data was indexed for versions saved on the branch. This is no longer true — search data corresponding to basic field settings is always indexed for all versions in the database. The new option means that the contents of model versions are not indexed. When such a branch is selected in the **Model Manager** window, for example, all filter options related to model contents will be disabled in the **Filters** menu. Other filter options remain enabled.

New Functionality in the Model Manager Server

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

MODELS AND FILES IN THE WEB INTERFACE

Version-controlled models and data files in a Model Manager server database are now available from the web interface of a Model Manager server. From the **Home** page, you can search versions in repositories and branches using the same Model Manager search functionality available in the COMSOL Desktop. Clicking on a version in the search results opens a corresponding **Model** or **File** page, where you can view and edit basic fields. This includes, for example, the title, description, and assigned tags of the item. You can also see the version history of an item on a branch or browse the version references for a specific item version. You can even create new or update existing data files by uploading new file resource contents.

IMPROVEMENTS TO THE ASSET MANAGEMENT SYSTEM

New Attribute for Linked Users

A new type of primitive attribute for linking to a user in a Model Manager database has been added.

Related Assets

Links to other related assets can be added on the **Asset** page. Clicking a link opens the corresponding page for that asset.

New Search Filters

Two new search filters have been added to the **Filters** menu when searching assets: **Last Modified By**, the user that last saved an asset, and **Owner**, the user that has ownership of an asset.

Automatic Table Filter

When an **Asset** page contains a composite attribute that has a primitive attribute with a set of allowed values using the **Table** mode, a list with filter options is automatically added to the page. Select check boxes in the list filters the table to only show rows with the corresponding attribute values.

IMPROVED STATUS REPORTING FOR MANAGED SERVER COMPONENTS

The status reporting for managed server components has been improved. You can now see, for example, the available disk space for a component's data directory, which Model Manager database is using a particular component, or the possible reason why a component has failed to start.

GENERAL IMPROVEMENTS

- If an account has been automatically mapped as a member of groups via a login configuration, those groups are now shown in the **External Group Memberships** field on the **My Account** page.
- Root administrators can assign an alias to a Model Manager server database. This enables users in COMSOL Multiphysics to connect to that database even if it is not set as the current default database.
- When running on Linux[®], the data directories for a managed SQL database server can now be placed on a network file system if using NFS mounted in hard mode. Using a physical disk is still the recommended setup for performance reasons.
- The username associated with authenticated requests now appears in the Model Manager server access log.
- The key aliases of certificates available in a **PKCS#12** keystore, a **Windows native certificate store**, or a **macOS native keychain** can now be selected from a list when adding a connector.
- If no **TLS host configuration** for a connector uses the **Default** option for the **Hostname**, the first configuration in the list will be used for all connections for which the hostname of a connecting client does not match any **Custom** hostname — effectively changing it to use the **Default** option.
- A Model Manager server now uses a temporary resource upload directory within the resources directory of each of its databases, instead of using the system temporary directory for all resource uploads. This reduces the disk space usage in the system temporary directory when, for example, saving large models to a server database.

COMPATIBILITY BETWEEN VERSIONS

Older Versions of COMSOL Multiphysics

COMSOL Model Manager server version 6.2 supports client connections from COMSOL Multiphysics version 6.0 and newer.

Minimum Version Requirement for External Microsoft® SQL Server® Database
Microsoft® SQL Server® 2012 is no longer supported as an external server component. The minimum version requirement is Microsoft® SQL Server® 2014.

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

File Types

You can create a new type of declaration, **File Type**, from: the **More Declarations** menu in the ribbon, the **Declarations** node's context menu, or the settings of **File Import** form objects. Such file types are available to use in **File Import** or as arguments to methods that open file browsers.

Timer Feature

A new **Timer** feature is available from the **Events** context menu. You can use it to regularly run an arbitrary command sequence without user interaction. The **Timer** feature makes it possible for apps to automatically respond to state changes outside of the model object.

Zooming and Scaling of Editor Windows

The **Form Editor** now supports zooming in and out. This functionality can be accessed through the context menu, keyboard shortcuts, the mouse wheel while holding the Ctrl key, or a new zoom field next to the scroll bar. The zoom level is set and saved for each form. There is also a **Fit to Space** action to set the zoom so that the contents fill the editor.

It was already possible to zoom in the **Method Editor**, but the user interface has now been changed so that it is similar to that of the **Form Editor**. A preference for the zoom level has been added, and it is now synchronized between method editors.

The **Main Window** editor has a new **Window Scale** menu, which has a similar user interface to that of zooming. Unlike zooming, window scaling does not change the size of the window contents, only the available area in the editor.

Graphics Window Toolbar

In simulation apps, **Ambient Occlusion** and **Direct Shadows** are now included in the standard **Graphics** window toolbar.

Opening Form Editor and Method Editor Windows

You can now open the **Form Editor** or **Method Editor** by pressing the Return key when a form or method node is selected in the application tree.

Test in Internet Explorer

The **Test in Internet Explorer** option has been removed in version 6.2.

IMPROVEMENTS TO THE FORM EDITOR

- Layout templates based on a **Subwindows** layout can now add forms directly to individual subwindows.
- **Ribbon Tabs** defined in the Application Builder can now be used in the Model Builder by including them in an add-in.
- You can specify a **Style** and an **Icon** in the **Settings** window for **File Import** form objects. These settings control the **Browse** button appearance. The style can be **Outlined**, **Raised**, or **Flat**, where **Outlined** is the default and **Raised** looks as it did in previous versions.
- The **Graphics** form object's toolbar has been extended with new options to choose **Direct Shadows** (shadow mapping), **Ambient Occlusion**, **Fresnel Transmittance** (uniform blending), and **Floor** (floor effect).
- The **Native Browser on Windows** setting for the **Video** form object has been removed.

IMPROVEMENTS AND CHANGES TO THE APPLICATION LANGUAGE

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

- You can now write method code to show plots from models other than the app's model. Use `loadModel` to load the other model. Then, you can use the method `useGraphics` to show a plot group from the loaded model. For example:

```
Model m = loadModel(...);  
useGraphics(m.result("pg1"), "/form1/graphics1");
```

Note that only plot groups from other models are supported by `useGraphics`. It is not possible to plot the geometry, mesh, or other entities.

The toolbar for these plots does not have all of the buttons that appear for plots from the app's model. This is because these actions are not supported in this case.

- There is a new utility method, `formattedDateTime`, to convert a time given in milliseconds since the epoch to a readable date and time string:

```
String dateString = formattedDateTime(long epochInMs);
```

New and Improved General Functionality in COMSOL Multiphysics

FILTERING THE MODEL TREE

At the top of the **Model Builder** window, just above the model tree, there is now a filter text field in which you can type any text to filter the model tree, showing only tree nodes with names that match the filter. Clear it to return to the full model tree. If you have added new nodes to the tree, click the **Refresh** button to the right of the text field to refresh the filtering. Also, there are options on the **Filter Options** menu to the right in the **Model Builder** window toolbar to control the display the subnodes to the nodes that match the filter, to filter to only match exact words, and to hide the text field for the filter text.

GROUP BY CONTINUOUS TANGENT

Activate **Group by Continuous Tangent** from the context menu when selecting applicable geometric entities or from the selection menu on the **Graphics** toolbar to select or deselect whole groups of boundaries (2D and 3D) or edges (3D) with a single click. Change the **Group by Continuous Tangent Tolerance** to control which entities are considered to be meeting with continuous tangent. These entities are highlighted when hovering over an entity.

FLOOR SHADOWS AND GAMMA CORRECTION

Floor shadows (projection shadows) of 3D geometries are now available for improved spatial understanding and more visually appealing plots. You can add a shadow plane (floor) under the model geometry onto which shadows are projected. Select the **Floor shadows** check box in the **View** node settings for 3D components to activate floor shadows and the associated settings for effects such as blurring and transparency.

You can also activate gamma correction for more realistic lighting, especially when shadow effects are used (ambient occlusion or direct shadows). Select the **Gamma**

correction check box in the settings for the **View** node in 3D components to activate gamma correction and settings for associated effects.

You can also switch floor shadows and gamma correction on and off from the **Scene Light** menu in the **Graphics** toolbar.

COMPARING WITH THE LATEST SAVED VERSION

There is a new **Compare with Saved** button in the **Compare** section of the **Developer** ribbon. Click it to directly compare the open model with the latest saved version of that model. For a model saved in a database, the software compares it with the last saved draft or version.

SHOWING ONLY APPROPRIATE VARIABLES

In the **Show More Options** dialog box, there is now an **All Variables** check box under **Physics** and **Results**. Clear it to only include appropriate variables in expressions for defining the model and results evaluation.

WEBVIEW2 IN WINDOWS VERSION

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

MULTIPLE UNDO AND REDO STEPS

The **Undo** and **Redo** toolbar buttons now include drop-down menus that show an item for each step in the undo or redo history. Selecting a menu item performs all steps up to and including the action it describes.

MARK-OF-THE-WEB SECURITY

Security has been improved when opening models and apps downloaded from the internet. On the **Security>Internet** page in the **Preferences** dialog box, you can choose to use a restrictive security policy (“Mark of the Web”) for these files. You can add trusted sites for which this policy will then not apply. If a model or app that originates from an untrusted site is opened in the COMSOL Desktop[®], a banner appears, notifying you that the security preferences have been temporarily restricted. To restore the normal values and close the banner, you can click the **Remove Restrictions** button in the banner or on the **Security** pages in the **Preferences** dialog box. This functionality is not available on Linux[®].

MULTISELECTION IN THE QUICK ACCESS TOOLBAR

When choosing which buttons to show in the Quick Access Toolbar, it is now possible to select multiple actions in the lists under **More Commands** in the toolbar menu or in the **Preferences** dialog box. All selected actions can then be added, removed, or moved at once.

LOAD DATA FROM DATABASES AND OTHER SOURCES

You can now load tables with data from a file using a fullscreen **Select File** window, where you can select a file from more sources, such as recent files, the current model, local and remote databases, or browsing the file system. This option is available in most settings where a **Load** button previously appeared under a table, such as in the settings of the **Parameters** and **Variables** nodes. The **Load** button now also includes a **Load From** option that opens the **Select File** window.

DATE, TIME, AND USERNAME FOR DATABASE ITEMS

Database items shown in the **Recent** list in the **Open** and **Select File** fullscreen windows now include the date and time when the item was saved and the username of the user that saved the item.

IMPROVED EXPRESSION CHOOSER IN THE CROSS-PLATFORM VERSION

In the cross-platform version of the COMSOL Desktop[®] (primarily for use on Linux[®] and macOS), the expression chooser — which appears when you type Ctrl+Space or click the **Replace Expression** button to insert or replace an expression — has been updated and does now look and work in the same way as the one in the COMSOL Desktop[®] version for Windows[®].

KEYBOARD SHORTCUT SUPPORT FOR CUT, COPY, AND PASTE

When the **Model Builder**, **Application Builder**, or **Physics Builder** tree is selected, the **Cut**, **Copy**, and **Paste** actions on nodes can now be performed using the common keyboard shortcuts (Ctrl+X, Ctrl+C, and Ctrl+V, respectively).

NEW ERROR INDICATORS IN TEXT FIELDS AND TABLES

In the Windows[®] version of the COMSOL Desktop[®], the content of a text field or table cell with an error or warning status is now indicated by a red or yellow squiggly underline, respectively, without changing the text color.

CURRENT WORKING DIRECTORY ON MACOS

On macOS, the current working directory is now preserved when launching COMSOL Multiphysics or a COMSOL batch command from the terminal.

NEW LOGICAL EXPRESSION SELECTION

Add a **Logical Expression** selection feature under **Definitions** to use a logical expression to combine other selections. For example, the expression `(se11 || se12) && !se13` means the union of selections `se11` and `se12` minus the selection `se13`.

ADD-IN IMPROVEMENTS

The **Add-in Libraries** window includes a new **Add Add-in** button. It opens a file browser dialog box where you can select an add-in that you want to add to the default **User Add-in Library**. There is also a new predefined add-in library for user-defined add-ins, located in `.comsol/6.2/addins`. When you use the **Add Add-in** button to add an add-in, the MPH file will be copied into `.comsol/6.2/addins` and show up in the tree under a **User-Defined Library** node.

IMPROVED ICONS FOR MATERIAL NODES

The icons for **Material** nodes now indicate by their appearance whether the material is assigned to domains, boundaries, edges, or points.

FIND AND REPLACE FOR HIDDEN SETTINGS

With the **Settings** filter in the **Find and Replace** window, it is now possible to include hidden settings in the search by selecting the new **Include hidden settings** check box. **Replace** also work for those hidden settings that can be altered. Those matches that are hidden get an extra **Hidden** label in the **Type** column, such as **Setting (Hidden)**. Finding and replacing hidden settings can be important when renaming variables.

IMPROVEMENT TO THE LOG WINDOW

The solver log and other information in the **Log** window can now be stored on file on the server, in `.comsol/v62/sessions/solverlog_[YYYY]-[MM][DD]_[HH][mm][SS]_[pid].log`.

You can add this functionality in the **Preferences** dialog box under **Preferences>User Interface>Log and Messages** using the **Store in file** check box. You can also specify retention time for old solver log files. In the **Log** window, you can then click the **Store in File** button to store the log in a file. You can also click the **Open Log File** button to open the log file on your computer in, for example, Notepad.

FASTER SAVING ON WINDOWS[®]

The writing of MPH files on Window[®]s, especially when saving to network drives, is now faster.

GENERAL FUNCTIONALITY

- The geometry and mesh file type descriptions **All 2D Importable Files** and **All 3D Importable Files** used in earlier versions of COMSOL Multiphysics have been replaced with descriptions that additionally distinguish between mesh and geometry files types: **All 2D Importable Geometry Files**, **All 2D Importable Mesh Files**, **All 3D Importable Geometry Files**, and **All 3D Importable Mesh Files**.
- There is now a **Selection List** button on the **Geometry**, **Mesh**, and **Definitions** ribbon tabs that opens the **Selection List** window.

GEOMETRY FUNCTIONALITY

- Use the new **Distance Measurement** and **Centroid Measurement** features to create parameters for the distance between entities or parameters for the centroid of selected vertices, respectively. The parameters can be used in downstream geometry construction and in physics. For a 2D geometry with constraints and dimensions enabled, it is better to use the **Distance** dimension feature instead of the **Distance Measurement** feature.
- The **Sweep** operation has been improved with better control of the twisting of a cross section by specifying a direction vector to follow. This makes it easy to create a tunnel where the floor has a slope only in the sweep direction, for example. You can now sweep points and edges, and it is possible to align the sweep to an existing object at the end of the spine curve. In the user interface, the **Settings** window has been reorganized for better usability, and the sweep direction is determined automatically by default.
- Exact geometric edges are now used, which is important in physics interfaces that require high accuracy in the normal, such as the Ray Optics interface. Previously, the edges were approximated when meshing, giving an approximation error that could not be decreased by refining the mesh.
- 3D Boolean operations are faster in COMSOL Multiphysics version 6.2, and the operations find intersections between faces that are tangent to each other at one of the endpoints of the intersection edge, for example, when intersecting two cylindrical faces.
- There is now an option to hide the visualization of the local coordinate system of a work plane.
- Add a **Logical Expression Selection** feature in the geometry sequence to use a logical expression to combine other selections. For example, the expression `(sel1 | |`

se12) && !se13 means the union of selections se11 and se12 minus the selection se13.

- Use the new **Propagate selections to resulting objects** check box to update selections on the input objects to corresponding entities on the output objects. If you do not want to propagate the selection to the resulting objects, clear the check box. This option is available for most geometry operations.
- Most geometry features now have an **Information** section containing information about the building of the resulting geometry objects, including time, date, and COMSOL version in which the feature was built. It also contains messages that were only written in the **Messages** window in previous versions.
- In the **Form Assembly** node, there is a new option to select the frame for the generated identity pairs.
- Measuring a selection of faces in 3D also displays the perimeter, and when measuring a selection of entities, the number of entities is displayed.
- There is a new option to specify whether to unite the input objects before conversion or to convert each object individually. This option is available in the **Convert to Solid**, **Convert to Surface**, **Convert to Curve**, or **Convert to Point** operations.

GEOMETRY PART IMPROVEMENTS

You can now load geometry parts stored in a database using a new **Load Part From** context menu option available under **Global Definitions>Geometry Parts** in the **Model Builder** tree. This option is also present under the **Geometry** ribbon toolbar. Selecting **Load Part From** opens a fullscreen **Select Model** window for selecting models, with a filter that is preapplied to show only models containing reusable geometry parts.

For a geometry part that is already loaded under **Global Definitions>Geometry Parts**, there is a new **Browse From** option using the **Browse** button under the filename. This new option makes it possible to browse for another geometry part using the same fullscreen **Select Model** window used for the **Load Part From** option.

For a geometry part loaded from a database, using either the new **Load Part From** or **Browse From** options, the field that refers to the filename now refers to the loaded part via an icon and the label of the database from which the geometry part was loaded.

Geometry parts that are referenced by the model object now show up in the **Auxiliary Data** window. For a geometry part that is loaded under **Global Definitions>Geometry Parts** (including those loaded from database), the **Auxiliary Data** window now shows a corresponding entry supporting the same types of actions as it does for other types of input files.

When you create a geometry part in version 6.2, it contains a **Local Parameters** node. Use this node to specify parameters that are used within the part but should not be specified by users of the part.

MESHING FUNCTIONALITY

- In 3D, a new option has been added to the **Free Triangular** operation: **Resolve narrow domain regions**. When it is activated, the **Free Triangular** operation creates the same mesh on selected faces as it would if the **Free Tetrahedral** operation were run on adjacent domains. This option is useful in sequences that define their own geometric model, where face mesh is edited before domain mesh is created.
- For the **Swept** operation, support for more general topology of linking faces has been added. For example, a linking face can now have a rectangular imprint in the middle of the face. This new functionality works as long as edges are either sufficiently tangent to or perpendicular to the local sweep direction.
- The **Identical Mesh** attribute lets you specify pairs of groups of faces or edges where you need identical mesh, for example, for periodic boundaries. It is supported for the **Free Tetrahedral**, **Boundary Layers**, **Free Triangular**, **Free Quad**, **Edge**, **Remesh Faces**, and **Remesh Edges** operations.
- It is now possible to specify how the source mesh of a **Copy** operation is transferred to the destination by adding an **Edge Map**, **Two-Point Map**, or **One-Point Map** attribute node. This functionality provides more control when copying mesh between geometries.
- The **Boundary Layers** operation has been significantly improved and now allows faster generation of thicker high-quality boundary layer meshes.
- Use the new **Corner Properties** attribute to gain full control of the corner handling in selected corners for your boundary layer mesh.
- The **Boundary Layer Properties** option **Adjust layer directions** has been removed and has been replaced by the **Method** option on the **Boundary Layers** node. It is no longer possible to control this option for individual boundaries.
- The **Distribution** attribute now has an **Equidistant** option when generating a fixed number of elements. With this option selected, the resulting mesh will have equally sized elements along edges (in 3D) or boundaries (in 2D).
- The **Adjust Edge Mesh** functionality in the mapped mesher has been improved to avoid small edge elements and to reduce skewness by moving points (if possible) along edges in the same edge group.

- The new operations **Remesh Edges**, **Remesh Faces** (3D), and **Remesh Domains** (2D) are used to remesh entities on meshes that define their own geometric models, such as imported meshes. Use the ordinary attributes for controlling the mesh size, and use the new attribute **Fixed Mesh** to prevent modification of the mesh on selected boundaries or edges.
- The new **Imprint** operation lets you make imprints of vertices, edges, and faces on another face in an imported mesh. This operation can be used to partition a face using a tool object or as a preparation for merging of meshes.
- You can now set up named selections of entities for imported meshes using **Explicit Selection**, **Adjacent Selection**, or by defining output selections in the following operations: **Intersect with Plane**, **Intersect with Line**, **Fill Holes**, **Create Faces**, **Create Domains**, **Create Edges**, and **Create Vertices**. The selections can be colored and used in subsequent meshing operations and when setting up the physics.
- The **Union**, **Intersect with Plane**, and **Intersect with Line** operations have a new option **Placement of mesh vertices**, with the choices **Linear** and **Curved**. When this option is active, the global shape of each face is used to obtain a more exact placement of new mesh vertices.
- When importing STL, PLY, or 3MF files, there is a new **Merge coinciding mesh vertices** option. Select this to merge mesh vertices that lie within the distance of a specified tolerance into a single mesh vertex.
- When specifying coordinates in the **Create Vertices** operation and the specified coordinates are adjacent to or inside a mesh element, the element is now partitioned to conform with the new vertex. Similarly, if you create vertices inside an unmeshed domain, it will become adjacent to the unmeshed domain.

New Functionality in the Physics Interfaces

UNIT DISPLAY FOR DEPENDENT VARIABLES

The units for dependent variables now appear within parentheses in the **Dependent Variables** section of the **Settings** window for the main physics interface nodes and under **Dependent Variables** on the **Select Physics** page in the Model Wizard.

EQUATION DISPLAY IN THE PDE INTERFACES

The equation display in the partial differential equation (PDE) interfaces now follow the equation form; for example, it does not include any time derivatives when you use a stationary equation.

USER-DEFINED NAME FOR GLOBAL EQUATIONS

There is a new **Name** field under the **Label** field in the settings for the **Global Equations** nodes. The text in the **Name** field must be a valid variable name, and it is used to set the field name attribute. This name is used as an identifier for the corresponding solver feature under **Dependent Variables**. This means that you can now set a deterministic name (identifier) than can also be more descriptive than the old default (ODE1, ODE2, and so on).

ENERGY BALANCE SECTION IN NONISOTHERMAL FLOW

In the **Nonisothermal Flow** multiphysics coupling node, the **Flow Heating** section has been replaced by the **Energy Balance** section. In addition to the **Include viscous dissipation** option, this section contains a new option to **Include kinetic energy**. When both options are selected, the conservative total energy equation is solved.

New and Updated Operators, Functions, and Definitions

DEEP NEURAL NETWORK FUNCTION

The new **Deep Neural Network** function provides training and validation using a deep neural network (DNN) for use with surrogate model training, for example. Deep neural networks form a class of machine learning algorithms similar to the artificial neural network and aim to mimic the information processing of the brain.

PARTIAL FRACTION FIT FUNCTION

The new **Partial Fraction Fit** function uses a modified adaptive Antoulas–Anderson (AAA) algorithm, AAA2, to compute a partial fractional fit. You can use this function for a rational approximation of frequency-domain responses. This approximation makes it possible to compute its inverse Fourier transform analytically and thus obtain the time-domain impulse response function. Doing so is useful in, for example, models using a Pressure Acoustics, Transient or Pressure Acoustics, Time Explicit interface.

POLYNOMIAL CHAOS EXPANSION FUNCTION

The new **Polynomial Chaos Expansion** function defines a polynomial chaos expansion (PCE, also called *polynomial chaos expansion* and *Wiener chaos expansion*), which is a method for representing a random variable in terms of a polynomial function of other random variables. PCE can be used, for example, to determine the evolution of uncertainty in a dynamical system when there is probabilistic uncertainty in the system parameters.

The **Polynomial Chaos Expansion** function is intended for use with the Uncertainty Quantification Module, which is required for creating and training the **Polynomial Chaos Expansion** function. If you already have a model with a trained **Polynomial Chaos Expansion** function, that function can be evaluated without a license for the Uncertainty Quantification Module.

NEW UNIT SYSTEMS WITH THE RANKINE TEMPERATURE UNIT

There are four new unit systems that use Rankine instead of Fahrenheit as the base unit for temperature. When using these special unit systems, you must provide all temperatures in Rankine unless you add [degF] after any number entered. The new unit systems are **British engineering units (Rankine)**, **FPS (Rankine)**, **IPS (Rankine)**, and **Gravitational IPS (Rankine)**. You typically need to use one of these unit systems whenever you use formulas intended for absolute temperatures (such as $\exp(-E_a / (k_B \text{const} * T))$) or material properties (such as thermal expansion) with the unit 1/temperature (for example, [1/K]) that are expected to be multiplied with a temperature difference.

NEW PHYSICAL CONSTANT: BOHR RADIUS

The Bohr radius, a_0 , is now available as a predefined physical constant: `a0_const`.

IMPROVED ROTATED COORDINATE SYSTEM

The **Rotated System** coordinate systems include a new **Rotation sequence** list, which you can use to choose from different Euler angle definitions.

INDETERMINISTIC OUTPUT FROM RANDOM FUNCTIONS

In **Random** and **Normal Distribution** function nodes, when you have selected the **Use random seed** check box, there is a new **Random seed type** list where you can choose **Fixed** (the default) or **Current computer time**, which is a new option that provides an indeterministic output.

LINE INTEGRATION COUPLING OPERATOR

The new **Line Integration** coupling operator evaluates the integral of a general operator argument expression, the integrand, on a line through a meshed source domain selection in 2D or 3D. The line passes through the destination point where the **Line Integration** operator is evaluated and the result of the integral is defined. The destination can be any point on the source selection where the coupling operator is defined as well as any point that is on any of the lower-dimensional entities adjacent to the source selection, such as the boundaries, edges, or points of the source selection.

NEW PARAMETER OPERATORS

There is a new set of parameter operators that makes it possible to easily perform simple pointwise operations over a set of solutions corresponding to different parameter values:

- `pint(p, p1, p2, expr)` — integrating the linear interpolation of `expr` evaluated at all parameters over the interval `p1` to `p2` (more or less the trapezoidal rule).
- `psum(p1, p2, expr)` — summing `expr` in all parameter values in the interval `p1` to `p2`
- `pmin(p, expr)` — finding the minimal value of `expr` over all parameters
- `pmax(p, expr)` — finding the maximal value of `expr` over all parameters
- `pmin(p, p1, p2, expr)` — finding the minimal value of `expr` in the interval `p1` to `p2`
- `pmax(p, p1, p2, expr)` — finding the maximal value of `expr` in the interval `p1` to `p2`
- `atpmin(p, p1, p2, expr1, expr2)` — evaluating `expr2` at the point where `expr1` has its minimal value on the interval `p1` to `p2`
- `atpmax(p, p1, p2, expr1, expr2)` — evaluating `expr2` at the point where `expr1` has its maximal value on the interval `p1` to `p2`

In these function calls, `p` is the name of the parameter. `p1` and `p2` must be real scalars.

LOGARITHMIC INTERPOLATION IN INTERPOLATION FUNCTIONS

It is now possible to use logarithmic interpolation for interpolation functions where input data point positions or data point values are transformed by a logarithm function before interpolated data output data are transformed back to the original space. In a new **Data Transformation for Interpolation** section in the settings for **Interpolation** functions, you can choose **Logarithm** instead of the default **None** for the function and the arguments.

PATCHEVAL OPERATOR

The new `patcheval(expr, exorder)` operator provides a continuously smoothed version of the expression `expr` as follows: For each mesh vertex, a patch of surrounding mesh elements is formed, and a polynomial of degree `exorder` is fitted to the values of `expr` in a set of sampling points in these elements. The value of `patcheval` at a given evaluation point is a weighted average of the values given by the polynomials corresponding to the vertices of the containing mesh element.

IMPROVED PLOTTING FUNCTIONALITY FOR USER-DEFINED FUNCTIONS

For functions that can have more than three arguments — that is, the **Analytic**, **Gaussian Process**, **Polynomial Chaos Expansion**, and **Deep Neural Network** functions — the table in the **Plot Parameters** section now includes a **Plot** column with check boxes so that you can choose which of the arguments (one, two, or three) to include in the plot.

IMPROVED UNIT HANDLING IN THE EXPRESSION OPERATOR

Unit handling in the **Expression Operator** feature has been improved. In the process, the default argument dimensions have changed from nondimensional (1) to undefined, specified as an empty unit string. When a formal argument has undefined dimensions, the dimensions of the actual argument used in each operator call are now forwarded to the unit evaluation. This means that expression operators no longer have fixed output dimensions but instead dimensions that may depend on the arguments where it is called. In addition, the new on-the-fly evaluation of dimensions adds correct evaluation for some cases that in earlier versions erroneously returned undefined dimensions, notably when coupling operators are involved in the defining expression.

These change may have implications for existing models and API code, in particular if you have compensated for undefined dimensions by explicitly introducing unit syntax in definitions as well as results and visualization.

DEFORMED MESH FUNCTIONALITY IN NEW RIBBONS

The **Deformed Mesh** section with the **Moving Mesh** and **Deformed Geometry** features has been removed from the **Definitions** ribbon. This functionality is now available in separate **Moving Mesh** and **Deformed Mesh** ribbons.

FUNCTION SWITCHES — NOT DEFINING ALL CASES

In the settings for **Function Switch** nodes, there is a new **Define all cases** check box (cleared by default). In previous versions of COMSOL Multiphysics, all functions in a **Function Switch** were active in the sense that all functions were declared and possible to use. This behavior might be convenient, but it can also be very costly. Select the **Define all cases** check box to revert to the behavior from earlier versions.

IMPORT WAVE AUDIO FILES INTO INTERPOLATION FUNCTIONS

It is now possible to choose **WAVE Audio File (*.wav)** as the file format to import WAVE audio files into **Interpolation** functions.

TOLERANCES FOR COMPATIBLE MESHES

For identity pairs and periodic boundary conditions, if you have selected **On** or **Require compatible meshes** from the **Elementwise mapping for compatible meshes** list, a **Tolerance for compatible meshes** field now appears, where you can specify the relative tolerance compared to the mesh element size for when mesh elements are to be considered as compatible. (Default: 0.001.)

IMPROVED HANDLING OF SMALL GAPS IN BEM MODELING

When using physics interfaces based on the boundary element method (BEM), the handling of small gaps has been improved by using singularity-weakening variable transformations. The new **Thin gap handling** check box is used to increase the accuracy of the quadrature between almost-touching mesh element pairs (that is, pairs that are extremely close with respect to element size). The **Threshold distance (relative to mesh size)** value controls the distance at which elements are considered almost touching. A larger value should provide more accurate results but at an increased computational cost. Under the quadrature settings, the **Integration order (very close elements)** field becomes active, making it possible to set a higher-order rule for such elements. In 3D, an additional variable transformation is applied to improve the accuracy even further.

IMPROVED SUPPORT FOR PRODUCT GEOMETRY

It is now possible to use the following operators in product geometry (for extra dimension):

- Average over extra dimension
- Integration over base dimension
- Average over base dimension
- General extrusion between product geometries

New Functionality in Studies and Solvers

SURROGATE MODEL TRAINING

Use the new **Surrogate Model Training** study node to add functionality for training a surrogate model. A surrogate model is a simplified and more computationally inexpensive model, which you can use to approximate the behavior of a more complex and computationally expensive model (the full finite element model). In COMSOL apps, for example, faster model evaluation using a surrogate model provides users with a more interactive app experience. You can create surrogate models by training a deep neural network (DNN), for example. Surrogate model training is typically based on

output data from a large parametric sweep of a full model. Training a surrogate model requires a number of inputs and outputs; the outputs are considered functions of the inputs. This model replaces the full finite element solution. A large number of data points are needed in order to fully describe how the inputs map to the outputs. The outputs can be captured using domain point probes, for example.

To see an example of how the computational speed of an app is increased by using a surrogate model, view the new Tubular Reactor Surrogate Model Application in the COMSOL Multiphysics Application Library.

PERFORMANCE IMPROVEMENTS

- More accurate time derivatives have been implemented for the backward differential formula (BDF) solver. This change leads to more efficient time stepping for some models.
- The ArmPL (Arm Performance Libraries) have been updated, providing improved performance on the `glnxarm64` platform, especially with a direct solver running with multiple CPU cores.
- The ArmPL is now available as an option on Apple silicon (using `-blas armpl`). ArmPL provides better performance than the default option but can sometimes be less stable.

STUDIES AND STUDY STEP FUNCTIONALITY

- You can now skip parameters in the **Parametric** solver when there is an error by selecting **Skip parameter step** from the **On error** list.
- The settings for study nodes now include a **Store in Output** section where you can specify for each physics interface in the model if you want to store the output: as controlled by the physics, all output, no output, or output based on a named selection.
- It is now possible to allow variables that are not solved for to be reused in a periodic manner for time-dependent solvers. This option is available in the **Advanced** section of the settings as a **Periodic values of variables not solved for** check box. It requires two inputs: the **Start time** and the **Interval length**. Together, they define the interval for which the input solution is periodic. When reusing variables for time instants that fall outside the specified time window, a periodic map is applied, allowing these variables to be used for times longer than one period.
- Support for converting to a domain decomposition solver has been added to the solver level and is available from the context menu for all solver nodes that support

direct and iterative solvers (such as **Stationary Solver** and **Time-Dependent Solver** nodes).

- The eigenvalue type of study steps and solvers have a number of improvements:
 - The **All (filled matrix)** method in the **ARPACK** solver in earlier versions of COMSOL Multiphysics has been replaced with the **LAPACK (filled matrix)** solver.
 - When you choose the **FEAST** solver, there is a new **Stochastic Estimation** button in the **Study Settings** section header, which you can use to estimate the number of eigenvalues within a defined ellipse or half ellipse, without actually computing the eigenvalues.
- Adaptive tolerance in the linear solver is now available for the fully coupled and segregated solvers. Based on an estimate of the global convergence, the algorithm tries to find an optimal tolerance for the linear solver. Three different convergence estimators are available from the **Method** list: **Full error estimate**, **Partial error estimate**, and **Nonlinear residual**. Using an adaptive tolerance avoids expensive oversolving of the linear system when still far away from global convergence. This functionality is only available for the **Constant (Newton)** nonlinear methods.
- Also, for domain decomposition solvers, there is a new **Only visualize the domains** option, which can be used for visualization of the domains during results processing and visualization (instead of the computed solution).
- New heuristics for adaptive tolerance have been added to the time-dependent solvers for the minimal Jacobian update of the constant Newton method. The new heuristics is turned on by default when the **Use linear heuristics for adaptive tolerance** check box is cleared. The new heuristics use a stricter tolerance at each Newton iteration step when the Jacobian is not updated. Therefore, more nonlinear iterations might be required in each time step, and more Jacobian updates might be needed. When the **Use linear heuristics for adaptive tolerance** check box is selected, the minimal Jacobian update behaves as in earlier versions of COMSOL Multiphysics.
- The **Modal Solver** and **Modal Reduction** solver now support using left eigenvectors. This method is preferable when you perform model reduction for nonsymmetric problems.
- The **Spread** setting in the **Parameter list type** list in the **Frequency Domain, Modal** study step now works as expected. Also, to make the spread parameter list type more useful, you can specify a **Minimum frequency** or **Maximum frequency**, which limits the output frequencies to solve for (By default there is no limit). An extra list of frequencies can also be supplied in a new **Additional frequencies** field. They will be

added to the output frequencies to solve for, which makes the spread option effectively an extension of the frequency parameter list type.

- There are new options in the CFL-number proportional integral derivative (PID) controller and for Anderson acceleration in the segregated solver settings when **Pseudo time-stepping** is chosen from the **Stabilization and acceleration** list:
 - The PID controller for the CFL number in pseudo time stepping has new options that can be used to tune the controller. The **PID controller** list has the values **Simple** (the old version of the controller) and **Interpolation** (a new version of the controller that blends different types of controllers when the error reaches the target error estimate). The **Target CFL number** was previously set to 10,000 but is now available as a setting. The solver does not converge unless the target CFL number is reached. The **Limit to target CFL number** check box controls how the CFL number is changed once the Target CFL number is reached. Previously, the CFL number continued to increase until the error tolerance was fulfilled. Now, by default, the CFL number is kept at the **Target CFL number**.
 - The Anderson acceleration includes new options. The **Above CFL threshold** list controls whether the solver should set the target CFL number once the threshold is reached (**Lock target CFL number**) or whether the PID controller should still control the CFL number based on the error estimate (**Use PID controller**). Previously, the solver would always set the CFL number to the target CFL number once Anderson acceleration was activated for pseudo time stepping. The **Threshold for Anderson step** field controls whether the Anderson step or the Newton step is used as the nonlinear step. If the norm of the new step is less than the threshold times the norm of the previous step, the Anderson step is used. Otherwise, the Newton step is used. Lowering the **Threshold for Anderson step** can improve robustness at the price of performance. Previously, the threshold for the Anderson step was set to 10.
 - There is now an option to use Anderson acceleration in the **Segregated Step** nodes. This can improve the convergence rate when the number of iterations of the step is greater than 1. This is, for instance, the case for the turbulence variables group in the CFD interfaces.
- It is now possible to use continuation for parametric solvers together with least-squares optimization. The **Use all for Continuation** check box is selected by default. It then expands all least-squares data points into the continuation sweep. If the least-squares data includes data points for the continuation parameter, it will be merged with the list provided by the user.

- In the settings for the **Advanced** node, you can now choose **Keep while expression is constant** from a new **Method** list under **Reuse sparsity pattern**. If you know a global expression for which the sparsity pattern needs to be recomputed when it is changed (for instance, t in a time-dependent problem), you can switch to **Keep while expression is constant**. Using this method, the pattern will be recomputed when the expression has changed value by more than 10^{-12} . Reusing the sparsity pattern this way can improve the solver performance.
- In the settings for the **Advanced** node, the new **Optimize element data initialization** check box is selected by default. Some elements (components of the discretization of the problem) then attempt to optimize the parallelization of their initialization. Currently, this default optimization is only implemented for contact problems and reduced-order modeling. You can turn it off by clearing the **Optimize element data initialization** check box.
- There is a new hybrid method for handling of constraints. Like the orthonormal method, the hybrid method handles constraint blocks in a parallel way. For constraint blocks whose estimated complexity of using singular value decomposition (SVD) factorization exceeds the orthonormal block limit, the Gaussian-elimination-based sparse method will be used, while other blocks are treated by SVD factorization. To use the new hybrid method, select **Hybrid** from the **Null-space function** list in the **Advanced** node. In addition, the **Orthonormal block limit** setting is now shared by the **Automatic** and **Hybrid** methods, and you can adjust its value using a slider.
- In the **Advanced** node, there are now improved ways to control the frequency of reading from and writing to recovery files. Using the **Recovery saving** list, you can choose to save on every iteration or use a saving ratio. You can also use a manual setting **Force save of recovery** to force the recovery file to save at a specified interval. The settings are also available as preference settings on the **Files>Recovery** page, and the default is to use the settings from the preferences.
- You can now choose a static distribution of assembly blocks by selecting the new **Use static distribution of assembly blocks** check box in the **Assembly Settings** section of the settings for the **Advanced** node.
- The parameter step size for continuation, `CMPpcontstep`, and the step size for time stepping, `timestep`, are now available as results processing variables. For continuation, the step size is also written to the **Log** window.

CLUSTER AND BATCH FUNCTIONALITY

- The LSF (load sharing facility) scheduler is now supported in the settings for the **Cluster Computing** node.
- When a **Parametric Sweep** is used as a job configuration, you can now select the **Postpone errors until end of sequence** check box to allow the sequence to continue even if an error is encountered. This option is useful when, for instance, you are interested in the solution even if there is an error.
- The default name in batch jobs is now chosen from the filename. Alternatively, it can be set as beforehand or as a preference on the **Computing>Cluster** page.

OTHER SOLVER IMPROVEMENTS

If there are control inputs in constraints when building a reduced-order model (ROM), the exported system from the ROM is now a reduced system in which the constrained states are eliminated. New eliminated-constraint-vector and eliminated-output-vector outputs are then available in the **Matrix** list in the **System Matrix** node under **Results>Derived Values**.

New and Improved Results and Visualization Functionality

GENERAL NEW RESULTS AND VISUALIZATION FUNCTIONALITY

- You can now use *configuration nodes* to simplify subsequent results evaluation and plotting. Under **Configurations**, you can add **Graph Plot Style**, **Multiselect Solution**, and **Single-Select Solution** configuration nodes, which you can refer to in plot nodes, for example, to take graph plot styles or specified solutions from these configurations. You can also right-click graph plot nodes and select **Create Style Configuration** from the context menu to create a **Graph Plot Style** style configuration based on the graph plot.
- Undo and redo operations are now supported for many more results and dataset actions.
- It is now possible to use your favorite plots in the **Add Predefined Plot** window. When you have made a plot you would like to use in other models, select **Add to Predefined Plot** from the plot's context menu. This option is available for plot groups, evaluation groups, derived values, and node groups under **Results**. The plot will then show up in the **Add Predefined Plot** window under **User Defined**.
- The **Information** section of a plot now includes information about the physical and virtual memory usage.

- You can now import data into results tables using a new **Import From** option on the **Import** menu in the **Data** section of the **Settings** window for **Table** nodes. Choosing the **Import From** option opens a fullscreen **Select File** window, where you can select a file from many sources, such as recent files, the current model, local and remote databases, or by browsing the file system.
- You can now hide the local coordinate system axes of a work plane by clearing the **Show in graphics** check box in the **Local Coordinate System** section of the settings for **Work Plane** nodes.
- For rendering of materials using built-in material types, the following types are new: **Glass (Tinted)**, **Aluminum (Matte)**, **Aluminum (Polished)**, **Sand**, **Titanium (Brushed)**, and **PCB**. The **PCB (Green)** and **PCB (Brown)** material types from previous versions of COMSOL Multiphysics have been removed.
- The LaTeX parser now supports the `\makebox` command for creating a box just wide enough to contain the specified text.
- DirectX[®] is no longer available as a rendering option on Windows[®] computers.

NEW AND IMPROVED DATASET AND EVALUATION FUNCTIONALITY

- Support for selections using **Selection** subnodes has been added to the **Array**, **Join**, **Mirror**, **Revolution**, and **Sector** datasets.
- The **Selection** plot subnode for plots that use a 2D revolution dataset now includes two check boxes: **Evaluate the start cap** and **Evaluate the end cap**. One use case of this functionality is to plot one expression on the start cap and another expression on the end cap.
- For numerical evaluation using evaluation features under **Derived Values** and **Evaluation Group** nodes, it is now possible to specify that the time or parameter tuple for which a maximum or minimum was attained is included in the results table.
- There is now an option for removing not-a-number (NaN) data points in **Data** export nodes.
- Extrusion datasets now include a **Separate levels** check box for plotting solutions with one slice of 2D data for each equidistant point in time, for example.
- You can now specify an azimuthal mode in **Revolution ID** datasets.

NEW AND IMPROVED GRAPHICS AND PLOT FUNCTIONALITY

- For most 3D plot types, a new **Visual Effects** subnode is available. In its settings, you can control whether the plot should be affected by lighting and also whether

ambient occlusion and direct shadows should be active and casting and receiving shadows.

- Graph plots now support interactive results extraction. Tooltips with values and plot data information appear when you move the cursor close to points in the graph. You can turn this functionality on and off using the **Enable Tooltip** button in the **Graphics** toolbar for 1D plots.
- A new **Comparison** subnode is available for **Global**, **Point Graph**, and **Line Graph** plots. You can use it to compare a graph plot with a table graph plot, using various options for the mapping and metrics in the comparison. Results of the comparison appear in a results table view.
- For titles in plot groups, it is now possible to use line breaks to make the title span multiple lines.
- The **More Plots** submenu has been split into several further submenus, such as **More Surface Plots** and **More Line Plots**.
- A new **Table Point** 2D plot makes it possible to plot the points in a table.
- There is now a **Function** plot for functions of three arguments that use a 3D slice plot to visualize such functions.
- Streamline surface plots now support curved surfaces.
- Improvements to the error bar functionality:
 - You can now add an **Error Bars** subnode to 1D **Function** and **Table Graph** plot nodes.
 - The error bars can now be plotted as filled regions.
- The 3D streamline plots have been improved for better positioning of the streamlines, better orientation of the ribbon style lines, and improved handling and efficiency.
- Mesh plots now support selections on dimensions higher than the evaluation dimension (for example, selecting a domain to plot the mesh on its boundaries).
- The displacements in plot arrays can now be defined as absolute displacements in addition to relative displacements.
- In the settings for 2D **Streamline** plot nodes, the **Starting distance factor** and **Terminating distance factor** values can now be specified for magnitude-controlled streamlines when the **Advanced parameters** list is set to **Manual**.
- In **Graph Marker** plot subnodes, a new **Show y-coordinate** check box is now available in the **Min and max** display mode to only show the x-coordinate, if desired.

- It is now possible to restrict **Table Graph** plots to a subset of the x - and y -axis data using range settings.
- You can now control the maximum width of the graph legend in 1D plot groups.
- It is now possible to add **Filter** subnodes to **Table Graph** plot nodes, with which you can exclude points from the plot using logical expressions.
- For animations of 1D graph plot groups where you have a time-dependent, parametric, or eigenvalue solution, you can now use the **Cumulative** list to control whether the animation should use the last time or parametric tuple or a cumulative list of such tuples.
- You can now include the plot's label in graph plot legends.
- For graph plots that include a color expression, it is now possible to use number format settings.
- **Pellets** plots now support a varying radius in the pellets plot to model shrinking core phenomena.
- For **Impulse Response**, **Line Segments**, and **Polarization plots**, you can now choose to include or omit the description of the solution from the automatic legend.
- For **Table Graph** and **Sobol Index** plots, there is now an option for whether the column headers should be included in graph legends.
- When applicable, the **Solution at angle (phase)** setting is now available in **Plot Group** nodes, not just in **Solution** datasets.
- **Octave Band** plots are now also available with a license for the Structural Mechanics Module or MEMS Module.
- A new nonlogarithmic option is available in the **Octave Band** plots. There is also a new **Domain** list with the options **Frequency** and **Time**, which means that **Octave Band** plots now also support time-domain expressions.
- **Graph Marker** plots now include the possibility to export the bandwidth, Q-factor, and damping ratio into a table.
- You can now use **Parameterized Surface** and **Embed 2D** datasets in **Streamline Surface** plots. For such plots, you can now also choose **On selected edges** to uniformly place seed points.

NEW AND IMPROVED REPORT AND PRESENTATION FUNCTIONALITY

- The report generator can now include images in portrait mode.
- Slide layout settings for the presentation generator are now available in **Section** nodes so that you can override the parent settings for slides with custom content.

- An **Images** section is now available in **Section** nodes for reports and presentations.
- There is now support in the report generator for adding custom captions to all model content features with images.
- The report generator can now handle a wide variety of formats of images embedded in Microsoft® Word® template files.
- The standard document properties in Microsoft® Word® reports and Microsoft® PowerPoint® presentations as well as HTML metadata, for example title and keywords, are now defined by settings in the report or presentation tree.

New Java® API Methods

The following new methods are now available:

MEASURING GEOMETRIC ENTITIES AND GEOMETRIC OBJECTS

Measuring geometric entities in `comp()`, `geom()`, and `mesh()` can now be done using the same methods:

```
model.component(<tag>).measure().MeasurementMethod;
model.component(<ctag>).geom(<tag>).measure().MeasurementMethod;
model.component(<ctag>).mesh(<tag>).measure().MeasurementMethod;
```

PREVIEW PLOTS OF DATASETS

There is now API support for creating preview plots of datasets, using the `createPlot` method:

```
DatasetFeature.createPlot();
```

CONTROLLING THE PLOT PREVIOUS, PLOT NEXT, AND SIMILAR BUTTONS

There are now the following new Boolean API methods for stepping and checking whether such operations are applicable: `stepPrevious`, `stepNext`, `stepFirst`, and `stepLast`; and `canStepPrevious`, `canStepNext`, `canStepFirst`, and `canStepLast`. All methods support an integer input argument `level` for the level to step in a time-dependent parametric sweep, for example. You can also determine the number of levels using `getStepLevels()`.

FIELD NAMES FOR ODES (GLOBAL EQUATIONS)

The field name attribute for ordinary differential equation (ODE) entities is new, and there are three new methods:

```
model().ode(<tag>).create(<tag>, <fname>)
```

```
model().ode(<tag>).field(<fname>)  
model().ode(<tag>).field()
```

These methods do the same things as the corresponding methods for `model().field()`.

NEW MODEL LOCATION METHODS

The so-called “save location” of a model has been added to `Model.java` in the COMSOL API via two new methods, `Model.location()` and `Model.locationUri()`. The return value of `Model.location()` is what can be provided as is, for example, `ModelUtil.load(String tag, String modelLocation)` to load a model from either the file system or a database.

New COMSOL Server Functionality

The **Security** preference for file system access now also applies to clients connected to COMSOL Server™ using the COMSOL API.

General Backward Compatibility Considerations

COMSOL Multiphysics version 6.2 can open MPH-files saved from COMSOL Multiphysics versions 4.0–6.1.

COMSOL Multiphysics version 6.2 can run Java® files saved from COMSOL Multiphysics versions 4.0–6.1. 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 6.2.

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`.

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.

Backward Compatibility with Version 5.6

PAIR FEATURES

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

- 1** Add a default pair feature with the **Disconnect pair** check box selected to ensure that the default pair feature does not contribute any new constraints or equations to the model.
- 2** Move all fallback features that were located under the pair features in 5.6 to the level below the physics interface. The selection of these moved features will typically get the same active selection as the pair feature they belonged to in 5.6. Because those moved features now have an overlapping selection with that pair feature, they will act as an equivalent fallback feature to the pair feature.
- 3** Optimize all moved fallback features by merging exact duplicates into one feature, including existing features such as the default features. In many situations, the result will be that only the default feature of the physics will remain because most fallback features under pair features in 5.6 were identical to the default feature of the physics interface.

Potential Issues

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

PHYSICS AND VARIABLES SELECTIONS

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

USER DOCUMENTATION NODES IN THE PHYSICS BUILDER

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

AC/DC Module

New and Improved Functionality in Version 6.2

NEW MAGNETIC MACHINERY, ROTATING, TIME PERIODIC INTERFACE

The Magnetic Machinery, Rotating, Time Periodic (mmtpr) interface, found under the **AC/DC>Electromagnetics and Mechanics** branch when adding a physics interface, is used for design and analysis of rotating magnetic machines such as electric motors and generators. It uses the time-periodic equation form. **Stationary, Frequency, and Time Dependent** studies are supported, for coupling to other physics. The interface is available in 2D with out-of-plane currents, and in-plane magnetic fields.

The time-periodic approach assumes temporal periodicity, but not necessarily single harmonic excitation (as is the case for a frequency-domain analysis): Any excitation can be used, including static excitation (permanent magnets) and pulse or wave forms containing many harmonics at once. Nonlinear material models are supported in the exact sense — as opposed to *effective* or *cycle-averaged* for a frequency-domain analysis. Unlike the time-dependent solver, the time-periodic approach solves for all time frames at once. This is a particularly powerful method for cases where the entire period needs to be solved many times, for instance, because the period is contained in a **Parametric Sweep** or an **Optimization** solver.

Another common use case is a magnetic machine model that needs to run many cycles to reach nominal operating conditions: An ordinary time-dependent solver would need to resolve the entire transient, while the time-periodic approach solves for the steady-state conditions directly.

NEW MULTIPHYSICS COUPLING AND NEW MULTIPHYSICS INTERFACE: MAGNETIC-ELASTIC INTERACTION IN ROTATING MACHINERY

The new multiphysics coupling **Magnetic Forces, Rotating Machinery** connects a Solid Mechanics interface to a Rotating Machinery, Magnetic interface at the domain level. It adds the loading caused by Maxwell stresses on a deformable and rotating structure.

The new Magnetic-Elastic Interaction in Rotating Machinery multiphysics interface combines a Solid Mechanics interface with a Rotating Machinery, Magnetic interface, using the **Magnetic Forces, Rotating Machinery** multiphysics coupling.

This multiphysics interface requires a license for the Structural Mechanics Module.

NEW MULTIPHYSICS COUPLING AND NEW MULTIPHYSICS INTERFACE: MAGNETIC–RIGID BODY INTERACTION IN ROTATING MACHINERY

A new multiphysics interface and a new multiphysics coupling for modeling electric motors and generators has been added. This multiphysics interface, Magnetic–Rigid Body Interaction in Rotating Machinery, combines a Multibody Dynamics interface with a Rotating Machinery, Magnetic interface through the new **Magnetic Forces, Rotating Machinery** multiphysics coupling. This functionality can be used to compute how deformations and stress originating from air gap forces are distributed in both the stator and rotor of an electric motor. Some common applications are magnetic bearings and machines with unbalanced rotors.

This multiphysics interface requires a license for the Multibody Dynamics Module.

RESTRUCTURED MAGNETOMECHANICS MULTIPHYSICS COUPLINGS

The **Magnetic Forces, Lorentz Coupling**, and **Magnetomechanical Forces** multiphysics couplings have been deprecated and replaced by a single multiphysics coupling called **Magnetomechanics**. The coupling can be used to combine Solid Mechanics with either Magnetic Fields or Magnetic Fields, No Currents.

If it is used to couple a Solid Mechanics interface with a Magnetic Fields, No Currents interface, the coupling becomes equivalent to the deprecated **Magnetic Forces** coupling. If it is used to couple a Solid Mechanics interface with a Magnetic Fields interface, a check box allows the user to choose between options that are equivalent to the deprecated **Lorentz Coupling** and the deprecated **Magnetomechanical Forces** coupling.

The new Magnetomechanics multiphysics coupling requires a license for the MEMS Module, the Structural Mechanics Module, or the Acoustics Module.

PERIODIC PAIR FEATURE IN 2D AXISYMMETRY

The Magnetic Fields interface now supports the **Periodic Pair** feature for the modeling of linear magnetic machinery also in 2D axisymmetry (previously available in planar 2D). This makes it possible to model rotationally symmetric linear (tubular) machinery in 2D.

NITSCHKE SUPPORT FOR PAIR CONDITIONS

For the **Continuity** and the **Sector Symmetry** pair features in the Electric Currents interface, the Magnetic Fields, No Currents interface, and the Rotating Machinery, Magnetic interface, a new constraint option is available for the scalar potential degree of freedom: the **Nitschke constraint** — note that for these features, the **Constraint Settings** section becomes visible only when **Advanced Physics Options** settings is enabled in the

Show More Options dialog box. The Nitsche constraints do not require conforming meshes and do not need a Lagrange multiplier either, simplifying the meshing sequence and allowing for a larger range of solver options.

Note: This implementation assumes the quasistatic approximation and is applicable only for pairs adjacent to domains with little or no displacement currents or induced eddy currents normal to the pair boundary. In addition, nonlinear material models in the neighboring domains are not yet supported.

IMPROVED FRAME HANDLING FOR THE MAGNETIC FIELDS INTERFACE

The **Ampère's Law** feature in the Magnetic Fields interface has been deprecated and replaced with **Ampère's Law in Solids** and **Ampère's Law in Fluids** features. The chosen reference frame — Lagrangian or Eulerian — affects how Maxwell's equations should be interpreted. (In particular, this applies to material properties and induced electric fields.) The two new features provide a more user-friendly implementation that helps to ensure that the appropriate reference frames are used for liquids, gases, and vacuum on one hand, and solid objects on the other.

The **Ampère's Law in Fluids** feature comes with a set of material models that is suitable for liquids (including, for example, the relative permittivity and permeability), while the **Ampère's Law in Solids** feature supports Jiles–Atherton hysteresis and remanent flux density, for example. Together, the two new features cover the same range as the original **Ampère's Law** feature. Furthermore, the features allow for a more intuitive setup of multiphysics couplings, with the **Ampère's Law in Fluids** feature allowing for couplings with magnetohydrodynamics or plasma physics, for example, and the **Ampère's Law in Solids** feature allowing for magnetomechanical couplings.

NEW DEFAULT FEATURE FOR THE MAGNETIC FIELDS INTERFACE

A new default feature **Free Space** has been added to the Magnetic Fields interface. It is used to specify the physical conditions in close proximity to your modeled device — typically in the air or a vacuum. The feature adds an Ampère's Law governing equation with a limited set of material models. It provides a canvas on top of which other features, such as the **Ampère's Law in Solids** and **Ampère's Law in Fluids** features can be added to locally specify material properties and excitation forms.

The **Free Space** feature comes with a built-in **Stabilization** option, improving numerical stability. The stabilization adds an artificial conductivity term to limit the contrast between the free space domains and the conductors typically included in the model. This conductivity term can be deduced automatically by choosing a skin depth that is an order of magnitude larger than the modeled device at its typical operating frequency

— a conductivity that results in such a large skin depth is assumed to have little or no impact on the result quality while still improving solver reliability and speed. A **User defined** option is available, too, for manual tuning of the stabilization.

Note: Replacing the **Magnetic Fields** default feature has an impact on backward compatibility. Any model method, Java® code, and MATLAB® code that has been saved using previous COMSOL versions may need to be revised in order to function properly.

LITZ WIRE SUPPORT FOR THE MULTITURN COIL DOMAIN

The **Wire Properties** section of the **Homogenized Multiturn** coil has been extended to support several options typically used for litz wire modeling. This includes the option to specify the (AC) resistance per unit length — either taken from a user-defined analytic model, from measurements, or from a specification sheet provided by the supplier. This option enables an easier workflow when modeling imperfect stranded conductors (such as litz wires operating at higher frequencies). The option to specify the total effective coil (or litz coil) resistance is available too.

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

PERIODICITY SUPPORT FOR THE SINGLE CONDUCTOR COIL AND THE MULTITURN COIL

The **Geometry Analysis** subfeature used for the **Coil** domain in the Magnetic Fields interface; the Magnetic and Electric Fields interface; and the Rotating Machinery, Magnetic interface in 3D has now been equipped with true (anti)periodicity support — through a **Periodic Boundaries** subfeature to the **Geometry Analysis** subfeature. The **Periodic Boundaries** subfeature is used together with the **Periodic Condition** feature under the physics node. This option is particularly useful for electric machine models that have some form of sector symmetry.

VOLTAGE EXCITATION FOR THE TERMINATED TERMINAL

The **Terminal** feature in the Electrostatics interface, the Electric Currents interface, and the Magnetic and Electric Fields interface is equipped with a **Terminated** option. This functionality makes it possible to connect the terminal to an impedance that might represent a load or a transmission line. The **Terminated** option has now been extended to support voltage excitation. Except for the means of excitation, this functionality is very similar to the existing power-fed terminated terminal.

BACKGROUND FIELD FEATURE FOR THE MAGNETIC AND ELECTRIC FIELDS INTERFACE

A new **Background Magnetic Flux Density** feature has been added to the **Magnetic and Electric Fields** interface. This provides a convenient way to add a background field when modeling liquid metals, for example.

Note that this feature only applies a constraint; it does not introduce (or require) a reduced field formulation.

DEBYE DISPERSION MODEL FOR ELECTRIC CURRENTS

New damping models have been added for (weakly) conducting materials. Under **Current Conservation** — when the material type is set to **Solid** — you can now use the **Dispersion** dielectric material model. In the **Dispersion** subnode, you can choose between the **Debye** and **Multipole Debye** dispersion models. This functionality is available for frequency-domain and time-dependent analysis.

Note that this material model requires a license for the AC/DC Module or the MEMS Module.

PIEZOELECTRIC MATERIAL, LAYERED MATERIAL MODEL

The **Piezoelectric Material** model in the Layered Shell interface (under **Structural Mechanics** when adding physics interface) is now also available as the **Piezoelectric Material, Layered** model in the Shell interface. As a result of this addition, thin piezoelectric composites can be analyzed using the Shell interface. This can provide significant savings in terms of computation time.

In addition to a license for the Structural Mechanics Module, this feature requires a license for either the AC/DC Module or the MEMS Module. 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.

FLOATING POTENTIAL FOR THE CONDUCTING SHELL AND THE PIEZORESISTIVE SHELL

For the Electric Currents in Shells interface and the Electric Currents in Layered Shells interface, a new **Floating Potential** subfeature has been added to the **Conductive Shell** and **Piezoresistive Shell** features. The **Floating Potential** node is used when modeling a metallic electrode at floating potential.

The Piezoresistive Shell feature requires a license the MEMS Module.

NEW PHASE SECTION IN PLOT GROUP SETTINGS

When creating plots for frequency-domain simulations, it is now possible to specify the **Solution at angle (phase)** directly in the plot group settings. Previously, this setting could only be specified on the datasets. Now, the same dataset can be used for different plot groups, displaying results for different phase settings.

VARIABLE REFACTORING FOR THE MAGNETIC FIELDS, NO CURRENTS INTERFACE

For the Magnetic Fields, No Currents interface, a new framework has been implemented for the definition of derived variables. These are the variables in the physics (including those used in results analysis) derived from the degrees of freedom solved for. The new framework enables quicker and more robust maintenance of the physics interface and better support for solid and liquid materials.

New and Updated Models in Version 6.2

PERMANENT MAGNET MOTOR WITH EFFICIENCY MAP

When electrical energy is transferred into mechanical work in an electric motor, some of the energy is lost in the form of heat. This modeling example demonstrates how to couple the Magnetic Machinery, Rotating, Time Periodic interface with the Heat Transfer in Solids interface in a two-way manner, where electromagnetic losses influence temperature, and the temperature influences torque and electromagnetic losses. A **Parametric Sweep** study over a range of rotor speeds and torque levels is performed in order to generate a motor efficiency map.

PERMANENT MAGNET MOTOR WITH CAMPBELL DIAGRAM

The noise emitted from an electrical machine can have many sources. This model studies acoustic noise from vibrations created by magnetic force variation is studied. Such noise is often audible at twice the excitation frequency but can also occur at higher frequencies or harmonics of the excitation frequency.

The model demonstrates how to couple the Magnetic Machinery, Rotating, Time Periodic interface with solid mechanics and acoustics to produce a Campbell diagram. The results provide insight into which of the magnetic force harmonics creates the most prominent motor noise at various speeds. This insight also correlates this with the natural frequencies of the motor structure.

ELECTROMAGNETIC AND MECHANICAL ANALYSIS OF AN INTERIOR PERMANENT MAGNET MOTOR

The magnets in an interior permanent magnet (IPM) motor are embedded in the rotor core, where they form narrow regions known as bridges. The thickness of the magnetic bridge is an important parameter to consider in the design, both from the electromagnetic and the mechanical perspective.

This model demonstrates the coupling between the Solid Mechanics and Rotating Machinery, Magnetic interfaces that is used to perform electromagnetic and mechanical analysis of an IPM motor. A motor with 10 rotor poles and 12 stator slots is modeled in 2D. The magnets are embedded in a V-shaped configuration inside the rotor core. The connection between the magnets and the rotor core is modeled as springs. The results provide insight into magnetic flux density and stress distribution in the system.

MAGNETIC-STRUCTURE INTERACTION IN A PERMANENT MAGNET MOTOR

This model demonstrates how the coupling between the Multibody Dynamics and Rotating Machinery, Magnetic interfaces can be used to perform electromagnetic and mechanical analysis. A permanent magnet motor with 10 rotor poles and 12 stator slots with surface mounted magnets is modeled in 2D. To model the magnetic-structure coupling integrated with a moving mesh, the electromagnetic force is transferred to the rotor, and the rotor motion is transferred to the moving mesh. A time-dependent problem computing the magnetic flux density and displacement is solved for three complete electrical periods.

RESONANT SPIRAL COIL

A self-resonating, five-turn spiral coil of copper wire with a 1-mm-square cross section is analyzed in 3D and 2D axisymmetry using the Magnetic and Electric Fields interface. The wire is equipped with a thin epoxy varnish insulation layer, tightly wound and mounted on a printed circuit board. The capacitive coupling through the thin varnish layer is modeled using a special boundary condition in 3D, thereby avoiding costly volumetric meshing. In the 2D axisymmetric model, a volumetric, swept mesh is used in the thin varnish layer.

HARTMANN FLOW IN LIQUID METAL BLANKET WITH HEAT TRANSFER

In nuclear fusion reactors, the conducting fluids, such as the fusion plasma and the liquid metal in the reactor blanket, will interact with the background magnetic field. This phenomenon is referred to as magnetohydrodynamics (MHD). The

electromagnetic phenomena and the fluid flow are fully coupled through the Lorentz force and the electromotive force. The example demonstrates the modeling of an MHD flow in the duct of a liquid metal blanket, coupled with the heat transfer.

IRON SPHERE IN A MAGNETIC FIELD TUTORIAL SERIES

An iron sphere in a magnetic field is a textbook example that demonstrates the effects of a magnetic field interacting with a permeable material. This tutorial series is designed as an introduction to the numeric modeling of electromagnetic effects with the COMSOL software.

The series investigates four different background magnetic field regimes consisting of a static background magnetic field and three time-varying fields at 60 Hz, 20 kHz, and 13.56 MHz. Each of these regimes shows unique behavior, has unique underlying assumptions, and utilizes different modeling techniques.

FEM RESISTOR IN CIRCUIT (USING THE EC EXTERNAL I-TERMINAL)

The **External Couplings** feature in the Electrical Circuit interface has two options that can be used for coupling currents: the **External I vs. U** and the **External I-terminal**. The former has two nodes (representing a differential external voltage measurement), and when coupling to an **Electric Currents Terminal** feature, you will need to have a local ground in the electric currents finite element model. The latter is a single node and, therefore, you will need two instances of it and two **Terminal** features in the Electric Currents interface. This simple tutorial model demonstrates how to use these features in practice.

UPDATED SYNCHRONOUS ELECTRIC DRIVE GEOMETRY PARTS

The geometry parts for the **Synchronous Electric Drive 3D** have been updated to match the latest released models in the *Motor Tutorial Series*.

Backward Compatibility with Version 6.1 and Earlier

- The switch from the old **Ampère's Law** feature to the new default **Free Space** feature in the Magnetic Fields interface means that this version of the Magnetic Fields interface 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 patch up old files, in most cases it suffices to add an instruction for creating the feature right after introducing the Magnetic Fields interface (although migrating to `AmperesLawSolid` or `AmperesLawFluid` is recommended):

```
model.physics().create("mf", "InductionCurrents", "geom1");  
model.physics("mf").create("al1", "AmperesLaw", 3);  
model.physics("mf").feature("al1").selection().all();
```

- The **Ampère's Law** feature in the Magnetic Fields interface has been deprecated and replaced with **Ampère's Law in Solids** and **Ampère's Law in Fluids** features.
- The **Magnetic Forces**, **Lorentz Coupling**, and **Magnetomechanical Forces** multiphysics couplings have been deprecated and replaced by a single multiphysics coupling called **Magnetomechanics**.
- The Electric Currents, Single Layer Shell interface has been deprecated and replaced by the Electric Currents in Shells interface.

Acoustics Module

New Functionality in Version 6.2

FREQUENCY-DEPENDENT IMPEDANCE CONDITIONS IN THE TIME DOMAIN

New functionality for specifying and setting up frequency-dependent impedance conditions in acoustic time-domain simulations. The functionality relies on a rational-fitting approach of the frequency-domain data, resulting in a system of memory ODEs solved in the time domain. A new **Partial Fraction Fit** fitting/interpolation function has been added to perform the transformation from the frequency-domain data to the time domain. The fitting relies on a variant of the so-called AAA algorithm. The functionality has been added to the Pressure Acoustics, Transient and Pressure Acoustics, Time Explicit interfaces.

In Pressure Acoustics, Transient and Pressure Acoustics, Time Explicit, the **Impedance** boundary condition can now be used to model realistic surface properties, such as those of an absorbing panel or any other surface that has frequency-dependent absorbing properties. This new functionality is essential for realistic full-wave room-acoustic simulations and more.

The **Partial Fractional Fit** function (available with the COMSOL Multiphysics base package) transforms frequency-domain data into a form that is suitable for the time-domain analysis. The function performs a rational approximation of frequency-domain responses. This makes it possible to compute its inverse Fourier transform analytically and thus obtain the time-domain impulse-response function. The fitting algorithm can be used for any data, but it is particularly important and useful for surface-impedance data in acoustic simulations.

ANISOTROPIC POROELASTIC MATERIAL IN THE POROELASTIC WAVES INTERFACE

The material model in the Poroelastic Waves interface has been extended with a new **Anisotropic Poroelastic Material** node. Many porous materials, such as fibrous materials, exhibit anisotropic properties. The anisotropic properties can now be defined for both the elastic matrix material properties and the relevant poroacoustic properties, that is, the flow resistivity, the tortuosity factor, and the viscous characteristic length.

RESTRUCTURE OF THE POROELASTIC WAVES INTERFACE

The Poroelastic Waves interface has been restructured so that conditions that apply to the porous elastic matrix and those applicable to the saturating fluid are located in separate menus. The conditions can simultaneously be applied on the same boundary for defining a multitude of mixed conditions.

PORT CONDITION FOR LINEARIZED POTENTIAL FLOW

A **Port** boundary condition has been added to the Linearized Potential Flow interface. The Port condition is used to excite and absorb specific acoustic modes that enter or leave waveguide structures, like a turbofan duct or other channel structures, in a convected acoustic simulation based on the linearized potential flow model. To provide the full acoustic description, several port conditions are applied on the same boundary. This allows setting up a modal decomposition of noise sources. All relevant propagating modes in the studied frequency range can be included (among the modes that have a cutoff frequency in the frequency range). The Linearized Potential Flow, Boundary Mode interface can be used to analyze and identify the propagating and nonpropagating modes.

IMPEDANCE CONDITION IN THE LINEARIZED POTENTIAL FLOW, BOUNDARY MODE INTERFACE

An **Impedance** condition can now be added in the Linearized Potential Flow, Boundary Mode interface when computing propagating and nonpropagating modes. This is important in combination with the **Port** boundary conditions when exciting a waveguide system with realistic outgoing and incident modes.

SLIP WALL FEATURE IN THERMOVISCOUS ACOUSTICS FOR NONIDEAL WALL CONDITIONS

The **Slip Wall** boundary condition is used to model the effective nonideal wall conditions that exist in the slip-flow regime, when the Knudsen number is in the range from 0.001 to 0.1. The **Slip Wall** condition is used for systems with very small geometrical dimensions or systems running at very low ambient pressures. This is, for example, relevant when modeling MEMS transducers and other microdevices. To model a slip wall on an interior condition, use the **Interior Slip Wall** condition.

SURFACE TENSION FEATURE IN THERMOVISCOUS ACOUSTICS

The **Surface Tension** feature adds the necessary condition to model an interface between two fluids including surface tension effects. This acoustic (perturbation) formulation of the Young–Laplace equation relies on a linearization around the stationary shape of

the fluid–fluid interface. The feature is important when modeling small curved interfaces between two different immiscible fluids, like microbubbles or microdrops.

NEW RCL OPTION FOR THE IMPEDANCE IN THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN

An **RCL** options has been added to the **Impedance** boundary condition in the Thermoviscous Acoustics, Frequency Domain interface. The condition is useful for modeling the interaction with simple spring-mass-damper systems using a lumped representation. An example is the flexible membrane of a microphone.

NEW AND IMPROVED MULTIPHYSICS COUPLINGS AND FUNCTIONALITY

Two new multiphysics couplings have been added and two existing multiphysics couplings now have the option to add subfeatures to simplify the modeling workflow.

New Pair Acoustic–Thermoviscous Acoustic Boundary Multiphysics Coupling

A new pair version of the existing Acoustic–Thermoviscous Acoustics Boundary coupling has been added, the **Pair Acoustic–Thermoviscous Acoustics Boundary**. This coupling is suited for modeling approaches that have assemblies with nonconforming meshes.

New Thermoviscous Acoustic–Thermal Perturbation Boundary Multiphysics Coupling

A new **Thermoviscous Acoustic–Thermal Perturbation Boundary** multiphysics coupling has been added to couple the acoustic temperature variation in a fluid to temperature fluctuations in solids. This is a coupling between the Thermoviscous Acoustics, Frequency Domain/Transient and Heat Transfer in Solids interfaces. The condition is, for example, useful for advanced acoustic simulations of thermoacoustic engines and pumps.

Interior Impedance for the Acoustic FEM–BEM Multiphysics Coupling

When coupling FEM- and BEM-based pressure acoustics models using the **Acoustic FEM–BEM Boundary**, the **Impedance** (subfeature) can be added to the multiphysics coupling to model an interior impedance between the two domains. This extends the applicability of the hybrid FEM–BEM modeling strategy, which is useful for large acoustic problems.

Thermoviscous Boundary Layer Impedance for the Acoustic–Structure Interaction Multiphysics Coupling

When coupling a vibrating structure to an acoustic domain with the **Acoustic–Structure Boundary**, the **Thermoviscous Boundary Layer Impedance** (subfeature) can be added to the multiphysics coupling. This simplifies the setup of large vibroacoustic models

where thermoviscous losses are included in the homogenized boundary condition formulation of the thermoviscous boundary layer impedance. This is, for example, important to speed up certain shape-optimization problems, or for faster approximate simulations.

NEW RECEIVER FEATURE IN RAY ACOUSTICS

A new physics-based **Receiver** feature in the Ray Acoustics interface drastically improves performance for analyzing the impulse response. The receiver feature is used to define the boundaries of a receiver sphere in the geometry when setting up the physics. The receiver collects information (arrival time and power) about the intersecting rays during the simulation. This information is then used to compute the impulse response in the results analysis.

RELEASE FROM PRESSURE FIELD IN RAY ACOUSTICS

The new **Release from Pressure Field** feature is used to create realistic sources extracted from a wave-based (near-field) simulation done with the Pressure Acoustics, Frequency Domain interface. This means that the classical point-source approximation of ray tracing is not always necessary. An example of a near-field source could be a loudspeaker placed in the dashboard of a car. This placement causes local reflections and diffraction, which are wave phenomena that cannot be captured by the ray tracing method. A local pressure-acoustics model can be solved and rays released with the **Release from Pressure Field** feature with magnitude and direction given by the intensity field in the pressure-acoustics model.

IMPROVEMENTS TO PLOTS AND IMPORT FOR ACOUSTIC MODELING

Import of WAVE Audio Files

WAVE audio files (.wav) can now be imported in an **Interpolation** functions. This is useful for many applications in acoustics, for example when comparing simulations with measured data, or when importing source signals for a transient analysis.

Function Input to Impulse Response Plot

The data source for the **Impulse Response** plot can now also be a function (and not only a receiver dataset). This means that an Impulse Response plot can be used to analyze user-defined impulse-response data, for example, data imported from a WAVE audio file. This allows analysis of measurement data as well as of data resulting from concatenation of low-frequency wave-based and high-frequency ray simulations.

Updates to the Octave Band Plot

The **Octave Band Plot** can now be used to analyze results based on transient simulations. The transient data is transformed to the frequency domain before being analyzed. The **Octave Band** plot now also has a **General (non-dB)** input type that can be used to analyze absorption data in acoustics or vibration velocity data in structural-vibrations models.

GRADIENT-BASED OPTIMIZATION WITH EXTERIOR FIELD OPERATOR IN 2D AXISYMMETRIC MODELS

Gradient-based optimization (shape or topology optimization) is now supported in 2D axisymmetric models when using the dedicated optimization exterior-field operator `pext_opt(r, z)` in the Pressure Acoustics, Frequency Domain interface. The optimization version of the exterior-field operator, similar to the already existing operator in 3D, is implemented such that its sensitivity can be computed analytically.

OUT-OF-PLANE SOURCE-TERM CONTRIBUTIONS IN THERMOVISCOUS ACOUSTICS, LINEARIZED NAVIER-STOKES, AND LINEARIZED EULER

Source terms in thermoviscous acoustics, linearized Navier–Stokes, and the linearized Euler interfaces now include out-of-plane contributions.

FIRST-ORDER MATERIAL CONTRIBUTIONS IN ACOUSTIC STREAMING

A new option to include the first-order material dependency of the viscosity has been added to the acoustic streaming multiphysics couplings. This effect is typically important in a rotating streaming flow generated by a combination of two resonances, generating a rotating acoustic wave.

LAGRANGIAN STEADY VELOCITY VARIABLES IN ACOUSTIC STREAMING

New predefined variables have been added for the Lagrangian steady velocity when modeling acoustic streaming. This velocity should be used when computing the trajectories of particles in a streaming flow. The variable is announced and can easily be selected as input for the viscous drag force in, for example, particle tracing for fluid-flow physics interfaces.

ADAPTIVE FREQUENCY SWEEP STUDY

A new frequency-domain study type called **Adaptive Frequency Sweep** has been added for the Pressure Acoustics, Frequency Domain interface. The study is useful for performing dense frequency sweeps in an efficient manner using the asymptotic waveform expansion (AWE) method. The study requires the input of a metric that keeps track of the acoustic response of the modeled system.

FREQUENCY MODAL FOR VIBROACOUSTIC MODELS

Both left and right eigenvectors are now computed when performing an eigenfrequency analysis. This allows the analysis of vibroacoustic (multiphysics) models using the modal solver.

PERFORMANCE IMPROVEMENTS FOR ACOUSTICS MODELS SOLVED WITH BEM

Several important improvements have been introduced when solving acoustic models with the boundary element method (BEM) using the Pressure Acoustics, Boundary Element interface.

- The evaluation of the BEM kernel for complex-valued wave numbers (models with damping) has been optimized. As an example, the evaluation of the Radiation Pattern in the Submarine Target Strength model in the Application Library is now 25% faster. The speedup depends on the model size.
- Load balancing for BEM models run on clusters has been significantly improved. As an example, solving the Submarine Target Strength model at 6 kHz on six nodes on a cluster is now 7.5 times faster in COMSOL 6.2 when compared with the previous version, now solving in 55 min instead of 7 h 30 min. The peak memory and memory balancing has also been greatly improved. This is a substantial speedup for solving large acoustic problems (the specific speedup is problem- and hardware-dependent).
- An improved solver is now available for solving models using the stabilized BEM method in noncluster configurations (on a regular workstation). As an example, the Submarine Target Strength model now solves in 16 min as compared to 25 min in the previous version. The actual speedup is problem- and hardware-dependent.
- There is now an option for improved thin gap integration handling.

NEW MODELS

The following models are new in version 6.2:

- Wave-Based Time-Domain Room Acoustics with Frequency-Dependent Impedance
- MEMS Microphone with Slip Wall
- Generation of Lamb Waves for Nondestructive Inspection of Plate Specimens
- Viscous Damping of a Microperforated Plate in the Slip Flow Regime
- Car Cabin Acoustics — Frequency-Domain Analysis
- Baffled Piston Radiation
- Loudspeaker Driver 3D — Frequency-Domain Analysis
- Eigenmodes of an Air Bubble with Surface Tension
- Scattered Field Formulation for Elastic Waves
- Elastic Cloaking with Polar Material
- Thermoacoustic Engine and Heat Pump
- Fuel Tank Vibration
- Nonlinear Transfer Impedance of a Tapered Orifice
- Type 4.3 Ear Simulator
- Transverse Isotropic Porous Layer
- Active Flame Validation

UPDATED MODELS

The following models have been updated in version 6.2:

- Small Concert Hall Acoustics
- Chamber Music Hall
- Porous Absorber
- Sound Transmission Loss Through a Concrete Wall
- Flow Duct
- Axisymmetric Condenser Microphone
- Optimizing the Shape of a Horn
- Tweeter Dome and Waveguide Shape Optimization

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
- Opto-Acoustophoretic Effect in an Acoustofluidic Trap
- Acoustic Trap in Glass Capillary with Bias Flow
- Room Impulse Response of a Smart Speaker
- Smartphone Microspeaker and Port Acoustics: Linear and Nonlinear Analysis
- Sound Transmission Loss Through a Window

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 6.2

GENERAL

- The **Electrode** domain node has been renamed **Current Conductor** in the current distribution interfaces.
- Contact resistance can now be included in the **Electric Ground**, **Electric Potential**, **Electrode Current**, and **Electrode Power** external boundary condition nodes in the Electrochemistry interfaces.
- An issue related to linearizing concentration dependence for the **Lumped Multistep** kinetics type in the Tertiary Current Distribution interface has been fixed.
- The **Knudsen diffusion** check box in the interface top node in the Transport of Concentrated Species interface has been renamed **Include pore-wall interaction** and moved to the **Fluid** subnode (under the porous-media nodes). Pore-wall interaction is now also supported for the Maxwell–Stefan Diffusion model. In addition, the **Kinetic gas theory** option has been renamed **Knudsen diffusivity**.
- A **Contributing Velocity** node can now be added to the **Porous Medium** node in Darcy’s law. This node is typically used together with pore-wall interactions to add the wall (diffusion slip) velocity stemming from the wall interactions to the Darcy fluid velocity. The wall velocity variable is only available when using the Maxwell–Stefan diffusion model in the Transport of Concentrated Species interface.
- The Transport of Concentrated Species interface now supports anisotropic **Tortuosity** in porous media. The new functionality allows for specifying different in-plane and through-plane effective gas diffusivities.
- To model diffusion in solid phase materials, a new **Solid** feature has been introduced in the Transport of Diluted Species interface. The new feature supports anisotropic diffusion as well as translation and deformation of the solid domain.
- The new Free and Porous Media, Darcy multiphysics interface is now available from the Model Wizard. The interface combines the Darcy’s Law and Laminar Flow interfaces and is used to compute fluid velocity and pressure fields of single-phase flow in models including both free flow and porous media.
- The Free and Porous Media interface has been renamed and is now available under the name Free and Porous Media, Brinkman.

BATTERY PACK INTERFACE

- The **Battery pack capacity** text field has been replaced by the text field **Initial battery cell capacity**. (This change was also implemented in Update 1 for version 6.1.)
- **Internal contact resistances** in the Battery Pack interface can now be added to the **Negative Connectors** and **Positive Connectors** nodes.
- The Battery Pack interface supports **From material** inputs for **Initial battery cell capacity**, Open circuit voltage related inputs, and overpotential related lumped parameters.
- A new node group, **Lumped Batteries**, has been added in the **Battery Material Library**. Included in this group is the sample material Lumped Battery Example, which includes the **Initial battery cell capacity** and Open circuit voltage related inputs.

LITHIUM-ION AND BATTERY WITH BINARY ELECTROLYTE INTERFACES

- The new **SOC and Initial Cell Charge Distribution** node automatically defines cell-level state-of-charge (SOC) and state-of-health (SOH) variables. The same node can also be used to set an initial cell-level charge distribution based on either an initial cell SOC, an open-circuit cell voltage, individual electrode potentials, or individual electrode charge inventories. The node is activated by selecting the **Enable state of charge (SOC) and initial charge distribution** check box in the interface top node.
- The **Tortuosity** in the **Porous Electrode**, **Separator**, **Porous Conductive Binder**, and **Highly Conductive Porous Electrode** nodes can now be defined using anisotropic diagonal or symmetric tensors. The setting can be used in the **Effective Transport Parameter Correction** section to specify, for instance, different in-plane and through-plane diffusivity and conductivity values.
- The **Particle Intercalation** node now supports setting an **Initial electrode potential**.
- The default domain has been changed to **Separator**.
- The default for **Electrical conductivity** has been changed to **User defined** with the default value 10 S/m in the **Porous Electrode** and **Porous Conductive Binder** nodes of the Lithium-Ion and Battery with Binary Electrolyte interfaces
- Fixed issue related to the migrative boundary flux by moving migrative boundary flux contributions from individual domain nodes to the interface node. The fix may impact results in models using the **Internal Electrode Surface** node.
- The **Initial Cell Charge Distribution** node is now a legacy node and has been removed from the context menu.

DEFAULT SOLVER CHANGES IN THE BATTERY INTERFACES

- The default **Initial time step** is now set to 1 s in the Time-Dependent Solver for all battery interfaces.
- The Battery Pack, Lumped Battery, and Single Particle Battery interfaces now enable **Stores solution before and after events** by default in the Time-Dependent Solver.

CHEMISTRY INTERFACE

- An issue related to converting concentrations to partial pressures has been fixed.
- An issue was fixed in the **Electrode Reaction** node. The issue resulted in the local current density variable not being properly defined if a user-defined equilibrium potential was combined with, for example, Butler–Volmer.

Reaction formulas entered in a **Reaction** feature are now automatically updated in the following manner:

- When the same species occurs several times in a formula, all contributions on the left-hand side and the right-hand side are separately summed.
- When the type of a species is changed to a surface species, the phase indication (ads) is added after the species name in all reactions in which it participates. When changing it to another type, the phase indication is removed.
- To make the formula easier to read, a space is added around all participating species.

New and Updated Models in Version 6.2

- 1D Isothermal Sodium-Ion Battery (*na_ion_battery_1d*)
- Silicon–Graphite Blended Electrode with Thermodynamic Voltage Hysteresis (*li_battery_sigr_hysteresis*)
- Battery Electrode with a Particle Size Distribution (*particle_size_distribution*)
- Surrogate Model Training of a Battery Rate Capability Model (*lib_rate_capability_surrogate*)
- The model Modeling Impedance in the Lithium-Ion Battery (*li_battery_impedance*) and the corresponding app, Lithium-Ion Battery Impedance (*li_ion_battery_impedance*), have been updated to use the IPOPT optimization solver
- The models *pouch_cell_utilization*, *lib_rate_capability*, *lib_internal_resistance*, *lib_drive_cycle*, *lib_diffusion_induced_stress*, *capacity_fade*,

li_battery_solid_electrolyte, and *li_battery_multiple_materials_1d* have all been updated to make use of the new SOC and Initial Cell Charge Distribution node.

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

New Functionality in Version 6.2

CAD IMPORT MODULE GEOMETRY KERNEL UPGRADE

The CAD Import Module, the Design Module, and the LiveLink™ products for CAD utilize the Parasolid® geometry kernel from Siemens PLM for solid modeling operations, geometry repair, and defeaturing. (Without these products, a COMSOL-native geometry modeling kernel is used.) The CAD Import Module released with COMSOL® version 6.2 includes an upgraded version of the Parasolid kernel. As a result, a number of stability issues have been fixed, which makes geometry modeling operations and the import of CAD models more robust.

- The CAD file import and export functionality included with these products has been extended to support new versions for some of the supported file formats (for details, see under Read from File, CAD on www.comsol.com/products/specifications/cad/).
- The Import node for 3D STEP files now includes the setting **Import body names** with the options **Automatic**, **On** and **Off** to control whether the body names are included in the object names after the import. With the **Automatic** option the body names are only included in the names of objects imported from multibody parts.
- The settings for the Cap Faces feature now include the option **Group adjacent edges** that will automatically select all edges in a loop when clicking on an edge.
- The CAD file import functionality now also supports the import of SOLIDWORKS® files on a supported Linux® operating system with an Intel® 64-bit processor.
- When using the CAD kernel, the automatic tolerance setting for the Form Assembly operation has been adjusted to a relative tolerance of 1e-5 that yields better pair detection for small overlaps.

New Functionality in the Design Module in Version 6.2

- The Loft operation now has an option **Remove redundant profile vertices** that can remove vertices that separate identical curves on the profile objects. The option

makes it easier to use the output of Interpolation Curve features as profiles for the loft.

- A new check box **Propagate selections to resulting objects** is available with the Loft, Offset Faces, Thicken, and Transform Faces operations. Clear this check box if you want that named selection refer only to the input objects when you choose to keep those.
- The settings for the Offset Faces feature now include a check box **Subtract input objects**. Select this check box to, after the faces are offset, subtract the input objects from the corresponding offset objects.

New Functionality in LiveLink™ for AutoCAD® in Version 6.2

The LiveLink™ interface now supports AutoCAD® 2024.

New Functionality in LiveLink™ for Inventor® in Version 6.2

- The LiveLink™ interface now supports Inventor® 2024.
- The Component Selection window has been added to the CAD software user interface for selecting assembly components for synchronization regardless of the visibility state of components. Previously, visible components were always synchronized with the assembly.
- The names of geometry objects after synchronization are now derived from the component names for CAD assemblies.

New Functionality in LiveLink™ for PTC Creo Parametric™ in Version 6.2

- The LiveLink™ interface now supports PTC Creo Parametric™ 10.0.
- The Component Selection window has been added to the CAD software user interface for selecting assembly components for synchronization regardless of the visibility state of components. Previously, visible components were always synchronized with the assembly.
- The names of geometry objects after synchronization are now derived from the component names for CAD assemblies.
- If you are using PTC Creo Parametric™ together with the Windchill® PLM software, synchronization is now supported for files that are not checked out. Note

however, that the associativity of the synchronized geometry can only be ensured for files that are checked out.

- The LiveLink™ interface now supports the synchronization of multibody parts that contain both solid and surface bodies.

LiveLink™ for PTC Pro/ENGINEER®

Starting with version 6.2 of COMSOL Multiphysics, we are no longer offering our LiveLink™ for PTC Pro/ENGINEER® product.

New Functionality in LiveLink™ for Revit® in Version 6.2

The LiveLink™ interface now supports Revit® 2024.

New Functionality in LiveLink™ for Solid Edge® in Version 6.2

- The Component Selection window has been added to the CAD software user interface for selecting assembly components for synchronization regardless of the visibility state of components. Previously, visible components were always synchronized with the assembly.
- The names of geometry objects after synchronization are now derived from the component names for CAD assemblies.

CFD Module

New Functionality in Version 6.2

- The Large Eddy Simulation (LES) and Detached Eddy Simulation (DES) interfaces are now also available for weakly compressible and compressible flows.
- Turbulent flow interfaces for all RANS models have been added under the **High Mach Number Flow** section in the **Fluid Flow** branch.
- Physics interfaces for Incompressible Potential Flow and Compressible Potential Flow have been added to the **Fluid Flow** branch. These interfaces can be used to model irrotational flows or to get initial values for more complex fluid-flow simulations.
- A new option, **Limit small time steps effect on stabilization time scale**, in the **Consistent Stabilization** section in the **Settings** window for the **Single-Phase Flow** interfaces, can be used to avoid the loss of pressure stabilization during initialization and for small time steps. This option is selected by default for the LES and DES interfaces.
- By default, the LES and DES interfaces get the **Maximum step constraint** set to **Expression** with the **Maximum step** set to `spf.dt_CFL` in the **Time-Dependent Solver** node when using automatic solver settings.
- The Free and Porous Media Flow interface has been renamed to Free and Porous Media Flow, Brinkman. Models created with earlier versions containing the Free and Porous Media Flow interface will keep the old name for the interface.
- The new Free and Porous Media Flow, Darcy predefined multiphysics interface under the **Porous Media and Subsurface Flow** section in the **Fluid Flow** branch couples a Darcy's Law interface and a Laminar Flow interface using the new **Free and Porous Media Flow Coupling** multiphysics coupling node.
- The new Multiphase Free and Porous Media Flow predefined multiphysics interface under the **Porous Media and Subsurface Flow** section in the **Fluid Flow** branch couples Laminar Flow, Darcy's Law, and the new Phase Transport in Free and Porous Media Flow interfaces.
- The new Thin-Film and Porous Media Flow interface, under the **Porous Media and Subsurface Flow** section in the **Fluid Flow** branch, couples the Thin-Film Flow and Darcy's Law interfaces using the **Thin-Film and Porous Media Flow** multiphysics coupling node.

- The new **Porous Interface** feature in the Laminar Flow, Turbulent Flow interface and Brinkman Equations interface, applicable on boundaries with different porosity on each side, accounts for changes in the stress caused by changes in porosity on each side and can include optional pressure losses.
- The **Porous slip** option is the new default setting for **Porous treatment of the no slip condition** in the Brinkman Equations interface or when **Enable porous media domains** is selected in the Laminar Flow interface.
- The **Wetted Wall** multiphysics feature, used together with the Two-Phase Flow, Level Set interface, now has the option to apply the **Porous slip** condition on walls adjacent to porous domains modeled with the Brinkman Equations interface.
- The mesh velocity from the **Free Surface** and **Fluid-Fluid Interface** features can now be overridden by alternative settings in the **Moving Mesh** node.
- A warning message is displayed in the **Log** window when **Use reduced pressure** or **Compensate for hydrostatic pressure/Compensate for hydrostatic pressure approximation** is selected in a fluid-flow interface that is part of a multiphase flow multiphysics coupling.

New Models in Version 6.2

TURBULENT FLOW AROUND A FACTORY CHIMNEY

This tutorial model simulates turbulent flow around a factory chimney and investigates the influence of helical strakes on the downstream vortices.

EPPLER AIRFOIL TRANSITION

This verification example simulates the flow around the Eppler 387 airfoil with the SST turbulence model and demonstrates how the transition model is able to capture the separation-bubble induced laminar-to-turbulent boundary-layer transition on the upper surface of the airfoil.

FDA BENCHMARK BLOOD PUMP

In this verification example, the turbulent flow in a centrifugal blood pump is simulated using the SST turbulence model, and the results are compared to experimental data.

SELF-LUBRICATING JOURNAL BEARING

This verification example describes how to model the self-lubricating phenomenon of a liquid lubricant in a journal bearing to which a porous bush has been press fitted and impregnated with the lubricant.

SQUEEZE FILM BETWEEN RECTANGULAR POROUS AND NONPOROUS PLATES

This verification example describes how to simulate the flow of a thin film of fluid in the gap between two rectangular plates, one of them with a porous facing, when the fluid is squeezed as a consequence of the relative motion between the plates.

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 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.

Backward Compatibility with Version 5.6 and Earlier

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

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

Chemical Reaction Engineering Module

New Functionality in Version 6.2

FEATURES FOR EVAPORATION, CONDENSATION, AND VAPOR TRANSPORT

Three new features for phase transfer and vapor transport in gaseous mixtures are available in the Transport of Concentrated Species interface:

The **Vapor Inflow** feature can be used to prescribe vapor inlet conditions injected from an upstream system involving liquids.

The **Vapor-Liquid Interface** prescribes equilibrium vapor conditions on a boundary representing a liquid surface, and can be used to model mass transfer due to evaporation and condensation.

When modeling liquid solutions, the **Vapor-Liquid-Mixture Interface** can be used to solve for the composition in the liquid phase as well as in the vapor phase.

VAPOR FLOW MULTIPHYSICS INTERFACES

Multiphysics interfaces, using the new features for vapor transport, are now available in the **Vapor Flow** branch under **Chemical Species Transport** when adding physics interfaces. These interfaces can be used to simulate evaporation and condensation at a vapor-liquid interface and to track the position of the interface.

The Laminar Vapor Flow, Concentrated Species multiphysics interface and the Laminar Vapor Flow, Concentrated Species, Moving Mesh multiphysics interface are available with a license for the Chemical Reaction Engineering Module. When your license also included the CFD Module or the Heat Transfer Module, interfaces that support turbulent flow are also available.

FLUID-FLUID INTERFACE

The **Fluid-Fluid Interface** feature in the fluid-flow interfaces is now available with a license for the Chemical Reaction Engineering Module. This feature defines the initial position of an interface separating two fluid phases and includes equations to track the evolution of the interface. It can be combined with vapor-liquid interface features to track the interface motion due to evaporation and condensation.

MASS-PRESERVING INITIALIZATION

When modeling equilibrium reactions, **Mass-preserving initialization** can be enabled in the **Initial Values** node of the Reaction Engineering interface. When enabled, the solver will first solve for the values of the species participating in equilibrium reactions, while disregarding any kinetic reactions. The solution to the equilibrium subproblem is then used as a consistent starting point for the time stepping.

PARAMETER ESTIMATION

Functionality for performing parameter estimation is now included with a license for the Chemical Reaction Engineering Module. The **Parameter Estimation** node and its **Global Least-Squares Objectives** subnode can be used to define objectives for parameter estimation. The **Parameter Estimation** study node can be used to perform the estimation.

SOLID FEATURE IN TRANSPORT OF DILUTED SPECIES

In order to model diffusion in solid phase materials, a new **Solid** feature has been introduced in the Transport of Diluted Species interface. The **Solid** feature supports anisotropic diffusion as well as translation and deformation of the solid domain.

IMPROVED MASS CONSERVATION IN REACTING FLOW

Conservation of mass has been significantly improved when coupling mass transport to fluid flow using conservation compensation. This functionality is included in the Reacting Flow, Diluted Species multiphysics interfaces when modeling an incompressible fluid. Using the Reacting Flow, Concentrated Species multiphysics interfaces, conservation compensation is used when the fluid is compressible.

REACTING FLOW COUPLING FEATURE SUPPORTS POROUS MEDIA

The **Reacting Flow** multiphysics coupling feature now supports porous media. When one of the coupled physics interfaces includes a porous media domain, the coupling feature checks that all coupled interfaces account for porous media in the domain.

ANISOTROPIC TORTUOSITY

It is now possible to use an anisotropic tortuosity when modeling mass transport in porous media with the Transport of Concentrated Species interface. An anisotropic tortuosity factor can be applied in the **Fluid** subnode when using any of the **Porous Medium**, **Porous Catalyst**, and **Packed Bed** features.

PORE-WALL INTERACTION REPLACES KNUDSEN DIFFUSION

The **Knudsen diffusion** check box on the interface top node of the Transport of Concentrated Species interface has been renamed **Include pore-wall interaction** and has been moved to the **Fluid** subnode (under porous media features). Pore-wall interaction is now also supported for the Maxwell–Stefan diffusion model. In addition, the **Kinetic gas theory** option has been renamed **Knudsen diffusivity**.

A **Contributing Velocity** subnode can now be added to the **Porous Medium** node in the Darcy’s Law interface. This subnode is typically used together with pore-wall interaction to add the wall (diffusion slip) velocity stemming from the wall interactions to the Darcy fluid velocity. The wall velocity variable is only available when using the Maxwell–Stefan diffusion model in the Transport of Concentrated Species interface.

SHRINKING CORE VISUALIZATION IN THE PELLETS PLOT

The **Pellets** plot has been extended with support for the shrinking core model. When enabled, the size of the solid core part is updated, and concentrations are plotted in the porous region surrounding the core.

NEW FREE AND POROUS MEDIA FLOW INTERFACES

The new Free and Porous Media, Darcy multiphysics interface is now available. The interface combines the Darcy’s Law and the Laminar Flow interfaces and is used to compute fluid velocity and pressure fields of single-phase flow where free flow is connected to flow through porous media.

The Free and Porous Media interface is now available under the name Free and Porous Media, Brinkman.

SHOWING COORDINATES OF GRAPH PLOTS

For graph plots in 1D plot groups, you can now dynamically inspect the value of a graph by hovering over it with the mouse. The x - and y -coordinate values of the current position are shown in a tooltip. The tooltip appears by default; click the **Enable Tooltip** button in the Graphics window toolbar to disable it.

REACTING FLOW, CONCENTRATED SPECIES

Multiphysics interfaces previously named Reacting Flow are now named Reacting Flow, Concentrated Species.

IMPROVED REACTION FORMULAS

Reaction formulas entered in a **Reaction** feature are now automatically updated in the following manner:

- When the same species occurs several times in a formula, all contributions on the left side and the right side are summed, respectively.
- When the type of a species is changed into a surface species, the phase indication (ads) is added after the species name in all reactions it participates in. If you change it into another type, the phase indication is removed.
- To make the formula easier to read, a space is added around all participating species.

New Models in Version 6.2

PRECIPITATION OF BARIUM SULFATE

Crystallization is an important separation process in the chemical industry. It is used for the production of pharmaceuticals and industrial chemicals. Crystallization can also be used in resource recovery as a way of separating valuable materials from waste. This model solves a discretized formulation of the population balance equation to obtain the crystal size distribution from precipitation of barium sulfate in 0D and in a 3D T-mixer.

PARAMETER ESTIMATION FOR PYROLYSIS OF WOOD

The pyrolysis of a centimeter-sized wood particle presents a fully coupled multiphysics problem with mass transfer, fluid flow, and heat transfer. This example model consists of two parts. The first part demonstrates how to set up a model describing the pyrolysis of a porous, anisotropic wood sphere. A reaction scheme consisting of five pseudospecies and five reactions is used. After that, parameter estimation is used to optimize the model using experimental data. The parameters to be estimated are one Arrhenius constant, two reaction heats, and one external heat transfer coefficient.

THERMAL DECOMPOSITION OF BETA-CAROTENE IN A FLOW REACTOR

The stability of beta-carotene in a fruit juice, which flows past a heated cylinder, is studied using **Uncertainty Quantification (UQ)** study types. The UQ studies identify the most influential parameters and quantify the importance of their respective uncertainties. The most likely outcome for the beta-carotene outflow, which is the quantity of interest (QoI), and the likelihood of it falling short of a lower limit is determined.

ACID-BASE EQUILIBRIA AND COPPER SPECIATION IN AMMONIA SOLUTION

Cupric ions show a strong affinity to ammonia in aqueous solutions, forming strongly colored deep blue complexes. The relative amounts of the different amine ligand complexes, with varying coordination numbers, are governed by the stability constants of the equilibria forming the coordination compounds. By entering a series of equilibrium reactions and an external source of ammonia, the **Time Dependent** solver can be used to produce a speciation diagram, which shows how the concentration of the different complexes vary with the total concentration of added ammonia.

Updated Models in Version 6.2

TURBULENT MIXING

This model has been updated to use the new **Periodic values of variables not solved for** option in the time-dependent solver. The fluid-flow field is solved in a time-dependent analysis until a quasi-steady flow field is developed. When modeling the turbulent mixing of a trace species, the fluid that flows from the last revolution of the impeller is sequentially reused by applying the new solver option.

STEFAN DIFFUSION

The new **Comparison** node, available for line graphs, is used to compute the root-mean square difference between the computed results and the imported experimental results.

PARAMETER ESTIMATION MODELS

Models that include parameter estimation have been updated to use the functionality now included in the Chemical Reaction Engineering Module. Those models now use the **Global Least-Squares Objective** node for defining objectives and the **Parameter Estimation** study node to perform the estimation. The following models have been updated:

- Finding Kinetic Arrhenius Parameters Using Parameter Estimation
- Degradation of DNA in Plasma
- Parameter Estimation for Nonideal Reactor Models

Composite Materials Module

New Functionality in Version 6.2

PIEZOELECTRIC MATERIAL, LAYERED MATERIAL MODEL

The **Piezoelectric Material** model in the Layered Shell interface is now also available as **Piezoelectric Material, Layered** in the Shell interface. As a result of this addition, thin piezoelectric composites can be analyzed using the Shell interface. This can give significant savings in terms of computation time.

This feature requires a license for the AC/DC Module or the MEMS Module. If the license also includes the Composite Materials Module, the material model can be used in multilayered shells, and the individual layers can have different material properties.

NEW MULTIPHYSICS INTERFACE: PIEZORESISTIVITY, LAYERED SHELL

The new Piezoresistivity, Layered Shell multiphysics interface is used to model piezoresistivity in layered structures. The interface combines an Electric Currents in Layered Shells interface, a Layered Shell interface, and new **Piezoresistivity, Layered** multiphysics coupling.

This feature requires a license for the MEMS Module.

PARTS FOR HOMOGENIZATION OF MICROSTRUCTURES

In the part libraries, an existing folder named **Representative Volume Elements** under the **COMSOL Multiphysics** branch has been renamed to **Unit Cells and RVEs**. New microstructure geometries for corrugated sheets, honeycombs, and gyroid have been added.

MISCELLANEOUS

- For the **Contact** feature in the Layered Shell interface, a predefined plot for contact forces is added. The predefined plot shows contact pressures and friction forces on the destination boundaries.
- The selection of variables representing the normal vector in various edge level features in Layered Shell interface is changed to the base selection.

HOMOGENIZED MATERIAL PROPERTIES OF PERIODIC MICROSTRUCTURES

In this app, homogenized material properties of various periodic microstructures can be computed. Such data can then be used in macroscopic analyses.

Ten different microstructures represented by unit cells are available. The dimensions and the properties of the constituent materials are given by the user of the app. The app calculates the homogenized density, elasticity matrix, coefficient of thermal expansion, heat capacity, and thermal conductivity.

The computed data can be stored in an MPH file or an XML file that can subsequently be imported into a COMSOL Multiphysics model.

STACKING SEQUENCE OPTIMIZATION

In this example, the stacking sequence of composite laminate is optimized based on the Hashin failure criterion. An optimization analysis is performed to find the optimum fiber orientation in each layer under specified loading conditions with the objective of minimizing the maximum failure index value in the laminate. The BOBYQA method is applied to find the optimum stacking sequence.

Corrosion Module

New Functionality in Version 6.2

GENERAL

- The **Electrode** domain node has been renamed **Current Conductor** in the current distribution interfaces.
- Contact resistance can now be included in the **Electric Ground**, **Electric Potential**, and **Electrode Current** external boundary condition nodes in the Electrochemistry interfaces.
- An issue related to linearizing concentration dependence for the **Lumped Multistep** kinetics type in the Tertiary Current Distribution interface has been fixed.
- To model diffusion in solid phase materials, a new **Solid** feature has been introduced in the Transport of Diluted Species interface. The new feature supports anisotropic diffusion as well as translation and deformation of the solid domain.
- The new Free and Porous Media, Darcy multiphysics interface is now available from the Model Wizard. The interface combines the Darcy's Law and the Laminar Flow interfaces and is used to compute fluid velocity and pressure fields of single-phase flow in models including both free flow and porous media.
- The Free and Porous Media interface has been renamed and is now available under the name Free and Porous Media, Brinkman.

CATHODIC PROTECTION AND SECONDARY CURRENT DISTRIBUTION INTERFACES

- The new **Impressed Current Point** node can be added at any point in an electrolyte domain to define the location of the active electrode of an impressed current protection system. The feature is typically used together with a **Connection Point** node for setting up a local impressed cathodic current protection system.
- The new **Sacrificial Point Anode** feature can be added at any point in an electrolyte domain to define a sacrificial anode.
- The **Connection Point** feature, available as a subnode to an **Edge Electrode** or a **Sacrificial Edge Anode** feature has been updated with a **Define reference electrode** check box which allows for setting a reference electrode potential at the same location. The reference electrode potential can typically be used by an **Impressed**

Current Point feature to actively control the electrode potential at the location of the connection point.

- A new **External Short** point feature is now available as subnode to an **Edge Electrode** for more flexible connection options between various points in a model.
- An issue related to duplicate parameter or variable name errors when using multiple **Connection Point** subnodes has been fixed.
- A current contribution sign error for the **Floating potential** electric potential option in the **Edge Electrode** and **Sacrificial Edge Anode** features has been fixed. (Also fixed in Update 2 for COMSOL Multiphysics version 6.1.)

CHEMISTRY INTERFACE

- An issue related to converting concentrations to partial pressures has been fixed.
- An issue was fixed in the **Electrode Reaction** node. The issue resulted in the local current density variable not being properly defined if a user-defined equilibrium potential was combined with, for example, Butler–Volmer.

Reaction formulas entered in a **Reaction** feature are now automatically updated in the following manner:

- When the same species occurs several times in a formula, all contributions on the left-hand side and the right-hand side are separately summed.
- When the type of a species is changed to a surface species, the phase indication (ads) is added after the species name in all reactions in which it participates. When changing it to another type, the phase indication is removed.
- To make the formula easier to read, a space is added around all participating species.

New and Updated Models in Version 6.2

- A new model named Pipeline Corrosion Protection Using Impressed Current Cathodic Protection (*pipeline_corrosion_protection_iccp*) has been added.
- A new Evans Droplet (*evans_droplet*) model has been added.
- The *co2_corrosion* model has been updated to more correctly define carbonic acid speciation and the resulting pH effects.

ECAD Import Module

New Functionality in Version 6.2

- The ECAD Import Module includes the new option **Metal layer between dielectric layers** for **Elevations**, when automatically generating layer stackup for imported IPC-2581, and ODB++ files. Previous options, including the manual specification of elevations and the automatic positioning of copper layers inside the dielectric layer above the copper, are also available.
- For the import of GDS files, the **All** option for **Grouping of geometries** has been removed as it is only applicable in very special cases.

Electrochemistry Module

New Functionality in Version 6.2

GENERAL

- The **Electrode** domain node has been renamed **Current Conductor** in the current distribution interfaces.
- **Contact Resistance** can now be included in the **Electric Ground**, **Electric Potential**, and **Electrode Current** external boundary condition nodes in the Electrochemistry interfaces.
- An issue related to linearizing concentration dependence for the **Lumped Multistep** kinetics type in the Tertiary Current Distribution interface has been fixed.
- To model diffusion in solid phase materials, a new **Solid** feature has been introduced in the Transport of Diluted Species interface. The new feature supports anisotropic diffusion as well as translation and deformation of the solid domain.
- The new Free and Porous Media, Darcy multiphysics interface is now available from the Model Wizard. The interface combines the Darcy's Law and the Laminar Flow interfaces and is used to compute fluid velocity and pressure fields of single-phase flow in models including both free flow and porous media.
- The Free and Porous Media interface has been renamed and is now available under the name Free and Porous Media, Brinkman.

CHEMISTRY INTERFACE

- An issue related to converting concentrations to partial pressures has been fixed.
- An issue was fixed in the **Electrode Reaction** node. The issue resulted in the local current density variable not being properly defined if a user-defined equilibrium potential was combined with, for example, Butler–Volmer.

Reaction formulas entered in a **Reaction** feature are now automatically updated in the following manner:

- When the same species occurs several times in a formula, all contributions on the left-hand side and the right-hand side are separately summed.

- When the type of a species is changed to a surface species, the phase indication (ads) is added after the species name in all reactions in which it participates. When changing it to another type, the phase indication is removed.
- To make the formula easier to read, a space is added around all participating species.

Electrodeposition Module

New Functionality in Version 6.2

GENERAL

- The **Electrode** domain node has been renamed **Current Conductor** in the current distribution interfaces.
- **Contact Resistance** can now be included in the **Electric Ground**, **Electric Potential**, and **Electrode Current** external boundary condition nodes in the Electrochemistry interfaces.
- An issue related to linearizing concentration dependence for the **Lumped Multistep** kinetics type in the Tertiary Current Distribution interface has been fixed.
- To model diffusion in solid phase materials, a new **Solid** feature has been introduced in the Transport of Diluted Species interface. The new feature supports anisotropic diffusion as well as translation and deformation of the solid domain.
- The new Free and Porous Media, Darcy multiphysics interface is now available from the Model Wizard. The interface combines the Darcy's Law and the Laminar Flow interfaces and is used to compute fluid velocity and pressure fields of single-phase flow in models including both free flow and porous media.
- The Free and Porous Media interface has been renamed and is now available under the name Free and Porous Media, Brinkman.

CHEMISTRY INTERFACE

- An issue related to converting concentrations to partial pressures has been fixed.
- An issue was fixed in the **Electrode Reaction** node. The issue resulted in the local current density variable not being properly defined if a user-defined equilibrium potential was combined with, for example, Butler–Volmer.

Reaction formulas entered in a **Reaction** feature are now automatically updated in the following manner:

- When the same species occurs several times in a formula, all contributions on the left-hand side and the right-hand side are separately summed.

- When the type of a species is changed to a surface species, the phase indication (ads) is added after the species name in all reactions in which it participates. When changing it to another type, the phase indication is removed.
- To make the formula easier to read, a space is added around all participating species.

New Model in Version 6.2

ALLOY DEPOSITION

A new Alloy Deposition (*alloy_deposition*) model has been added.

Fatigue Module

New Functionality in Version 6.2

HARMONIC VIBRATION IMPROVEMENT

For frequency-sweep computations, the list of excitation frequencies entered in the preceding frequency-domain analysis no longer needs to be monotonic. The Fatigue Module internally considers the frequencies in ascending order. Additionally, the sign of the entered **Rate of change** under **Harmonic Vibration** is disregarded.

New Model in Version 6.2

BRACKET FATIGUE HARMONIC VIBRATION

A new tutorial for fatigue using **Harmonic Vibration** has been added. The *bracket_fatigue_harmonic_vibration* tutorial considers a steel bracket subjected to harmonic base excitation, and it includes the effect of a static load case (mean stress).

Fuel Cell & Electrolyzer Module

New Functionality in Version 6.2

GENERAL

- The **Electrode** domain node has been renamed **Current Conductor** in the current distribution interfaces.
- **Contact Resistance** can now be included in the **Electric Ground**, **Electric Potential**, **Electrode Current**, and **Electrode Power** external boundary condition nodes in the Electrochemistry interfaces. The *nonisothermal_pem_fuel_cell* and *pemfc_serpentine_flow_field* models have been updated to use the new features.
- An issue related to linearizing concentration dependence for the **Lumped Multistep** kinetics type in the Tertiary Current Distribution interface has been fixed.
- The **Knudsen diffusion** check box in the interface top node in the Transport of Concentrated Species interface has been renamed **Include pore-wall interaction** and moved to the **Fluid** subnode (under the porous-media features). Pore-wall interaction is now also supported for the Maxwell–Stefan diffusion model. In addition, the **Kinetic gas theory** option has been renamed **Knudsen diffusivity**.
- A **Contributing Velocity** node can now be added to the **Porous Medium** node in Darcy’s law. This node is typically used together with pore-wall interactions to add the wall (diffusion slip) velocity stemming from the wall interactions to the Darcy fluid velocity. The wall velocity variable is only available when using the Maxwell–Stefan diffusion model in the Transport of Concentrated Species interface.
- The Transport of Concentrated Species interface now supports anisotropic **Tortuosity** in porous media. The new functionality allows for specifying different in-plane and through-plane effective gas diffusivities.
- To model diffusion in solid phase materials, a new **Solid** feature has been introduced in the Transport of Diluted Species interface. The new feature supports anisotropic diffusion as well as translation and deformation of the solid domain.
- The new Free and Porous Media, Darcy multiphysics interface is now available from the Model Wizard. The interface combines the Darcy’s Law and the Laminar Flow interfaces and is used to compute fluid velocity and pressure fields of single-phase flow in models including both free flow and porous media.
- The Free and Porous Media interface has been renamed and is now available under the name Free and Porous Media, Brinkman.

HYDROGEN FUEL CELL AND WATER ELECTROLYZER INTERFACES

- **Pore-wall interaction** can now be enabled in the **Gas Diffusion Layer** and **Gas Diffusion Electrode** nodes in the Hydrogen Fuel Cell and Water Electrolyzer interfaces. The wall diffusivity can either be defined using Knudsen diffusivities or user-defined values. The *sofc_unit_cell*, *soec* and *soec_thermodynamics* models have been updated to use the new feature.
- **Anisotropic tortuosities** can now be used in the **Gas Diffusion Layer** and **Gas Diffusion Electrode** nodes to compute effective diffusion coefficients in the Hydrogen Fuel Cell and Water Electrolyzer interfaces. The *pem_gdl_species_transport_2d* model has been updated to use the new feature.
- **Film resistance** functionality has been added to **H₂/O₂ Electrode Surface**, **H₂/O₂ Internal Electrode Surface**, and **H₂/O₂ Thin Gas Diffusion Electrode** nodes in the Hydrogen Fuel Cell and Water Electrolyzed interfaces.
- The new **Internal Electrode Contact Resistance** subnode can now be added to the **Electronic Conducting Phase** node.
- The new **Internal Electrolyte Contact Resistance** subnode can now be added to Electrolyte Phase node.
- The default value for the **Anodic transfer coefficient** has been changed to 1.5 in electrode reactions when running in electrolysis mode for Proton Exchange, Solid Oxide, and Molten Carbonate electrolyte types.

CHEMISTRY INTERFACE

- An issue related to converting concentrations to partial pressures has been fixed.
- An issue was fixed in the **Electrode Reaction** node. The issue resulted in the local current density variable not being properly defined if a user-defined equilibrium potential was combined with, for example, Butler–Volmer.

Reaction formulas entered in a **Reaction** feature are now automatically updated in the following manner

- When the same species occurs several times in a formula, all contributions on the left-hand side and the right-hand side are separately summed.
- When the type of a species is changed to a surface species, the phase indication (ads) is added after the species name in all reactions in which it participates. When changing it to another type, the phase indication is removed.
- To make the formula easier to read a space is added around all participating species.

New Model in Version 6.2

SHUNT CURRENTS IN AN ALKALINE ELECTROLYZER STACK

A new Shunt Currents in an Alkaline Electrolyzer Stack (*aec_shunt_currents*) tutorial has been added.

Geomechanics Module

New Functionality in Version 6.2

ENHANCEMENTS TO CONCRETE

- A new **Coupled damage-plasticity** model is available under the **Concrete** node.
- The **Mazars** damage model is available under the **Concrete** node.

ENHANCEMENTS TO SOIL PLASTICITY

A new flow-rule formulation based on the multiplicative decomposition of deformation gradients is available for the material models in the **Plasticity**, **Soil Plasticity**, and **Elastoplastic Soil Material** nodes. This means that these material models now better accommodate large strains.

ENHANCEMENTS TO HARDENING SOIL MATERIAL MODEL

The **Hardening Soil Material** model includes several enhancements:

- A new control **Failure criterion** has been added. You can choose between the **Mohr–Coulomb**, **Matsuoka–Nakai**, and **Panteghini–Lagioia** criteria.
- A new control named **Mobilized dilatancy angle** has been added. You can choose between the expressions **Soreide**, **Rowe**, **Modified Rowe**, **Wehnert**, **Li–Dafalias**, **Rowe–Li–Dafalias**, and **User Defined**.
- A new material property **Reference stiffness for primary loading** has been added.
- The cap of the yield surface is now centered at the tip of the cone of the p-q curve instead of at the (0,0) coordinate.
- The combination of yield functions has been smoothed.

HARDENING SOIL SMALL STRAIN MATERIAL MODEL

Under the **Elastoplastic Soil Material** node, **Hardening Soil Small Strain** has been added as a new material model.

SMALL STRAIN OVERLAY MATERIAL MODEL

Under the **Nonlinear Elastic Material** node, **Small Strain Overlay** has been added as a new material model.

ENHANCEMENTS TO PLASTICITY

- The user interface for the **Plasticity** node has been updated. The **Formulation** list has been removed; instead, the formulation is now controlled from the parent node.
- Kinematic hardening is available also for large-strain plasticity.
- A simplified user interface for **Set Variables** has been introduced. For the large-strain formulation, you now directly set the components of the plastic strain tensor. In models created in a previous version, in which the option **Plastic Deformation Gradient Inverse** was used, the new user input will be shown. The values of old user inputs will, however, not be converted. You must enter the new data.

NON-ZERO OUT-OF-PLANE INELASTIC SHEAR STRAINS

For 2D problems involving plasticity or creep, it is possible that the out-of-plane inelastic shear strains are nonzero if the material is anisotropic. This can now be accommodated by selecting the **Include out-of-plane strains** check box in the **Advanced** section for the corresponding features.

NEW MULTIPHYSICS COUPLING: PHASE FIELD DAMAGE

This multiphysics coupling connects a Phase Field in Solids interface and a Solid Mechanics interface. The stresses, strains, or strain energy drive the phase-field value, and the phase-field value provides the degree of damage to the elastic material model. The settings for the **Phase Field Damage** node are similar to those in a **Damage** node in Solid Mechanics.

NEW MULTIPHYSICS INTERFACE: PHASE FIELD DAMAGE

The new multiphysics interface Phase Field Damage combines a Solid Mechanics interface, a Phase Field in Solids interface and a **Phase Field Damage** multiphysics coupling. It is found under the **Structural Mechanics** branch in the **Add Physics** window.

New Models in Version 6.2

HYSTERESIS IN SOIL USING THE SMALL STRAIN OVERLAY MODEL

In this example, cyclic tensile and shear tests show the stiffness degradation and hysteresis effect with the small strain overlay model. The small strain overlay material model captures the effect of high stiffness at low strain as well as hysteresis under cyclic loading, which is a common effect for most soils. The formulation allows stiffness degradation with an increase in the shear strain and the recovery of the full stiffness at load reversal.

TRIAxIAL TEST WITH THE HARDENING SOIL SMALL STRAIN MATERIAL MODEL

In this example, monotonic and cyclic triaxial tests are simulated using the Hardening Soil Small Strain material model. The model captures the effects of small strain stiffness and hysteresis under cyclic loading. The stress–strain relationship matches the hyperbolic curve reported in a reference.

CONCRETE DAMAGE-PLASTICITY MATERIAL TESTS

This example shows the behavior of the coupled damage-plasticity material model for concrete when subjected to different loading conditions.

FAILURE OF A CONCRETE BEAM USING COUPLED DAMAGE-PLASTICITY

Failure of reinforced concrete structures involves advanced material behavior and interaction between materials. This example demonstrates how to model various stages of the failure of a reinforced concrete beam by including a coupled damage-plasticity material model for the concrete, metal plasticity for the rebars, and a nonlinear bond-slip law for the interaction between concrete and the rebar.

Heat Transfer Module

New and Improved Functionality in Version 6.2

SST TURBULENCE MODEL ADDED TO THE HEAT TRANSFER MODULE

The list of turbulence models available with the Heat Transfer Module has been extended with the Menter shear stress transport (SST) turbulence model. It combines the accuracy of the $k-\omega$ model and the robustness of the $k-\epsilon$ model. It is suitable for many external flow cases and internal flows with sudden expansions.

IMPROVED COMPUTATIONAL PERFORMANCE FOR ORBITAL THERMAL LOADS AND SURFACE-TO-SURFACE RADIATION

The evaluation of the heat flux of a source at infinity is now avoided on elements with very small view factors. In the Surface-to-Surface Radiation interface, this concerns the **External Radiation Source** feature when the **Source position** is set to **Infinite distance**. In the Orbital Thermal Loads physics interface, both the **Sun Properties** and **Planet Properties** features are concerned, since they can be considered as infinite distance sources. It results in a decrease of the CPU time and memory requirements, particular when used with the hemicube method. The performance gains increase with the number of external sources at infinite distance and interior mesh elements.

In the Surface-to-Surface Radiation and Orbital Thermal Loads interfaces, a new **Lower integration order for irradiation** option is available with the **Hemicube** algorithm in 3D and 2D. This option, which is activated by default in the **Jacobian Contribution** section, reduces the integration order and, therefore, the number of evaluation for irradiation. This speeds up the computation while having a minimal effect on results.

NONISOTHERMAL REACTING FLOW IN POROUS MEDIA

The **Nonisothermal Reacting Flow** multiphysics coupling is now available with porous media. For the coupling to be applicable, the domain must exclusively be either a fluid or porous medium in all three transport interfaces. In the Transport of Concentrated Species in Porous Media interface, the **Porous Medium** and **Porous Catalyst** features are supported. For both features, reactions in the fluid are supported. For the latter, reactions on the surface of the porous matrix are supported as well. In the Heat Transfer in Porous Media interface, the **Local thermal equilibrium** and **Local thermal nonequilibrium** models are supported.

ORBITAL THERMAL LOADS

The **Orbital Thermal Loads** physics interface now does not require an extra **Events** interface. The **Generate Events Interface** feature has been removed and replaced by the **Events Timeline** feature, which handles events internally to the interface. The use of the **Orbital Thermal Loads** interface is now simplified thanks to this feature.

DISCRETE ORDINATES METHOD IN 2D AXISYMMETRY

In the Radiation in Participating Media and Radiation in Absorbing-Scattering Media interfaces, the discrete ordinates method (DOM) is now available for 2D axisymmetric configurations with isotropic scattering.

THERMAL CONNECTION BETWEEN FACES

The new **Thermal Connection** feature, available in the Heat Transfer physics interfaces, is designed to connect two boundary selections by a thermal resistor, a thermal capacitor, or a Lumped Thermal System interface. This is used 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.

This feature greatly simplifies the setup of such models by eliminating the need to create a separate Lumped Thermal System interface for a thermal resistor or capacitor between two boundary selections.

For more advanced connection where a Lumped Thermal System interface is needed between the two boundary selections, only one **Thermal Connection** feature is now required instead of two **Lumped system connector** features.

THERMAL INSULATION CONDITION ON EDGES

A new **Thermal Insulation, Edge** feature has been added to the Heat Transfer physics interfaces and is available for 3D, 2D axisymmetric, and 2D components. It is designed to cancel the default continuity condition when two 3D objects are in contact along an edge, or when 2D objects are in contact at a point.

SURFACE-TO-SURFACE RADIATION AND ORBITAL THERMAL LOADS IMPROVEMENTS

In the Surface-to-Surface Radiation and Orbital Thermal Loads interfaces, the **Check consistency** option for the hemicube and ray-shooting methods has been updated to account for external radiation sources at finite or infinite distance. A solver warning is issued when external radiation sources irradiates a surface radiating toward the other side.

In the Surface-to-Surface Radiation and Orbital Thermal Loads interfaces, you can now set the opacity, layer opacity and radiation direction for all bands from one input when **User defined for all bands** is selected, regardless of the number of spectral bands. This option can be used if the transparency or opacity properties of the material are not wavelength dependent.

In the Surface-to-Surface Radiation and Orbital Thermal Loads interfaces, radiation directions are now displayed on boundaries on which opacity is the same for all wavelengths.

In the Orbital Thermal Loads interface, a new predefined **Trajectory Following Visualization** plot has been added. It makes it possible to visualize the trajectory of the spacecraft from the spacecraft's point of view.

DEFAULT AND PREDEFINED PLOTS

Default plots have been reworked for all interfaces in the Heat Transfer Module, and many predefined plots are now available depending on the physics interfaces and features present in the study. Several plots that were default plots are now instead optional and can be added from the **Add Predefined Plot** window. A default plot for the **Nonisothermal Flow** multiphysics coupling has also been added.

VISUALIZATION TOOLS FOR WAVELENGTH-DEPENDENT PROPERTIES

In the Surface-to-Surface Radiation, Orbital Thermal Loads, Radiation in Participating Media, and Radiation in Absorbing-Scattering Media interfaces, predefined functions have been added to help visualize the wavelength dependency of material properties. When selecting **Solar and ambient** or **Multiple spectral bands** wavelength dependence, numerous functions are available for surface and domain material properties and can be used under the **Results** node. Predefined plots have also been added to visualize both the continuous wavelength dependency of properties and the band-averaged values used in the computation.

NONISOTHERMAL FLOW

The **High viscous dissipation at wall** thermal wall function, available in the **Nonisothermal Flow** coupling, has been updated to automatically calculate the critical wall distance. The critical wall distance, which depends on turbulence parameters, gives the location of the switch between the linear and logarithmic behaviors in the boundary layer. This will improve the accuracy of the viscous heat flux calculation by the thermal wall function with nondefault turbulence parameters.

HEAT AND MOISTURE TRANSPORT IN ANISOTROPIC POROUS MEDIA

Support for anisotropic properties has been added for the following material properties in applicable physics interfaces:

- Heat Transfer interfaces: **Building Material: Vapor permeability**
- Moisture Transport interfaces: **Building Material: Moisture diffusivity**
- Moisture Transport interfaces: **Building Material: Vapor permeability.**
- Moisture Transport interfaces: **Hygroscopic Porous Medium: Moisture diffusivity**
- Moisture Transport interfaces: **Thin Moisture Barrier: Moisture diffusivity**

NEW OPTION FOR WEATHER STATION SELECTION

A new **Around location** option for choosing a weather station in **Ambient Properties** has been added to the **Ambient Properties** node. Once the latitude and longitude of a specific location are given, this option displays the 100 nearest weather stations using the haversine formula.

THIN MOISTURE BARRIER ON EXTERIOR BOUNDARIES

The **Thin Moisture Barrier** feature in the Moisture Transport interfaces is extended to be applicable on exterior boundaries.

NEW OPTIONS FOR PAIR THERMAL CONTACT FEATURE

In the Heat Transfer interfaces, the **Pair Thermal Contact** feature has new options to specify the **Equivalent thin resistive layer**. It is now possible to specify the **Total resistance** or **Total conductance** of the thermal contact. With these options, the resistance or conductance of the thermal contact is independent of the contact surface.

MISCELLANEOUS IMPROVEMENTS

- In the Radiation in Participating Media and Radiation in Absorbing-Scattering Media interfaces, the **Pair Continuity** and **Periodic Condition** features have been corrected in 2D axisymmetric geometries to ensure continuity or periodicity, respectively.
- The model input used in the Surface-to-Surface Radiation, Orbital Thermal Loads, Radiation in Participating Media, and Radiation in Absorbing-Scattering Media interfaces has been changed to wavelength instead of length.
- All features of the physics interfaces in the Heat Transfer Module with a **Constraint Settings** section available have been harmonized and now follow the same naming as interfaces in other modules. Notably, **Classic constraint** has been changed to

Pointwise constraints, and **Discontinuous Galerkin constraints** has been changed to **Nitsche constraints**.

Updated Models in Version 6.2

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

- The *Nonisothermal Turbulent Flow over a Flat Plate* model is now using the SST turbulence model.
- The *Lumped Composite Thermal Barrier* model is now using the new **Thermal Connection** feature.
- The *Parasol*, *Orbit Calculation*, *Spacecraft Thermal Analysis*, and *Orbit Thermal Loads* models are now using the **Define properties for all bands** option.
- The *Condensation Risk in a Wood-Frame Wall* model is now using the **Around location** option for **Ambient Properties**.
- The *Radiation in a Cavity* model is now using the **Thermal Insulation, Edge** feature.
- The *Orbit Calculation*, *Spacecraft Thermal Analysis*, and *Orbit Thermal Loads* models are now using the new **Orbital Thermal Loads** interface with the **Events Timeline** feature.
- All models have been updated with the new default plots, and many have been updated to use the new predefined plots.

Backward Compatibility with Version 6.1

- In COMSOL Multiphysics® version 6.2, the **Isothermal domain** check box 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** check box 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 check box 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 check box 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.

- In version COMSOL Multiphysics® 6.0, when the **From material** option was selected in **Shape Memory Alloy** feature for the heat capacity at constant pressure, C_p , the property was taken from the material **Basic** group. In version 6.1, the material properties C_{p_A} and C_{p_M} are taken from the groups **Austenite Phase** and **Martensite Phase**, respectively. Identity rules enable to ensure backward compatibility in most cases. However, if different values were set for C_p and C_{p_A} or C_{p_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 check box **Apply symmetry to external radiation sources** is selected by default in version 6.1. API script need to be updated by clearing this check box 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 ($Idiff$, $Idiffu_band$, $Idiffd_band$, and $Idiff_band$) are set to 0 when the **Include diffuse irradiance** check box is not selected.
- In the settings of the **Surface-to-Surface Radiation** interface, the **Mesh element self-irradiation** check box 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** check box 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** check box selected, the behavior in version 6.1 is the same as when this check box 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 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.

Backward Compatibility with Version 5.6

- In the **Prescribed Radiosity** feature in the **Surface-to-Surface Radiation** interface, if the graybody model is selected to define the radiative intensity, the emissivity is temperature dependent, and the temperature model input is automatically defined from a multiphysics coupling in an API script, then you must update the script to define T_{ext} . In other cases, T_{ext} is set to the value that was set to the model input in version 5.6 or earlier so that the model behavior is unchanged.
- In **Heat Transfer** interfaces, in the **Deposited Beam Power** boundary condition, for **User defined** beam profiles, the beam intensity is now defined from its value in the beam cross section, which corresponds to the deposited beam power divided by the cosine of the incidence angle. Nevertheless, for **User defined** beam profiles, the models and API scripts created in version 5.6 or earlier behave identically in version 6.0.
- In the **Opaque Surface** feature of the Surface-to-Surface Radiation interface, the coordinate system choice is restricted to the boundary system. When opening a model created in version 5.6 or earlier with another coordinate system selected, this input should be checked for consistency.
- In the Surface-to-Surface Radiation interface, when the **Expression** or **Tolerance** settings had been modified in the **View Factor Update** section of a model created in version 5.6 or earlier, these settings are reported in the new **View Factor** section.
- A modification of the numerical results within the solver tolerance range may be observed when computing surface-to-surface radiation models created in version 5.6 or earlier, due to the introduction in the Surface-to-Surface interface of the **Jacobian contribution** option set by default to **Only local contributions to radiosity**.

When opening a model created in version 5.6 or earlier, a warning is displayed under the Surface-to-Surface Radiation interface. Note that the accuracy level remains the same because the convergence criteria and the solver tolerance are unchanged.

- The Local Thermal Nonequilibrium predefined multiphysics interface and the **Local Thermal Nonequilibrium** multiphysics coupling are now obsolete. When opening a model created in version 5.6 or earlier, this feature is not migrated. It should be manually replaced by the **Porous Medium** feature with the **Porous medium type** set to **Local thermal nonequilibrium**. Support for features marked as obsolete is not ensured in future versions; thus it is strongly recommended to transfer this feature to the new feature.
- In the Moisture Transport interface, the **Hygroscopic Porous Medium** feature not supporting the **Porous Material** functionality is now obsolete. When opening a model created in version 5.6 or earlier, this feature is not migrated. It should be manually replaced by the new **Hygroscopic Porous Medium** feature. Support for features marked as obsolete is not ensured in future versions; thus it is strongly recommended to transfer this feature to the new feature.

MEMS Module

New Functionality in Version 6.2

NEW MULTIPHYSICS INTERFACE: PIEZORESISTIVITY, LAYERED SHELL

The new Piezoresistivity, Layered Shell multiphysics interface is used to model piezoresistivity in multilayer structures. This new multiphysics interface combines an Electric Currents in Layered Shells interface, a Layered Shell interface, and the new **Piezoresistivity, Layered** multiphysics coupling. This interface requires the Composite Materials Module.

NEW MULTIPHYSICS COUPLING: THERMAL EXPANSION, THIN LAYER

For details, see [New Multiphysics Coupling: Thermal Expansion, Thin Layer](#) in the release notes for the Structural Mechanics Module.

NEW FEATURE IN SHELL INTERFACE: PIEZOELECTRIC MATERIAL, LAYERED

Piezoelectric Material, Layered is a new feature in the Shell interface. This addition saves assembly and computation time when solving thin piezoelectric composites with the Shell interface. This new feature requires the Structural Mechanics Module and Composite Materials Module.

NEW FEATURE IN THERMOVISCOUS ACOUSTICS: SLIP WALL FOR NONIDEAL WALL CONDITIONS

The **Slip Wall** boundary condition is used to model the effective nonideal wall conditions that exist in the slip-flow regime, when the Knudsen number is in the range of 0.001 to 0.1. The **Slip Wall** condition is used for systems with very small geometrical dimensions or systems running at very low ambient pressures. This is relevant when modeling, for example, MEMS transducers and other microdevices. To model a slip wall on an interior condition, use the **Interior Slip Wall** condition. This feature requires the Acoustics Module.

STABILIZATION OF UNCONSTRAINED CONTACT PROBLEMS

For details, see [Stabilization of Unconstrained Contact Problems](#) in the release notes for the Structural Mechanics Module.

COMPUTATION OF WARPAGE

For details, see [Computation of Warpage](#) in the release notes for the Structural Mechanics Module.

LIMITED DISPLACEMENT

For details, see [Limited Displacement](#) in the release notes for the Structural Mechanics Module.

SPECIMEN SIZE IN TEST MATERIAL

For details, see [Specimen Size in Test Material](#) in the release notes for the Structural Mechanics Module.

ACTIVATION OF VISCOELASTIC MATERIALS

For details, see [Activation of Viscoelastic Materials](#) in the release notes for the Structural Mechanics Module.

THIN LAYER ENHANCEMENTS

For details, see [Thin Layer Enhancements](#) in the release notes for the Structural Mechanics Module.

THIN ELASTIC LAYER DEPRECATED

For details, see [Thin Elastic Layer Deprecated](#) in the release notes for the Structural Mechanics Module.

ENERGY DISSIPATION COMPUTATION

For details, see [Energy Dissipation Computation](#) in the release notes for the Structural Mechanics Module.

GEOMETRIC NONLINEARITY SETTINGS

For details, see [Geometric Nonlinearity Settings](#) in the release notes for the Structural Mechanics Module.

LINEAR KINEMATICS USING EXTERNAL STRESS-STRAIN RELATION

For details, see [Linear Kinematics Using External Stress–Strain Relation](#) in the release notes for the Structural Mechanics Module.

RESULTANT LOADS IN SYMMETRY PLANES

For details, see [Resultant Loads in Symmetry Planes](#) in the release notes for the Structural Mechanics Module.

APPLICATION POINT FOR RESULTANT IN GLOBAL COORDINATE SYSTEM

For details, see [Application Point for Resultant in Global Coordinate System](#) in the release notes for the Structural Mechanics Module.

COMBINING FRICTION WITH ADHESION AND DECOHESION

For details, see [Combining Friction with Adhesion and Decohesion](#) in the release notes for the Structural Mechanics Module.

CHANGES IN SETTINGS FOR CONTACT AND CONTINUITY

For details, see [Changes in Settings for Contact and Continuity](#) in the release notes for the Structural Mechanics Module.

IMPROVED SYMMETRY CONDITION

For details, see [Improved Symmetry Condition](#) in the release notes for the Structural Mechanics Module.

SYMMETRY PLANE IMPROVED FOR TWISTING MOTION

For details, see [Symmetry Plane Improved for Twisting Motion](#) in the release notes for the Structural Mechanics Module.

LOCAL COORDINATE SYSTEM IN ROLLER CONDITION

For details, see [Local Coordinate System in Roller Condition](#) in the release notes for the Structural Mechanics Module.

MIXED FORMULATION IMPROVEMENTS

For details, see [Mixed Formulation Improvements](#) in the release notes for the Structural Mechanics Module.

LINEARIZATION POINT TEMPERATURE

For details, see [Linearization Point Temperature](#) in the release notes for the Structural Mechanics Module.

THERMOELASTICITY USING FINITE STRAINS

For details, see [Thermoelasticity Using Finite Strains](#) in the release notes for the Structural Mechanics Module.

INERTIAL LOADS ON RIGID FEATURES

For details, see [Inertial Loads on Rigid Features](#) in the release notes for the Structural Mechanics Module.

CELL PERIODICITY ENHANCEMENTS

For details, see [Cell Periodicity Enhancements](#) in the release notes for the Structural Mechanics Module.

ENERGY VARIABLES IN FREQUENCY DOMAIN

For details, see [Energy Variables in Frequency Domain](#) in the release notes for the Structural Mechanics Module.

CHANGED DEFAULT VARIABLES IN RESULT PRESENTATION

For details, see [Changed Default Variables in Result Presentation](#) in the release notes for the Structural Mechanics Module.

GROUPING OF ODE VARIABLES IN SOLVER

For details, see [Grouping of ODE Variables in Solver](#) in the release notes for the Structural Mechanics Module.

RENAMED FEATURES

Several features have been renamed in order to get a more consistent naming structure. These are:

- **Piezoelectric Effect** renamed to **Piezoelectricity**
- **Piezoresistive Effect, Domain Currents** renamed to **Piezoresistivity, Domain Currents**
- **Piezoresistive Effect, Boundary Currents** renamed to **Piezoresistivity, Boundary Currents**
- **Piezoresistive Effect, Shell** renamed to **Piezoresistivity, Shell**

New Models in Version 6.2

ALUMINUM NITRIDE LAMB WAVE RESONATOR

Lamb wave resonators are used as filters in wireless communication systems and as sensors. This tutorial shows how to create a 3D model of an aluminum nitride lamb wave resonator and perform eigenfrequency and frequency-response analyses to characterize the device.

CAPACITIVE MICROMACHINED ULTRASONIC TRANSDUCER WITH LUMPED MODEL

A capacitive micromachined ultrasonic transducer (CMUT) converts ultrasound to electrical signal for imaging applications. This tutorial demonstrates how to derive a

lumped model of a CMUT from its physical model. The lumped model is created using the Lumped Mechanical System interface, with the lumped parameters obtained from the **Parameter Estimation** study available in the Optimization Module.

SOLIDLY MOUNTED RESONATOR WITH UNCERTAINTY QUANTIFICATION

A solidly mounted resonator (SMR) is a piezoelectric resonator made from multiple layers of materials. This tutorial shows how to use the Uncertainty Quantification Module to analyze the impact of manufacturing variation on an SMR's resonance frequency. The tutorial shows how a surrogate model is used in sensitivity analysis, uncertainty propagation, and reliability analysis to obtain statistical data on a physical model.

THIN-FILM BAW RESONATOR WITH EQUIVALENT CIRCUIT

This tutorial demonstrates how an electrical circuit model can be derived from the physical model of a resonator using the Optimization Module. The equivalent circuit model is a modified Butterworth–Van Dyke circuit created using the Electrical Circuit interface, with the lumped parameters obtained from the **Parameter Estimation** study available in the Optimization Module.

Metal Processing Module

New Functionality in Version 6.2

IMPORT OF MATERIAL PROPERTIES

For general steels, magnetic permeability and electrical conductivity can now be imported from JMatPro[®].

CALCULATION OF TRANSFORMATION TEMPERATURES

Certain transformation temperatures are now calculated based on the chemical composition of hypoeutectoid steels. Examples include the martensite start temperature, M_s , as well as A_{e1} and A_{e3} lines in an Fe-C diagram. These temperatures can be used to define lower and upper temperature limits in phase transformation nodes.

CALCULATION OF EQUILIBRIUM PHASE FRACTIONS

Temperature-dependent equilibrium phase fractions of different metallurgical phases can now be calculated based on the chemical composition and different carbon solubility levels in an iron-carbon (Fe-C) diagram. These phase fractions can be used in the definitions of phase transformations for hypoeutectoid steels.

CALCULATION OF HARDNESS

Functionality for calculating Vickers hardness (HV), after quenching, has been added. The hardness calculation is based on the chemical composition of the steel and the rate of cooling.

PERFORMANCE IMPROVEMENTS

The calculations of TRIP strains and plastic recovery have been improved by local integration of the underlying ODEs. Memory consumption is reduced, and a significant speedup typically ensues.

MULTIPHYSICS COUPLING SUBNODE FOR ANNEALING

It is now possible to reset degrees of freedom related to plasticity in the metallurgical phases once a specified temperature is exceeded. This is accomplished using the **Annealing** subnode to the **Phase Transformation Strain** multiphysics coupling.

Microfluidics Module

New Functionality in Version 6.2

- The new Incompressible Potential Flow interface has been added to the **Fluid Flow** branch. This can be used to model solenoidal, irrotational flows, or to obtain initial values for more complex fluid flow simulations.
- The new **Limit small time step effects on stabilization time scale** option (located in the **Consistent Stabilization** section of the **Settings** window for the **Single-Phase Flow** interfaces) can be used to avoid the loss of pressure stabilization during initialization and for small time steps.
- The Free and Porous Media Flow physics interface has been renamed to Free and Porous Media Flow, Brinkman. Models created with earlier software versions containing the Free and Porous Media Flow interface will keep the old name for the interface.
- The new predefined multiphysics interface, Free and Porous Media Flow, Darcy, under the **Porous Media and Subsurface Flow** section in the **Fluid Flow** branch couples a Darcy's Law interface and a Laminar Flow interface using the new **Free and Porous Media Flow Coupling** multiphysics coupling node.
- The new **Porous Interface** feature in the Laminar Flow and Brinkman Equations interfaces, applicable on boundaries with different porosity on each side, accounts for changes in the stress caused by changes in porosity on each side and can include optional pressure losses.
- The new default setting for **Porous treatment of the no slip condition**, in the Brinkman Equations interface or when **Enable porous media domains** is selected in the Laminar Flow interface, is **Porous slip**.
- The mesh velocity from the **Free Surface** and **Fluid-Fluid Interface** features can now be overridden by alternative settings in the **Moving Mesh** node.
- A warning message is displayed in the **Log** when **Use reduced pressure** or **Compensate for hydrostatic pressure/Compensate for hydrostatic pressure approximation** is selected in a fluid flow interface that is part of a multiphase flow multiphysics coupling.

Updated Models in Version 6.2

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 6.2

A **Mixing Plane** feature, applicable to interior boundaries at the intersection of rotating and nonrotating domains, has been added in the Mixer Module. This is an approach, alternative to and often more accurate than the frozen rotor approach, in which averaging in the direction of rotation is performed such that the flow on each side is influenced by mixed flow conditions on the other side. Hence, variations in the flow topology on the other side are accounted for in an averaged sense. The mixing plane approach is an efficient method for modeling sectors of centrifugal pumps, turbines, and compressors.

Multibody Dynamics Module

New Functionality in Version 6.2

MULTIPHYSICS INTERFACE FOR MODELING MAGNETIC–STRUCTURE INTERACTION IN ROTATING MACHINERY

A new multiphysics interface and a multiphysics coupling for modeling electric motors and generators have been added. This multiphysics interface, Magnetic-Rigid Body Interaction in Rotating Machinery combines a Multibody Dynamics interface with a Rotating Machinery, Magnetic interface through the new **Magnetic Forces, Rotating Machinery** multiphysics coupling. This functionality can be used to compute how deformations and stresses originated from air gap forces are distributed in both the stator and rotor of an electric motor. Some of the applications are magnetic bearings, unbalanced rotors, etc.

RADIAL CONSTRAINT IN PLANAR JOINT

A new option to add a radial constraint is available in the **Constraints** subnode under **Planar Joint**. This option can be used to restrict the planar motion of a body within a circle of a given center and radius.

EMBEDDED REINFORCEMENT

Using the **Embedded Reinforcement** multiphysics coupling, it is possible to embed elements from a Wire interface into a flexible solid domain to act as reinforcements. When using a Multibody Dynamics interface with a license for the Structural Mechanics Module, it is also possible to insert edge elements from the Truss or Beam interfaces as well as surface elements from the Membrane interface into a domain.

STABILIZATION OF UNCONSTRAINED CONTACT PROBLEMS

For details, see [Stabilization of Unconstrained Contact Problems](#) in the release notes for the Structural Mechanics Module.

INERTIA RELIEF ANALYSIS

For details, see [Inertia Relief Analysis](#) in the release notes for the Structural Mechanics Module.

LIMITED DISPLACEMENT

For details, see [Limited Displacement](#) in the release notes for the Structural Mechanics Module.

ENERGY DISSIPATION COMPUTATION

For details, see [Energy Dissipation Computation](#) in the release notes for the Structural Mechanics Module.

GEOMETRIC NONLINEARITY SETTINGS

For details, see [Geometric Nonlinearity Settings](#) in the release notes for the Structural Mechanics Module.

CHANGES IN SETTINGS FOR CONTACT AND CONTINUITY

For details, see [Changes in Settings for Contact and Continuity](#) in the release notes for the Structural Mechanics Module.

LOCAL COORDINATE SYSTEM IN ROLLER CONDITION

For details, see [Local Coordinate System in Roller Condition](#) in the release notes for the Structural Mechanics Module.

MIXED FORMULATION IMPROVEMENTS

For details, see [Mixed Formulation Improvements](#) in the release notes for the Structural Mechanics Module.

INERTIAL LOADS ON RIGID FEATURES

For details, see [Inertial Loads on Rigid Features](#) in the release notes for the Structural Mechanics Module.

ELASTIC WIRE SETTINGS RESTRUCTURED

For details, see [Elastic Wire Settings Restructured](#) in the release notes for the Structural Mechanics Module.

CHANGED DEFAULT VARIABLES IN RESULT PRESENTATION

For details, see [Changed Default Variables in Result Presentation](#) in the release notes for the Structural Mechanics Module.

ENERGY VARIABLES IN FREQUENCY DOMAIN

For details, see [Energy Variables in Frequency Domain](#) in the release notes for the Structural Mechanics Module.

GROUPING OF ODE VARIABLES IN SOLVER

For details, see [Grouping of ODE Variables in Solver](#) in the release notes for the Structural Mechanics Module.

New Models in Version 6.2

MAGNETIC-STRUCTURE INTERACTION IN A PERMANENT MAGNET MOTOR

In this example, the coupling between Multibody Dynamics and Rotating Machinery, Magnetic for performing electromagnetic and mechanical analysis is demonstrated. A permanent magnet motor with 10 rotor poles and 12 stator slots with surface mounted magnets is modeled in 2D. To model magnetic-structure coupling integrated with moving mesh, the electromagnetic force is transferred to the rotor, and the rotor motion is transferred to the moving mesh. A time-dependent problem, computing the magnetic flux density and displacement, is solved for three complete electrical periods.

DYNAMICS OF A HOPPING HOOP

In this example, the dynamics of a hopping hoop is simulated. The different types of motion, which a thin rolling ring-point mass system can exhibit, are investigated. It is shown that under certain conditions, the rolling ring can disconnect from the ground and lift into the air.

Nonlinear Structural Materials

Module

New Functionality in Version 6.2

PARAMETER ESTIMATION

New functionality for parameter estimation has been implemented. The addition of a larger family of solvers, including, for example, the Levenberg–Marquardt and IPOPT solvers, can greatly improve performance for some parameter estimation studies for uniaxial, biaxial, or cyclic load cases. Three models have been added to showcase this new functionality.

NEW POLYMER VISCOPLASTICITY NODE

A new **Polymer Viscoplasticity** subnode has been added to the **Hyperelastic Material** in Solid Mechanics.

- Three new viscoplastic models are introduced: the **Bergstrom–Boyce** model, the **Bergstrom–Bischoff** model, and the **Parallel network** model. It is also possible to define **User defined** viscoplastic models.
- The new framework can handle large viscoplastic strains, and it is based on the multiplicative decomposition of deformation gradients.

HYPERELASTICITY IMPROVEMENTS

The following improvements have been made to hyperelastic materials:

- It is now possible to define the strain energy density for the **Arruda–Boyce** model in terms of the inverse Langevin function.
- Plasticity has been added to **Hyperelastic Material**, **Layered** material model in the Shell interface.

ENHANCEMENTS TO PLASTICITY

- The user interface for the **Plasticity** node has been updated. The **Formulation** combo box has been removed from the **Plasticity** and **Porous Plasticity** nodes. The formulation is now controlled in the parent node.
- Kinematic hardening is also available for large strain plasticity.

- Large strain plasticity is available in the Membrane and Shell interfaces.
- A simplified user interface for **Set Variables** had been introduced. For the large strain formulation, you now directly set the components of the plastic strain tensor. In a model created in a previous version where the **Plastic Deformation Gradient Inverse** option was used, the new user input will be shown. The values of old user inputs will, however, not be converted. You must enter the new data.

NONZERO OUT-OF-PLANE INELASTIC SHEAR STRAINS

For 2D problems involving plasticity, creep, or viscoplasticity, where the material is anisotropic, it is possible that the out-of-plane inelastic shear strains are nonzero. This can now be accommodated by selecting the **Include out-of-plane strains** check box in the **Advanced** section of the corresponding features.

NEW MIXED FORMULATION

When using a mixed formulation under plane stress conditions, a new option called **Implicit formulation** is available. In this case, the plane stress equations are used to simplify the problem. The following features have this option:

- **Nonlinear Elastic Material** in the Solid Mechanics interface (available in 2D plane stress)
- **Hyperelastic Material** in the Solid Mechanics interface (available in 2D plane stress)
- **Hyperelastic Material** in the Membrane interface (available for all space dimensions)
- **Hyperelastic Material, Layered** in the Shell interface (available for all space dimensions)

ANAND–NARAYAN MATERIAL MODEL

A new material model, **Anand–Narayan**, has been added under the **Viscoplasticity** node. This model targets the properties of lithium in battery applications.

NEW MULTIPHYSICS COUPLING: PHASE FIELD DAMAGE

This multiphysics coupling connects a Phase Field in Solids interface and a Solid Mechanics interface. The stresses, strains, or strain energy drive the phase field value, and the phase field value provides the degree of damage to the elastic material model. The settings for the **Phase Field Damage** node are similar to those in a **Damage** node in the Solid Mechanics interface.

NEW MULTIPHYSICS INTERFACE: PHASE FIELD DAMAGE

The new Phase Field Damage multiphysics interface combines a Solid Mechanics interface, a Phase Field in Solids interface, and a **Phase Field Damage** multiphysics coupling. It is found under the **Structural Mechanics** branch in the **Add Physics** window.

IMPROVEMENTS TO SHAPE MEMORY ALLOY

There are several enhancements to the **Shape Memory Alloy** material models:

- A new predefined plot shows the stress-temperature phase diagram. The plot shows the austenite to martensite transition, and it is useful for understanding the behavior of the SMA with respect to changes in material parameters.
- The penalty method for enforcing the upper bound for the transformation strain has been improved. There is a new setting in the **Advanced** section for both the Lagoudas and the Souza–Auricchio models for better control of the penalty constraint in the pseudoplastic domain.
- The Prager–Lode yield surface is now available for the Souza–Auricchio model. The new formulation makes possible it to model a situation where the martensite start stress differs between tension and compression.
- Large strain plasticity is available for the Souza–Auricchio model.
- The material parameters for the phase transformation can now be entered as start and finish stresses or start and finish temperatures.

IMPROVEMENTS TO FIBER

- Compressible fibers are available in the Holzapfel–Gasser–Ogden hyperelastic material model.
- **Thermal Expansion** is available as a subnode under **Fiber** for hyperelastic materials.
- The **Uniaxial data** material model for nonlinear stress–strain relations is available for **Fiber** under **Linear Elastic Material** and **Nonlinear Elastic Material**.

EXTENDED BUILT-IN MATERIAL FOR SOLDER

The built-in material **Solder, 60Sn-40Pb** has been augmented to include Anand viscoplasticity data.

LARGE SWELLING IN POLYMER HYDROGELS

In this example model, it is demonstrated how to implement a coupled nonlinear field theory of large deformation and solvent diffusion for polymer hydrogels. Polymer hydrogels consist of a cross-linked network of long-chained polymers that has imbibed a large number of solvent molecules. Such materials are used in many biomedical applications, such as targeted drug delivery, tissue engineering, and stimuli-sensitive actuators.

UNIAXIAL LOADING OF A SHAPE MEMORY ALLOY USING SOUZA-AURICCHIO MODEL

The model uses the Souza–Auricchio model and shows the thermomechanical properties of a NiTi alloy sample subjected to uniaxial tension-compression loading.

The first parametric study displays the pseudoelasticity effect at different temperatures. In the second study, a partial loading–unloading cycle is added. The third study shows the shape memory effect in a low-temperature loading cycle followed by a temperature increase.

CHLOROPRENE RUBBER COMPRESSION TEST

In this example, the Bergstrom–Boyce material model is used to capture the nonequilibrium behavior of carbon-black-filled chloroprene rubber under a strain history that alternates compression with relaxation. Results are verified against experimental and numerical results taken from literature.

SMALL PUNCH TEST FOR UHMWPE POLYETHYLENE

This example demonstrates the use of the Bergstrom–Bischoff material to model ultra-high-molecular-weight polyethylene (UHMWPE), a material commonly employed in knee and hip joint replacements. The small punch test is designed to assess mechanical properties using very small samples.

BUCKLING OF HDPE LINERS

In this example, the Bergstrom–Bischoff viscoplastic material is used to model the temperature and strain dependent behavior of high-density polyethylene (HDPE) liners.

HDPE is used to make liners for damaged pipes in oil and gas applications and to manufacture hydrogen storage vessels for fuel cells. In these application areas, the

material is prone to collapse due to gas penetration between the liner and the hosting structure.

TIRE INFLATION

This example showcases how to use fibers in thin layers to model thin anisotropic composites embedded in a solid, without explicitly drawing either the layer of material or the reinforcing fibers. In particular, this is used to model steel cords in tire belts that are used to provide structural support to the tire below the tread. A curvilinear coordinate system is used to define the anisotropic material properties of the carcass ply.

The model also makes use of the “twist” degree of freedom in 2D axisymmetry to capture the azimuthal displacement induced by the orientation of the cords during inflation of the tire.

UNIAXIAL STRETCHING OF A RECTANGULAR SHEET

This example demonstrates the wrinkling of a thin rectangular sheet stretched uniaxially. First, a static analysis is performed to determine the region of compressive stresses without wrinkling. Next, a prestressed buckling analysis is carried out to find out the potential linearized buckling modes. Finally, a nonlinear postbuckling analysis is carried out to investigate the evolution of wrinkles.

PARAMETER ESTIMATION OF HYPERELASTIC MATERIALS

This example demonstrates how to set up a parameter estimation study for fitting the parameters of an Ogden hyperelastic model from experimental data. The process considers multiple load cases under large deformations, which is often necessary in order to obtain constitutive models with good predictive capabilities.

PARAMETER ESTIMATION OF ELASTOPLASTIC MATERIALS

This example demonstrates how to estimate the material parameters of an elastoplastic material model given cyclic shear data. The elastoplastic material model includes mixed isotropic and kinematic hardening parameters.

PARAMETER ESTIMATION OF VISCOPLASTIC POLYMERS

This example demonstrates how to estimate the parameters of a viscoplastic material suitable for the modeling of rubber-like materials. The data used for parameter estimation of a Bergstrom–Boyce model consists of cyclic uniaxial tension and compression tests at two different strain rates. After estimating the primary parameters,

the formulation is extended to include temperature dependence using additional data from tensile experiments at different temperatures.

Optimization Module

New Functionality in Version 6.2

IMPROVED GRADIENT-BASED OPTIMIZATION

Gradient-based optimization for eigenvalue problems has been extended with the **Stationary Then Eigenfrequency** study step. This study step makes it possible to use the Shape Optimization and Topology Optimization interfaces when optimizing eigenvalue problems. Both interfaces have also been extended with mirror and sector symmetry features to treat such eigenvalue problems, which often call for design symmetry.

This functionality is demonstrated in three new or updated Application Library models: Maximizing the Eigenfrequency of a Beam (*beam_eigenfrequency_topology_optimization*), Maximizing the Eigenfrequency of a Shell (*shell_eigenfrequency_shape_optimization*), and Bracket — Eigenfrequency Shape Optimization (*bracket_eigenfrequency_shape_optimization*).

EXTENDED PARAMETER ESTIMATION

Parameter estimation has been extended with support for the covariance matrix, which gives more information about the uncertainty of the estimated parameters than the confidence intervals. This functionality is available with the **Levenberg–Marquardt** solver, which now supports bounds for the control parameters. Both the **Global Least-Squares** feature and the **Parameter Estimation** study step now include support for specifying the variance of individual measurements by providing a variance column in the **Settings** window.

ADDITIONAL IMPROVEMENTS

- Shape optimization was previously limited to symmetry box bounds for the maximum displacement, but now it is also possible to have a circular or Euclidean bound or to specify asymmetric bounds on individual components, optionally in a custom coordinate system.
- The **Control Variable Field** feature has been extended with a discretization that is constant over geometric entities, and it is now possible to group these entities when vertices are shared. The user interface for the boundary conditions of the **Control Function** feature has been changed to increase the number of options and improve

the consistency when changing between polynomial and Helmholtz (maximum slope) regularization.

- Material sweeps over one of the optimization study steps is now supported. Derivative-free optimization also supports optimization over the material sweep, similar to the parametric sweep.
- The ribbon layout has been split with new dedicated **Shape Optimization**, **Topology Optimization**, and **Parameter Estimation** ribbons, while the **Control Variable Field** and **Control Function** features remain in the **Definitions** ribbon.
- Topology optimization with milling constraints now supports setting a diffusion length if the discretization is linear, which can avoid numerical artifacts for certain angles between the mesh and the milling directions.
- The default solver suggestion has been improved to put optimization first in segregated solvers, which simplifies the setup of the solver configuration for 3D models. Also, the adjoint dependent variables can be initialized with the forward variables, which can lead to dramatic speedup for certain self-adjoint problems that use iterative solvers, as often seen for large-scale topology optimization of structural mechanics.
- An error message now appears for the case where the **Times to store** list is set to **Steps taken by solver** while performing transient optimization of least-squares objectives. This combination was never fully supported.

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 in version 6.1, the continuity of the slope is preserved.

Particle Tracing Module

New and Improved Functionality in Version 6.2

NEW OPTION FOR FRICTION FORCE CALCULATION

It is now possible to accurately calculate the friction force in the low electric field limit. The electric field limit is defined as the ratio of the magnitude of the electric field to the background gas number. When the electric field limit is low, the kinetic energy of charged particles is mostly thermal, that is, the thermal energy of charged particles is much greater than the energy acquired from the electric field between collisions. The **Friction Force** feature in the Charged Particle interface has a new option for calculating the friction force in the low electric field limit using the Mason–Schamp equation.

Plasma Module

New Functionality in Version 6.2

HEAT EQUATION IN GLOBAL MODELS

It is now possible to solve the heavy species energy conservation equation in global models. This new functionality can be enabled in the settings of the Plasma interface, in the **Heavy Species Energy Balance** section. In the Plasma Model settings, there are options to set the initial gas temperature, heat transfer to the surfaces, and gas temperature of the feed.

New model that solves the heavy species energy conservation equation:

Hydrogen Global Model Coupled with the Two-Term Boltzmann Equation

REACTION GROUP FEATURE

The **Reaction Group** feature can be used to add heavy species reactions in a table. 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.

New models that showcase the **Reaction Group** feature:

- *Hydrogen Global Model Coupled with the Two-Term Boltzmann Equation*
- *Microwave Cavity Plasma*

FREQUENCY-STATIONARY STUDIES FOR MICROWAVE PLASMAS

It is now possible to solve microwave plasma models using a **Frequency-Stationary** study when the system is excited using input power instead of deposited power.

The new model *Microwave Cavity Plasma Reactor* uses a **Frequency-Stationary** study step to parameterize the input power.

New and Updated Applications in Version 6.2

MICROWAVE CAVITY PLASMA REACTOR

This tutorial shows how to prepare a model of a hydrogen plasma sustained in a microwave cavity at moderate pressures. The model solves the plasma transport equations fully coupled with Maxwell's equations, fluid flow, and heat transfer. A microwave cylindrical chamber contains a bell jar where a hydrogen plasma is created.

The reactor is carefully designed so that the electric field has its maximum intensity above a substrate and much lower intensity at the bell jar boundary.

HYDROGEN GLOBAL MODEL COUPLED WITH THE TWO-TERM BOLTZMANN EQUATION

This tutorial studies the chemistry of a hydrogen plasma at moderate pressures. The main goals are to show how to prepare a global model to study a plasma chemistry and the influence of the electron energy distribution function. The plasma global model, which includes the heavy species heat equation, is solved self-consistently with the Boltzmann equation in the two-term approximation.

HYDROGEN BOLTZMANN ANALYSIS

This model solves the Boltzmann equation in the two-term approximation for a background of molecular and atomic hydrogen. Electron mobility and source terms are computed by suitable integration of the electron energy distribution function over electron impact cross sections.

Backward Compatibility with Version 6.0 and Earlier

There is a new version of the Plasma, Time Periodic interface. When opening models created with version 6.0 and earlier, a warning sign appears saying, “This physics interface is obsolete and will be removed in a future version. A new version of the physics interface is available from the Add Physics window.” These models can still be changed and will run as before, but it is encouraged to recreate the models using COMSOL Multiphysics® version 6.2.

Porous Media Flow Module

New Functionality in Version 6.2

NEW PHYSICS INTERFACE: MOISTURE TRANSPORT IN SOLIDS

A new physics interface, Moisture Transport in Solids, has been added. The new interface is useful, for example, in conjunction with deformations in solids. The new interface accounts for large strains.

NEW MULTIPHYSICS INTERFACE: UNSATURATED POROELASTICITY

The new multiphysics interface Unsaturated Poroelasticity combines a Solid Mechanics interface, a Moisture Transport in Solids interface, and an **Unsaturated Poroelasticity** multiphysics coupling. It is found under the **Structural Mechanics** branch in the **Add Physics** window.

This multiphysics interface requires a license for the Structural Mechanics Module.

NEW MULTIPHYSICS COUPLING: UNSATURATED POROELASTICITY

This multiphysics coupling connects a Moisture Transport in Solids interface and a Solid Mechanics interface. The moisture pressure acts as a load in the pores, and structural deformations modify the storage coefficients and porosity. The settings for the **Unsaturated Poroelasticity** node are similar to those in the **Poroelasticity** node.

BIPHASIC POROELASTICITY

A new **Biphasic** poroelasticity model has been added to the **Poroelasticity** multiphysics coupling node. The new option is suitable for modeling the coupling between fluid flow and deformation in hydrogels and soft biological tissues for which both solid and fluid constituents can be assumed materially incompressible.

The default **Elastic Predeformation** node that was added with the **Poroelasticity, Large Deformation, Solid** interface has been removed as the default feature.

NEW FEATURE: DUAL PERMEABILITY

A new **Dual Permeability** node has been added to the Darcy's Law and Richards' Equation interfaces. This feature makes it possible to model saturated fluid flow within a dual permeability approach, where two distinct and interconnected porous systems are present. The dual permeability approach can be applied to situations where both porous systems are fully saturated and have varying permeabilities.

NEW FEATURE: UNSATURATED DUAL PERMEABILITY

A new **Unsaturated Dual Permeability** node has been added to the Darcy's Law and Richards' Equation interfaces. The feature allows for modeling unsaturated fluid flow within two distinct and interconnected porous systems.

Fluid flow through macropores and micropores is described by the Richards' Equation interface and is solved separately for both systems. This accounts for the heterogeneous nature of unsaturated porous media flow, where the presence of two distinct and interconnected systems significantly affects fluid flow.

NEW FEATURE: DUAL POROSITY

A new **Dual Porosity** node has been added to the Darcy's Law and Richards' Equation interfaces. This feature allows for the modeling of saturated fluid flow where two distinct and interconnected porous systems are present.

One system possesses significantly larger pores (macropores), in which the fluid flow takes place. This system is interconnected with a second system of smaller pores (micropores).

NEW PHYSICS INTERFACE: PHASE TRANSPORT IN FREE AND POROUS MEDIA FLOW

A new physics interface, Phase Transport in Free and Porous Media Flow, has been added. The new interface allows for easier modeling of multiphase transport in coupled free and porous media flow.

NEW MULTIPHYSICS INTERFACE: FREE AND POROUS MEDIA FLOW, DARCY

The new Free and Porous Media Flow, Darcy multiphysics interface combines a Darcy's Law interface, a Laminar Flow interface, and the new **Free and Porous Media Flow Coupling** node. It is found under the **Porous Media and Subsurface Flow** branch under **Fluid Flow** in the **Add Physics** window.

RENAMED INTERFACE

The Free and Porous Media Flow physics interface has been renamed to **Free and Porous Media Flow, Brinkman**. Models created with earlier versions containing the **Free and Porous Media Flow** interface will keep the old name.

NEW DENSITY OPTIONS

The options for the **Fluid type** in the **Fluid** subnode to **Porous Medium** and **Unsaturated Porous Medium** in the Darcy's Law and Richards' Equation interfaces have been

extended to better describe the equation of state used for the fluid. Two new options, **Incompressible** and **Compressible, linearized** have been added, and the previous default option **Gas/Liquid** has been renamed to **Compressible**.

When opening an old model, these new options will be used. When building a model using the API, the new default option will also be used. This is true even if, for example, a Java® file created from an older version is used when the **Porous Medium** or **Unsaturated Porous Medium** nodes uses the **Linearized** or **Poroelastic** storage model and the default gas/liquid `fluidType` is used. If you want to achieve full backward compatibility using the API, and you have code such as

```
model.component("comp1").physics("d1").feature("porous1").set("storageModelType", "poroelastic");

model.component("comp1").physics("d1").feature("porous1").feature("fluid1").set("rho_mat", "userdef");

model.component("comp1").physics("d1").feature("porous1").feature("fluid1").set("rho", "1000[kg/m^3]");
```

it will need to be changed to

```
model.component("comp1").physics("d1").feature("porous1").set("storageModelType", "poroelastic");

model.component("comp1").physics("d1").feature("porous1").feature("fluid1").set("fluidType", "compressibleLinearized");

model.component("comp1").physics("d1").feature("porous1").feature("fluid1").set("rhoref_mat", "userdef");

model.component("comp1").physics("d1").feature("porous1").feature("fluid1").set("rhoref", "1000[kg/m^3]");
```

ADDITIONAL VELOCITY CONTRIBUTIONS

A new **Contributing Velocity** subnode is available under **Porous Medium**, **Unsaturated Porous Medium**, and **Dual Porosity Medium** in the Darcy's Law and Richards' Equation interfaces. This new subnode makes it possible to add external velocity contributions to the Darcy velocity, which can, for example, arise due to solute concentration gradients in liquid flow or Knudsen diffusion in gas flow applications.

REFACTORIZING OF VELOCITY VARIABLES

For results evaluation, the velocity variables have been updated such that `d1.u` now includes possible **Contributing Velocity** fields. The Darcian/non-Darcian contribution is available in the new variable `d1.ud`. The latter has been corrected to include the correct frame transformation, which has an effect during results evaluation for large

deformations. For consistency, the corresponding material frame Darcy velocity variable has been renamed to `d1.Ud`.

REFACTORING OF GRAVITY

The user input for **Acceleration** of gravity in the **Gravity Effects** section in the Darcy's Law, Richards' Equation, Fracture Flow, and Layered Darcy's Law interfaces has been removed. Instead, the variable `d1.g` for the acceleration of gravity is by default set to `g_const` if gravity effects are excluded in the physics interface. If gravity effects are included, the acceleration of gravity can be edited in the **Gravity** feature.

BOUSSINESQ APPROXIMATION

Boussinesq approximation for the **Nonisothermal Flow** multiphysics couplings is now available. The approximation assumes that density variations are only contributing to buoyancy effects.

CONTEXT MENUS FOR BOUNDARY CONDITIONS

Boundary conditions have been organized into context menus to enhance the user interface. The **Interior Surfaces** menu groups **Flux Discontinuity**, **Interior Wall**, and **Thin Barrier** under a single context menu.

New and Updated Models in Version 6.2

PAPERBOARD ROLL

This example demonstrates how to model moisture transfer and induced swelling in a paperboard roll subjected to varying ambient conditions. The material properties of the paperboard are anisotropic.

Polymer Flow Module

New Functionality in Version 6.2

- Two new viscoelastic models are introduced in the Viscoelastic Flow Interface: the finitely extensible nonlinear elastic model with the Chilcott and Rallison closure (**FENE-CR**) and the Rouse linear entangled polymers model (**Rolie-Poly**).
- Two different formulations of the rheological constitutive equation can now be selected in the Viscoelastic Flow interface. In addition to the stress formulation, where the dependent variable is the extra stress, the conformation formulation, where the conformation tensor is an intermediate structural-dependent variable, is introduced.
- The new Incompressible Potential Flow interface has been added in the **Fluid Flow** branch. This can be used to model solenoidal, irrotational flows, or to obtain initial values for more complex fluid flow simulations.
- The new **Limit small time steps effect on stabilization time scale** option can be used to avoid the loss of pressure stabilization during initialization and for small time steps. This option can be found in the **Consistent Stabilization** section of the **Settings** window for the Laminar Flow and Brinkman Equations interfaces.
- The Free and Porous Media Flow physics interface has been renamed to Free and Porous Media Flow, Brinkman. Models created with earlier versions containing the Free and Porous Media Flow interface will keep the old name for the interface.
- The new **Porous Interface** feature in the Laminar Flow and Brinkman Equations interfaces, applicable on boundaries with different porosity on each side, accounts for changes in the stress caused by changes in porosity on each side and can include optional pressure losses.
- The new default setting for **Porous treatment of the no slip condition**, in the Brinkman Equations interface or when **Enable porous media domains** is selected in the Laminar Flow interface, is **Porous slip**.
- The mesh velocity from the **Free Surface** and **Fluid-Fluid Interface** features can now be overridden by alternative settings in the **Moving Mesh** node.
- A warning message is displayed in the **Log** when **Use reduced pressure** or **Compensate for hydrostatic pressure/Compensate for hydrostatic pressure approximation** is selected in a fluid flow interface that is part of a multiphase flow multiphysics coupling.

Backward Compatibility with Version 6.0

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.

Ray Optics Module

New and Improved Functionality in Version 6.2

RELEASE GAUSSIAN BEAM WITH ELLIPTICAL CROSS SECTION

The **Gaussian Beam** feature now has options for defining the cross section of released beam of rays. You can choose between a circular and elliptical cross section. The **Elliptical** option can be used if the desired beam is not perfectly axisymmetric. For this option, specify the aspect ratio of the cross section of the beam, which is the ratio of the major axis of the ellipse to the minor axis. The major axis is always equal to the diameter of a corresponding circular cross section. In addition, the orientation of the major axis in the plane perpendicular to the beam axis can be specified as required.

OPTICAL MATERIAL LIBRARY UPDATES

In the Optical material library, 93 new glasses from CDGM Glass Co. Ltd have been added. These added glasses have all the material properties required to perform structural-thermal-optical performance (STOP) analyses.

SPECIFY DIFFERENT VALUES FOR S AND P POLARIZED RAYS

The absorption and reflection coefficients can have different values for s and p polarized rays when either intensity or power or both are computed for released rays. Now, a dedicated **Specify different values of s- and p-polarization** check box is added, which, when selected, makes it possible to specify different values of absorption coefficients or reflection coefficients for s- and p-polarized rays. This check box has been added to the following features:

- **Wall**, when **Wall Condition** is set to either **Specular reflection**, **Diffuse scattering**, **Isotropic scattering**, **Mixed diffuse and specular reflection**, or **General reflection**
- **Mirror**
- **Illuminated Surface**, when **Rays to release** is set to **Reflected** and **Reflection coefficient calculation** is set to **Using absorption coefficient** in the **Ray Direction Vector** section

MICHELSON INTERFEROMETER

This tutorial model demonstrates how to plot the interference pattern from the combination of two rays with slightly different optical path lengths. A simple Michelson interferometer is used to achieve a change of optical path length by slightly moving one of the mirrors.

RF Module

New Functionality in Version 6.2

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

A new feature, **Cable Shield**, has been integrated into the Electromagnetic Waves, Frequency Domain interface. This feature allows for the efficient simulation of intricate shields, such as braided or perforated types, by utilizing a streamlined boundary condition that reduces computational demands.

NEW ELECTRIC DISPLACEMENT FIELD MODE

The **Wideband Debye** electric field displacement model is now available in the **Wave Equation, Electric** feature. This model can be used to accurately describe losses and dispersive effects in PCB substrates in the Electromagnetic Waves, Frequency Domain and Electromagnetic Waves, Boundary Elements interfaces.

AUTOMATIC PATH PARAMETERIZATION FOR LIGHTNING AND ELECTROSTATIC DISCHARGE (ESD) APPLICATIONS

The **Edge Current** feature in the Electromagnetic Waves, Transient interface can adaptively determine a parameterized path based on the selected geometry's unique shape. This enhancement simplifies the modeling process for lightning and ESD applications.

REFERENCE-EDGE CONTROLLED LUMPED PORT DESIGN

The **Lumped Port** feature now includes the **Reference-edge controlled** type. This allows users to designate extra edge selections, ensuring the proper direction of voltage flow between two conductive boundaries where the lumped port is positioned.

ENHANCED MATERIAL OPTIONS FOR MMWAVE APPLICATIONS

The RF Material Library has been expanded to include Alumina Ribbon Ceramic from Corning Incorporated; WavePro® WP025LDf, WavePro® WP025, WavePro® WP030, WavePro® WP050, WavePro® WP108, WavePro® WP120, and WavePro® WP150 from Garlock, Radix™ Printable Dielectric by Rogers Corporation; and Zetamix ε Filaments, White Zirconia Zetamix Filament, and Alumina Zetamix Filament, all sourced from Zetamix.

ASSESSMENT OF SPECIFIC ABSORPTION RATE (SAR) FOR 1G AND 10G MASS

In the Electromagnetic Waves, Frequency Domain interface, when using the **Specific Absorption Rate** feature, postcomputation provides the predefined variables SAR1g and SAR10g, representing SAR values for 1 g and 10 g of mass, respectively. These values are commonly used in industrial applications to measure radiation exposure levels.

ABCD PARAMETERS FOR TWO-PORT DEVICES

Analysis of a two-port device includes the option to compute two-port ABCD parameters. Using a port sweep, a comprehensive set of ABCD parameters can be derived from a full 2 x 2 S-parameter matrix. Moreover, even with just a single port excitation, the system can generate ABCD parameters by presuming the device operates as a reciprocal network.

NEW BOUNDARY CONDITIONS ADDED TO THE ELECTROMAGNETIC WAVES, BOUNDARY ELEMENTS INTERFACE

Both the **Impedance Boundary Condition** and the **Layered Impedance Boundary Condition** have been added to the **Electromagnetic Waves, Boundary Elements** interface. The **Impedance Boundary Condition** typically represents a metallic bulk material outside the simulation domain. With the **Layered Impedance Boundary Condition**, additional layers of different materials and thicknesses can be added between the interior simulation domain and the exterior domain.

ELECTRIC DISPLACEMENT FIELD MODELS ADDED TO THE ELECTROMAGNETIC WAVES, BOUNDARY ELEMENTS INTERFACE

Now, for the **Wave Equation, Electric** feature in the **Electromagnetic Waves, Boundary Elements** interface it is possible to choose between several different material models, like **Relative permittivity**, **Refractive index**, **Dielectric loss**, and so on, when specifying the dielectric properties in the domain solved for. The options are also available for the new **Impedance Boundary Condition** and **Layered Impedance Boundary Condition** nodes.

SYMMETRY FUNCTIONALITY FOR THE ELECTROMAGNETIC WAVES, BOUNDARY ELEMENTS INTERFACES

In the settings for the **Electromagnetic Waves, Boundary Elements** interface it is now possible to select symmetry planes to reduce the computational domain. The symmetry settings also control the far-field calculations and the physics-controlled meshing.

PHYSICS-CONTROLLED MESHING FOR THE TIME-DOMAIN INTERFACES

The time-domain physics interfaces, **Electromagnetic Waves, Transient** and **Electromagnetic Waves, Time Explicit**, now provide physics-controlled mesh suggestions based on the frequency/wavelength content in the simulation.

CYCLIC SYMMETRY OPTION FOR PERIODIC CONDITIONS

Cyclic symmetry has been added as an additional option for the **Periodic Condition**. This allows you to perform simulations of a smaller structure, if the problem is cyclically symmetric.

HIGHER-ORDER DISCRETIZATION ELEMENTS

Up to seventh-order curl elements can now be used with the **Electromagnetic Waves, Frequency Domain** and **Electromagnetic Waves, Transient** interfaces.

API COMMAND FOR THE ADD DIFFRACTION ORDER BUTTON ACTION

Now, it is possible to perform the same action as when clicking the **Add Diffraction Orders** button for periodic ports, using the COMSOL Java API. Use the Java statement

```
model.component("comp1").physics("emw").feature("port1")  
.runCommand("addDiffractionOrders");
```

where “comp1”, “emw”, and “port1” are the tags for the model component, the physics interface, and the excited port, respectively, and `model` is a model object.

ELECTRICAL CONDUCTIVITY CAN BE SPECIFIED WITH THE DRUDE-LORENTZ AND DEBYE DISPERSION MODELS

Now, when using the **Drude-Lorentz dispersion model** and the **Debye dispersion model Electric displacement field model** options, the **Electrical conductivity** can now be specified in a separate input field. This functionality is applicable to the **Wave Equation, Electric, Impedance Boundary Condition, Transition Boundary Condition, Layered Impedance Boundary Condition**, and **Layered Transition Boundary Condition** nodes for the **Electromagnetic Waves, Frequency Domain** and **Electromagnetic Waves, Boundary Elements** interfaces.

PHYSICS-CONTROLLED MESHING HANDLES MORE FUNCTIONS AND VARIABLES

The algorithm for building physics-controlled meshes has been improved to better handle material property expressions, including functions and material property variables. For instance, a material property can be defined as $f(\text{freq})$, where f is the function name and freq is the model input for the frequency. However, it is not

possible to specify functions that depend on spatial coordinates, like $g(x)$, where x is a spatial coordinate. Material property variables, like `mat1.rfi.n11`, for example, can now be used as part of other user-defined material property expressions. Here, `mat1` is the material tag, `rfi` is a material property group identifier, and `n11` is a tensor element (in this case, a refractive index element).

HEXAGONAL UNIFORM ARRAY FACTOR

The hexagonal uniform array factor quickly estimates the far-field pattern of antenna arrays on a triangular grid. The hexagonal antenna arrays provide lower sidelobes, more robust performance with better resolution, lower spatial noise, and wider coverage.

INSTANTANEOUS NORM VARIABLES FOR VECTOR QUANTITIES

New variables have been introduced of the form

`phys.normXi = sqrt(real(Xx)^2+real(Xy)^2+real(Xz)^2)`, where `phys` is the physics tag, such as `emw`, and `X` is a physical quantity like electric field (E), magnetic field (H), and so on. These variables are especially useful when visualizing time-harmonic vector waves.

USER-DEFINED SURFACE IMPEDANCE OPTION

For the **Impedance Boundary Condition** and **Layered Impedance Boundary Condition**, the surface impedance can now be directly entered, instead of being indirectly calculated from the given material parameters.

ONLY ISOTROPIC MATERIAL PROPERTIES ALLOWED FOR SOME BOUNDARY CONDITIONS

The UI has been updated to reflect that the **Transition Boundary Condition**, **Impedance Boundary Condition**, **Layered Transition Boundary Condition**, and **Layered Impedance Boundary Condition** nodes only support the entry of isotropic material properties. Thereby, the UI better reflects the assumptions used in the implementation of these boundary conditions.

UPDATED ALGORITHM FOR PHYSICS-CONTROLLED MESHING FOR MODE ANALYSIS

The algorithm for physics-controlled meshing for mode analysis has been slightly changed. A slightly different mesh can be obtained when running an old model in COMSOL Multiphysics® version 6.2.

EXPORT EXPRESSIONS SUBNODE ADDED TO DEFAULT RADIATION PATTERN PLOTS

To simplify the export of data from **Radiation Pattern** (far-field) plots, when default **Radiation Pattern** plots are generated, there is now an **Export Expressions** subnode. Use the **Export Expressions** subnode to add expressions for additional quantities to include in plot data export for the **Radiation Pattern** plot.

REUSE SPARSITY PATTERN ENFORCED WHEN PERFORMING FREQUENCY AND WAVELENGTH SWEEPS

To improve performance, the automatic solver suggestions now checks the **Reused sparsity pattern** check box in the **Assembly Settings** section of the **Advanced** solver feature node, when performing frequency or wavelength domain sweeps.

NEW PHASE SECTION IN PLOT GROUP SETTINGS

When creating plots for frequency-domain simulations, it is now possible to specify the **Solution at angle (phase)** directly in the plot group settings. Previously, this setting could only be set on the datasets. Now, the same dataset can be used for different plot groups, displaying results for different phase settings.

REVISED SOLVER CONFIGURATIONS FOR THE ELECTROMAGNETIC WAVES, BOUNDARY ELEMENTS INTERFACE

The **Direct Preconditioner** now uses the **MUMPS** solver and adopts the **Block low rank factorization** with a default tolerance of $1e-6$ for the **Block low rank factorization**. These enhancements are especially advantageous when processing models of a substantial size.

New and Updated Models in Version 6.2

LIGHTNING SURGE ON A POWER TRANSMISSION TOWER

This model aims to address these concerns by simulating the effects of lightning surges on high-voltage transmission line towers. By examining the lightning-induced voltage on three-phase conductors, it allows for the assessment of potential overvoltages and helps in designing effective protection measures. This model specifically focuses on lightning carrying a current of 10 kA that strikes one of the tower's shielded wires. The induced voltage on the three-phase conductors is computed through rigorous simulation techniques.

LIGHTNING SURGE ANALYSIS OF AN OFFSHORE WIND FARM

This model simulates the effects of a lightning strike on one turbine and its resulting induced electric fields on adjacent turbines. The simulation aids in understanding the potential impacts of lightning on interconnected turbines within an offshore wind farm. Through analysis of these fields, engineers and researchers can gauge potential overvoltages and transient effects, thereby informing design for protection measures, wind turbine layout optimization, and continuous farm operation.

MODELING OF DIPOLE ANTENNA ARRAY USING THE BOUNDARY ELEMENT METHOD

This example of a dipole antenna array demonstrates a cost-effective analysis using the boundary element method (BEM). When dealing with a large array made of metallic radiators, the finite element method (FEM) would necessitate greater computational resources. The simulation results depict the radiation patterns of a 12-by-1 array, consisting of metallic half-wave dipole antennas resonant at 1 GHz.

RCS OF A METALLIC SPHERE USING THE BOUNDARY ELEMENT METHOD

This model illustrates the process of evaluating the radar cross section (RCS) of a metallic sphere through the use of the boundary element method (BEM). By taking advantage of a vertical symmetry plane that is parallel to the polarization of an incident background field, the model reduces computational expenses. The computed RCS values are compared with analytical values within the Mie RCS region.

SAR OF A HUMAN HEAD NEXT TO A WI-FI ANTENNA

This example has been revised to demonstrate the application of the SAR1g variable within a simulated model of a human head.

LIGHTNING-INDUCED VOLTAGE OF A WIRE IN AN AIRPLANE

This example has been revised to illustrate the use of an automatically parameterized path representing a lightning channel.

Rotordynamics Module

New Functionality in Version 6.2

HYDRODYNAMIC BEARING IMPROVEMENTS

The functionality in the Hydrodynamic Bearing interface has been subject to the following changes:

- A new predefined bearing type for modeling step thrust bearings has been implemented in the **Hydrodynamic Thrust Bearing** feature.
- For a thrust bearing, the equilibrium position can now be determined based on a force balance between an external load and the resulting force from the pressure field. This functionality is available by selecting **Load** under **Collar Properties** in the settings for the **Hydrodynamic Thrust Bearing** node.
- It is now possible to specify a **User defined** bearing center in all the different bearing nodes in the Hydrodynamic Bearing interface.

NEW PREDEFINED PLOT

In the Hydrodynamic Bearing interface, a set of new unwrapped plots has been added to the list of predefined plot. These new plots are present when the **Hydrodynamic Journal Bearing** node is used. The functionality unwraps the journal bearing surface for easier interpretation of pressure and velocity plots.

CHANGED INPUT IN ROTOR CROSS SECTION

In the Beam Rotor interface, the settings in the **Rotor Cross Section** node have been simplified by removing one level of settings.

GEOMETRIC NONLINEARITY SETTINGS

For details, see [Geometric Nonlinearity Settings](#) in the release notes for the Structural Mechanics Module.

INERTIAL LOADS ON RIGID FEATURES

For details, see [Inertial Loads on Rigid Features](#) in the release notes for the Structural Mechanics Module.

ENERGY VARIABLES IN FREQUENCY DOMAIN

For details, see [Energy Variables in Frequency Domain](#) in the release notes for the Structural Mechanics Module.

GROUPING OF ODE VARIABLES IN SOLVER

For details, see [Grouping of ODE Variables in Solver](#) in the release notes for the Structural Mechanics Module.

Semiconductor Module

New Functionality in Version 6.2

FINITE ELEMENT FORMULATIONS

Significant enhancements have been made to the stability, accuracy, and efficiency of finite element formulations, including logarithmic, quasi-Fermi level, and density-gradient formulations. These improvements encompass various aspects, such as improvements in weak form equations, constraint settings, and default solver settings. As a result, the majority of library models can now be solved more efficiently using finite element formulations. For instance, solving a 3D bipolar transistor model with the **Finite volume** formulation would typically take a day. With the improved **Finite element, log formulation**, this model can now be solved on a standard PC in 15 minutes. Several factors contribute to this achievement:

- The finite-element, log formulation with linear shape function significantly reduces the degree of freedom required for tetrahedral meshes in 3D.
- Unlike the finite volume formulation, the 3D finite element formulation does not necessitate a swept mesh.
- In general, continuation studies with the finite element formulation are more efficient compared to the finite volume formulation.
- The finite element formulation demands less stringent initial conditions for current terminals (metal contact).

DOPING FEATURES

The **Analytic Doping Model** and **Geometric Doping Model** features now include preview buttons for visualizing doping profiles before solving the model equations. Two preview buttons, **Plot Doping Profile for Selected** and **Plot Net Doping Profile for All** are accessible in both the toolbar and the context menu. **Plot Doping Profile for Selected** illustrates the dopant concentration added by the selected doping feature, while **Plot Net Doping Profile for All** displays the absolute value of the net dopant concentration. P-type and n-type regions are easily distinguishable by their red and blue coloring, respectively.

You can explore this new functionality in tutorial models such as *bipolar_transistor*, *bipolar_transistor_3d*, *mosfet*, *trench_gate_igbt_3d*, and *gaas_pn_junction_infrared_led_diode*.

METAL CONTACT

For finite element formulations, the accuracy of using current-type terminals in metal contact has been significantly improved. This enhancement is achieved by setting the default constraint to **Weak Constraints** when the **Contact Type** is **Ideal Ohmic** and the **Terminal Type** is **Current**, **Circuit (Current)**, or **Power**.

OPTICAL TRANSITIONS

An information message will now appear if optical physics are provided by the **Semiconductor Electromagnetic Waves** multiphysics coupling. Additionally, a schematic image has been added to the **Equation** section within the **Semiconductor Electromagnetic Waves Coupling** feature.

INFINITE ELEMENT DOMAIN

An **Infinite Element Domain** can now be added to a model containing the **Schrödinger Equation** interface with only a Semiconductor Module license.

You can view this new feature in the *solving_hydrogen_atom* model.

IMPACT IONIZATION GENERATION

The **Okuto–Crowell model** in the **Impact Ionization Generation** feature has been extended to a more general expression that can account for different dependencies on the electric field.

DEFAULT PLOTS

A plot group visualizing the net doping profile has been added to the default plots for the Semiconductor interface. This net doping profile displays the absolute value of the net dopant concentration.

CORRECTIONS AND BACK COMPATIBILITY

- For finite element formulations, the **Constraint Settings** now include the dependent variable *Electric Potential*.
- The finite element (linear, quadratic shape function) formulation using electron and hole concentrations as the dependent variables is deprecated, and old models solved with this formulation will be converted to the finite volume formulation.
- The **Streamline diffusion** for the **finite element, log formulation**, found under the **Consistent Stabilization** section, is now turned off by default.

ELECTROLYTE-GATED ORGANIC FIELD-EFFECT TRANSISTOR

This model demonstrates how to simulate an electrolyte-gated organic field-effect transistor using a general drift–diffusion model. The model uses the Stabilized Convection–Diffusion Equation interface and the Electrostatics interface. It allows for the visualization of transistor characteristics and showcases the formation of the electric double layers (EDLs) in the simulated device, highlighting key features of EGOFETs.

SOLVING THE HYDROGEN ATOM

This model provides guidance on solving the hydrogen atom using the Schrödinger Equation interface. It offers visualizations of the hydrogen atom's orbitals and demonstrates that the simulated eigenenergies and probability densities align closely with analytical results.

UPDATED MODELS

Several models have been updated to showcase the advantages of the improved finite element formulations. These updated models are not only easier to set up but also demonstrate more efficient solutions. These models include *bipolar_transistor*, *bipolar_transistor_3d*, *bipolar_transistor_thermal*.

Structural Mechanics Module

New Functionality in Version 6.2

NEW PHYSICS INTERFACE: PHASE FIELD IN SOLIDS

A new physics interface, Phase Field in Solids, has been added. For consistency, the existing physics interface Phase Field has been renamed to Phase Field in Fluids. The new interface is useful, for example, in conjunction with some damage models for solids. In the **Add Physics** view, you can find it under **Mathematics**, in the subfolder **Moving Interface**.

NEW PHYSICS INTERFACE: TRANSPORT IN SOLIDS

A new physics interface, Transport in Solids, has been added. It is intended for analysis of diffusion-type problems in deforming solid media. When the diffusion is stress-driven, it can easily be coupled with a Solid Mechanics interface. In the **Add Physics** view, you find it under **Chemical Species Transport**.

NEW MULTIPHYSICS COUPLING: MAGNETIC FORCES, ROTATING MACHINERY

Using the new **Magnetic Forces, Rotating Machinery** multiphysics coupling, you can combine a Solid Mechanics interface with a Rotating Machinery, Magnetic interface at the domain level. The coupling adds the loading on a deformable and rotating structure, caused by Maxwell stresses.

NEW MULTIPHYSICS INTERFACE: MAGNETIC-ELASTIC INTERACTION IN ROTATING MACHINERY

The Magnetic-Elastic Interaction in Rotating Machinery combines a Solid Mechanics interface with a Rotating Machinery, Magnetic interface, using the **Magnetic Forces, Rotating Machinery** multiphysics coupling.

NEW MULTIPHYSICS COUPLING: THERMAL EXPANSION, THIN LAYER

Using the new **Thermal Expansion, Thin Layer** multiphysics coupling, you can couple the thermal expansion in boundaries having a **Thin Layer** material model with the temperature field on the same boundaries (computed with a heat transfer interface).

NEW MULTIPHYSICS COUPLING: UNSATURATED POROELASTICITY

This multiphysics coupling connects a Moisture Transport in Solids interface and a Solid Mechanics interface. The moisture pressure acts as a load in the pores, and structural deformations modify the storage coefficients and porosity. The settings for the **Unsaturated Poroelasticity** node are similar to those in the **Poroelasticity** node.

This feature requires a license for the Porous Media Flow Module.

NEW MULTIPHYSICS INTERFACE: UNSATURATED POROELASTICITY

The new Unsaturated Poroelasticity multiphysics interface combines a Solid Mechanics interface, a Moisture Transport in Solids interface, and the new **Unsaturated Poroelasticity** multiphysics coupling. It is found under the **Structural Mechanics** branch in the **Add Physics** window.

This feature requires a license for the Porous Media Flow module.

STABILIZATION OF UNCONSTRAINED CONTACT PROBLEMS

When modeling contact problems, it is common that some parts of the structure do not have sufficient constraints until contact has been established. This would lead to a singular stiffness matrix. A new automatic stabilization of such problems has been added. It can be controlled using the new **Stabilization** subnode under **Contact**.

INERTIA RELIEF ANALYSIS

Inertia relief analysis is a special type of static analysis of unconstrained structures that are accelerated by external loads. There will then be a force balance between the external loads and the inertial forces.

In all structural mechanics interfaces, a new feature, **Inertia Relief**, has been added. Using this feature, you can automate the setup of a special study sequence that computes the acceleration field, the corresponding inertia forces, and the resulting stresses.

COMPUTATION OF WARPAGE

In some applications, it is important to determine to what extent an originally planar surface deviates from its intended shape after deformation. Using the new **Warpage** node, available in the Solid Mechanics, Shell, and Layered Shell interfaces, the deviation from the original shape of a boundary can be automatically computed.

IMPROVED FORMULATION OF AVERAGE ROTATION

In the **Average Rotation** feature, the formulation for small rotations has been changed. The rotation is now computed using analytical expressions, so that extra dependent variables are no longer added. As an effect of this change, the user interface has been simplified.

FLUID-FILLED CAVITIES

A new feature, **Enclosed Cavity**, has been added to the Solid Mechanics interface. It can be used to model closed, fluid-filled cavities without the need to mesh the cavity itself. The pressure in the cavity acts as a load on the structure, and the volume of the cavity is controlled by the structural deformations. There are several available equations of state for the contents of the cavity, such as isothermal or adiabatic gas, or incompressible fluid.

VIRTUAL CRACK EXTENSION METHOD

The possibility to compute energy release rate and stress intensity factors using the virtual crack extension method has been added. To do this, you add a **Virtual Crack Extension** subnode under a **Crack** node. The method is an alternative to a J-integral, with the advantage that it can also be used in the presence of, for example, body loads or thermal expansion.

LIMITED DISPLACEMENT

In the **Prescribed Displacement** node in the Solid Mechanics, Multibody Dynamics, Shell, Layered Shell, and Membrane interfaces, the **Limited Displacement** option has been added. Using this option, it is possible to prescribe a maximum allowed displacement in a certain direction.

SPECIMEN SIZE IN TEST MATERIAL

Some material models have an inherent length scale. When using the **Test Material** feature for studying the behavior of a certain material model, it is now possible to specify a specimen size. In previous versions, it was assumed that the material model was length-scale independent.

ACTIVATION OF VISCOELASTIC MATERIALS

The **Activation** feature, in which elements can be added during the simulation, can now also be used for viscoelastic materials.

THIN LAYER ENHANCEMENTS

The **Thin Layer** feature is now also available for pairs.

In the **Spring Material**, new options have been introduced for prescribing a predeformation and for adding viscous damping.

THIN ELASTIC LAYER DEPRECATED

The **Thin Elastic Layer** feature in the Solid Mechanics interface has been deprecated. The same functionality is available using the more general **Thin Layer** feature.

In models created in versions prior to 6.2, **Thin Elastic Layer** nodes are still functional, but it is not possible to add such nodes from the UI. You are encouraged to replace **Thin Elastic Layer** by **Thin Layer** in existing models.

ENERGY DISSIPATION COMPUTATION

In **Spring Foundation**, **Thin Elastic Layer**, and **Thin Layer**, energy dissipation can now be computed when damping is present. A new **Energy Dissipation** section has been added to the settings.

RESULTANT LOAD IN SHELL INTERFACE

The possibility to specify load distribution by resultant has been added to the **Face Load**, **Edge Load**, and **Point Load** features in the Shell interface.

BIPHASIC MODEL IN POROELASTICITY

In the **Poroelasticity** multiphysics coupling, a new poroelasticity model named **Biphasic** has been added. It is suitable when modeling, for example, biological tissues and hydrogels.

This feature requires a license for the Porous Media Flow Module.

ENHANCEMENTS TO FIBER FUNCTIONALITY

The possibility to add a distribution of fibers to a material by adding a **Fiber** subnode was introduced in version 6.0. In the current release, there are several extensions to this functionality.

- The functionality is now also available in the Shell interface. In this context, fibers can also be assigned a bending stiffness and a through-thickness location.
- The material model of the fiber can now be a general nonlinear function between stress and strain.
- The **Fiber** subnode can now also be added to material models in a **Thin Layer**.
- The variable that is, by default, plotted for fibers has been changed from second Piola–Kirchhoff stress to Cauchy stress.

GEOMETRIC NONLINEARITY SETTINGS

The control over the formulation of geometric nonlinearity in the **Geometric Nonlinearity** section of various material models has been revamped. The purpose is to make the behavior more transparent, and also to enforce better consistency between a parent material and its subnodes.

LINEAR KINEMATICS USING EXTERNAL STRESS-STRAIN RELATION

The user-defined material model **External Stress-Strain Relation** no longer forces study steps to be geometrically nonlinear. The kinematic behavior is now controlled in the same way as for other material models. The default behavior is, however, the same as before.

RESULTANT LOADS IN SYMMETRY PLANES

When a load is given on a resultant form, it is now possible to account for symmetry and antisymmetry planes in the model. All load features having the **Resultant** option now have a new **Symmetry** section for specifying symmetry properties.

APPLICATION POINT FOR RESULTANT IN GLOBAL COORDINATE SYSTEM

When the **Resultant** option is used for input of a load, and **Application point defined using** is selected as **Coordinates**, the given coordinates are now always interpreted in the global coordinate system, irrespective of the settings in the **Coordinate System Selection** section.

When opening a model created in version 6.1, the conversion is automatic. However, it is not possible to perform an automatic conversion when models are created using the API. If this is the case, you may need to modify the script in order to adhere to the new convention.

WIRE AS EMBEDDED REINFORCEMENT

Using the **Embedded Reinforcement** multiphysics coupling, it is now also possible to embed elements from a Wire interface as reinforcements.

CROSS SECTIONS FOR BEAM AND TRUSS

In the Beam and Truss interfaces, the settings in the **Cross-Section Data** node have been simplified by removing one level of settings.

COMBINING FRICTION WITH ADHESION AND DECOHESION

Friction and adhesion are now by default considered as additive contributions. With this behavior, a smooth transition from bonded to frictional contact is obtained when an adhesive bond is broken by decohesion.

CHANGES IN SETTINGS FOR CONTACT AND CONTINUITY

As part of an effort to homogenize the terminology for Nitsche type constraints, there have been some changes to the settings for **Contact** and **Continuity**. In particular, the **Nonsymmetric** formulation has been renamed to **Incomplete**.

IMPROVED SYMMETRY CONDITION

When a symmetry condition is applied in the Solid Mechanics interface, the normal to the symmetry plane is now evaluated in an average sense rather than per element. This remedies problems that could occur when not all nodes on a symmetry plane of an imported mesh lie exactly on the same plane.

PERIODIC CONDITION FOR AXISYMMETRIC SHELLS

In the Shell interface, the **Periodic Condition** boundary condition is now also available in 2D axisymmetry.

SYMMETRY PLANE IMPROVED FOR TWISTING MOTION

In the **Symmetry Plane** node, available in the Solid Mechanics interface in 2D axisymmetry, it is now possible to choose between two interpretations of the symmetry. If you have enabled twisting degrees of freedom by selecting **Include circumferential displacement** in the physics interface settings, the new section **Circumferential Condition** will be available in the **Symmetry Plane** node. There, you can choose between symmetry and antisymmetry.

LOCAL COORDINATE SYSTEM IN ROLLER CONDITION

In the settings for **Roller** in the Solid Mechanics interface, a **Coordinate System Selection** section has been added. The local coordinate system can be used to specify directions when **Normal orientation** is selected as **Plane** or **Cylinder**.

EQUIVALENT STRESS IN STRESS LINEARIZATION

In the **Stress Linearization** feature, it is now also possible to evaluate Tresca and von Mises equivalent stresses along the stress classification line based on the linearized stress tensors.

MIXED FORMULATION IMPROVEMENTS

- The mixed formulations have been improved so that they will always produce symmetric contributions to the stiffness matrix.
- When using a mixed formulation under plane stress conditions, a new option called **Implicit formulation** is available. With this option, the plane stress equations are used to simplify the problem.

THERMAL EXPANSION FOR CONSTRAINTS

It is now possible to add a **Thermal Expansion** subnode under essentially all constraints. This functionality, in which a constraint deforms due to temperature changes, has been available since several releases for the more common types of constraints. From version 6.2, it is also present in more specialized constraints in, for example, the Beam and Shell interfaces.

LINEARIZATION POINT TEMPERATURE

When performing a coupled structural–thermal analysis in the frequency domain, it is now possible to set the temperature for the linearization point without a preceding stationary heat transfer solution. Select the **Use volume reference temperature as linearization point** check box in the settings or the **Thermal Expansion** multiphysics coupling to use the reference temperature as a linearization point.

THERMOELASTICITY USING FINITE STRAINS

In case of large deformations, the Thermoelasticity multiphysics interface uses a new formulation based on multiplicative decomposition and the logarithmic Hencky strains.

INERTIAL LOADS ON RIGID FEATURES

Frame acceleration loads are now also applied to mass contributions given by the **Mass and Moment of Inertia** subnode under several features. These features are:

- **Rigid Connector**
- **Rigid Domain**
- All types of gears in the Multibody Dynamics and Solid Rotor interfaces

In the **Mass and Moment of Inertia** nodes, there is now a new section named **Frame Acceleration Forces**, in which you can select whether or not to include this contribution using the **Exclude contribution** check box. When you add a new node, the default is that the force contribution is included. If, however, you open a model created in version 6.1 or earlier, the **Exclude contribution** check box is selected. The reason is that in a

correct model built in an earlier version, separate point loads modeling the inertial loads should already be present.

The frame acceleration forces can come from any of these features:

- **Gravity**
- **Rotating Frame**
- **Base Excitation**
- **Linearly Accelerated Frame**

CELL PERIODICITY ENHANCEMENTS

The following changes have been made to the **Cell Periodicity** feature:

- The possibility to use homogeneous boundary conditions has been added, both for displacements and for tractions. Using the new **Type** list, you can now choose between **Periodic** and **Homogeneous** boundary conditions.
- It is now also possible to define a parametric sweep. This is controlled from the **Advanced** section.
- In the special study generated from **Cell Periodicity**, there is now also a solver suggestion for an iterative solver.

TOTAL REACTION FORCES FROM RIGID CONNECTORS

In the Shell and Beam interfaces, any reaction forces acting on rigid connectors are now also included in the variable for total reaction force.

ELASTIC WIRE SETTINGS RESTRUCTURED

The **Advanced** section has been removed from the **Elastic Wire** node in the Wire interface. The contents of this section are now available in the **Wire Properties** section under **Compressive stiffness**. The **Basic Section Properties** section has been renamed to **Cross-Section Data**.

INERTIAL TERMS IN BEAM INTERFACE

In the Beam interface, contributions to the mass matrix have been improved. A bending–torsion inertial coupling has been added to the **Linear Elastic Material** and **Section Stiffness** material models.

ENERGY VARIABLES IN FREQUENCY DOMAIN

In the frequency domain, the stored energy density, W_h , and the strain energy density, W_s , are no longer identical. W_h is defined as the cycle average elastic strain density,

whereas W_s is a complex-valued variable that acts as a potential for the instantaneous stress state.

PREDEFINED PLOTS

Several of the multiphysics couplings between different structural mechanics physics interfaces have options for automatically selecting coupled regions. New predefined plots indicating the regions that are actually connected have been added.

The predefined stress plot created by an **Embedded Reinforcement** node has been moved from the **Solid Mechanics** branch in the plot tree to the **Multiphysics** branch.

CHANGED DEFAULT VARIABLES IN RESULT PRESENTATION

When using the menus for adding variables (**Add/Replace Expression** or Ctrl+Space), most stress and strain variables will now have different default definitions depending on where in the model tree you are working. When modeling in the physics interfaces, or generally under **Definitions**, the standard definition of variables is used whereas when working under **Results**, variables that are determined from a least-squares fit to the Gauss point values are used. For example, if you want to add the stress in the x -direction, the suggested variable is `solid.sxx` in the first case and `solid.sGpxx` in the latter case.

The variables that are visible in a certain context can be controlled using the **Show All Variables** check boxes in the **Show More Options** dialog box.

GROUPING OF ODE VARIABLES IN SOLVER

In several features, a set of ordinary differential equation (ODE) degrees of freedom representing rigid body motion can be created. Up to version 6.1, some of these nodes would add two or three nodes under **Dependent Variables** in the solver sequence. For models with many such features, this list could become long. In version 6.2, the default is that such variables are grouped together instead.

The features where this new possibility has been introduced are:

- **Rigid Material**
- All gears
- All joints

In version 6.1, the same functionality was introduced for:

- **Rigid Connector**
- **Attachment**

You can control whether or not to use the grouping. This setting can be done in two places:

- The default behavior can be controlled from the **Advanced Settings** section in the settings for the physics interface.
- The default behavior can, however, be overridden for each individual feature by using a selection in the **Advanced** section in the feature itself.

When opening an old model, the grouping will be turned off in order to maintain full compatibility with the previous version. When building a model using the API, the new default will be used. This is true even if, for example, a Java® file created from an older version is used. If you want to achieve full backward compatibility using the API, you need to add lines similar to

```
model.component("comp1").physics("solid").prop("AdvancedSettings").set("GroupPhysOdesRd", false);
```

OVERRIDE PROPERTIES FOR CONSTRAINTS IN THE BEAM INTERFACE

The **Pinned** and **No Rotation** nodes in the Beam interface now contribute to each other. Previously, they would override each other.

In the unlikely situation that the previous behavior was used in an old model, the boundary conditions that are applied will change if it is solved in version 6.2. You will have to restructure the constraints or use the **Modify model configuration for study step** control in the settings for the study.

POINT MASS DAMPING REMOVED

Point Mass Damping, which existed as an optional subnode under **Point Mass** in some physics interfaces, has been removed. It is replaced by a new section named **Damping** in the settings for **Point Mass**. Old models are automatically converted when opened.

GENERAL NOTATION FOR PRESCRIBED DISPLACEMENT REMOVED

The **General notation** option, which was present in the **Prescribed Displacement** node for several physics interfaces, has been removed. This option was almost never used, and there are several better ways to include the same effect in a model.

RENAMED FEATURES

Several features have been renamed in order to get a more consistent naming structure. These are:

- **Layered Linear Elastic Material** renamed to **Linear Elastic Material**, **Layered**

- **Layered Hyperelastic Material** renamed to **Hyperelastic Material, Layered**
- **Layered Adiabatic Heating** renamed to **Adiabatic Heating, Layered**
- **Piezoelectric Effect** renamed to **Piezoelectricity**
- **Piezomagnetic Effect** renamed to **Piezomagnetism**
- **Piezoresistive Effect, Domain Currents** renamed to **Piezoresistivity, Domain Currents**
- **Piezoresistive Effect, Boundary Currents** renamed to **Piezoresistivity, Boundary Currents**
- **Piezoresistive Effect, Shell** renamed to **Piezoresistivity, Shell**

PART LIBRARIES

The **Representative Volume Elements** folder has been renamed to **Unit Cells and RVEs**. The structure inside this folder has been changed, and a number of new geometries have been added. These geometries include five different types of corrugated sheets and a gyroid.

New Models in Version 6.2

ELASTIC CLOAKING WITH POLAR MATERIAL

In this example, the **External Stress** feature is used to set the material model with nonsymmetric stress required in the design of an elastic invisibility cloak. This is a device whose aim is to shield a region of space from both P and S waves.

SCATTERED FIELD FORMULATION FOR ELASTIC WAVES

This model showcases how to solve for the scattered field when knowing the incident field for three different types of scatterers: an infinitely rigid object, a cavity, and an elastic inclusion.

This formulation can be useful when the scatterer is in the far field of the source, such that the probing wave resembles a plane wave. In this case, including the source would require an unnecessarily huge computational domain to be meshed. Both P and S plane waves are thus used as incident fields in the model.

Moreover, the model shows how to numerically compute the field emitted by a point source, and then use the solution as the known incident field for a subsequent study where the scattering problem is solved.

HOMOGENIZED MODEL OF A CORRUGATED SHEET

This example presents a homogenized numerical model of a corrugated sheet based on a unit cell, where numerically obtained equivalent stiffness matrices are compared with various analytical models. Two corrugation profiles, trapezoidal and round, are considered in this example.

MICROMECHANICAL MODEL OF A PIEZOELECTRIC FIBER COMPOSITE

In this example, the micromechanical properties of a piezoelectric fiber composite are studied. The homogenized electromechanical properties of the composite are derived from the individual microscopic properties of the matrix and fiber.

MICROMECHANICAL MODEL OF A TRIPLY PERIODIC MINIMAL SURFACE-BASED COMPOSITE

In this example, the homogenized elastic and thermal properties of a composite material based on a triply periodic minimal surface (TPMS) are computed.

A gyroid TPMS-based unit cell is subjected to periodic boundary conditions to get the homogenized material properties. The effects of negative Poisson's ratio and different volume fractions on the homogenized properties are analyzed.

EIGENFREQUENCY SHIFTS CAUSED BY TEMPERATURE CHANGES

This example explores the shift in natural frequencies that results from change of the temperature. One study investigates a doubly clamped beam where both ends are fixed, while the other study looks at a cantilever beam where only one end is fixed.

The following effects are studied:

- Stress stiffening
- Change in size
- Constraint effects
- Temperature-dependent Young's modulus

Results shows a very good fit to analytical values based on the small changes in geometry caused by the thermal expansion.

LADDER FRAME

This model illustrates how to perform eigenfrequency and static analyses of a ladder frame structure for a light truck. The example shows how to convert a solid geometry to a shell model, implement a distributed load by specifying the corresponding resultant, and verify a weld junction.

POSTBUCKLING ANALYSIS USING AN INCREMENTAL ARC LENGTH METHOD

For slender structures, buckling is a catastrophic instability if the service load is above the critical limit. For such structures, it can be important to study the behavior of the structure beyond the critical buckling load, which is known as postbuckling analysis. Tracing the equilibrium path in postbuckling analysis is not easy, as it leads to numerical difficulties such as limit points. Using an arc-length method is a well-known strategy to trace equilibrium paths in such situations.

In this example model, an incremental arc-length method combined with cubic extrapolation is used for postbuckling analysis of a hinged cylindrical panel subjected to a point load at its center. The results are compared with a similar example in which a global equation approach based on monotonically increasing average displacement is used.

MOISTURE TRANSPORT IN A PAPERBOARD ROLL

This example demonstrates how to model moisture transfer and induced swelling in a paperboard roll subject to varying ambient conditions. The material properties of the paperboard are anisotropic.

VACANCY ELECTROMIGRATION IN IC INTERCONNECT LINES

As integrated circuit (IC) technology advances, with circuits becoming more powerful and compact, it is increasingly important to identify and prevent any cause of circuit failure.

One particularly critical factor contributing to circuit failure is electromigration within the interconnects, stemming from the accumulation of vacancies within the metal.

Electromigration denotes the migration of vacancies within the metal, driven by electric fields, concentration, hydrostatic stress, and temperature gradients. This example illustrates how this highly coupled phenomenon can be modeled and analyzed in COMSOL Multiphysics.

HYDROGEN DIFFUSION IN METALS

This model shows how to simulate the uptake and diffusion of hydrogen in a notched metal sample from an aqueous electrolyte. It uses the Transport in Solids interface to model both the concentration-driven and stress-driven diffusion in the solid domain.

ELECTROMAGNETIC AND MECHANICAL ANALYSIS OF AN INTERIOR PERMANENT MAGNET MOTOR

The magnets in an interior permanent magnet (IPM) motor are embedded in the rotor core, where they form narrow regions known as bridges. The thickness of the magnetic bridge is an important parameter to consider in design, both from the electromagnetic and mechanical perspectives. As the magnetic saturation in bridge areas affect the electromagnetic characteristics of an IPM, the thickness of the bridge should be kept minimum to reduce losses. However, during high-speed rotation, these narrow bridges also experience high stress, caused by centrifugal forces. These two conflicting aspects makes the design and analysis of an IPM motor challenging.

In this example, the coupling between Solid Mechanics and Rotating Machinery, Magnetic for performing electromagnetic and mechanical analysis of an IPM motor is demonstrated. A motor with 10 rotor poles and 12 stator slots is modeled in 2D. The magnets are embedded in a V-shaped configuration inside the rotor core. The connection between magnets and rotor core is modeled as springs. The results give insight into magnetic flux density and stress distribution in the system.

BRACKET — INERTIA RELIEF ANALYSIS

This is a tutorial example, showing how to perform an inertia relief analysis.

Subsurface Flow Module

New Functionality in Version 6.2

NEW FEATURE: DUAL PERMEABILITY

A new **Dual Permeability** node has been added to the Darcy's Law and Richards' Equation interfaces. The feature allows for the modeling of saturated fluid flow within a dual permeability approach, where two distinct and interconnected porous systems are present. The dual permeability approach can be applied to situations where both porous systems are fully saturated and have varying permeabilities.

NEW FEATURE: UNSATURATED DUAL PERMEABILITY

A new **Unsaturated Dual Permeability** node has been added to the Darcy's Law and Richards' Equation interfaces. The feature allows modeling unsaturated fluid flow within two distinct and interconnected porous systems.

Fluid flowing through macropores and micropores is described by the Richards' Equation interface and is solved separately for both systems. This accounts for the heterogeneous nature of unsaturated porous media flow, where the presence of two distinct and interconnected systems significantly affects fluid flow.

NEW FEATURE: DUAL POROSITY

A new **Dual Porosity** node has been added to the Darcy's Law and Richards' Equation interfaces. This feature allows for the modeling of saturated fluid flow where two distinct and interconnected porous systems are present.

One system possesses significantly larger pores (macropores) in which the fluid flow takes place. This system is interconnected with a second system of smaller pores (micropores).

NEW PHYSICS INTERFACE: PHASE TRANSPORT IN FREE AND POROUS MEDIA FLOW

A new physics interface, **Phase Transport in Free and Porous Media Flow**, has been added. The new interface allows for the easier modeling of multiphase transport in coupled free and porous media flow.

NEW MULTIPHYSICS INTERFACE: FREE AND POROUS MEDIA FLOW, DARCY

The new multiphysics interface **Free and Porous Media Flow, Darcy** combines a Darcy's Law interface, a Laminar Flow interface, and the new **Free and Porous Media Flow Coupling** node. It is found under the **Porous Media and Subsurface Flow** branch under **Fluid Flow** in the **Add Physics** window.

RENAMED INTERFACE

The **Free and Porous Media Flow** physics interface has been renamed to **Free and Porous Media Flow, Brinkman**. Models created with earlier versions containing the **Free and Porous Media Flow** interface will keep the old name.

NEW DENSITY OPTIONS

The options for the **Fluid type** in the **Fluid** subnode to the **Porous Medium** and **Unsaturation Porous Medium** nodes in the Darcy's Law and Richards' Equation interfaces have been extended to better describe the equation of state used for the fluid. Two new options, **Incompressible** and **Compressible, linearized** have been added, and the previous default option, **Gas/Liquid**, has been renamed to **Compressible**.

When opening an old model, the new implementation will be used. When building a model using the API, the new default will also be used. This is true even if, for example, a Java® file created from an older version is used when the **Porous Medium** or **Unsaturation Porous Medium** node uses the **Linearized** or **Poroelastic** storage model and the default gas/liquid `fluidType` is used. If you want to achieve full backward compatibility using the API, and you have code such as

```
model.component("comp1").physics("d1").feature("porous1").set("storageModelType", "poroelastic");

model.component("comp1").physics("d1").feature("porous1").feature("fluid1").set("rho_mat", "userdef");

model.component("comp1").physics("d1").feature("porous1").feature("fluid1").set("rho", "1000[kg/m^3]");
```

it will need to be changed to

```
model.component("comp1").physics("d1").feature("porous1").set("storageModelType", "poroelastic");

model.component("comp1").physics("d1").feature("porous1").feature("fluid1").set("fluidType", "compressibleLinearized");

model.component("comp1").physics("d1").feature("porous1").feature("fluid1").set("rhoref_mat", "userdef");
```

```
model.component("comp1").physics("d1").feature("porous1").feature("fluid1").set("rhref", "1000[kg/m^3]");
```

ADDITIONAL VELOCITY CONTRIBUTIONS

A new **Contributing Velocity** subnode is available under **Porous Medium**, **Unsaturated Porous Medium**, and **Dual Porosity Medium** in the Darcy's Law and Richards' Equation interfaces. The new subnode allows for adding external velocity contributions to the Darcy velocity, which can, for example, arise due to solute concentration gradients in liquid flow or Knudsen diffusion in gas flow applications.

REFACTORING OF VELOCITY VARIABLES

For results evaluation, the velocity variables have been updated such that `d1.u` now includes possible **Contributing Velocity** fields. The Darcian or non-Darcian contribution is available in the new variable `d1.ud`. The latter has been corrected to include the correct frame transformation, which has an effect in results evaluation for large deformations. For consistency, the corresponding material frame Darcy velocity variable has been renamed to `d1.Udi`.

REFACTORING OF GRAVITY

The user input for **Acceleration** of gravity in the **Gravity Effects** section in the Darcy's Law, Richards' Equation, Fracture Flow, and Layered Darcy's Law interfaces has been removed. Instead, the variable `d1.g` for the acceleration of gravity is by default set to `g_const` if gravity effects are excluded in the physics interface. If gravity effects are included, the acceleration of gravity can be edited in the **Gravity** feature.

BOUSSINESQ APPROXIMATION

Boussinesq approximation for the **Nonisothermal Flow** multiphysics couplings is now available. The approximation assumes that density variations are only contributing to buoyancy effects.

CONTEXT MENUS FOR BOUNDARY CONDITIONS

Boundary conditions have been organized into context menus enhancing the user interface. The **Hydraulics** menu combines the **Pressure Head**, **Hydraulic Head**, **Atmosphere/Gauge**, **Pervious Layer** and **Precipitation** boundary conditions, whereas the **Interior Surfaces** menu groups the **Flux Discontinuity**, **Interior Wall**, and **Thin Barrier** boundary conditions under a single context menu.

PREDEFINED PLOTS AND VARIABLES FOR WELL FEATURE

The **Well** feature has been enhanced to include predefined global quantities for assessing injection and production rates. Additionally, for time-dependent simulations, predefined plots for injection and production rates are available.

New and Updated Models in Version 6.2

MODELING GROUNDWATER CONTAMINATION

A simple model for groundwater contamination is developed containing a well that is bored into an aquifer. The aquifer consists of a partially saturated layer and a fully saturated layer. The model investigates how much of the contaminant ends up in the well after a contaminant is introduced in the aquifer. It employs the **Free and Porous Media Flow** and the **Reacting Flow, Diluted Species** coupling nodes, which link Creeping Flow, Darcy's Law, and Transport of Diluted Species in Porous Media interfaces.

SEAWATER INTRUSION IN A COASTAL AQUIFER

Seawater intrusion is a critical issue in coastal regions, where it endangers both freshwater supplies and the overall quality of groundwater.

This example showcases seawater intrusion in a coastal aquifer where a pumping well is located at some distance from the shoreline. The model incorporates a dual porosity approach in order to account for the distinctive condition, where a low-conductivity aquifer intersects with highly conductive “lava tubes”.

FURROW IRRIGATION — DUAL PERMEABILITY

This example shows how to set up a model of furrow irrigation in a nonuniform soil column. It employs the **Unsaturated Dual Permeability** feature, which links two Richards' Equation interfaces through a fluid transfer function. This scenario can be regarded as a benchmark problem for dual permeability modeling.

Uncertainty Quantification Module

New Functionality in Version 6.2

DEPENDENT INPUT PARAMETERS FOR UQ STUDIES

Settings for input parameters now support dependent input parameters. The dependent input parameters can be selected in the same correlation group with a specified correlation matrix. The dependent input parameters are sampled using the Gaussian copula method, which uses the user-specified marginal distributions and correlation matrix for data sampling. When surrogate function settings are used to specify the surrogate-based Monte Carlo analysis' input parameters and surrogate model verification input parameters, the same correlation settings used in training the surrogate function are used to sample data for the Monte Carlo analysis and verification.

SURROGATE FUNCTIONS ADDED TO SURROGATE-BASED UQ STUDIES

The surrogate-based UQ study types, including sensitivity analysis, uncertainty propagation, reliability analysis, and inverse uncertainty quantification now create a global surrogate function after you run the study. The function can be evaluated without a license for the Uncertainty Quantification Module.

Wave Optics Module

New Functionality in Version 6.2

NEW BOUNDARY CONDITIONS ADDED TO THE ELECTROMAGNETIC WAVES, BOUNDARY ELEMENTS INTERFACE

Both the **Impedance Boundary Condition** and the **Layered Impedance Boundary Condition** have been added to the Electromagnetic Waves, Boundary Elements interface. The **Impedance Boundary Condition** typically represents a metallic bulk material outside the simulation domain. With the **Layered Impedance Boundary Condition**, additional layers of different materials and thicknesses can be added between the interior simulation domain and the exterior domain.

ELECTRIC DISPLACEMENT FIELD MODELS ADDED TO THE ELECTROMAGNETIC WAVES, BOUNDARY ELEMENTS INTERFACE

Now, for the **Wave Equation, Electric** feature in the Electromagnetic Waves, Boundary Elements interface it is possible to choose between several different material models, like **Relative permittivity, Refractive index, Dielectric loss**, and so on, when specifying the dielectric properties in the domain solved for. The options are also available for the new **Impedance Boundary Condition** and **Layered Impedance Boundary Condition** nodes.

SYMMETRY FUNCTIONALITY FOR THE ELECTROMAGNETIC WAVES, BOUNDARY ELEMENTS INTERFACES

In the settings for the Electromagnetic Waves, Boundary Elements interface, it is now possible to select symmetry planes to reduce the computational domain. The symmetry settings also control the far-field calculations and the physics-controlled meshing.

PHYSICS-CONTROLLED MESHING FOR THE TIME-DOMAIN INTERFACES

The time-domain physics interfaces, Electromagnetic Waves, Transient and Electromagnetic Waves, Time Explicit, now provide physics-controlled mesh suggestions based on the frequency or wavelength content in the simulation.

CYCLIC SYMMETRY OPTION FOR PERIODIC CONDITIONS

Cyclic symmetry has been added as an additional option for the **Periodic Condition**. This allows you to perform simulations of a smaller structure, if the problem is cyclically symmetric.

HIGHER-ORDER DISCRETIZATION ELEMENTS

Up to seventh-order curl elements can now be used with the Electromagnetic Waves, Frequency Domain and Electromagnetic Waves, Transient interfaces.

API COMMAND FOR THE ADD DIFFRACTION ORDER BUTTON ACTION

Now, it is possible to perform the same action as when clicking the **Add Diffraction Orders** button for periodic ports, using the COMSOL Java API. Use the Java statement

```
model.component("comp1").physics("ewfd").feature("port1").runCommand("addDiffractionOrders");
```

where “comp1”, “ewfd”, and “port1” are the tags for the model component, the physics interface, and the excited port, respectively, and `model` is a model object.

ELECTRICAL CONDUCTIVITY CAN BE SPECIFIED WITH THE DRUDE-LORENTZ AND DEBYE DISPERSION MODELS

Now, when using the **Drude-Lorentz dispersion model** and the **Debye dispersion model** **Electric displacement field model** options, the **Electrical conductivity** can now be specified in a separate input field. This functionality is applicable to the **Wave Equation, Electric; Impedance Boundary Condition; Transition Boundary Condition; Layered Impedance Boundary Condition;** and **Layered Transition Boundary Condition** nodes for the Electromagnetic Waves, Frequency Domain; Electromagnetic Waves, Beam Envelopes; and Electromagnetic Waves, Boundary Elements interfaces.

PHYSICS-CONTROLLED MESHING HANDLES MORE FUNCTIONS AND VARIABLES

The algorithm for building physics-controlled meshes has been improved to better handle material property expressions, including functions and material property variables. For instance, a material property can be defined as $f(\text{freq})$, where f is the function name and freq is the model input for the frequency. However, it is not possible to specify functions that depend on spatial coordinates, like $g(x)$, where x is a spatial coordinate. Material property variables, like `mat1.rfi.n11` can now be used as part of other user-defined material property expressions. Here, `mat1` is the material tag, `rfi` is a material property group identifier, and `n11` is a tensor element (in this case, a refractive index element).

INCIDENT WAVE DIRECTION INPUT FIELD FOR GAUSSIAN BEAMS

When using the **Gaussian beam** input option in the **Scattering Boundary Condition** and **Matched Boundary Condition** in the **Electromagnetic Waves, Beam Envelopes** interface, there is a new **Incident wave direction** input field. This input field specifies the main

propagation direction for the incident Gaussian beam. Previously, it was the wave vector defined by the physics interface that specified the propagation direction. This solution could be problematic if the wave vector distribution was inhomogeneous due to a complicated phase distribution.

HEXAGONAL UNIFORM ARRAY FACTOR

The hexagonal uniform array factor quickly estimates the far-field pattern of antenna arrays on a triangular grid. The hexagonal antenna arrays provide lower sidelobes, more robust performance with better resolution, lower spatial noise, and wider coverage.

INSTANTANEOUS NORM VARIABLES FOR VECTOR QUANTITIES

New variables have been introduced of the form

$\text{phys.normXi} = \sqrt{\text{real}(Xx)^2 + \text{real}(Xy)^2 + \text{real}(Xz)^2}$, where *phys* is the physics tag, such as *ewfd*, and *X* is a physical quantity like electric field (E), magnetic field (H), and so on. These variables are especially useful when visualizing time-harmonic vector waves.

USER-DEFINED SURFACE IMPEDANCE OPTION

For the **Impedance Boundary Condition** and **Layered Impedance Boundary Condition**, the surface impedance can now be directly entered instead of being indirectly calculated from the given material parameters.

ONLY ISOTROPIC MATERIAL PROPERTIES ALLOWED FOR SOME BOUNDARY CONDITIONS

The UI has been updated to reflect that the **Transition Boundary Condition**, **Impedance Boundary Condition**, **Layered Transition Boundary Condition**, and **Layered Impedance Boundary Condition** nodes only support the entry of isotropic material properties. Thereby, the UI better reflects the assumptions used in the implementation of these boundary conditions.

UPDATED ALGORITHM FOR PHYSICS-CONTROLLED MESHING FOR MODE ANALYSIS

The algorithm for physics-controlled meshing for mode analysis has been slightly changed. A slightly different mesh can be obtained when running an old model in COMSOL Multiphysics® version 6.2.

EXPORT EXPRESSIONS SUBNODE ADDED TO DEFAULT RADIATION PATTERN PLOTS

To simplify the export of data from **Radiation Pattern** (far-field) plots, when default **Radiation Pattern** plots are generated, there is now an **Export Expressions** subnode. Use the **Export Expressions** subnode to add expressions for additional quantities to include in plot data export for the **Radiation Pattern** plot.

REUSE SPARSITY PATTERN ENFORCED WHEN PERFORMING FREQUENCY AND WAVELENGTH SWEEPS

To improve performance, the automatic solver suggestions now checks the **Reused sparsity pattern** check box in the **Assembly Settings** section of the **Advanced** solver feature node, when performing frequency-domain or wavelength-domain sweeps.

NEW PHASE SECTION IN PLOT GROUP SETTINGS

When creating plots for frequency-domain simulations, it is now possible to specify the **Solution at angle (phase)** directly in the plot group settings. Previously, this setting could only be set on the datasets. Now, the same dataset can be used for different plot groups, displaying results for different phase settings.

REVISED SOLVER CONFIGURATIONS FOR THE ELECTROMAGNETIC WAVES, BOUNDARY ELEMENTS INTERFACE

The **Direct Preconditioner** now utilizes the **MUMPS** solver and adopts the **Block low rank factorization** functionality with a default tolerance of $1e-6$. These enhancements are especially advantageous when processing models of a substantial size.

New and Updated Models in Version 6.2

NEW APPLICATION LIBRARY CATEGORIES

Two new categories — **Couplers, Filters, and Mirrors** and **Modulators and Switches** — have been added to the Wave Optics Module Application Library. The **Waveguides and Couplers** category has been renamed **Waveguides**. Some existing models have been moved to the new categories. For instance, the Directional Coupler model is now found in the **Couplers, Filters, and Mirrors** category.

IN-PLANE SWITCHING OF A LIQUID CRYSTAL CELL

This model demonstrates the switching capability of a liquid crystal (LC) display cell in an in-plane switching (IPS) configuration. The Oseen–Frank model is used to solve for the LC director (optical axis) distribution when a static electric field is applied. A

Weak Form PDE interface is used to define the Oseen–Frank weak equation, whereas the **Electrostatics** interface is used to solve the electric potential distribution. For the given inhomogeneous anisotropic LC material, a full-wave simulation is performed using the **Electromagnetic Waves, Frequency Domain** interface.

METASURFACE BEAM DEFLECTOR

This model demonstrates how to simulate a metasurface beam deflector that uses anomalous refraction. The structure itself is a repeated array of six posts (so-called meta elements). The periodicity of the individual posts is 500 nm, so the full structure of six posts is 3 μm wide. The posts are made of silicon and are 1 μm tall. The substrate is SiO_2 . The structure is designed so that incident light coming through the substrate at a normal angle of incidence will be refracted at prescribed angle (the anomalous refraction angle).

WAVEGUIDE S-BEND

This model demonstrates how to simulate the propagation of guided waves in a dielectric S-bend optical waveguide. The model demonstrates that the phase approximation, required by the **Electromagnetic Waves, Beam Envelopes** interface, can be numerically calculated by solving an additional partial differential equation.

RCS OF A METALLIC SPHERE USING THE BOUNDARY ELEMENT METHOD

This model illustrates the process of evaluating the radar cross section (RCS) of a metallic sphere through the utilization of the boundary element method (BEM). By taking advantage of a vertical symmetry plane that is parallel to the polarization of an incident background field, the model reduces computational expenses. The computed RCS values are compared with analytical values within the Mie RCS region.

SIMULATION OF METAL-AIR SURFACE PLASMON POLARITON PROPAGATION AND DISPERSION

Electromagnetic waves that are confined to propagate along a surface, such as surface plasmon polaritons (SPPs), are of great research interest due to their potential applications in nanoscale manipulation of light. This model demonstrates how to set up a simulation of the frequency-wave vector dispersion relationship of SPPs propagating along an interface between air and bulk silver.

THRESHOLD GAIN CALCULATIONS FOR VERTICAL-CAVITY SURFACE-EMITTING LASERS

This model has been extended to also include a far-field calculation.

TAPERED WAVEGUIDE

The mesh sequence has been improved to better handle different taper length sweep ranges.

Material Library

New and Updated Material Data in Version 6.2

In version 6.2, the Material Library contains 15,412 materials and 139,553 material property datasets. The following improvements and additions have been made:

- Data has been added for over 600 glasses and over 500 polymers.
- Data has been added for several solders, graphites, and thermoelectric materials.
- Data has been added for several glasses, several PMMAs, Colmonoy, Haynes 6B, Haynes 6K, Hastelloy HYBRID-BC1, Haynes G-35, Haynes NS-163, Haynes HR-224, Haynes 233, Haynes HR-235, and Haynes 25.
- Data has been added for several superalloys, magnetic materials, and sealing glasses.
- Data has been added for CMSX-2 Plus, IN-939, and Inconel 718.
- The refractive index versus wavelength and temperature are now available as new properties.
- New references for H₂O and KCl were incorporated, and some of the values changed by a few percent.
- Added stress–strain curves for Al 6005, Al 6111, Al 7075, 1035 steel, Ti-6Al-4V, C37700, and 34CrNi3MoV.
- A new reference has been added for H11 tool steel thermal properties.
- The expansion data for several Corning glasses (1415, 1416, 1417, 1990, 7046, 7056, 7070, 7555, 7556, 7567, 7568, 7570, 7572, 7574, 7575, 7576, 7578, 7580, 7583, 7585, 7586, 7589, 7590, 7595, 7597, 7598, 7599, 7723, 7732, 7761, 8161, 8445, 8463, 9010, 9013, 9108, and 9119) were high by a factor of 10. These values have been fixed.
- The ϵ -N fatigue data for 314 stainless steel, the UTS/YS data for Mg alloy HM31 F temper, and the thermal diffusivity for the 4 compacted graphitic irons were incorrect. These values have been fixed.
- The thermal expansion data for Ti-4Al-4Mo-2Sn (IMI550) was incorrect and has been corrected. The correct values are 5% to 10% lower.
- The creep data for Hytrel 4556 were incorrect. These values have been fixed.
- The thermal conductivity of Inconel HX/Hastelloy X/Nimonic PE13 was too high by a factor of 100. This value has been fixed.

- The compressive stress–strain data for Sn - 3.8 Ag - 4.8 Bi, UNS 26000, 300M, SX Rene N5, and CMSX-4 were incorrect. These have all been fixed.
- The compressive stress–strain data for Al 7075 were incorrect due to an error in the reference. A correction was released by the authors and the data has been updated accordingly.
- The expansion data for Ti-150A was incorrect and has been fixed.
- The expansion data for Timetal 834 was incorrect in the reference. It was replaced with data from a new reference.
- Data for Mg alloy AE44 was incorrectly labeled as being for AE33. This error has been fixed.
- Several sets of viscosity data have been refitted to the Arrhenius equation.
- The stress rupture for Waspaloy age hardened bar was in error and has been revised.
- The density data for several graphites were incorrect and have been corrected: Graphite CZR, Graphite HPD, Graphite SFG-1, Graphite DFP-1, Graphite PLS-1, and Graphite SCF.
- New references have been added for the viscosity and expansion of some liquid metals and solders. Any changes in values are only a few percent.

LiveLink™ for Excel®

New Functionality in Version 6.2

- The **COMSOL** ribbon tab inside the Excel® user interface now includes functionality to evaluate, and insert into a worksheet, the results from **Evaluation Group** nodes. A corresponding VBA method is also available.
- It is now possible to browse models saved on a Model Manager database when opening models with the LiveLink™ add-in inside the Excel® user interface.

LiveLink™ for MATLAB®

New Functionality in Version 6.2

IMPROVED PLOTTING

When plotting using `mphplot`, `mphgeom`, and `mphmesh`, the plots now show the axes labels for the x -, y - and z -axes. In addition, the following view settings are used: lighting and hiding. This leads to plots in MATLAB® that are closer to the plots seen inside of COMSOL Multiphysics. This behavior could previously be obtained using the `view` property when calling these functions, but the behavior is now on by default.

CHANGING PARAMETERS

Changing parameter values and re-solving a model is probably the most used feature of LiveLink™ for MATLAB®. The `mphsetparam` function has been added to make it easier to set such parameters and to pass parameter values between functions. The `setparam` function permits setting parameter values (real or complex) using the numerical types in MATLAB® as well as using strings. Values can be set using cell arrays, structs, and dictionaries, which all make it easy to transfer parameters from one function to another all, contained in a single variable name. Parameter groups and parameter cases are supported.

MODEL MANAGER SUPPORT

The Model Manager is a tool that can be used to store and manage COMSOL Multiphysics® models and other related files in a database. An API has been added to the Model Manager that can be used to access and work with such models and files. Some wrapper functions have been added that can be used to load, save, and search for models. You can use the `mmsearch` function to search for models in a specific branch. It returns results in MATLAB® struct arrays that can be easily modified and displayed in MATLAB®.

OTHER NEW FUNCTIONS

The `mphselectball` function is a new function in LiveLink™ for MATLAB® that has functionality similar to the existing `mphselectcoords` function but uses the built-in **Ball Selection** from COMSOL Multiphysics®, which have more options and finer control.

The new `mphbatchinfo` function gives information about the `model.batch` model node, which contains settings for parametric sweeps and cluster operations.

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.

Backward Compatibility with Version 5.6

COLOR TABLES

All functions that have to do with color tables (for example, `cividis`, `cyclic`, and `disco`) except for `colortable` have been deprecated. When using the `colortable` function, you can use autocompletion to get a list of available color tables. As an alternative to using, for example, `cividis` you can instead use

```
table = colortable('cividis');
```

REPLACED AND DEPRECATED FUNCTIONS

The `mphint`, `mphgetp`, and `mphgetmodelnode` functions are deprecated. `mphgetmodelnode` has been replaced by `mphcomponentinfo` where new features have been added. The deprecated functions still exist but will result in a warning when used, and they may be removed in a future release of COMSOL Multiphysics. Such warnings can be disabled using this command

```
warning('off', 'COMSOL:DEPRECATED')
```

LiveLink™ for Simulink®

New Functionality in Version 6.2

A new block has been added, which makes it easier to use reduced-order models exported from COMSOL Multiphysics®.

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 6.2 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 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 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. Now, 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 new 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().setType(<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().  
    getType();
```

COMSOL 6.0 API Changes

PAIR FEATURES

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

Java[®] or MATLAB[®] program that accessed fallback features under a pair feature will not work anymore. This is a necessary limitation because any attempt to support such API backward compatibility will be both unintuitive and unsafe. For most physics created from the API, the default feature from the physics will act as 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);

```

COMSOL 5.6 API Changes

MESH PART UNITS

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

THE ADAPT FUNCTION — THE EXPRTYPE PROPERTY

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

This concludes the release notes for COMSOL Multiphysics version 6.2.

I n d e x

- A** AC/DC Module
 - new and updated models in 50
 - new functionality in 45
- AC/DC Module New and Updated Models in Version 6.2 50
- AC/DC Module New Functionality in Version 6.2 45
- Acoustics Module
 - new functionality in 54
- Application Builder, new functionality in 16
- B** backward compatibility, general considerations 40
- Battery Design Module
 - new and updated models in 65
 - new functionality in 63
- C** CAD Import Module
 - new functionality in 67
- CFD Module
 - new functionality in 70
 - new models in 71
- Chemical Reaction Engineering Module
 - new functionality in 74
- Composite Materials Module
 - new functionality in 79
 - new models in 80
- COMSOL API changes 177
- COMSOL Multiphysics, new functionality in 18
- Corrosion Module
 - new and updated models in 82
 - new functionality in 81
- D** Design Module
 - new functionality in 67
- E** ECAD Import Module
 - new functionality in 83
- Electrochemistry Module
 - new functionality in 84
- Electrodeposition Module
 - new models in 87
- F** Fatigue Module
 - new functionality in 88
 - new models in 88
- Fuel Cell & Electrolyzer Module
 - new functionality in 86, 89
 - new models in 91
- G** general new functionality 10
- Geomechanics Module
 - new functionality in 92
- geometry and mesh, new functionality for 22
- H** Heat Transfer Module
 - backward compatibility 99, 101–102, 111, 132
 - new functionality in 95
 - updated models in 99
- L** LiveLink™ for Excel®
 - new functionality in 173
- LiveLink™ for MATLAB®
 - backward compatibility 175
 - new functionality in 174
- LiveLink™ for Simulink®
 - new functionality in 176
- LiveLink™ products for CAD
 - new functionality in 67
- M** Material Library
 - new and updated material data in 171
 - new material data in 171
- MEMS Module

- new functionality in 104
 - new models in 107
- Metal Processing Module
 - new functionality in 109
 - new models in 115
- Microfluidics Module
 - new functionality in 110
 - updated applications in 111
- Mixer Module
 - new functionality in 112
- Model Manager Server, new functionality in 14
- Model Manager, new functionality in 10
- Multibody Dynamics Module
 - new functionality in 113
- N** Nonlinear Structural Materials Module
 - new functionality in 116
 - new models in 119
- O** operators, functions, and definitions, new and updated 26
- Optimization Module
 - backward compatibility 123
 - new functionality in 122
- P** Particle Tracing Module
 - new functionality in 124
- Plasma Module
 - backward compatibility 126
 - new applications in 125
 - new functionality in 125
- Porous Media Flow Module
 - new functionality in 127
- Porous Media Module
 - new applications in 130
- R** Ray Optics Module
 - new functionality in 133
 - new models in 134
 - results and visualization, new functionali-
ty in 35
- RF Module
 - new functionality in 135
- Rotordynamics Module
 - new functionality in 141
- S** Structural Mechanics Module
 - new functionality in 146
 - new models in 156
 - studies and solvers, new functionality in 30
- Subsurface Flow Module
 - new and updated models in 163
 - new functionality in 160
- U** Uncertainty Quantification Module
 - new functionality in 164
- W** Wave Optics Module New and Updated Models in Version 6.2 139, 168
- Wave Optics Module New Functionality in Version 6.2 165