

COMSOL Multiphysics Release Notes



COMSOL Multiphysics Release Notes

© 1998–2022 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 6.1

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/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/forum
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/videos
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM010001

Contents

Chapter I: Release Notes

COMSOL Multiphysics 10
General New Functionality
New Functionality in the Model Manager
New Functionality in the Model Manager Server
New Functionality in the Application Builder $\ . \ . \ . \ . \ . \ . \ . \ . \ . \ $
New Functionality in COMSOL Compiler $^{\texttt{TM}}$
New and Improved General Functionality in COMSOL Multiphysics $\ . \ . \ . \ 17$
New Geometry and Mesh Functionality
New Functionality in the Physics Interfaces
New and Updated Operators, Functions, and Definitions $\ . \ . \ . \ . \ . \ . \ 25$
New Functionality in Studies and Solvers
New and Improved Results and Visualization Functionality
New Java [®] API Methods
New COMSOL Server Functionality . <t< td=""></t<>
General Backward Compatibility Considerations
Backward Compatibility with Version 6.0
Backward Compatibility with Version 5.6
Backward Compatibility with Version 5.5
AC/DC Module 37
New Functionality in Version 6.1
Updated Functionality in Version 6.1 $\ .$ $\ .$ $\ .$ $\ .$ $\ .$ $\ .$ $\ .$ $\ .$ $\ .$ $\ .$ $\ .$ $\ .$ 40
New and Updated Models in Version 6.1
Backward Compatibility with Version 6.0 and Earlier $\ . \ . \ . \ . \ . \ . \ . \ . \ 43$
Acoustics Module 44
New Functionality in Version 6.1
New and Updated Model Examples in 6.1 $\ . \ . \ . \ . \ . \ . \ . \ . \ . \ $
Important Fixes in Version 6.1
Backward Compatibility with Version 6.0

Battery Design Module	53
New Functionality in Version 6.1.	. 53
New and Updated Applications and Models in Version 6.1	. 56
CAD Import Module, Design Module, and LiveLink™ Products	
for CAD	57
New Functionality in Version 6.1	. 57
New Functionality in the Design Module in Version 6.1	. 58
New Functionality in LiveLink [™] for AutoCAD® in Version 6.1	. 58
New Functionality in LiveLink ^{TM} for Inventor [®] in Version 6.1	. 58
New Functionality in LiveLink [™] for PTC® Creo® Parametric [™] in Version 6.1.	58
New Functionality in LiveLink [™] for Revit® in Version 6.1	
New Functionality in LiveLink [™] for SOLIDWORKS® in Version 6.1	
New Functionality in LiveLink [™] for Solid Edge® in Version 6.1	
	,
CFD Module	60
New Functionality in Version 6.1	. 60
New and Updated Models in Version 6.1.	. 62
Backward Compatibility with Version 6.0 and Earlier	. 62
Backward Compatibility with Version 5.6 and Earlier	. 63
Backward Compatibility with Version 5.5 and Earlier	. 63
Chemical Reaction Engineering Module	64
New Functionality in Version 6.1.	. 64
New Models in Version 6.1	
Updated Models in Version 6.1	. 68
Composite Materials Module	70
New Functionality in Version 6.1	. 70
New Models in Version 6.1	
Corrosion Module	72
New Functionality in Version 6.1	. 72
New and Updated Models in Version 6.1.	. 74
ECAD Import Module	75
New Functionality in Version 6.0.	. 75
•	

Electrochemistry Module	76
New Functionality in Version 6.1	76
Electrodeposition Module	78
New Functionality in Version 6.1	78
Fatigue Module	80
New Functionality in Version 6.1.	80
Fuel Cell & Electrolyzer Module	81
New Functionality in Version 6.1	81
New and Updated Models in Version 6.1	84
Geomechanics Module	85
New Functionality in Version 6.1.	85
Heat Transfer Module	86
New and Improved Functionality in Version 6.1	86
New Applications in Version 6.1	89
Updated Applications in Version 6.1	89
Backward Compatibility with Version 6.0.	90
Backward Compatibility with Version 5.6.	91
Backward Compatibility with Version 5.5.	92
Liquid & Gas Properties Module	94
Improved Functionality in Version 6.1	94
New Model in Version 6.1	94
MEMS Module	95
New Functionality in Version 6.1	95
New Models in Version 6.1	99
Metal Processing Module	101
New Functionality in Version 6.1	101
New Models in Version 6.1	102
Microfluidics Module	103
New Functionality in Version 6.1	103

Updated Applications in Version 6.1		•							103
Backward Compatibility with Version 6.0									104
Backward Compatibility with Version 5.5 and Earlier	•	•	•		•		•	•	104
Molecular Flow Module									105
New Functionality in Version 6.1		•	•						105
Updated Applications in Version 6.1		•							105
Backward Compatibility with Version 6.0 and Earlier	•	•	•	•	•	•	•	•	105
Multibody Dynamics Module									106
New Functionality in Version 6.1	•	•	•	•		•		•	106
Nonlinear Structural Materials Module									109
New Functionality in Version 6.1									109
New Models in Version 6.1	•	•	•	•	•	•	•	•	111
Optimization Module									113
New Functionality in Version 6.1									113
New and Updated Models in Version 6.1									114
Backward Compatibility with Version 6.0	•	•	·	•	•	•	•	•	115
Particle Tracing Module									116
New and Improved Functionality in Version 6.1 \therefore									116
New Applications in Version 6.1									117
Important Fixes in Version 6.1									117
Backward Compatibility with Version 6.0 and Earlier	•	•	•		•		•	•	118
Pipe Flow Module									119
New Functionality in Version 6.1									119
Updated Models in Version 6.1		•			•			•	119
Plasma Module									121
New Functionality in Version 6.1									121
New and Updated Applications in Version 6.1.									123
Backward Compatibility with Version 6.0 and Earlier									124
Backward Compatibility with Version 5.5 and Earlier		•	•						124

Porous Media Flow Module	125
New Functionality in Version 6.1	125
New Models in Version 6.1	126
Polymer Flow Module	128
New Functionality in Version 6.1	128
New Models and Updated Models in Version 6.1	129
Backward Compatibility with Version 6.0.	129
Ray Optics Module	130
New and Improved Functionality in Version 6.1	130
New And Updated Applications in Version 6.1	131
Important Fixes in Version 6.1.	131
Backward Compatibility with Version 6.0 and Earlier	132
RF Module	133
New Functionality in Version 6.1	133
New and Updated Models in Version 6.1	136
Backward Compatibility with Version 6.0 and Earlier	138
Rotordynamics Module	139
Rotordynamics Module New Functionality in Version 6.1	139 139
New Functionality in Version 6.1.	139
New Functionality in Version 6.1.	39 40
New Functionality in Version 6.1. .	139 140 141
New Functionality in Version 6.1. .	139 140 141 141
New Functionality in Version 6.1. .	139 140 141 141 150
New Functionality in Version 6.1. .	139 140 141 141 150 153
New Functionality in Version 6.1. .	139 140 141 141 150 153
New Functionality in Version 6.1. .	139 140 141 141 150 153 154
New Functionality in Version 6.1. .	139 140 141 141 150 153 153 154
New Functionality in Version 6.1. . . . New Models in Version 6.1. . . . Structural Mechanics Module . . . New Functionality in Version 6.1. . . . New and Updated Models in Version 6.1. . . . Subsurface Flow Module . . . New Functionality in Version 6.1. . . . New Functionality in Version 6.1. . . . New Functionality in Version 6.1. . . . New Models in Version 6.1. . . . New Functionality in Version 6.1. . . .	139 140 141 150 153 153 154 155

Backward Compatibility with Version 6.0 and Earlier	160
Material Library	162
New and Updated Material Data in Version 6.1	162
LiveLink™ for Excel [®]	165
New Functionality in Version 6.1	165
LiveLink [™] for MATLAB [®]	166
New Functionality in Version 6.1	166
Backward Compatibility with Version 6.0.	167
Backward Compatibility with Version 5.6	167
LiveLink™ for Simulink [®]	168
New Functionality in Version 6.1	168
General Backward Compatibility	168
The COMSOL API for Use with Java $^{{ m I}\!{ m B}}$	169
COMSOL 6.1 API Changes	169
COMSOL 6.0 API Changes	170
COMSOL 5.6 API Changes	171
Index	173

Release Notes

COMSOL Multiphysics[®] version 6.1 includes new functionality and improvements across the COMSOL[®] products. These *Release Notes* provide information regarding new functionality in version 6.1 for all COMSOL[®] software products.

COMSOL Multiphysics

General New Functionality

INTERFACE TO MICROSOFT WORD

A new interface to Microsoft Word[®] is now available, providing similar functionality as the interface to Microsoft PowerPoint[®]. When you install this interface, a COMSOL ribbon will be added to Microsoft Word[®], which provides functionality for interactively creating and updating Word documents with content, such as plots and tables, from COMSOL Multiphysics[®] models.

New Functionality in the Model Manager

Also see the Model Manager Reference Manual for more information.

GENERAL IMPROVEMENTS

- Comments for previously saved commits can now be edited.
- Components, geometry sequences, materials, and physics can now be inserted into a model opened in the COMSOL Desktop using the **Insert into Opened Model** button in the **Contents** section of the **Settings** window for a model version.
- A model can be opened with the current node selected using the **Open Node** button in the **Contents** section of the **Settings** window for a model version.
- Date filters now support shorthands for common date intervals. These shorthands are also supported in the Model Manager search syntax. Examples include @lastModified:TODAY and @lastModified:YESTERDAY.
- The default repository when saving models or data files can now be changed from the initial repository. The most recently used repository is also remembered when opening models or selecting data files.
- The **Copy Location** option in the context menu of a model version can be used to quickly open the model in another COMSOL Multiphysics process. A new **Clipboard** option appears in the list of options in the **Open** window when such a location has been copied.

COMPATIBILITY BETWEEN VERSIONS

Local and server databases can be shared between COMSOL Multiphysics 6.0 and COMSOL Multiphysics 6.1 since both the database format and the server connection

are designed for backward and forward compatibility. All database configurations in COMSOL Multiphysics are automatically migrated as part of the preferences migration when upgrading. Some new Model Manager functionality in COMSOL Multiphysics is only available when connected to the newest version of the Model Manager server.

Secure Credentials Entry When Running in Client-Server Mode

The Model Manager no longer allows writing a password in the **Connect to Server Database** window when running COMSOL Multiphysics in client–server mode unless the client–server connection is over a secure channel. Instead, the password is provided via a secure password prompt in the console window for the started COMSOL Multiphysics server process.

DEMO DATABASE FOR MODEL MANAGER

A local demo database for Model Manager can now be downloaded and opened using the **Download Demo Database for Model Manager** option in the **Help** menu. On the Windows[®] operating system, this menu is available as a submenu to the **File** menu. The demo database contains an imported copy of the COMSOL Application Libraries.

This functionality was released already with COMSOL Multiphysics 6.0 update 2.

VERSION CONTROL OF RELATED FILES AS FILESETS

Model Manager now supports storing related data files as *filesets* in a database. This enables version control of, for example, CAD assemblies that reference multiple external component files or HTML documents referencing external image files. The file contents of a fileset can be viewed and updated via the **Settings** window in the Model Manager workspace.

Improved CAD Support

All CAD formats can now be imported to Model Manager databases and used with the CAD Import Module functionality in Model Builder. When importing a CAD assembly file to the database, any external component files that the assembly references are imported as well.

Save Model Output to Databases

Data files generated as output from a model can now be saved directly to a Model Manager database. This includes, for example, export of geometries, meshes, plots, animations, and reports. The output files are version controlled in the database in exactly the same way as other data files. Model Manager also stores a reference between a saved file version and the corresponding model version. These version references are shown in the **References** window with a new **Output File** reference type.

Improvements to Tag Assignments and Tag Trees Several usability improvements have been made to tags:

- Assigned tags are now shown in a compact format in the Settings window.
- Tag assignment in the Model Manager workspace now supports selection of multiple items.
- A tag can be assigned to the current selection of models and data files when creating the tag from the **New Tag** dialog box.
- The tag tree in the **Model Manager** window now shows all tags when searching in a branch without having to repeatedly click the **Show More** toolbar button. Each tag node shows a count of the number of item hits matching that tag assignment.
- Models and data files shown under tag nodes in tag trees are grouped into batches of 100 items, with the next batch of items obtained by expanding a **Show More** node.
- Tags can be assigned to the model that is opened in the COMSOL Desktop already from the **Save** window. New tags can also be created as needed from that window.

New Functionality in the Model Manager Server

Also see the Model Manager Server Manual for more information.

GENERAL IMPROVEMENTS

- The Model Manager server is now available for ARM64 architectures on macOS and Linux[®].
- A **Hyperlink** primitive attribute has been added to the asset management system. The attribute can be used to link to a web resource.
- A **File version** primitive attribute has been added to the asset management system. The attribute can be used to link to a file version stored in the database from an asset.

CONNECT WITH CPU AND NSL LICENSES

COMSOL Multiphysics can now connect to a Model Manager server with CPUlocked licenses (CPU) and named single-user licenses (NSL) in addition to floating network licenses (FNL).

This functionality was released already with COMSOL Multiphysics 6.0 update 2.

SWITCHING BETWEEN DATABASES

The Model Manager server now supports switching between databases in the web interface via a list in the top menu bar. Selecting a database from the list sets it as the *current database* shown in the web interface. The list is hidden if only one database is available. COMSOL Multiphysics always connects to the database set as the *default database*.

WEB-BASED ASSET MANAGEMENT SYSTEM

The Model Manager server now includes a web-based asset management system that COMSOL Multiphysics users can use to link models and simulation results to various documents, presentations, project notes, slides, and other supplementary files and metadata. The asset management system can also be used to share such files and metadata with coworkers who may not have access to the COMSOL Multiphysics software

This functionality was released already with COMSOL Multiphysics 6.0 update 2.

BUILT-IN SUPPORT FOR HTTPS, EXTERNAL AUTHENTICATION, AND REVERSE PROXY-BASED AUTHENTICATION

Setting up Transport Layer Security (HTTPS), external authentication with AD/LDAP, and reverse proxy-based authentication is now supported directly from the system administration pages of the web interface.

This functionality was released already with COMSOL Multiphysics 6.0 update 2.

USER MANAGEMENT AND ACCESS CONTROL ADMINISTRATION

Setting up user management and access control for a server database is now possible from the database administration pages of the web interface.

This functionality was release already with COMSOL Multiphysics 6.0 update 2.

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

Apps With Resizable Subwindows

Apps built with the Application Builder no longer have to have just one big window. You can now create applications with resizable subwindows just like the COMSOL Desktop itself. The **Settings** window for the **Main Window** node includes a new **Main window type** list, where you can choose between the old **Single window** type or the new type, **Subwindows**. The **Main Window** editor, which you open by double-clicking the **Main Window** node, now includes a new grid editor, similar to the **Form** editor. Each cell in the editor is a subwindow where you can add one or more forms to appear as windows when running the application. The sizes of the rows and columns can be specified interactively by dragging them. They can also be entered manually in a new section called **Window Layout**, which is added to the **Settings** window for **Main Window** when **Main window type** is set to **Subwindows**. You can also control if dragging between subwindows is enabled. There are also new form layouts for apps with subwindows; see below.

Running Apps on Linux ARM Computers

It is now possible to run compiled apps on Linux ARM computers.

Localizing Apps

Apps can now be localized using embedded language resource files. The Tubular Reactor app in the COMSOL Multiphysics Application Library has been updated to include a demonstration of this feature. There is a new **Languages** library node that can be added under **Libraries** for managing the localization, and there is a new **Language Localization** section in the **Settings** window for **Main Window** for selecting the language to use.

Replacing Libraries

Entries in the settings for the **Files**, **Images**, **Sounds**, and **Languages** library nodes can now be replaced. Using the **Replace File** button in the table toolbar, you can select a new file to replace an existing entry in the table. The replace action only changes the file data; the name of the entry remains unchanged.

Showing Errors and Warnings

There is a new **Show Errors** button on the ribbon toolbar when using the **Form** editor or **Main Window** editor. When enabled, any objects that are in an error or warning state (such as having a missing source) will show an icon in the corner of the object in the editor. In the **Application Builder** tree, icons of window editor objects now show their error or warning status, and form icons show if they contain any object with such status. This display happens regardless of whether you have enabled the **Show Errors** setting or not.

IMPROVEMENTS TO THE FORM EDITOR

- Improvements to the **New Form** wizard:
 - The new **Basic** and **Subwindows** layouts are based on subwindows and appear below **Blank form**. **Basic** replaces the Ribbon, sections, and graphics template, and **Subwindows** replaces the Ribbon, logo, sections, and graphics template, which were available in earlier versions. The template previously named Basic has been renamed to **Single Form** and now appears after the **Subwindows** template.
 - The **Inputs/Outputs** tab in the **Basic** template in earlier versions has been split into two tabs: **Inputs** and **Outputs**.
- A local form can now be converted to a global form or vice versa, either by pasting or by drag-and-drop from a global to a local form location or vice versa.
- You can add a vertical scrollbar to forms by selecting the new **Vertically scrollable** check box, which is available for:
 - Forms in a subwindow
 - Forms in a tabbed form collection
 - The dialog settings for a form
- The **Image** form objects now include a **Stretch image** setting that determines if the image is scaled to fill the available space. When the form alignment is set to **Fill**, the image will grow or shrink depending on available grid cell space.
- The **Table** form objects now include an **Insert Row** action. Also, you can now select which of the properties in **Table** form objects that will be used to determine the number of rows in the table.
- In Graphics form objects, it is now possible to use all kinds of selections as the source. This includes selections generated by a geometry or mesh sequence. Selections in the geometry sequence are not accessible directly, only the ones generated by the geometry features. Also, the previous Target for Data Picking section is now called Data Picking. The previous Data picking check box has moved to this section and is now called Enable data picking.
- The Internet Explorer-based .NET WebBrowser used to display in-line web and HTML pages for Video and Web Page form objects in Windows has been replaced with the Chromium[™]-based CefSharp browser component. In the Settings windows for those form objects, use the new Native browser on Windows list to choose Chromium[™] (CEF) (the default) or Internet Explorer®, which was the only option in earlier versions.

- Applicable form objects now have an **On focus gained** event. The event occurs when the user control associated with the form object becomes focused by the user.
- It is now possible to select individual components in a vector as the source of form objects that support scalar data types. It is also possible to refer to a double array property or double array declaration as the source for an input field.

IMPROVEMENTS TO THE METHOD EDITOR

- You can now press F8 to launch the **Run** action, available on the context menus for **Method** nodes.
- You can now choose **Copy as Code to Clipboard** from the context menu when rightclicking multiple selected **Method** nodes, if applicable. The copied code will be the concatenation of the code of each of the nodes.

IMPROVEMENTS AND CHANGES FOR THE APPLICATION LANGUAGE

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

- The saveApplicationAs and saveApplicationOnServerAs methods now have a boolean return value. If the return value is false, the user canceled the operation and the application was not saved.
- The application theme and image export theme properties found in the **Themes** node settings are now available in the application language and support data access:

```
// Set dark application theme
app.mainWindow().set("theme", "$dark");
// Set light image export theme
app.mainWindow().set("imagetheme", "$light");
```

When running the app in the cross-platform user interface or in a web browser, the app must be saved and uploaded for the application theme change to take effect. The image export theme does not have this limitation.

The AppModel.declaration() method is a new method for retrieving a list object containing global DataSource objects. Additionally, the new
 Form.declaration() method retrieves a list object containing DataSource objects defined under a given form. This list object is of type Declaration, and this type exposes the same list methods as the FormObjectList, FormList, and ItemList types. The DataSource.getType() method is a new method for retrieving the same type of DataSource object and functions as the FormObject.getType() method.

- Two new overload methods for saveApplication and saveApplicationOnServer are now available. They take a boolean argument that determines if the user is prompted with a confirmation dialog box before saving. The existing methods without an argument remain and function as before.
- The *double conversion* toString(double, int) and toString(double, int, boolean) methods have been changed to use a format based on significant digits rather than number of decimals. This change affects existing apps. If you require the decimal behavior, you can use the toString(double,String) method instead.

New Functionality in COMSOL CompilerTM

For the COMSOL Compiler output, the platforms you can choose from now include Linux, ARM. The macOS, ARM platform in previous versions has been changed to macOS, Apple silicon, and the macOS platform in previous versions has been renamed as macOS, Intel.

New and Improved General Functionality in COMSOL Multiphysics

MULTIPHASE MATERIALS AND EFFECTIVE MATERIALS

New Multiphase Materials and Effective Materials nodes are available under Materials in the model tree. Using the Settings window for these materials, you can define effective material properties based on a set of mixing rules and the volume fractions and material properties of each Phase or Compound. The Multiphase Material is intended to be used in transport problems, and is especially useful for multiphase flow applications. You can use the Effective Material node to model other types of mixed, averaged, and composite materials.

NEW FIND AND REPLACE WINDOW

The **Find and Replace** window appears when you select the **Find** button in the Quick Access Toolbar, or just by pressing Ctrl+F. You can search using a text string or a regular expression, and use various search options. You can then choose for which occurrences you want to replace the text. You will also see a preview of the updated text.

CREATE AND EDIT PARAMETERS

You can now right-click in text fields and tables cells that contain numbers and expression in settings for many nodes in the **Model Builder**. It opens a **Create Parameter**

dialog box where you can create a parameter to be inserted instead of the number or expression. It is also added as a new parameter in a **Parameters** node of your choice.

Input fields and table cells in setting with the new **Create Parameter** option also includes an **Edit Parameter** option in the context menu. The **Edit Parameter** option shows up if the current text selection or cursor is inside an existing parameter name, or (available in the context menu of a table cell not in edit mode) if the cell contains only such a name. It opens an **Edit Parameter** dialog box, similar to the **Create Parameter** dialog box, where you can edit the expression, description, and name of the parameter.

Input fields and table cells containing expressions now also include tooltips that show the evaluated value of the expression.

MEMORY USAGE INFORMATION FOR APPLICATION LIBRARIES

When you hover over the **Computation time** value for a selected model in the **Application Libraries** window, you can now see the maximum amount of physical memory (RAM) used while solving. You will also see information about the computer that was used when logging the values for the memory usage and computation time.

NEW AND IMPROVED MATERIALS IN THE BUILT-IN MATERIAL LIBRARY

For the **Built-in** material library, the Structural Steel material has been extended with data for plasticity, creep, viscoplasticity, and temperature-dependent properties. Also, a new generic Rubber material has been added.

PERFORMANCE IMPROVEMENTS

Several performance and scalability improvements for the assembly process and cluster functionality have been made to improve the handling of large models.

IMPROVED SELECTION LIST WINDOW

The Selection List window has been improved in several ways:

- For named selections under **Boundary selections**, for example, the context menu now includes a **Create Selection** submenu with a number of selections types that you can create based on the chosen named selections.
- With an active selection (in a selected node in the model tree) it is now possible to add to or remove the selection in the **Selection List** window to or from that active selection using a context menu like **Add to Geometric Entity Selection for Variables I**, for example.

- The **Create Selection** toolbar button in earlier versions has been renamed **Create Explicit Selection** and now also works when you are in the **Geometry** branch of the Model Builder.
- There are now Collapse All and Expand All buttons in the Selection List toolbar.
- By default, the **Selection List** window now appears to the right in the COMSOL Desktop.

IMPROVEMENTS TO THE AUXILIARY DATA WINDOW

The **Auxiliary Data** window now supports showing output files generated from, for example, data export features and reports. In connection with this improvement, the **Filter Type** toolbar button now includes new filter options: **Input** and **Output**. If **Input** is selected, only input file items will appear in the window. If **Output** is selected, only output file items will appear. Both button can also be selected or deselected at the same time to show both input and output files or to show nothing at all.

The actions available to perform on input versus output file items in the Auxiliary Data window are almost identical, with a few exceptions. The Import to Database action is only available for input files, and a new action is available for output files: Save as Version, which is available both in the context menu and in the toolbar. This action saves an existing output database file as a new version in a database. Click a data item in the Auxiliary Data window, and the Save as Version action should be available. Clicking it will bring up a Save Fileset window, where you can edit certain aspects of the file, such as its title and tags, and see what underlying file resources are associated with the file.

With the support to also show output files in the **Auxiliary Data** window, a new **Unsaved changes** status has been added. This status shows up for database files whose working copy changes have not yet been saved to a database. The **Unsaved changes** status means that the output feature has written new data to a file in a temporary folder on the server file system (the working copy of the database file), but that it is not yet saved to the database. You will have to manually save the file to the database in this case, either via the **Save as Version** option in the **Auxiliary Data** window or when saving the model via the **Save To** option in the main **File** menu. The unsaved output files will show up in the **Auxiliary Data** section in the fullscreen window.

In version 6.1, the **Status** window can include more than one status message for each data row.

Some table settings have new names in version 6.1:

- The Property column has been renamed to Setting.
- The Source column has been renamed to Location.

The Set Source action has been renamed to Browse From, and the Show Source Location has been renamed to Show Location.

NEW FULLSCREEN SELECT FILE, EXPORT, AND WRITE WINDOWS

A new **Browse From** option is now available for selected features that can generate output files in COMSOL Multiphysics. The **Browse From** option opens a fullscreen **Select File** window for selecting the output file, which can be either a new or existing file. When you select **Browse From** to use the **Select File** window, you can select a file from other sources than the file system, such as selecting a file from a database.

New **Export To** and **Write To** options are now available for features that export data — such as geometry, mesh, and results data — and for writing reports and presentations. They open fullscreen **Export** and **Write** windows, respectively, for selecting the location and filename for the exported or written data. In these windows, you can export and write files to other locations other than the file system. You can, for example, store the files in a database.

OTHER IMPROVEMENTS IN THE COMSOL DESKTOP

- In the **Preferences** dialog box, the categories on the left side are now displayed in a tree instead of as a list. Many preferences pages have been reorganized into different, often smaller, pages.
- The Model Builder and other trees that include Collapse All and Collapse Selected actions in their toolbars now also include a Collapse Other action. This action collapses all tree branches except the selected ones.
- When you have selected **Click and Hide** in the **Graphics** window toolbar, there is a new **Exit Click and Hide** option in the **Graphics** window context menu.
- Certain tables now include an **Insert Row** action. The action can be found in the table context menu and its toolbar. This action is supported by tables in the settings for **Parameters**, **Variables**, **Interpolation** and **Piecewise** functions, and **Polygon** and **Interpolation Curve** geometry features.

GEOMETRY FUNCTIONALITY

- Update of selections after a geometry rebuild (geometry associativity) has been improved.
- This release provides more robust 3D Boolean operations.
- The COMSOL Multiphysics part libraries have been upgraded to include additional and more versatile pipe geometries and 2D and 3D geometries for common repetitive microstructures, intended for homogenization using the representative volume element (RVE) technique (also called *representative element volume* or the *unit cell*).
- In the settings for **Revolve**, it is now possible to specify a straight edge as the revolution axis.
- You now enter the **Sketch** mode directly when you create a 2D component, geometry part, or work plane. With this, domains are not visualized and you can drag vertices and edges in the graphics. Click the **Sketch** button on the ribbon toolbar to exit the **Sketch** mode.
- Visualization of a construction geometry is now also available outside of the **Sketch** mode as a dashed edge visualization.
- Measuring the objects of a geometry now also lists the number of finite voids in the geometry.
- The operations **Extrude**, **Revolve**, and **Sweep** now also support virtual entities with collapsed edges and merged vertices.
- It is now possible to select between displaying a warning or an error when the condition of a **Parameter Check** is true.
- In the **Sweep** geometry feature, the direction of the sweep was wrong (reversed) in version 6.0 and earlier versions when the edge was created by a Boolean operation as an intersection of two faces using the COMSOL kernel. This bug has been fixed.
- The new tutorial *Busbar Assembly Geometry with Geometry Parts* has been added to demonstrate how to use geometry parts as a way to organize, parameterize, and reuse geometries that you create in the software.
- The new tutorial *Busbar Assembly Geometry with Group Nodes* has been added to demonstrate how to use group nodes in the model tree as a way to organize the geometry nodes that belong together.

- The new tutorial *Light Bulb Geometry* has been added to demonstrate how to create a geometry using Sketch mode to draw complex shapes.
- The new tutorial *Steam Reformer Geometry* has been added to demonstrate how to use selections when creating a parametric geometry for analysis and how to set up the selections needed for the physics definitions.

MESHING FUNCTIONALITY

- The new menu item **Cleanup and Repair** contains the operations **Fill Holes**, **Collapse Entities**, and **Merge Entities**. Use the new operation **Merge Entities** to merge points, edges, or boundaries in a mesh. This can be used for closing a gap between two disconnected components and for other types of repair and cleanup in a mesh.
- The new **Collapse Entities** can collapse edges or boundaries in a mesh. Use this operation to clean up a mesh of short edges or small faces.
- A new parameter, **Maximum angle for trimming**, enables you to control whether or not a boundary layer mesh is trimmed in narrow corners. This parameter is available regardless of the **Handling of sharp edges** setting.
- For the **Boundary Layer Properties** node, the new **Adjust directions for each layer** option makes it possible to change the direction layer by layer. This setting is on by default.
- The **Mapped** operation is now supported for meshes that define their own geometric model.
- Use the new **Vertex** operation to generate vertex mesh. This is useful for isolated points.
- The **Swept** operation is now supported for meshes that define their own geometric model. The operation has a new option, **Remesh source faces**, which enables remeshing of the source faces, when possible.
- The input tables to the **Create Vertices** and **Create Edges** operations are now automatically updated if the input mesh is changed. Selected chains of edges are mapped to new chains of edges, if possible.
- You can now use named selections for mesh operations in a mesh that defines its own geometric model. Named selections can be imported from a file, defined in a geometry sequence that you import into the mesh, or defined and used in the physics defining a **Filter** or **Partition** dataset that you import into the mesh.
- When importing mesh files of the STL, 3MF, PLY, or VRML formats, the new **Create domains** check box will make sure that domains (without mesh) are

automatically created within all finite void regions in the mesh. The same check box is also available for the **Fill Holes** and **Create Faces** repair operations.

- The Import operation now supports the import of geometry into a mesh. The Free Triangular and Visualization options specify how the faces of the geometry will be meshed before being imported.
- For the Fill Holes operation, Fill holes tolerance has changed to Maximum hole perimeter and the old Maximum hole perimeter has changed to Perimeter.
- For exporting mesh to the NASTRAN and COMSOL Multiphysics formats, the settings **Export as linear elements** and **Export as second-order elements** in earlier versions of COMSOL Multiphysics have been changed to use only an **Export as second-order elements** check box (clear it for linear elements).
- It is now possible to lock a mesh using the Model Builder context menu for mesh. The **Lock Mesh** option converts the current meshing sequence to a single **Mesh** node containing only its output mesh. The mesh is independent of any other data in the model. Use the **Disconnect from Geometry** option to break any connection with the geometry.
- The exact representation of circular edges are now used when creating geometry from mesh.
- For meshes that define their own geometric model, the smooth evaluation of curved and planar edges has been improved.
- The PLY file format import and export of quad elements have been improved.

New Functionality in the Physics Interfaces

PERIODIC CONDITIONS - DESTINATION SELECTION

The Settings window for all Periodic Condition and Periodic Flow Condition nodes has a new option in the context menu, where you can add a Destination Selection section to select the destination boundaries manually. This section is hidden when the automatic detection is used, but you can activate it for a manual selection by right-clicking the Periodic Condition or Periodic Flow Condition node and selecting Manual Destination Selection. You can select it again to switch off the manual destination selection.

There is also a new **Orientation of Destination** section that becomes active when the **Transform to intermediate map** settings under the **Orientation of Source** section is something other than **Automatic**.

EQUATION FORM FOR PDE AND ODE INTERFACES

The PDE and ODE interfaces now have an **Equation form** setting similar to the one found in physics interfaces as well as an accompanying equation display. This new setting does not affect the form of the equation itself. It only controls the interpretation of time derivatives of the dependent variables. Time derivatives are expanded to multiplications by 2*pi*i*freq when the equation form is set to **Frequency domain** or when it is set to **Study controlled** and the study step is of frequencydomain type. See also Equation Form for PDE and ODE Interfaces in the backward compatibility section below.

MOVING INTERFACE — PHASE FIELD THIN-FILM FLOW

The Phase Field Thin-Film Flow, Shell (pftffs) and Phase Field Thin-Film Flow, Edge (pftffs) interfaces have been combined and renamed Phase Field Thin-Film Flow (pftff).

MOVING MESH AND DEFORMED GEOMETRY

Boundary condition features controlling the boundary of a **Deforming Domain** node under **Moving Mesh** or **Deformed Geometry** now include the possibility to exclude lowerdimensional selections from the condition. This can be useful for resolving conflicts between adjacent boundary conditions at common edges or points, which can otherwise prevent finding a solution of the model or result in undesirable mesh deformation patterns.

A new **Prescribed Mesh Velocity** boundary condition is now available in the Moving Mesh and Deformed Geometry interfaces. You can use it on boundaries of domains with free displacement to specify the velocity of the boundary. The spatial frame in the adjacent domains moves in accordance with the velocity.

IMPROVED PDE INTERFACES

- Dirichlet Boundary Condition and Constraint nodes are now available on edge and point levels in 2D and 3D PDE interfaces.
- In the **More** context submenu for the domain level, a **Prescribed Solution** node is now available for prescribing the value of the solution on some domains in PDE interfaces.
- Curl elements (shcurl and shcurl2) are now available in 1D PDE interfaces.

UPDATED PERIODIC CONDITIONS

The **Periodic Condition** settings now include a new **Destination Selection** section where you can select the destination boundaries. This section is hidden when the automatic

detection is used, but the manual selection is activated using the **Manual Destination Selection** context menu, which can be switched on or off.

There is also a new **Orientation of Destination** section that becomes active when the **Transform to intermediate map** settings under the **Orientation of Source** section is something other than **Automatic**.

See also Periodic Conditions — Destination Selection in the *Backward Compatibility with Version 6.0* section below.

New and Updated Operators, Functions, and Definitions

LEAST-SQUARES FIT FUNCTION

You can now add **Least-Square Fit** function nodes under **Global Definitions** and under **Definitions** on the component level. A **Least-Square Fit** node defines a least-squares fit of a function based on the input data, which can be a file, a local table, or a result table. The function can be a general mathematical function with parameterized expressions.

GAUSSIAN PROCESS FUNCTION

The **Gaussian Process** function is intended for use with the Uncertainty Quantification Module. If you have a license for the Uncertainty Quantification Module, you can now add **Gaussian Process** function nodes under **Global Definitions** and under **Definitions** on the component level. The **Gaussian Process** function defines a Gaussian process, which is a surrogate model that gives an approximate prediction of the quantities of interest in an uncertainty quantification analysis.

INTERPOLATION FUNCTION IMPROVEMENTS

- In **Interpolation** function nodes, you can now choose **Function** from the **Data source** list to use another user-defined function as the data source. This functionality can be useful for creating primitive and inverse functions or for downsampling.
- For Interpolation functions defined on a component level, when requesting to define variables using spatial coordinates for a mesh frame, you can now select a new **Reinterpolate interpolation data on computational mesh** check box. Doing so makes it possible to evaluate variables using Lagrange interpolation with cached values evaluated at Lagrange points on the computational mesh when using an interpolation function to define spatially dependent variables.

NEW BUILT-IN OPERATORS AND FUNCTIONS

- The new built-in mathematical functions gcd and 1cm are now available for the greatest common divisor and least common multiple, respectively.
- A new built-in withparam operator is now available for accessing the solution parameters for the solution defined by a given solver tag.

New Functionality in Studies and Solvers

STUDIES AND STUDY STEP FUNCTIONALITY

- Improvements to the Time-Dependent Solver:
 - The RK34, Cash-Karp 5, Dormand–Prince 5, Adam-Bashforth 3, and Classical Runge–Kutta methods now support coupled systems of mixed-order time derivatives.
 - The time-dependent solver now also supports explicit time-stepping methods.
 - Note that the **Time-Explicit Solver** is no longer available in the Model Builder (unless you open a model that was created in an earlier version of COMSOL Multiphysics and that uses it).
- Improvements to the reduced-order model features and study steps:
 - An **Equation form** setting has been added to the stateful reduced-order model features and study steps.
 - Model control inputs in constraints are now also available for stateless reducedorder models.
- Improvements to the adaptive mesh refinement and error estimation:
 - The design for both stationary and time-dependent adaptation has been improved so that more specialized settings have been moved to the Adaptive Mesh Refinement and, when applicable, Error Estimation subnodes under the stationary, eigenvalue, and time-dependent solver nodes.
 - A new Advanced section is available in the settings for the Adaptive Mesh
 Refinement node. It includes a Use classic method for vector elements check box.
 If this is selected, the old 6.0 adaptive method will be used in which higher-order
 DOF names for vector elements are ignored in the residual calculation.
 Otherwise, a new adaptive method with a more rigorous calculation is used. The classical method exhibits faster mesh convergence for lumped parameters and is

therefore the default setting for the predefined adaptive mesh study in the RF Module and can be used for models using linear curl elements.

- A new option to not store error estimates for L2 and Functional adaptation is available in the Adaption and Error Estimates section in the settings for the Stationary study step.
- The Values of linearization point settings now support both physics controlled and user-controlled linearization points. It is also now available for the following study types: Eigenvalue, Eigenfrequency, Mode Analysis, Boundary Mode Analysis, Frequency Domain Perturbation, and Linear Buckling.
- There is now support for computing and storing left eigenvectors using the Eigenvalue solver.
- For the GCRO-DR methods in the iterative solver, there is a new Minimum number of restarts field, which is available when **Reuse subspace** is set to **Automatic**. In this field, you can specify a minimum value for the number of restarts. This can be useful to control in situations where a subspace exists between solver calls and needs to be cleared between solver calls for instance, in a frequency sweep.

CLUSTER AND BATCH FUNCTIONALITY

• New COMSOL cluster options on Linux:

The -mpiofiprovider option is now available for choosing an OFI (Open Fabrics Interface). This option is not supported for ARM processors.

A new option for the command-line option -scalapack: scalapack. It uses Netlib's ScaLAPACK.

• For COMSOL Hydra commands, fiinfo and fipingpong are now available for running fi_info and fi_pingpong commands, respectively.

New and Improved Results and Visualization Functionality

GENERAL NEW RESULTS AND VISUALIZATION FUNCTIONALITY

- The new **Add Predefined Plot** window contains, if applicable, plots that are predefined by the physics interfaces in the model but are not created directly when computing the solution. You can add such plots to plot groups under **Results** from the **Add Predefined Plot** window.
- In version 6.1, only the relevant postprocessing features, dependent on the physics in the current COMSOL Multiphysics simulation model, are visible. To show all features under **Results**, select the **All Dataset Types**, **All Plot Types**, **All Numerical**

Evaluation Types, and **All Export Types** check boxes in the **Show More Options** dialog box.

- Direct shadows are now supported when you run the COMSOL Desktop with OpenGL rendering, optimized for quality. You can enable direct shadows using shadow mapping in the Scene Light menu in the Graphics window's toolbar or in the Visual Effects section in 3D View nodes Settings window, where you can also control, among other properties, the shadow strength and softness. All lights in the 3D Graphics view cast shadows by default. You can turn it off by clearing the Cast shadows check box.
- Improved rendering performance, both in general and in particular for effects such as transparency and ambient occlusion.
- Anisotropic specular highlighting is now available in the **Material** nodes' **Settings** windows under **Appearance**. You add it by adding a value in the **Anisotropy** field (or using the associated slider). You can use it to give an effect that could be the result of machining or polishing in a certain direction, which is determined by the **Anisotropy axis** setting.
- You can now browse, search, and manage color tables using the new **Color Table** window, which you open by clicking the **Change Color Table** button next to the **Color table** setting in plot nodes. Using the **Color Table** window, you can search for color tables and get previews of the available color tables, which are grouped into categories, before switching to another color table. It is now also possible to store and load custom color tables in the model using the option to add **Color Table** nodes under **Results>Color Tables**. Such color tables appear under **In Model** in the **Color Table** window.
- The Internet Explorer-based .NET WebBrowser used to display in-line web and HTML pages in the .NET UI has been replaced with the Chromium[™]-based CefSharp browser component. This upgrade should be seamless and not affect you as a user. See Backward Compatibility with Version 6.0 for some compatibility information.
- For inserting Greek and other characters, a number of Greek symbols from the *textgreek* LaTeX package have been added for display as text in plots: \textalpha, \textbeta, and so on; \textAlpha, \textBeta, and so on; and some additional symbols.

NEW AND IMPROVED DATASET AND EVALUATION FUNCTIONALITY

• There is now support for adding selections to Cut Line datasets.

- For **Shell** datasets, there is now a **Displacement of normals** array input, where you can enter an additive correction to the shell's orientation that should be applied when evaluating in the spatial frame.
- The trees with predefined expressions that you open by clicking the **Insert Expression** or **Replace Expression** button now contain variables defined by the datasets.
- Using the new **Array Factor** dataset, you can define array factors for use in radiation pattern plots in RF simulations of antenna arrays, simplifying the analysis for a periodic antenna array. You can choose an **Array Factor** dataset as the dataset for the input data in **Radiation Pattern** plots.
- In the **Spatial FFT** dataset, you can now change the sign convention to suit the ones used for electromagnetic waves, for example.
- It is now possible to export WAV audio files with 32-bit floating point data.
- There is now an option for disabling interpolation between stored time steps in the **Time Integral** and **Time Average** datasets.
- You can now apply an out-of-plane wave number in Extrusion 2D datasets.

NEW AND IMPROVED GRAPHICS AND PLOT FUNCTIONALITY

- For 2D **Surface** plots, it is now possible to create discrete (banded) plots by choosing **Discrete** instead of **Continuous** from the new **Color table type** list. You can then also specify the number of bands.
- A new **Pellets** plot type is available for visualizing a flow of pellets or similar objects as spheres or as three orthogonal filled circles (slices).
- A new **Coordinate System Point** plot is available for showing the orientation of a source in postprocessing, such as plotting the coordinate system in a source point.
- You can now use the **Plot Previous**, **Plot Next**, **Plot First**, and **Plot Last** buttons for any level of a parameter or time in a multiparametric sweep. You can choose the level to switch to from a menu to the right of those buttons.
- The algorithm for finding and drawing the best streamlines has been improved for flat streamline plots in the 2D and 3D plot groups; that is, the rendering of streamlines has been improved for 2D **Streamline** and 3D **Streamline Surface** and **Streamline Multislice** plots.
- It is now possible to inherit the transparency and Fresnel transmittance between **Transparency** plot subnodes in a plot group.
- You can now place legends for 1D graph plots outside the graph. In the settings for **ID Plot Group** nodes, you can choose **Outside graph axis area** from the **Layout** list to choose a position outside the graph.

- A Show units option in now available in graph plot groups with Color Expression subnodes to show a unit above the color legend.
- For **Graph Marker** subnodes, you can now choose **Line intersection** from the **Display mode** list to display the intersections with vertical, horizontal, identity, or general straight lines, including the display of those lines.
- You can now add **Graph Marker** subnodes to **Line Graph**, **Octave Band**, and **Table Graph** plots.
- Color table files with RGB data can now use tabs, commas, or semicolons as separator characters, in addition to spaces.
- It is now possible to specify the point size in plots and plot subnodes that plot markers: Annotation plots; Max/Min Line, Max/Min Surface, Max/Min Volume plots; Table Annotation plots; Graph Marker plot subnodes; and Marker plot subnodes in 2D and 3D plot groups.
- There is now a setting for controlling the plot position and whether to show maximum and minimum values and units for color legends in graph plot groups.
- You can now specify a default for the line width in graph plots on the **Graph Plots** page in the **Preferences** dialog box. Doing so can be useful as a personal preference or for presentations using a projector; being able to increase the line width will improve the visibility, for example. The default line width is **Default from preferences** for a plot created from the COMSOL user interface and 1 for a plot created from the API. The default value of the preference is 1.
- In the **Selection** subnode for plot nodes, you can now make selections at geometric entity levels that are lower than the plot group's dimension (for example, selecting boundaries for a 3D plot). The selection is mapped from the chosen geometric entity level to the plot group's geometric entity level using adjacency mapping.
- It is now possible to disable inclusion of parameter tuples for solutions in automatic legends in the following graph plots: **Global**, **Point Graph**, and **Octave Band**. The new **Solution** check box in the **Legends** section is selected by default; clear it to make the legend text shorter.
- The Coordinate System plots can also be used for plotting an arbitrary matrix variable. For that purpose, it is now possible to choose **Columns** or **Rows** from the **Vectors to plot** list, when the coordinate system's plot source is a matrix variable. For example, plotting the deformation gradient's columns as a triplet of arrows gives a good picture of the local deformation and rotation of the material.

- The default marker position has been changed from **Interpolated** to **In data points**. The new default position, **In data points**, presents the data in a form that is closer to the underlying numerical solution than **Interpolation**.
- For Animation export features where the target is a file, a new HD option for the image size is available and provides animations with 1280-by-720 pixels.
- The line width in graph plots can now be specified as a preference on the **Results>Plot>Graph Plots** page in the **Preferences** dialog box.
- Surface Slit plots can now take any dataset as their input.
- There is now a **Rounded end caps** check box (selected by default), which you can clear to remove the rounded tube end caps in 2D and 3D **Line** plots using tubes and 2D and 3D **Tube Data** plots.
- To improve graph plot coloring, a **Long** graph cycle that contains 19 colors is now available. The graph cycle used in 6.0 and earlier versions is now called **Default**.
- The **Particle Trajectories** and **Ray Trajectories** plots now accept the following datasets as inputs:
 - Particle and Ray
 - Particle Bin and Ray Bin
 - Intersection Point 2D and Intersection Point 3D
 - Receiver 2D and Receiver 3D
 - Transformation 3D and Receiver 3D
- A Color table transformation list has been added to the settings for Whirl plots.
- The norm for the arrows (the arrow length) is now available as a built-in expression, Arrow length, for the Color data in Color Expression subnodes added to Arrow plots and to Particle Trajectories, Point Trajectories, and Ray Trajectories plots where the Point style is set to Arrow.

NEW AND IMPROVED REPORT AND PRESENTATION FUNCTIONALITY

- There is now support in the report generator for linked images in reports on the Microsoft Word[®] format.
- It is now possible to use custom labels on the title page of reports.
- The presentation of matrix-valued properties has been improved.
- Only material properties that are used in the model are included in the report.

- Reports on Method Call features now include the method code, if desired.
- The allocation of vertical space on presentation slides with mixed nonimage contents, for example text and equations, has been improved.

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 if such operations are applicable: stepPrevious, stepNext, stepFirst, stepLast; 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 get the number of levels using getStepLevels().

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

COMSOL Multiphysics version 6.1 can run Java[®] files saved from COMSOL Multiphysics versions 4.0–6.0. 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 6.1.

Backward Compatibility with Version 6.0

NEW CEFSHARP BROWSER

The Chromium[™]-based CefSharp browser component replaces the Internet Explorerbased .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\6.1\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 in 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 ID AND OD 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 either.

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:

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

Potential Issues

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

PHYSICS AND VARIABLES SELECTIONS

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

USER DOCUMENTATION NODES IN THE PHYSICS BUILDER

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

Backward Compatibility with Version 5.5

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

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

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

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

New Functionality in Version 6.1

MAGNET FEATURE FOR ELECTROMAGNETIC MACHINE MODELING

The **Magnet** feature can be used to model a magnetized domain or a regular pattern of magnetized domains. The direction of the magnetization can be specified by means of an expression, or by simply selecting the north and south pole boundaries. The feature comes with pattern recognition logic, making it easier to set up Halbach arrays and complicated rotor patterns. For formulations that support electrical currents, this feature makes it possible to impose electrical insulation on interior boundaries, making it easier to compute circulating currents and losses in segmented magnets.

The Magnet feature comes in two variants: a Nonconducting Magnet feature for the Rotating Machinery, Magnetic interface and for the Magnetic Fields, No Currents interface (for the latter, simply called Magnet), and a Conducting Magnet feature for the Rotating Machinery, Magnetic interface and for the Magnetic Fields interface (for the Magnetic Fields interface, simply called Magnet). In addition to support for interior electric insulation, the Conducting Magnet feature supports the Loss Calculation subfeature. Both variants support the Remanent flux density and Nonlinear permanent magnet constitutive relations.

MULTIPHASE WINDING FEATURE FOR ELECTROMAGNETIC MACHINE MODELING

The **Multiphase Winding** feature is used to model a regular pattern of coils, as typically used in electric motor modeling in 2D. Within this pattern, a coil or group of coils carrying the same current at the same phase angle is commonly referred to as a *phase*. The **Multiphase Winding** feature simplifies the excitation of a system of multiple phases by enforcing standard winding layouts. The feature uses automated pattern recognition logic to detect inconsistent configurations. For cases that do not follow a standard pattern, the feature supports a **User Defined** layout configuration.

The **Multiphase Winding** feature supports the **Loss Calculation** subfeature for automated determination of the resistive losses. It is available for the Rotating Machinery, Magnetic physics interface in 2D.

PASSIVE CONDUCTOR FEATURE FOR THE MAGNETIC FIELDS INTERFACE

The **Passive Conductor** feature in the Magnetic Fields interface makes it possible to impose electrical insulation on interior boundaries, making it easier to compute circulating currents and losses in segmented conductors. The feature shows many similarities with the **Conducting Magnet**, with the difference that this feature only supports B-H constitutive relations that do not include the magnetization: **Relative permeability**, **B-H curve**, and **Effective B-H curve**. Like the magnet, the **Passive Conductor** feature supports the **Loss Calculation** subfeature for determining the resistive losses.

PASSIVE CONDUCTOR FEATURE FOR THE MAGNETIC FIELDS, CURRENTS ONLY INTERFACE

The **Passive Conductor** feature in the Magnetic Fields, Currents Only interface is a simplified version of the **Conductor** feature from the same interface. It does not require the specification of terminal and ground boundaries and is not swept over by automated source sweep studies; it is used to model a conductor without feeding. The feature supports an **Electric Insulation** subfeature, which imposes electrical insulation on interior boundaries. This functionality makes it easier to compute circulating currents and losses in segmented conductors.

MAGNETOHYDRODYNAMICS MULTIPHYSICS INTERFACES

The new Magnetohydrodynamics Multiphysics interface consists of the Magnetic Fields interface (or the Magnetic and Electric Fields interface), the Laminar Flow interface, and a **Magnetohydrodynamics** multiphysics coupling. The coupling applies the Lorentz force from the electromagnetics interface to Laminar Flow, and — in return — applies the Lorentz velocity term from Laminar Flow to the electromagnetics interface.

The multiphysics interface comes in three variants: 2D with out-of-plane electric currents, 2D with in-plane electric currents, and 3D. The 2D variant with out-of-plane currents uses the Magnetic Fields interface, the other two use the Magnetic and Electric Fields interface. All three variants are available with the AC/DC Module only. The multiphysics coupling feature can be used separately as well, and is available with the AC/DC Module and the Plasma Module.

LIQUID METAL MATERIAL LIBRARY FOR MAGNETOHYDRODYNAMICS

The AC/DC material library has been extended with a **Liquid Metals** folder for magnetohydrodynamics (MHD) modeling. This new folder contains materials with material properties for the thermal conductivity, electrical conductivity, dynamic

viscosity, and density of various liquid metals, including titanium, steel, iron, nickel, copper, aluminum, magnesium, tin, lithium, sodium, and others.

TIME-DOMAIN SUPPORT FOR THE MAGNETIC AND ELECTRIC FIELDS

The Magnetic and Electric Fields interface now has support for the **Time Dependent** study type. Furthermore, the default exterior boundary condition has been updated from **Magnetic Insulation** with **Electric Insulation**, to **Magnetic Insulation** with **Ground**, making it consistent with the default **Magnetic Insulation** boundary condition in the Magnetic Fields interface.

COVARIANT FORMULATION IN 2D AXISYMMETRY

The Magnetic Fields interface and the Magnetic and Electric Fields interface now use the *covariant formulation* in 2D axisymmetry. The covariant formulation maps the solution on the shape functions differently and is much better able to handle the inherently singular properties of the axis in cylindrical coordinate systems. The covariant formulation gives better performance, better numerical stability, and better accuracy.

MAGNETIC FIELDS FORMULATION MAGNETIC FIELDS NO CURRENTS COUPLING

The Magnetic Fields Formulation Magnetic Fields No Currents Coupling assures continuity of the normal magnetic flux density and the tangential magnetic field across boundaries between the Magnetic Fields, No Currents interface and the Magnetic Field Formulation interface. This new multiphysics coupling is especially suitable within the context of *superconductor modeling*.

LAYERED IMPEDANCE BOUNDARY CONDITION FOR THE MAGNETIC FIELDS INTERFACE

The Layered Impedance Boundary Condition is an extension of the Impedance Boundary Condition, which makes it possible to model a sequence of geometrically thin layers on top of a substrate. It is used on exterior boundaries where the field is known to penetrate only a short distance outside the boundary. In brief, this feature combines a Layered Transition Boundary Condition with an Impedance Boundary Condition. This new feature is available for the Magnetic Fields interface.

CIRCUIT EXTRACTION IMPROVEMENTS

The **Circuit Extractor** add-in converts matrices of lumped quantities into electrical circuits. These circuits can then be used as a lumped representation of an electromagnetic device. Typically, a finite element model is subjected to a **Source Sweep** study and the lumped matrices are extracted. These matrices are then fed into the **Circuit Extractor**. The resulting circuit is validated by comparing it with the original finite element model. Once validated, the circuit can be used as a lumped representation for cases where the finite element model would be computationally demanding. The method can be seen as a physics-based alternative to *reduced-order modeling* (ROM) or *vector fitting*.

The Electrostatics; Electrostatics, Boundary Elements; and Electric Currents interfaces now produce capacitance (and resistance) matrices in a format that is directly compatible with the **Circuit Extractor** add-in. The Magnetic and Electric Fields interface now supports the **Stationary Source Sweep** and the **Frequency Domain Source Sweep** study types and generates impedance, resistance, and inductance matrices in a format that is compatible with the **Circuit Extractor** add-in. The **Circuit Extractor** itself has now been extended to support impedance matrices.

MAGNETOSTRICTION IMPROVEMENTS

The old **Magnetostriction** multiphysics coupling has been split into a **Nonlinear Magnetostriction** coupling and a **Piezomagnetic Effect** coupling (the latter is also known as *linear magnetostriction*).

Together with the new multiphysics couplings, two new Ampère's Law variants (Ampère's Law, Nonlinear Magnetostrictive and Ampère's Law, Piezomagnetic) and two new multiphysics interfaces (Nonlinear Magnetostriction and Piezomagnetism) have been introduced. The new multiphysics interfaces are based on the Magnetic Fields interface together with the Solid Mechanics interface. The new Ampère's Law, Nonlinear Magnetostrictive feature supports the Loss Calculation subfeature. This subfeature allows for automated determination of resistive and magnetic losses in laminated iron, using empirical loss models, such as Steinmetz or Bertotti.

A new constitutive relation, **Analytic magnetization curve**, is available in the ordinary **Ampère's Law** feature (given that the **Material type** of this feature has been set to **Solid**). The new multiphysics couplings, new **Ampère's Law** features, and new constitutive relation are available with the Magnetic Fields and the Rotating Machinery, Magnetic interfaces. The couplings and the dedicated features require the AC/DC Module,

together with either the Structural Mechanics Module, Acoustics Module, or MEMS Module. The new constitutive relation only requires the AC/DC Module.

COIL IMPROVEMENTS

For the **Coil** feature, the **Homogenized multiturn** conductor model has been updated. The coil wire cross-sectional area can now be derived from the slot **Filling factor**. The thickness of the wires is then derived from the area of the selected domains and the desired relative amount of copper in the coil cross section. This setting is frequently used in motor modeling.

New and Updated Models in Version 6.1

GENERAL APPLICATION LIBRARY RESTRUCTURING

The AC/DC Application Library has been restructured to provide a better overview of models per application area, per level of difficulty, or per used physics.

ELECTRIC FIELD FROM POWER LINES

Power lines are commonly used as a means of transmitting electrical power across large distances. In this tutorial, towers transmitting high-voltage three-phase AC power are modeled, and the resulting electric field is computed. The towers have two shielding lines above the phase lines, which protect the tower from lightning strikes.

This new model also illustrates the strength of using the boundary element method (BEM) for modeling, which is used by the Electrostatics, Boundary Elements interface.

MAGNETIC FIELD FROM POWER LINES

Power lines are commonly used as a means of transmitting electrical power across large distances. In this tutorial, towers transmitting high voltage three-phase AC power are modeled, and the resulting magnetic field is computed. The towers have two shielding lines above the phase lines, which protect the tower from lightning strikes.

This new model uses the Magnetic Fields interface to compute the magnetic field.

ELECTRIC FIELD BETWEEN CONCENTRIC CYLINDERS

This new introductory model treats the electrostatics problem of two concentric cylinders of infinite length, which is commonly found in textbooks. Since the problem can be solved analytically, the model can be used to compare theory with numerical simulation results. Two cases are considered: one with a fixed potential on each

cylinder and the other with a surface charge density on one cylinder and a fixed potential on the other.

MAGNETIC FIELD FROM AN INFINITE CONDUCTOR

This new introductory model creates a simple model of the magnetostatics problem with a wire of infinite length carrying current, which is commonly found in textbooks. Since there is an analytical solution to this problem, the model can be used to compare theory with numerical results from the simulation. The wire still has a finite radius in this example, giving different solutions for the regions inside and outside of the wire.

HARTMANN BOUNDARY LAYER

This new introductory magnetohydrodynamics model illustrates the effect of an external magnetic field on the laminar flow profile of a conducting fluid. The fluid velocity transverse to the magnetic field induces volumetric currents via the electromotive force, creating a Lorentz force on the fluid in the direction opposite the flow direction. The fluid velocity profile between two no-slip walls are compared with analytical values for different values of the *Hartmann number*.

MAGNETOHYDRODYNAMICS PUMP

This new introductory model illustrates the action of a magnetohydrodynamics pump. A moving magnetic field is created by three-phase coils, and the contrary induced magnetic field due to eddy currents in the liquid creates a repelling force on the liquid in the column, generating motion. This type of induction pump is used in, for example, high-temperature cooling systems, where the lack of moving parts and the pumped liquid being kept in a hermetically sealed system are clear advantages.

MAGNETIC BRAKE

This model has been updated to use the new time-domain support for the Magnetic and Electric Fields interface.

PERMANENT MAGNET MOTOR IN 2D

This model has been updated to use the new **Magnet** feature (with the **Loss Calculation** subfeature), and the new **Multiphase Winding** feature. It also uses the new **Filling Factor** option for specifying wire thickness.

ADDITIONAL UPDATED MODELS

The following models have been updated to use the new **Evaluation Groups**, used for lumped matrix extraction (see *Circuit Extraction Improvements* above for more details):

- Modeling a Capacitive Position Sensor Using FEM/BEM
- Tunable MEMS Capacitor

The following models have been updated to use the new **Magnet** feature (and in most cases the corresponding **Loss Calculation** subfeature):

- Generator in 2D
- Linear Motor in 2D
- One-Sided Magnet and Plate
- Permanent Magnet
- Permanent Magnet Motor in 3D
- Quadrupole Lens
- Rotating Machinery 3D Tutorial,
- Modeling of an Electric Generator in 3D
- Static Field Modeling of a Halbach Rotor

Backward Compatibility with Version 6.0 and Earlier

The old Magnetostriction multiphysics coupling has been split into a Nonlinear Magnetostriction coupling and a Piezomagnetic Effect coupling (the latter is also known as *linear magnetostriction*). Old Magnetostriction multiphysics couplings and old Ampère's Law, Magnetostrictive features in existing models will continue to work but will show an *obsolete functionality* warning.

Acoustics Module

New Functionality in Version 6.1

NEW ACOUSTIC STREAMING MULTIPHYSICS

The two new Acoustic Streaming from Pressure Acoustics and Acoustic Streaming from Thermoviscous Acoustics multiphysics interfaces are used to model the physical phenomenon acoustic streaming, which is when an acoustic field induces a fluid flow. The multiphysics couplings **Acoustic Streaming Domain Coupling** and **Acoustic Streaming Boundary Coupling** are used to compute the forces, stresses, and boundary slip velocities that the acoustic field induces in the fluid, generating flow. The interfaces couple a frequency-domain acoustic field to a stationary or time-dependent fluid flow. Acoustic streaming is important in microfluidics and lab-on-a-chip systems for applications such as particle handling, mixing of fluids, and microfluidic pumps.

PHYSICS-CONTROLLED MESH FOR ACOUSTICS PHYSICS

The generation of a physics-controlled mesh has been extended to more acoustics interfaces. A physics-controlled mesh results in the generation of a good initial mesh that fulfills best meshing practices, resolves the wave phenomena, and creates boundary layers. Physics-controlled mesh has been added for:

- · Pressure Acoustics, Boundary Element
- Pressure Acoustics, Kirchhoff-Helmholtz
- Pressure Acoustics, Asymptotic Scattering
- Pressure Acoustics, Time Explicit
- Nonlinear Pressure Acoustics, Time Explicit
- Elastic Waves, Time Explicit
- Thermoviscous Acoustics, Frequency Domain
- Thermoviscous Acoustics, Transient

THERMOVISCOUS-THERMOELASTICITY BOUNDARY MULTIPHYSICS COUPLING

Two new multiphysics interfaces, the Thermoviscous Acoustic-Thermoelasticity Interaction, Frequency Domain and the Thermoviscous Acoustic-Thermoelasticity Interaction, Transient interface, combine the Thermoviscous Acoustics, Frequency Domain or Transient interface with the Thermoelasticity multiphysics interface using the new **Thermoviscous Acoustic–Thermoelasticity Boundary** multiphysics coupling. The multiphysics interfaces solves for and has predefined multiphysics couplings between the displacement and temperature field in the solid and the acoustic variations in pressure, velocity, and temperature in the fluid domain.

It can be used to, for example, model the vibrating response of MEMS devices, including detailed description of the damping. The multiphysics interfaces are available in 3D, 2D, and 2D axisymmetric geometries.

The functionality requires both the Acoustics Module and the MEMS Module

LUMPED SPEAKER BOUNDARY AND INTERIOR LUMPED SPEAKER BOUNDARY FOR PRESSURE ACOUSTICS, TRANSIENT

The Lumped Speaker Boundary and the Interior Lumped Speaker Boundary conditions have been added to the Pressure Acoustics, Transient interface to model hybrid lumped–FEM loudspeaker setups. The condition sets up couplings between the boundary condition and an Electrical Circuit interface. This makes it possible to set up models that can include the large signal parameters like $C_{MS}(x)$, BL(x), or $R_{MS}(v)$ in a simplified way. Predefined global variables exist for the axial position and velocity.

LUMPED SPEAKER BOUNDARY AND INTERIOR LUMPED SPEAKER BOUNDARY FOR THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN AND TRANSIENT

The two new boundary conditions, **Lumped Speaker Boundary** and **Interior Lumped Speaker Boundary**, have been added to Thermoviscous Acoustics in the frequency domain and in the time domain. This completes and extends the existing conditions available in pressure acoustics. The setup and modeling of microspeakers with hybrid lumped–FEM representations is now simple in thermoviscous acoustics. The lumped representations are also often accurate over a larger frequency range for microtransducers as breakup effects occur outside the intended frequency range. In the time domain, nonlinear effects can be included through large signal parameters like $C_{MS}(x)$, BL(x), or $R_{MS}(v)$.

LUMPED PORT IN THERMOVISCOUS ACOUSTICS

The **Lumped Port** feature connects a waveguide or duct inlet or outlet to a lumped representation element. This can be an **Electrical Circuit** (lumped electroacoustic representation), a two-port network defined through a transfer matrix, or a lumped representation of a waveguide. Basically, it couples the end of a waveguide with an exterior system that has a given acoustic lumped representation. Several representations and sources exist for describing the lumped system. When using the

lumped port representation, it is assumed that only plane pressure waves — that is, the (0,0)-mode — propagate in the acoustic waveguide. The condition ensures a mathematically and physically consistent coupling, including the thermoviscous boundary layers in the waveguide.

NEW CONVECTED ACOUSTIC-STRUCTURE BOUNDARY, TIME EXPLICIT MULTIPHYSICS COUPLINGS

The new **Convected Acoustic-Structure Boundary, Time Explicit** and **Pair Convected Acoustic-Structure Boundary, Time Explicit** multiphysics couplings are used to couple the Convected Wave Equation, Time Explicit interface with the Elastic Waves, Time Explicit interface. The conditions are added on the boundary or pair selection between the fluid domain and the solid domain for modeling acoustic–structure interaction (ASI) or vibroacoustic problems in the presence of a stationary background flow.

NEW FRACTURE BOUNDARY CONDITION IN ELASTIC WAVES, TIME EXPLICIT

The new **Fracture** condition, available with the Elastic Waves, Time Explicit physics interface, is used to treat two elastic domains with imperfect bonding. The fracture can be, for example, a thin elastic layer, a fluid-filled layer, or it can be a discontinuity in elastic materials (an interior boundary). Several options exist for specifying the properties of the thin elastic domain. Typical applications are for modeling nondestructive testing (NDT) applications, like the response of delamination regions or other defects, or for modeling wave propagation in fractured solid media in the oil and gas industry.

PIEZOELECTRIC WAVES, TIME EXPLICIT NEWS

Improved accuracy and efficiency when solving models involving piezoelectric effects using the time-explicit method. The performance when solving large models on a cluster architecture has also been highly improved to take full advantage of the distributed computing.

PRESSURE ACOUSTICS, TIME EXPLICIT NEWS

The **Impedance** conditions now use an improved numerical flux formulation for stability, ensuring that both acoustically hard and soft boundaries result in a stable solution.

Two new conditions have also been added to model a transfer impedance setup: the **Interior Impedance** and the **Pair Impedance** conditions. Both conditions also take advantage of the improved numerical flux from the impedance condition.

ELASTIC WAVES, TIME EXPLICIT NEWS

In 2D models, the underlying equations have been reformulated to be more efficient for these specific cases. In 2D models, there is now a plane strain option for the out-of-plane component, which you activate by selecting the **Include out-of-plane** components check box in the **2D Approximation** section. The formulation in 2D axisymmetric models has also been improved. As an example, the number of degrees of freedom (DOFs) solved for in the Seismic Waves Earth tutorial model have been reduced from $17 \cdot 10^6$ to $12 \cdot 10^6$.

NEW TRANSFER MATRIX COUPLING BOUNDARY CONDITION IN PRESSURE ACOUSTICS, FREQUENCY DOMAIN

The new **Transfer Matrix Coupling** boundary feature is used to couple two boundaries (source and destination) using a transfer matrix representation. The transfer matrix is a reduced or lumped representation of the physical domain, connecting the two boundaries. The feature has two options, a so-called pointwise coupling and a lumped representation.

EXTENDED THERMAL CONDITIONS IN THE THERMOVISCOUS BOUNDARY LAYER IMPEDANCE CONDITION

A new **Thermally Conductive Wall** option exists in the **Thermoviscous Boundary Layer Impedance** condition. The new option makes it possible to model nonideal thermal isothermal or adiabatic wall condition using different analytical representations of walls of finite thickness or infinite walls.

New variables for the evaluation of the combined dissipated and transported energy (including convective terms) in the boundary layers have been added. The variables are useful when modeling heating and not only dissipation.

SPEEDUP OF EVALUATIONS BASED ON THE KIRCHHOFF-HELMHOLTZ KERNEL

Features that rely on the evaluation of the Kirchhoff–Helmholtz integral are now evaluated up to 50% faster compared to the previous version (the value depends on the hardware and the complexity of the plot — more gain is achieved as complexity increases). This is in the **Exterior Field Calculation** feature that is used in pressure acoustics for plotting the exterior field in results. In the high-frequency acoustics interfaces — Pressure Acoustics, Asymptotic and Pressure Acoustics, Kirchhoff-Helmholtz — this is, in particular, important as the actual computation time lies in the evaluation of the kernel. As an example, the evaluation time of the last plot in the Submarine Asymptotic Scattering tutorial has decreased by 25%.

NEW POROUS LAYER IMPEDANCE OPTIONS FOR PRESSURE ACOUSTICS

The **Porous Layer** option in the **Impedance** boundary condition has been updated to include options to handle a certain degree of angle dependency of the impedance. The angle of incidence can be controlled to be normal to the surface or to use a specific angle or direction. An automatic option assigns an effective angle of incidence, which is useful when simulating, for example, room acoustics simulations with diffuse acoustic fields.

RAY ACOUSTICS IMPROVEMENTS

Export of Balloon Data with Method Call

The Application Library model *Loudspeaker Driver in a Vented Enclosure* now includes a **Method** and a **Method Call** that enable the export of the loudspeaker radiation data (balloon plot) in a format suited for further use. This is an example of how custom export can be made using a method and the Application Builder.

Improved Pseudorandom Number Generation

The Ray Acoustics interface includes several features that rely on pseudorandom number generation (PRNG), such as conditional ray–boundary interaction and diffuse or isotropic scattering at boundaries. For such features, the use of pseudorandom number generators have been extensively reviewed and revised. The new expressions are much less likely to incur unwanted correlations between random numbers that should be completely uncorrelated. This includes random boundary conditions acting on different rays, as well as unwanted correlations between different randomly generated vector components.

Option to Only Store Accumulated Variables in Solution

Depending on your application, the accumulated variables (such as sound pressure level on boundaries) might be more valuable information than the position and direction of individual rays. To reduce file size, you now have the option to only retain the accumulated variables in the solution while discarding the degrees of freedom associated with the rays.

Release From Exterior Field

The **Release From Exterior Field** feature can now pick up exterior fields from **Pressure Acoustics, Kirchhoff Helmholtz** and **Pressure Acoustics, Asymptotic Scattering**. Moreover, the feature now handles solving for several frequencies in a parametric sweep.

FLOW-INDUCED NOISE IMPROVEMENTS

The excess pressure terms are now included in the Lighthill stress tensor. These describe the deviations from linear isentropic behavior, for example, if strong nonlinear effects occur in the source region or if a heat source is present in the flow simulation.

The **Aeroacoustic Flow Source Coupling** multiphysics feature now allows the source to be taken from the new Detached Eddy Simulation (DES) fluid flow interface. The Large Eddy Simulation (LES) fluid flow interface has several new features, including a **Synthetic Turbulence** option for inlets. See further details in the CFD Module release text.

PREDEFINED PLOT SUGGESTIONS

Several predefined acoustics plots have been to the **Add Predefined Plot** menu in the **Results** section. The new predefined plots automatically set up useful plots for several situations:

- Plots for multiphysics configurations showing the pressure or sound pressure level, for coupled pressure and thermoviscous acoustic or pressure acoustic and poroelastic waves models.
- A plot of the cell-wave time scale, when solving models based on the discontinuous Galerkin time-explicit formulation. This plot is useful for identifying problematic mesh regions that limit the internal time stepping.

SOLVER SUGGESTION IMPROVEMENTS

Several new iterative solver suggestions have been added and improvements have been made to the existing generated solver suggestions. The most important are:

- For Pressure Acoustics, Frequency Domain models, the suggested iterative solver based on the shifted Laplace method has been improved for faster convergence.
- For Thermoviscous Acoustics models, the iterative solver suggestion based on the domain-decomposition method has been improved to use the latest solver technology. The solver is now a good choice for solving larger models.

Solver suggestions have also been added for when you are solving 3D vibroelectroacoustic models such as loudspeakers and other transducers. In particular, an efficient iterative solver suggestion is given when coupling acoustics (pressure and thermoviscous acoustics), structures (solid and/or shell), and the Magnetic Fields physics using the **Lorentz Coupling** or the **Magnetomechanical Forces** multiphysics couplings.

The following models are new in version 6.1:

- · Prestressed Micromirror Vibrations: Thermoviscous-Thermoelasticity Coupling
- One Family House Acoustics
- Acoustic Microfluidic Pump
- Acoustic Streaming in a Microchannel Cross Section
- Lumped Loudspeaker Driver Transient Analysis with Nonlinear Large Signal

UPDATED MODELS

The following models have been updated:

- Angle Beam Nondestructive Testing: The model uses the new **Fracture** feature to represent the defect.
- Flowmeter with Piezoelectric Transducers: The model now includes a background flow and uses the new **Pair Convected Acoustics-Elastic Waves Boundary, Time Explicit** multiphysics coupling.
- Small Concert Hall: The model includes an additional **Source with Directivity** feature that imports the directivity from external data.
- Loudspeaker Driver in a Vented Enclosure: The model includes a **Method** and **Method Call**, which make it possible to export the loudspeaker radiation data (balloon plot) in a format suited for further use.

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:

- Type 4.3 Ear Simulator
- Cavity Flow Noise
- Electrostatic Speaker Driver
- 3D Acoustic Trap and Thermoacoustic Streaming in Glass Capillary
- Simple Thermoacoustic Engine
- Transient Sound Pressure Level
- Topology Optimization of a Sound Partition Considering Acoustic-Structure Interaction

- In certain cases, a wrong sign could be used in the **Perfectly Matched Boundary**, leading to bad absorption performance; this has now been fixed.
- The coupling between the Thermoviscous Acoustics, Frequency Domain and Transient physics interfaces and the Shell and Layered Shell physics interfaces has been improved. In particular, the coupling is now done to the top or bottom displacement components instead of simply to the midplane.
- The **Biot-Allard (thermal and viscous losses)** option in the Poroelastic Waves physics interface has been improved to handle low-porosity materials correctly.
- The area in the **Lumped Port** and **Port** conditions now takes symmetries into account for a correct scaling of transfer matrices and other area-dependent postprocessing variables.
- The default for the **Typical wave speed for perfectly matched layers** values, in all the acoustic physics interfaces, now uses a value taken from the physics (which comes from the material data).
- Physics symbols have been introduced for the symmetry axis in the **Sector symmetry** options in the **Exterior Field Calculation** feature.
- The export to WAV file format now supports export with 32-bit float encoding.

In the **Impulse Response** plot, the 1/3 and 1/6 octave filters were slightly misaligned, leading to small spectral errors when reconstructing the impulse response; this has been fixed.

Backward Compatibility with Version 6.0

- In Pressure Acoustics, Frequency Domain, using the **Port** and **Lumped Port** conditions together with symmetries works automatically since version 6.0 update 2. The correct effective port area is automatically calculated and used. In version 6.0 update 1 (and earlier), an additional area scaling factor had to be added manually to a range of postprocessing variable, like the total power, volume flow, or the two-port parameters, to get correct results. This manual scaling is no longer needed.
- An automatic calculation of the speaker area multiplication factor has been introduced in the Lumped Speaker Boundary and Interior Lumped Speaker Boundary features in Pressure Acoustics, Frequency Domain. To retrieve the old behavior when Use symmetries is selected, set the Speaker area multiplication factor to User defined:

model.component("comp1").physics("acpr").feature("lsb1").set("Mul tiplicationFactor", "UserDefined");

- In Pressure Acoustics, Frequency Domain, the **Porous Layer** option in the **Impedance** boundary condition is no longer compatible with the **Anisotropic Acoustics** material model. Model the boundary condition as a domain with Poroacoustics instead.
- The default **Reference pressure for the sound pressure level** has changed in the Pressure Acoustics interfaces; the Linearized Navier-Stokes, Frequency Domain interface; and Linearized Potential Flow, Frequency Domain interface. To retrieve the old behavior for interface acpr, add the following line after creating the interface:

```
model.component("comp1").physics("acpr").prop("cref").set("cref",
"343[m/s]");
```

Battery Design Module

New Functionality in Version 6.1

GENERAL NEW FUNCTIONALITY IN THE ELECTROCHEMISTRY INTERFACES

- Fixed an issue with the **Electrode Power** boundary feature for the **Current Distribution Initialization** study step if the dependent variable name of phis has been changed. Also, the field name of the Lagrange multiplier has been changed to include the physics tag. This name change may cause backward compatibility issues when running the Java API.
- The **Continuity** Pair condition has been improved in the Primary/Secondary Current Distribution, Lithium-Ion Battery, and Battery with Binary Electrolyte interfaces.
- The charge settings have been moved to the new **Species Properties** node for the Tertiary Current Distribution, Nernst–Planck interface.
- Nonideal activities are now supported in the Tertiary Current Distribution, Nernst– Planck interface. Functionality has also been added for computing the nonideal activities using the Debye–Hückel equation, based on the ionic strength of the solution.
- For all electrochemistry interfaces, the new 0.25 V (Seawater Ag/AgCl vs. SHE) option has been added to the Physics vs. Materials Reference Electrode Potential section of the physics interface node.

NEW BATTERY-SPECIFIC FUNCTIONALITY

- Added the entries Electrolyte>LiPF6 in 1:1 EC:DMC (Liquid, Li-ion Battery) and Electrode>NMC 532, LiNi0.5Mn0.3Co0.2O2 (Positive, Li-ion Battery) to the Battery Material Library.
- New Battery Pack interface.
- New Specific Media>Battery Layers domain node in the Heat Transfer interfaces.
- Added new postprocessing variables (surface and average particle concentrations weighted over porous material) in the Lithium-Ion and Battery with Binary Electrolyte interfaces.
- Modified handling of the Battery Cell Capacity variable in the Lumped Battery interface. With the new formulation, the available capacity for cycling will be reduced as the battery capacity is reduced.
- The **Parameter Estimation** study step, and the BOBYQA, Levenberg-Marquardt, and IPOPT optimization solvers are now available with a Battery Design Module license.
- Fixed an issue with **Charge-Discharge Cycling** boundary node when using a **Current Distribution Initialization** study step if dependent variable name of phis has been changed. This fix was also included in version 6.0 update 2.

ADVANCED CHEMICAL FORMULAS IN THE CHEMISTRY INTERFACE

The process of adding new chemical species, both when manually creating **Species** features and when taken from **Reaction** features, has been significantly improved and extended.

- The **Name** section of a **Species** node now includes both the species name and the variable name used for postprocessing. These names differ for charged species, species with parentheses, and for species with a phase indication.
- The **Species** node now includes a **Chemical Formula** section where the composition of the species can be defined in terms of elements from the periodic table (by selecting the **Enable formula** check box). The molar mass and species charge are automatically computed from the formula.
- The **Chemical Formula** section supports repeating units using three types of enclosing marks: (), [], and {}. One example of this is [Al(OH)(OH2)5]^2+.
- The **Chemical Formula** section supports the following charge representations: ++, (++), ^(++), ^++, ^2+, (2+), and ^(2+).

- For intricate molecular structures, a simplified (trivial) name can now be used, while the composition and charge are detailed using the **Chemical Formula** input. When a **Chemical Formula** is defined, it is used in place of the simplified name when balancing a reaction formula.
- The reaction formula parser used in the **Reaction** node has been extended to also support species names with repeating units and the extended set of charge representations.
- The error messages shown for invalid species names have been improved.

IMPROVEMENTS TO THERMODYNAMICS

- Constant properties defined by the **Thermodynamics** functionality can now be evaluated as parameters.
- Evaluation of thermodynamic properties is now faster. Models where a significant part of the solution time is spent evaluating functions from the **Thermodynamics** functionality can now be solved in 30% less time.

The filter search field, used when adding species to a thermodynamic system, has been improved. Filtered species can now be entered one at a time, from the top, using the Enter key. Also, it is no longer necessary to update the filter each time a species is added to the **Selected species** list.

GENERAL NEW FUNCTIONALITY IN THE CHEMICAL TRANSPORT INTERFACES

- The charge and molar mass settings have been moved to the new **Species Properties** node for the Transport of Concentrated Species interface. Java API back comp is provided for the charge setting commands, but not for the molar mass setting commands.
- The charge settings have been moved to the new **Species Properties** node for the Nernst–Planck Equations and Transport of Diluted Species interfaces.
- Nonideal activities are now supported in the Nernst–Planck Equations and Transport of Diluted Species interfaces. Functionality has also been added for computing the nonideal activities using the Debye–Hückel equation, based on the ionic strength of the solution.
- Consistent stabilization is now available when using the **Maxwell-Stefan** diffusion model in the Transport of Concentrated Species interface.
- The Maxwell–Stefan diffusion model can be combined with turbulent flow when using the Reacting Flow multiphysics interfaces.

- The **Periodic Condition** for mass transport can now be combined with wall conditions for fluid flow when using the **Reacting Flow** or **Reacting Flow**, **Diluted Species** coupling feature. In the case of turbulent flow, the wall concentration is made periodic by applying the constraint to the wall function.
- Interior fan conditions are now supported by the **Reacting Flow** and **Reacting Flow**, **Diluted Species** coupling features.
- A new **Fracture** feature, consistent with other features for porous media transport, is now available. The feature includes two subnodes, **Fluid** and **Fracture Material**, including settings for the respective phases.

New and Updated Applications and Models in Version 6.1

- New Jelly roll (*jelly_roll*) and Jelly Roll on a Flattened Geometry (*jelly_roll_flattened*) tutorials.
- New Thermal Runaway Propagation in a Battery Pack (thermal_runaway_propagation) tutorial.
- New Zinc Bromine Redox Flow Battery (*znbr_flow_battery*) tutorial.
- The Thermal Distribution in a Pack of Cylindrical Batteries Tutorial has been updated (and the filename has changed from *lumped_li_battery_pack_6s2p* to *lumped_li_battery_pack_6s2p*). The updated tutorial now uses the new Battery Pack interface, which includes the potential drop in the electric conductors, and the associated Joule heating. The model now also uses the new **Battery Layers** node in the Heat Transfer interface. The meshing sequence has also been improved.
- The Liquid-Cooled Lithium-Ion Battery Pack (*li_battery_pack_3d*) tutorial has now been updated to use the new Battery Layers node in Heat Transfer.
- The Lithium Battery Pack Designer (*li_battery_pack_designer*) app has been updated to use the new **Battery Layers** node in the Heat Transfer interface.

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

New Functionality in Version 6.1

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 COMSOLnative geometry modeling kernel is used.) The CAD Import Module released with COMSOL[®] version 6.1 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 settings for the import of 3D CAD files now include an option to **Simplify curves and surfaces** that converts, when possible, the underlying curve and surface manifolds of the imported geometric entities to analytical form. Importing objects with this option may improve both the performance and reliability of geometric operations on some imported geometry.
- The defeaturing operation **Delete Faces** has been renamed to **Replace Faces** to better reflect that new faces are created to heal the wounds after removing the faces.
- For the defeaturing operations Replace Faces (Delete Faces in version 6.0) and Detach Faces, the heal method options Create capping faces, Extend adjacent faces, Extend adjacent faces from child, and Extend adjacent faces from parent are equivalent to the previously available Fill, Patch, Patch from child, and Patch from parent, respectively. When using the Create capping faces option, the operations can now create replacement faces in more cases when compared to previous versions.
- The defeaturing operations Delete Fillets and Delete Holes now have the options to enter the Minimum hole radius and Minimum fillet radius, respectively, in addition to the previously available options for entering the maximum radii for the search.

- The new **Offset Faces** operation enables offsetting of faces on 3D objects in the normal direction. This functionality is especially useful when modifying geometry imported from 3D CAD files. For example, it makes it possible to reparameterize certain geometric features of the imported objects.
- The new **Transform Faces** operation enables applying a linear transform (consisting of displacement, rotation, and isotropic scaling) on a selection of faces on 3D geometry objects. This functionality is also especially useful when modifying geometry imported from 3D CAD files. For example, it makes it possible to reparameterize certain geometric features of the imported objects.
- The new tutorials in the *Optimization of an Imported Bracket Geometry Tutorial Series* have been added to demonstrate how to use the **Offset Faces** and **Transform Faces** operations to reparameterize and optimize the size and position of throughholes on an imported geometry of a bracket.

New Functionality in LiveLink[™] for AutoCAD® in Version 6.1

- The LiveLink[™] interface now supports AutoCAD[®] 2023.
- The Synchronize Offline window in the CAD software user interface now contains a button to preview the geometries for the parameter combinations in the synchronization request file. Use this functionality to ensure that valid geometries will be generated and saved in the synchronization file.

New Functionality in LiveLink[™] for Inventor® in Version 6.1

- The LiveLink[™] interface now supports Inventor[®] 2023.
- The Synchronize Offline window in the CAD software user interface now contains a button to preview the geometries for the parameter combinations in the synchronization request file. Use this functionality to ensure that valid geometries will be generated and saved in the synchronization file.

New Functionality in LiveLinkTM for PTC® Creo® ParametricTM in Version 6.1

- The LiveLink[™] interface now supports PTC[®] Creo[®] Parametric[™] 9.0.
- The Synchronize Offline window in the CAD software user interface now contains a button to preview the geometries for the parameter combinations in the

synchronization request file. Use this functionality to ensure that valid geometries will be generated and saved in the synchronization file.

New Functionality in LiveLink[™] for Revit® in Version 6.1

- The LiveLink[™] interface now supports Revit[®] 2023.
- The new offline synchronization functionality enables synchronization when COMSOL Multiphysics[®] and the CAD software cannot be run side by side on the same computer. A synchronization file, which can contain one or several versions of the geometry, is prepared in advance using the LiveLink add-in inside the CAD software. The synchronization file is then loaded by the LiveLink feature in a COMSOL [®] model. Synchronization files can also be saved in response to a request saved to file from the LiveLink feature. Offline synchronization supports parametric sweeps when using mass parameters in Revit[®], and can be used on all supported operating systems. In addition to being installed by the COMSOL installer, the LiveLink add-in is also installed by the COMSOL Client installer, available for download at https://www.comsol.com. Such an installation enables the functionality for preparing designs for synchronization, saving files for offline synchronization, and running simulation apps that synchronize geometry.

New Functionality in LiveLink[™] for SOLIDWORKS® in Version 6.1

Starting with version 6.1 of COMSOL Multiphysics, we are no longer able to offer the one-window interface in our LiveLinkTM for SOLIDWORKS[®] product. LiveLinkTM for SOLIDWORKS continues to contain its bidirectional interface.

New Functionality in LiveLink[™] for Solid Edge® in Version 6.1

- The LiveLink[™] interface now supports Solid Edge[®] 2023.
- The Synchronize Offline window in the CAD software user interface now contains a button to preview the geometries for the parameter combinations in the synchronization request file. Use this functionality to ensure that valid geometries will be generated and saved in the synchronization file.

CFD Module

New Functionality in Version 6.1

- Physics interfaces for Detached Eddy Simulation, a hybrid RANS-LES method, have been added under the Single-Phase Flow section in the Fluid Flow branch. The three new interfaces — DES RBVM, Spalart–Allmaras; DES RBVMWV, Spalart– Allmaras; and DES Smagorinsky, Spalart–Allmaras — are applicable to threedimensional, time-dependent, single-phase, incompressible flows.
- Predefined High Mach Number, Reacting Flow multiphysics interfaces have been added in the Chemical Species Transport branch. Physics interfaces for laminar flow, and turbulent flow using either the k-ε or the Spalart–Allmaras turbulence models are available for the transport of diluted species. The corresponding physics interfaces for transport of concentrated species are available with a license for the Chemical Reaction Engineering Module.
- The new predefined multiphysics interface, Nonisothermal Flow, Viscoelastic Flow, under the **Nonisothermal Flow** section in the **Fluid Flow** branch couples a Viscoelastic Flow interface and a Heat Transfer in Fluids interface.
- The new predefined multiphysics interface, Two-Phase Flow, Level Set, Brinkman Equations, under the **Multiphase Flow** section in the **Fluid Flow** branch couples a Brinkman Equations interface and a Level Set in Porous Media interface using the **Two-Phase Flow**, **Level Set** multiphysics coupling node. It also adds a **Wetted Wall** multiphysics feature.
- New multiphysics interfaces, Dispersed Two-Phase Flow with Species Transport, are now available in the Chemical Species Transport branch for laminar flow, and turbulent flow using either the k-ε, k-ω, Low Re k-ε, or SST model. These couple a Mixture Model interface, Continuous Phase Transport of Diluted Species interface, and Dispersed Phase Transport of Diluted Species interface using the Dispersed Two-Phase Flow, Diluted Species multiphysics coupling node.
- The new predefined multiphysics interface, Solid Thin Film Damping, for modeling fluid-induced damping in solid structures, is now available under the Fluid-Structure Interaction section in the Structural Mechanics branch. It couples a Solid Mechanics interface and a Thin-Film Flow interface using the Structure Thin-Film Flow Interaction multiphysics coupling node.
- All transport-equation RANS turbulence models, except for the Spalart–Allmaras model, are now applicable in porous media domains. There are three options for the

Porous medium turbulence model: the **Nakayama–Kuwahara**, **Pedras–de Lemos**, and a **Default** option, which combines the other two models.

- For the Large Eddy Simulation and Detached Eddy Simulation interfaces, the **Inlet** condition has a new option, **Include synthetic turbulence**, for generating spatiotemporal velocity fluctuations based on a model energy spectrum.
- The new **Thin Barrier** node in the Multiphase Flow in Porous Media multiphysics interface makes it possible to add thin layers that act as a resistance for the flow fields of all phases without meshing through the layer's thickness.
- Laminar-to-turbulent boundary layer transition can be modeled by enabling the new **Include transition modeling** option in the Turbulent Flow, SST interface.
- The Mixture Model multiphysics coupling node in the Phase Transport, Mixture Model multiphysics interface has a new Include bubble-induced turbulence option in the Turbulence section. This option is available for all two-equation turbulence models when the Dispersed phase in the Physical Model section is set to Liquid droplets/bubbles.
- The Two-Phase Flow, Level Set; Two-Phase Flow, Phase Field; and Three-Phase Flow, Phase Field multiphysics couplings have a new option to get the effective material properties from a Multiphase Material node with built-in mixing rules. This is especially effective when coupling these multiphysics interfaces with other physics, such as heat transfer or electrostatics, since the multiphase material will use appropriate mixing rules for nonfluid material properties. In older versions, this would require you to write user-defined expressions based on the volume fraction of each fluid phase to compute the effective material properties used in each physics interface.
- Dedicated material property groups have been added for all available inelastic non-Newtonian models. The material property group contains all necessary material parameters and the apparent viscosity expression. It picks up the shear rate from the fluid flow interface to define the dynamic viscosity in the **Basic (def)** material model by means of a synchronization rule. Thus, an inelastic non-Newtonian model can be selected directly by adding the corresponding **Material Properties** group as a subnode to a material node.
- The Thin-Film Flow, Shell (tffs) and Thin-Film Flow, Edge (tffs) interfaces have been renamed Thin-Film Flow (tff). Similarly, the Phase Field, Thin-Film Flow, Shell (pftffs), and Phase Field, Thin-Film Flow, Edge (pftffs) interfaces have been renamed Phase Field, Thin-Film Flow (pftff). Moreover, the Two-Phase Thin-Film Flow, Shell, Phase Field and Two-Phase Thin-Film Flow, Edge, Phase Field interfaces have been renamed Two-Phase Thin-Film Flow, Phase Field.

Several applications using the Two-Phase Flow, Level Set; Two-Phase Flow, Phase Field; or Three-Phase Flow, Phase Field interface have been updated to include a **Multiphase Material** to compute the effective material properties from volume fractions and the material properties of each phase.

RESIN TRANSFER MOLDING OF A WIND TURBINE BLADE

This example shows the modeling of a resin transfer molding (RTM) process for a wind turbine blade using the Two-Phase Flow, Level Set, Brinkman Equations interface. Resin is injected into a preform consisting of different composites with different anisotropic permeabilities.

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.

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

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

Backward Compatibility with Version 5.5 and Earlier

TURBULENT INTENSITY AND TURBULENCE LENGTH SCALE

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

```
model.component("comp1").physics("spf").feature("inl1").
set("LT_list", "user_defined");
```

in the Model Java-file to retrieve the old length scale.

EULER-EULER MODEL

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

PHASE TRANSPORT, MIXTURE MODEL

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

TWO-PHASE FLOW

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

Chemical Reaction Engineering Module

New Functionality in Version 6.1

SHRINKING CORE MODEL FOR PACKED BEDS

The **Packed Bed** feature in the Transport of Diluted Species interface now includes the shrinking core model. Use this model to describe situations where a solid particle is consumed and shrinks, leaving an outer porous layer, and an inner shrinking core. The core consuming reactions, the transport and reactions in the porous layer, as well as the interparticle mass transfer can be studied in detail.

The shrinking core model is enabled using the new **Pellet Type** setting in the **Pellets** subnode of the **Packed Bed** node. A new **Transport of Diluted Species, Shrinking Core Model** entry s also available under the Reacting Flow in Porous Media interface in the Model Wizard and the **Add Physics** window.

DISPERSED TWO-PHASE FLOW WITH SPECIES TRANSPORT

A new multiphysics interface, Dispersed Two-Phase Laminar Flow with Species Transport, is now available for modeling two-phase flow coupled with chemical species transport. A liquid containing a dispersed phase (droplets or bubbles) is modeled using a Mixture Model interface. Mass transport of solutes in each phase, as well as solute extraction between the phases, can be solved for. Turbulent flow versions of the multiphysics interface are available with a license for either the CFD Module or the Heat Transfer Module.

The Mixture Model, Laminar Flow interface is now available with a license for the Chemical Reaction Engineering Module.

ADVANCED CHEMICAL FORMULAS IN THE REACTION ENGINEERING AND CHEMISTRY INTERFACES

The process of adding new chemical species, both when manually creating **Species** features and when taken from **Reaction** features, has been significantly improved and extended.

- The **Name** section of a **Species** node now includes both the species name and the variable name used for postprocessing. These names differ for charged species, species with parentheses, and for species with a phase indication.
- The **Species** node now includes a **Chemical Formula** section where the composition of the species can be defined in terms of elements from the periodic table (by selecting the **Enable formula** check box). The molar mass and species charge are automatically computed from the formula.
- The **Chemical Formula** section supports repeating units using three types of enclosing marks: (), [], and {}. One example of this is [Al(OH)(OH2)5]^2+.
- The **Chemical Formula** section supports the following charge representations: ++, (++), ^(++), ^++, ^2+, (2+), and ^(2+).
- For intricate molecular structures, a simplified (trivial) name can now be used, while the composition and charge are detailed using the **Chemical Formula** input. When a **Chemical Formula** is defined, it is used in place of the simplified name when balancing a reaction formula.
- The reaction formula parser used in the **Reaction** node has been extended to also support species names with repeating units and the extended set of charge representations.
- The error messages shown for invalid species names have been improved.

NEW FRACTURE FEATURE FOR TRANSPORT OF DILUTED SPECIES IN FRACTURES

A new **Fracture** feature, consistent with other features for porous media transport, is now available. The feature includes two subnodes, **Fluid** and **Fracture Material**, including settings for the respective phases.

NEW HIGH MACH NUMBER REACTING FLOW MULTIPHYSICS INTERFACES

The new High Mach Number Reacting Flow multiphysics interface is available for users with licenses for both the CFD Module and the Chemical Reaction Engineering Module. The new High Mach Number Reacting Flow, Diluted Species multiphysics interface is available for users with a license for the CFD Module.

POSTPROCESSING FUNCTIONALITY

- It is now possible to place line graph legends outside of the graph axis area. This is useful for avoiding that graph lines are covered by the legend when plotting many curves in the same graph.
- An extended color cycle, including 19 colors, is now available for graph plots.
- A new **Pellets** plot is available for visualizing the concentration inside porous pellets (or similar particles) of a packed bed. Spheres or three orthogonal filled circles can be used to visualize the concentration in 3D and 2D plot groups.

NEW AND IMPROVED FUNCTIONALITY

- Evaluation of thermodynamic properties is now faster. Models where a significant part of the solution time is spent evaluating functions from the **Thermodynamics** functionality can now be solved in 30% less time.
- The molar mass definition is now synchronized by the **Species Matching** section in the Chemistry interface. Selecting a species solved for, the molar mass in the corresponding mass transport interface is updated to the one defined in **Chemistry**.
- Definitions of postprocessing variables available in the Transport of Diluted Species interface have been added to the documentation.
- Constant properties defined by the **Thermodynamics** functionality can now be evaluated as parameters.
- The molar mass and charge settings have been moved to the new **Species Properties** node for the Transport of Concentrated Species interface. The node is named **Species Molar Mass** when **Migration in electric field** is not selected. Java API backward compatibility is provided for the charge setting commands, but not for the molar mass setting commands.
- The charge settings have been moved to the new **Species Properties** node for the Transport of Diluted Species, and the Nernst–Planck Equations interfaces. The node is named **Species Charges** when ideal species activities are used.
- Nonideal activities are now supported in the Nernst–Planck Equations and Transport of Diluted Species interfaces. Functionality has also been added for computing the nonideal activities using the Debye–Hückel equation, based on the ionic strength of the solution.

- When importing reactions from a CHEMKIN file, separate **Species** nodes are now also created for species that does not occur in the reactions.
- Consistent stabilization is now available when using the **Maxwell-Stefan** diffusion model in the Transport of Concentrated Species interface.
- The **Maxwell-Stefan** diffusion model can be combined with turbulent flow when using the Reacting Flow multiphysics interfaces.
- The **Periodic Condition** for mass transport can now be combined with wall conditions for fluid flow when using the **Reacting Flow** or **Reacting Flow**, **Diluted Species** coupling feature. In the case of turbulent flow, the wall concentration is made periodic by applying the constraint to the wall function.
- Interior fan conditions are now supported by the **Reacting Flow** and **Reacting Flow**, **Diluted Species** coupling features.
- A new Adsorption/Desorption of bulk species check box is available in the Adsorbed Species section in the Porous Catalyst feature.
- Separate species are now used in the solver sequence for adsorbed species and surface species (defined by the **Porous Catalyst** and **Packed Bed** features). This means that individual settings can be applied to the variables in the solver sequence.
- It is now possible to add user-defined contributions to the equations for pellet mass transport in a **Packed Bed** feature. To do so, use the **Show More Options** window and select the **Equation-Based Contributions** check box under **Physics**. Settings for the pellet extra dimension are available in features for user-defined contributions, such as the **Weak Contribution** node, when selecting **Attached Dimensions** for the selection.
- The filter search field, used when adding species to a thermodynamic system, has been improved. Filtered species can now be entered one at a time, from the top, using the Enter key. Also, it is no longer necessary to update the filter each time a species is added to the **Selected species** list.

New Models in Version 6.1

CONTINUOUS MIXER

This model simulates a tank with an asymmetrically positioned impeller. The tank and impeller geometry are defined using parts from the Mixer Module Part Library. A liquid is continuously fed into the tank near the top and exits through an outlet at the bottom. The turbulent flow field resulting from the impeller agitation is solved for using a frozen-rotor analysis. The model shows how to visualize the mixing and compute the residence-time distribution using particle tracing. A pulse containing

50,000 massless particles is injected and tracked as it flows through the mixer. The residence-time distribution is computed by counting the number of particles that have reached the outlet.

CRYSTALLIZATION OF BENZOIC ACID IN A MIXED SUSPENSION, MIXED PRODUCT REMOVAL CRYSTALLIZER

Crystallization is a key separation process in pharmaceuticals production, for example. It is the process by which a chemical species is separated from solution by forming a crystal. To achieve the required product properties, control over the crystal particle size distribution is necessary. This 0D model implements a population balance equation to determine the crystal size distribution and mean particle size for benzoic acid crystallizing in a mixed-suspension, mixed-product-removal crystallizer.

EVAPORATION OF ETHANOL AND WATER FROM A WINE GLASS

During professional wine tastings, several samples of wine are judged in one session. It is important that each sample is covered since evaporation of ethanol and water will change the taste. This model simulates the evaporation of ethanol and water from a wine glass. Evaporation of multiple species from a nonideal liquid mixture is modeled using the extended Raoult's law. The evaporation induces free convection in the surrounding vapor phase, both due to the change in composition and due to the heat of vaporization. The model is set up in 2D axisymmetry using coupled Laminar Flow, Transport of Concentrated Species, and Heat Transfer in Fluids interfaces. Accurate thermodynamic data is provided by the **Thermodynamics** functionality.

Updated Models in Version 6.1

SYNGAS COMBUSTION IN A ROUND-JET BURNER

The model has been updated to use a gas phase thermodynamic system (under the **Thermodynamics** node). Thermodynamics properties and transport properties of the gas mixture are automatically defined by a Chemistry interface, eliminating the need for interpolation functions.

POROUS CATALYTIC REACTOR WITH INJECTION NEEDLE

The model has been substantially reworked to illustrate the modeling of a porous catalytic reactor for hydrogen oxidation on a noble metal Rh catalyst dispersed on an alumina support (Rh/Al_2O_3). The surface reaction kinetics in the reactor are implemented using a Porous Catalyst feature. Adsorption and desorption of bulk

species as well as the development of surface species attached to the catalyst are solved for.

DRUG RELEASE FROM A BIOMATERIAL MATRIX

The model uses a single Transport of Diluted Species in Porous Catalysts interface to study drug release from a biomaterial matrix into damaged cell tissue. The surface species kinetics in the biomaterial is modeled using a **Porous Catalyst** feature.

Composite Materials Module

New Functionality in Version 6.1

IMPROVED CONTACT FUNCTIONALITY

The **Contact** node in the Layered Shell interface is improved, and is now similar to other physics interfaces.

Two new formulations have been added, **Penalty, dynamic** and **Augmented Lagrangian**, **dynamic**, which are essential for modeling dynamic contact events. The **Friction**, **Slip Velocity**, **Adhesion**, and **Decohesion** subnodes are now available for modeling different contact phenomena efficiently.

The **Source external to current physics** check box has been removed, and contact with objects outside of the interface is now handled automatically. If a contact pair is present in the model, then a default **Contact** feature is automatically added.

PERFORMANCE IMPROVEMENT IN LAYERED LINEAR ELASTIC MATERIAL

When the **Layered Material Link** or **Layered Material Stack** material type contains transformations of symmetry, antisymmetry, or repetition types, the symmetry properties of the structure are employed to speed up the generation of matrices as long as the middle layer is not merged. A combination of numerical and analytical extra dimension integration is used to compute the stiffness matrix in the **Layered Linear Elastic material**. This improvement applies to the **Layered Linear Elastic Material** node when there is a single product selection in the Shell and Membrane interfaces and when weak equations are written in terms of the section forces. As a result of this improvement, assembly time and computation time are significantly reduced.

MULTIPLICATIVE DECOMPOSITION IN THE LAYERED LINEAR ELASTIC MATERIAL

- In the Shell interface, the **Layered Linear Elastic Material** node now supports the multiplicative decomposition of strains. This formulation is important in the case of finite deformations, and it is now the default for geometric nonlinear analyses.
- The Automatic option in the Shear Correction Factor section sets the shear correction factors to 5/6. The new Based on 3D elasticity theory option can restore the old behavior of the Automatic option.

NEW OPERATORS IN LAYERED MATERIALS

New physics-scoped operators have been added to the Layered Shell interface and to the layered materials in the Shell and Membrane interfaces. These operators help to compute the integration, average, or summation of variables over layers or interfaces in the thickness direction.

REDUCED NUMBER OF DEFAULT PLOTS

For details, see Reduced Number of Default Plots in the release notes for the Structural Mechanics Module.

MISCELLANEOUS

- The Add Material button in the Layered Linear Elastic Material node in the Shell interface is now consistent with the same button available in other layered features.
- The length scale factor (LSF) variable in the Layered Shell interface is now defined in the edge product selection, which gives the correct through-thickness variation.
- The reaction force and moment variables in the Layered Shell interface are available in lower dimensions. The total reaction force and moments are corrected for the **Continuity** node.
- In the **Safety** node under **Layered Linear Elastic Material**, the mode C index for Puck failure criterion is corrected. The safety factor are now nonnegative for the Puck and LaRC03 criteria.

The layers' thickness are correctly used to compute the initial stresses from section forces in the **Initial Stress and Strain** subnode under the **Layered Linear Elastic Material** node in the Membrane interface.

New Models in Version 6.1

MICROMECHANICS OF FAILURE: MULTISCALE ANALYSIS OF A COMPOSITE STRUCTURE

In this example, the structural integrity of a cylinder made by a fiber composite is assessed both at the macroscale and microscale levels.

Analyses both at the macroscale and microscale, so-called multiscale analyses, offer an in-depth insight into the composite structure as well as its constituent's response. They also help you to study the micromechanics of failure based on the global scale loading.

Corrosion Module

New Functionality in Version 6.1

GENERAL NEW FUNCTIONALITY IN THE ELECTROCHEMISTRY INTERFACES

- Fixed an issue with the **Electrode Power** boundary feature for the **Current Distribution Initialization** study step if the dependent variable name of phis has been changed. Also, the field name of the Lagrange multiplier has been changed to include the physics tag. The name change may cause backward compatibility issues when running the Java API.
- The **Continuity** Pair condition has been improved in the Primary/Secondary Current Distribution and Cathodic Protection interfaces.
- The charge settings have been moved to the new **Species Properties** node for the Tertiary Current Distribution, Nernst–Planck interface.
- Nonideal activities are now supported in the Tertiary Current Distribution, Nernst– Planck interface. Functionality has also been added for computing the nonideal activities using the Debye–Hückel equation, based on the ionic strength of the solution.
- The new 0.25 V (Seawater Ag/AgCl vs. SHE) option has been added to the Physics vs. Materials Reference Electrode Potential section of the physics interface node for all electrochemistry interfaces.

NEW CORROSION MODULE-SPECIFIC FUNCTIONALITY

A new **Sacrificial Anode Surface** boundary node is now available under the **Cathodic Protection Surfaces** node in the Cathodic Protection and Secondary Current Distribution interfaces.

ADVANCED CHEMICAL FORMULAS IN THE CHEMISTRY INTERFACE

The process of adding new chemical species, both when manually creating **Species** features, and when taken from **Reaction** features, has been significantly improved and extended.

• The **Name** section of a **Species** node now includes both the species name and the variable name used for postprocessing. These names differ for charged species, species with parentheses, and for species with a phase indication.
- The **Species** node now includes a **Chemical Formula** section where the composition of the species can be defined in terms of elements from the periodic table (by selecting the **Enable formula** check box). The molar mass and species charge are automatically computed from the formula.
- The **Chemical Formula** section supports repeating units using three types of enclosing marks: (), [], and {}. On example of this is [Al(OH)(OH2)5]^2+.
- The **Chemical Formula** section supports the following charge representations: ++, (++), ^(++), ^++, ^2+, (2+), and ^(2+).
- For intricate molecular structures, a simplified (trivial) name can now be used, while the composition and charge are detailed using the **Chemical Formula** input. When a **Chemical Formula** is defined, it is used in place of the simplified name when balancing a reaction formula.
- The reaction formula parser used in the **Reaction** node has been extended to also support species names with repeating units and the extended set of charge representations.
- The error messages shown for invalid species names have been improved.

GENERAL NEW FUNCTIONALITY IN THE CHEMICAL TRANSPORT INTERFACES

- The charge settings have been moved to the new **Species Properties** node for the Nernst–Planck Equations and Transport of Diluted Species interfaces.
- Nonideal activities are now supported in the Nernst–Planck Equations and Transport of Diluted Species interfaces. Functionality has also been added for computing the nonideal activities using the Debye–Hückel equation, based on the ionic strength of the solution.
- The **Periodic Condition** for mass transport can now be combined with wall conditions for fluid flow when using the **Reacting Flow, Diluted Species** coupling feature. In the case of turbulent flow, the wall concentration is made periodic by applying the constraint to the wall function.
- Interior fan conditions are now supported by the **Reacting Flow** and **Reacting Flow**, **Diluted Species** coupling features.
- A new **Fracture** feature, consistent with other features for porous media transport, is now available. The feature includes two subnodes, **Fluid** and **Fracture Material**, including settings for the respective phases.

- New Atmospheric Corrosion with Mass Transport (*atmospheric_corrosion_mass_transport*) tutorial.
- New Corrosion Inhibitor Transport (*corrosion_inhibitor_transport*) tutorial.
- New Alternating Current-Induced Corrosion (*ac_corrosion*) tutorial.

ECAD Import Module

New Functionality in Version 6.0

The ECAD Import Module includes two new options for ignoring unnecessary vertices, thereby eliminating short edges when generating the 2D geometries from the layouts imported from IPC-2581, ODB++, and GDS files. When importing any of these file formats, the settings for the import now include, in the new **Ignore vertices** in layers section, the **Ignore vertices with continuous tangent** and **Eliminate short edges** check boxes.

Electrochemistry Module

New Functionality in Version 6.1

GENERAL NEW FUNCTIONALITY IN THE ELECTROCHEMISTRY INTERFACES

- Fixed an issue with the **Electrode Power** boundary feature for the **Current Distribution Initialization** study step if the dependent variable name of phis has been changed. Also, the field name of the Lagrange multiplier has been changed to include the physics tag. The name change may cause backward compatibility issues when running the Java API.
- The **Continuity** pair condition has been improved in the Primary/Secondary Current Distribution interfaces.
- The charge settings have been moved to the new **Species Properties** node for the Tertiary Current Distribution, Nernst–Planck interface.
- Nonideal activities are now supported in the Tertiary Current Distribution, Nernst– Planck interface. Functionality has also been added for computing the nonideal activities using the Debye–Hückel equation, based on the ionic strength of the solution.
- The new **0.25 V (Seawater Ag/AgCl vs. SHE)** option has been added to the **Physics vs. Materials Reference Electrode Potential** section of the physics interface node in all electrochemistry interfaces.

ADVANCED CHEMICAL FORMULAS IN THE CHEMISTRY INTERFACE

The process of adding new chemical species, both when manually creating **Species** features and when taken from **Reaction** features, has been significantly improved and extended.

- The Name section of a Species node now includes both the species name and the variable name used for postprocessing. These names differ for charged species, species with parentheses, and for species with a phase indication.
- The **Species** node now includes a **Chemical Formula** section where the composition of the species can be defined in terms of elements from the periodic table (by selecting the **Enable formula** check box). The molar mass and species charge are automatically computed from the formula.

- The **Chemical Formula** section supports repeating units using three types of enclosing marks: (), [], and {}. On example of this is [Al(OH)(OH2)5]^2+.
- The **Chemical formula** section supports the following charge representations: ++, (++), ^(++), ^++, ^2+, (2+), and ^(2+).
- For intricate molecular structures, a simplified (trivial) name can now be used, while the composition and charge are detailed using the **Chemical Formula** input. When a **Chemical Formula** is defined, it is used in place of the simplified name when balancing a reaction formula.
- The reaction formula parser used in the **Reaction** node has been extended to also support species names with repeating units and the extended set of charge representations.
- The error messages shown for invalid species names have been improved.

GENERAL NEW FUNCTIONALITY IN THE CHEMICAL TRANSPORT INTERFACES

- The charge settings have been moved to the new **Species Properties** node for the Nernst–Planck Equations and Transport of Diluted Species interfaces.
- Nonideal activities are now supported in the Nernst–Planck Equations and Transport of Diluted Species interfaces.
- The **Periodic Condition** for mass transport can now be combined with wall conditions for fluid flow when using the **Reacting Flow** or **Reacting Flow**, **Diluted Species** coupling feature. In the case of turbulent flow, the wall concentration is made periodic by applying the constraint to the wall function.
- Interior fan conditions are now supported by the **Reacting Flow**, and the **Reacting Flow**, **Diluted Species** coupling features.
- A new **Fracture** feature, consistent with other features for porous media transport, is now available. The feature includes two subnodes, **Fluid** and **Fracture Material**, including settings for the respective phases.

Electrodeposition Module

New Functionality in Version 6.1

GENERAL NEW FUNCTIONALITY IN THE ELECTROCHEMISTRY INTERFACES

- Fixed an issue with the **Electrode Power** boundary feature for the **Current Distribution Initialization** study step if the dependent variable name of phis has been changed. Also, the field name of the Lagrange multiplier has been changed to include the physics tag. The name change may cause backward compatibility issues when running the Java API.
- The **Continuity** Pair condition has been improved in the Primary/Secondary Current Distribution interfaces.
- The charge settings have been moved to the new **Species Properties** node for the Tertiary Current Distribution, Nernst–Planck interface.
- Nonideal activities are now supported in the Tertiary Current Distribution, Nernst– Planck interface. Functionality has also been added for computing the nonideal activities using the Debye–Hückel equation, based on the ionic strength of the solution.
- The new **0.25 V (Seawater Ag/AgCl vs. SHE)** option has been added to the **Physics vs. Materials Reference Electrode Potential** section of the physics interface node in all electrochemistry interfaces

ADVANCED CHEMICAL FORMULAS IN THE CHEMISTRY INTERFACE

The process of adding new chemical species, both when manually creating **Species** features, and when taken from **Reaction** features, has been significantly improved and extended.

- The Name section of a Species node now includes both the species name and the variable name used for postprocessing. These names differ for charged species, species with parentheses, and for species with a phase indication.
- The **Species** node now includes a **Chemical Formula** section where the composition of the species can be defined in terms of elements from the periodic table (by selecting the **Enable formula** check box). The molar mass and species charge are automatically computed from the formula.

- The **Chemical Formula** section supports repeating units using three types of enclosing marks: (), [], and {}. On example of this is [Al(OH)(OH2)5]^2+.
- The **Chemical Formula** section supports the following charge representations: ++, (++), ^(++), ^++, ^2+, (2+), and ^(2+).
- For intricate molecular structures, a simplified (trivial) name can now be used, while the composition and charge are detailed using the **Chemical Formula** input. When a **Chemical Formula** is defined, it is used in place of the simplified name when balancing a reaction formula.
- The reaction formula parser used in the **Reaction** node has been extended to also support species names with repeating units and the extended set of charge representations.
- The error messages shown for invalid species names have been improved.

GENERAL NEW FUNCTIONALITY IN THE CHEMICAL TRANSPORT INTERFACES

- The charge settings have been moved to the new **Species Properties** node for the Nernst–Planck Equations and Transport of Diluted Species interfaces.
- Nonideal activities are now supported in the Nernst–Planck Equations and Transport of Diluted Species interfaces. Functionality has also been added for computing the nonideal activities using the Debye–Hückel equation, based on the ionic strength of the solution.
- The **Periodic Condition** for mass transport can now be combined with wall conditions for fluid flow when using the **Reacting Flow** or **Reacting Flow**, **Diluted Species** coupling feature. In the case of turbulent flow, the wall concentration is made periodic by applying the constraint to the wall function.
- Interior fan conditions are now supported by the **Reacting Flow** and **Reacting Flow**, **Diluted Species** coupling features.
- A new **Fracture** feature, consistent with other features for porous media transport, is now available. The feature includes two subnodes, **Fluid** and **Fracture Material**, including settings for the respective phases.

Fatigue Module

New Functionality in Version 6.1

RANDOM VIBRATION IMPROVEMENT

A **Life cutoff** input has been added to the **Random Vibration** node. It is used to facilitate the visualization of fatigue life so that extremely long lifetimes can be removed.

Fuel Cell & Electrolyzer Module

New Functionality in Version 6.1

GENERAL NEW FUNCTIONALITY IN THE ELECTROCHEMISTRY INTERFACES

- Fixed an issue with the **Electrode Power** boundary feature for the **Current Distribution Initialization** study step if the dependent variable name of phis has been changed. Also, the field name of the Lagrange multiplier has been changed to include the physics tag. The name change may cause backward compatibility issues when running the Java API.
- The **Continuity** Pair condition has been improved in the Primary/Secondary Current Distribution interfaces.
- The charge settings have been moved to the new **Species Properties** node for the Tertiary Current Distribution, Nernst–Planck interface.
- Nonideal activities are now supported in the Tertiary Current Distribution, Nernst– Planck interface. Functionality has also been added for computing the nonideal activities using the Debye–Hückel equation, based on the ionic strength of the solution.
- The new 0.25 V (Seawater Ag/AgCl vs. SHE) option has been added to the Physics vs. Materials Reference Electrode Potential section of the physics interface node in all electrochemistry interfaces.

NEW FUEL CELL AND ELECTROLYZER-SPECIFIC FUNCTIONALITY

- A **Consistent Stabilization** section has been added to the settings for gas-phase diffusion equations in the Hydrogen Fuel Cell and Water Electrolyzer interfaces.
- The default solver generation for 3D models using the Hydrogen Fuel Cell and the Water Electrolyzer interfaces has been significantly improved, with the default generated solver depending on the physics settings and the number of degrees of freedom. For large problems, **Algebraic Multigrid** iterative solvers are now generated by default, which typically reduces memory usage and computational time considerably.
- Correction of the normal boiling value of H2O species in the Hydrogen Fuel Cell and Water Electrolyzer interfaces. This update affects the built-in thermal

conductivity value calculated by the interface. This fix was also included in version 6.0 update 2.

- A new **Auxiliary Species** option was added to the Hydrogen Fuel Cell and Water Electrolyzer interfaces. This provides the possibility to add an arbitrarily defined gaseous species on the O2 and/or H2 sides.
- Improved formulation of the mass flow rates boundary condition using the **Total mass flow rate** option in the Hydrogen Fuel Cell and Water Electrolyzer interfaces. The new formulation is used for the species-dependent variables for the case when gas-phase diffusion is active in combination with Darcy's law.

ADVANCED CHEMICAL FORMULAS IN THE CHEMISTRY INTERFACE

The process of adding new chemical species, both when manually creating **Species** features and when taken from **Reaction** features, has been significantly improved and extended.

- The **Name** section of a **Species** node now includes both the species name and the variable name used for postprocessing. These names differ for charged species, species with parentheses, and for species with a phase indication.
- The **Species** node now includes a **Chemical Formula** section where the composition of the species can be defined in terms of elements from the periodic table (by selecting the **Enable formula** check box). The molar mass and species charge are automatically computed from the formula.
- The **Chemical Formula** section supports repeating units using three types of enclosing marks: (), [], and {}. On example of this is [Al(OH)(OH2)5]^2+.
- The **Chemical Formula** section supports the following charge representations: ++, (++), ^(++), ^++, ^2+, (2+), and ^(2+).
- For intricate molecular structures, a simplified (trivial) name can now be used, while the composition and charge are detailed using the **Chemical Formula** input. When a **Chemical Formula** is defined, it is used in place of the simplified name when balancing a reaction formula.
- The reaction formula parser used in the **Reaction** node has been extended to also support species names with repeating units and the extended set of charge representations.
- The error messages shown for invalid species names have been improved.

IMPROVEMENTS TO THERMODYNAMICS

- Constant properties defined by the **Thermodynamics** functionality can now be evaluated as parameters.
- Evaluation of thermodynamic properties is now faster. Models where a significant part of the solution time is spent evaluating functions from the **Thermodynamics** functionality can now be solved in 30% less time.
- The filter search field, used when adding species to a thermodynamic system, has been improved. Filtered species can now be entered one at a time, from the top, using the Enter key. Also, it is no longer necessary to update the filter each time a species is added to the **Selected species** list.

GENERAL NEW FUNCTIONALITY IN THE CHEMICAL TRANSPORT INTERFACES

- The charge and molar mass settings have been moved to the new **Species Properties** node for the Transport of Concentrated Species interface. Java API backward comp is provided for the charge setting commands, but not for the molar mass setting commands.
- The charge settings have been moved to the new **Species Properties** node for the Nernst–Planck Equations and Transport of Diluted Species interfaces.
- Nonideal activities are now supported in the Nernst–Planck Equations and Transport of Diluted Species interfaces. Functionality has also been added for computing the nonideal activities using the Debye–Hückel equation, based on the ionic strength of the solution.
- Consistent stabilization is now available when using the **Maxwell-Stefan** diffusion model in the Transport of Concentrated Species interface.
- The **Maxwell-Stefan** diffusion model can be combined with turbulent flow when using the Reacting Flow multiphysics interfaces.
- The Periodic Condition boundary condition for mass transport can now be combined with wall conditions for fluid flow when using the Reacting Flow or Reacting Flow, Diluted Species coupling feature. In the case of turbulent flow, the wall concentration is made periodic by applying the constraint to the wall function.
- Interior fan conditions are now supported by the **Reacting Flow** and **Reacting Flow**, **Diluted Species** coupling features.
- A new **Fracture** feature, consistent with other features for porous media transport, is now available. The feature includes two subnodes, **Fluid** and **Fracture Material**, including settings for the respective phases.

• A new Fuel Cell Stack Cooling (*stack_cooling*) tutorial has been added.

Geomechanics Module

New Functionality in Version 6.1

ENHANCEMENTS TO PLASTICITY

- The **Plasticity**, **Elastoplastic Soil**, **Soil Plasticity**, **Rocks**, and **Concrete** nodes now have an **Advanced** section where it is possible to control the number of iterations and relative tolerance used by the plasticity algorithm.
- For the Drucker–Prager and Mohr–Coulomb models available in the **Soil Plasticity**, **Porous Plasticity**, and **Safety** nodes, the intersection between the elliptic cap and the failure surface has been corrected.
- The UI and some API names for the **Plasticity** node have been updated.

ENHANCEMENTS TO DAMAGE

- Some user inputs in the **Damage** node can now be directly imported from the **Materials** node.
- The default solver suggestions for Phase field damage have been improved.
- It is now possible to combine the **Damage** node with the **Plasticity** or **Soil Plasticity** nodes.
- For the **Scalar** damage, the expression for the damaged stress variable is corrected when used in combination with the mixed formulation.

TESTING OF SOIL MODELS

A new feature, **Test Material**, has been added to the Solid Mechanics interface. There, it is possible to set up various numerical tests, including the **Isotropic**, **Oedometer**, and **Triaxial** tests for soil material models.

Heat Transfer Module

New and Improved Functionality in Version 6.1

SPACECRAFT THERMAL ANALYSIS

The new **Orbital Thermal Loads** physics interface provide ready-made features to model radiative loads on spacecraft, in particular radiation from the Sun and the Earth for satellites orbiting around Earth.

The user provides the spacecraft radiative properties, orbit and orientation, orbital maneuvers, and planet properties. From this information, the physics interface computes and generates graphics showing the direct solar radiation, albedo, and planet infrared flux as well as the radiative heat transfer between the different spacecraft parts. Combined with a heat transfer physics interface, it also accounts for heat conduction in the solid parts of spacecraft.

THERMAL CONNECTORS BETWEEN SHELLS AND DOMAINS

The Thermal Connection, Layered Shell, Edges; Thermal Connection, Layered Shell, Surfaces; and Thermal Connection, Nonlayered Shell multiphysics couplings are designed to define a continuity condition between the shared or facing surfaces between temperature fields computed by a domain heat transfer interface and a shell heat transfer interface. The connector can be used for shells that are in contact through an edge, a boundary shared with the domain interface, or a boundary that is facing another boundary from the domain interface. These multiphysics couplings greatly simplifies the use of combined domain and shell interfaces in models.

SYMBOLS FOR RADIATION WITH THE OPACITY CONTROLLED OPTION

The symbols representing the emitted radiation direction are now also displayed when the **Opacity controlled** option is used. The symbols are only available when the **Wavelength dependence of radiative properties** option is set to **Constant**.

VERIFICATION TOOLS FOR SURFACE-TO-SURFACE RADIATION MODELS

In the Surface-to-Surface Radiation physics interface settings, in the **View Factor** section, the **Check consistency** option has been added for the hemicube and ray shooting methods. It is selected by default. This verification may have a noticeable extra computational cost when the hemicube method is used. If so, a warning message

will inform you about it. In this case, once the model setup has been verified, the verification can be disabled.

SURFACE-TO-SURFACE RADIATION IMPROVEMENTS

The ray shooting method have been improved so that small surfaces are detected even when a coarse resolution is used for the view factor evaluation. This is obtained by shooting additional rays, one for each group of boundaries detected in the geometry. Combined with the resolution adaption, this improves the view factor accuracy with an optimal number of rays. This may affect the results in case some small boundaries were not detected previously due to a coarse resolution.

Also, for all view factor computation methods, expressions used to define variables and equations are now preprocessed to remove the previously used if statements testing the radiation direction. Now the expressions readability is facilitated and their evaluations are much faster during the assembly step.

FLUENCE RATE

In the **Surface-to-Surface Radiation** interface it is now possible to add a **Fluence Rate Calculation** node in order to select the domains where the fluence rate has to be computed. The variable for fluence rate is *<phys>*.E0, where *<phys>* is the name of the **Surface-to-Surface Radiation** physics interface, for example, rad.E0.

The fluence rate variable indicates the exposure to radiation if you were to place a small object in a cavity.

CLIMATE DATA: ASHRAE 2021

ASHRAE Weather Data 2021 has been integrated into COMSOL Multiphysics[®]. C ASHRAE 2021 contains additional locations; there are more than 8500 stations in the 2021 edition. You can select any of them in the **Ambient Properties** node. The database provides typical climate data for the temperature, humidity, precipitation, and solar radiation.

ENHANCED THERMAL WALL FUNCTIONS FOR VISCOUS DISSIPATION

In the Nonisothermal Flow coupling, when the Turbulence model type is set to RANS in the single-phase flow interface, there is a new option, Thermal wall function, in the Heat Transfer Turbulence section. It is set by default to Standard, which corresponds to the legacy implementation and which is suitable for most of the configurations. The other option, High viscous dissipation at wall, accounts for viscous dissipation in the boundary layer, which is not the case with the Standard option. This is needed for accurate results in case of fast internal flow, especially if the paths are narrow or the fluid is very viscous.

EVAPORATION MASS SOURCE IN HYGROSCOPIC POROUS MEDIUM

The **Moisture Flow** multiphysics coupling has been updated so that the evaporation rate variable computed by the **Moisture Transport** interface is accounted for in the mass balance computed by **Brinkman Equations** interface.

INFLOW AND OPEN BOUNDARY CONDITIONS FOR HYGROSCOPIC POROUS MEDIUM

The **Open Boundary** and **Inflow** boundary condition are now applicable to exterior boundaries adjacent to domains where a **Hygroscopic Porous Medium** node is active.

USER-DEFINED PHASE TRANSITION FUNCTION FOR PHASE CHANGE MATERIAL

A **User-defined** phase transition function is now available in the **Phase Change Material** feature. When it is selected, it is possible to define the phase transition function in the corresponding user input.

IMPLEMENTATION OF SOME HEAT TRANSFER FEATURES ON THE MATERIAL FRAME

The Solid, Building Material, Biological Tissue, Battery Layers, and Shape Memory Alloy features are now defined on the material frame. This change has no effect unless the spatial and material frame differ. This is the case when moving mesh features are used or when structural mechanics physics interfaces are combined with the heat transfer physics interface.

SYNCHRONIZE MATERIAL PROPERTIES IN THE IDEAL GAS GROUP

Synchronization rules have been added for material properties for the **Ideal gas** group. Thanks to this, if a property like **Heat capacity at constant pressure** is defined in the **Basic** group, the **Ideal gas** group definition is synchronized. Also, some rules are available to deduce a property from other properties available in the same group. For example, if the **Mean molar mass** property is defined, then the **Specific gas constant**, $R_s = R_{const}/M_n$ is deduced from it, so you no longer need to define both.

SOLVER SUGGESTIONS FOR LUMPED THERMAL SYSTEMS

Solver suggestions for the Lumped Thermal System physics interface have been improved to be automatically merged with the solver suggestions from heat transfer interfaces when **Lumped Thermal Connector** features are active. This is particularly important for large models using an iterative solver. In this case, hybrid preconditioners are defined.

ORBIT CALCULATION

This model is an introduction to defining and verifying a satellite orbit and computing the solar, albedo, and Earth infrared (IR) thermal loads. A 1U CubeSat is in a circular orbit at 400 km altitude, with an inclination of 50° and a longitude of ascending node of 0° . The satellite is rotating slowly about its nadir-pointing axis. One orbital period is analyzed, during the winter solstice. The total irradiation and the radiative loads from all environmental sources are computed. This type of analysis is usually performed prior to computing the temperature evolution within the structure of the satellite.

ORBIT THERMAL LOADS

A satellite in orbit experiences solar, albedo, and planetary IR loads, where albedo and planetary IR can vary with latitude and longitude. This model demonstrates how to define Earth properties that are spatially varying. It shows how to read spreadsheet and image data and how to define and use different functions. The total irradiation and flux onto the satellite are evaluated over several orbits and the albedo flux is plotted over the surface of the Earth.

SPACECRAFT THERMAL ANALYSIS

This model shows the workflow for computing the satellite temperature on multiple orbit periods. The direct solar, albedo, and Earth infrared thermal loads are precomputed in an **Orbit Thermal Loads** study and reused on multiple orbit periods in an **Orbital Temperature** study.

Updated Applications in Version 6.1

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 Disk Stack Heat Sink model is now using the new Thermal Connection, Layered Shell, Edges multiphysics coupling.
- The *Continuous Casting ALE* and *Friction Stir Welding* models are now using the new **Solid with Translational Motion** feature.

- The *Brake Disk* model now uses **Moving Mesh** features instead of **Translational Motion** to improve the computational performance.
- The Thermal Bridges in Building Construction 2D Composite Structure, Thermal Bridges in Building Construction — 2D Square Column, Thermal Bridges in Building Construction — 3D Iron Bar, and Thermal Bridges in Building Construction — 3D Two Floors models have been updated to match the 2017 version of the ISO norm 10211.

Backward Compatibility with Version 6.0

- 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, Cp, the property was taken from the material **Basic** group. In version 6.1, the material properties Cp_A and Cp_M are taken from the groups **Austenite Phase** and **Martensite Phase**, respectively. Identity rules enable to ensure backward compatibility in most cases. However, if different values were set for Cp and Cp_A or Cp_M , the results may differ between versions 6.0 and 6.1.
- In the Symmetry for Surface-to-Surface Radiation feature of the Surface-to-Surface Radiation interface, the 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 an Open Boundary condition on boundaries of Shape Memory Alloy or Building Material domains.
- In the Surface-to-Surface Radiation interface features, the diffuse irradiance variables (Idiff, Idiffu_band, Idiffd_band, Idiff_band) are set to 0 when the Include diffuse irradiance check box is not selected.
- In the settings of the Surface-to-Surface Radiation interface, the Mesh element selfirradiation 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 towards the center of curvature.
- The speed of sound variable definition has been corrected in the **Fluid** and **Moist Ai**r 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 $T_{\rm ext}$. In other cases, $T_{\rm ext}$ is set to the value that was set to the model input in version 5.6 or earlier so that the model behavior is unchanged.
- In Heat Transfer interfaces, in the Deposited Beam Power boundary condition, for User defined beam profiles, the beam intensity is now defined from its value in the beam cross section, which corresponds to the deposited beam power divided by the cosine of the incidence angle. Nevertheless, for User defined beam profiles, the models and API scripts created in version 5.6 or earlier behave identically in version 6.0.
- In the **Opaque Surface** feature of the Surface-to-Surface Radiation interface, the coordinate system choice is restricted to the boundary system. When opening a

model created in version 5.6 or earlier with another coordinate system selected, this input should be checked for consistency.

- In the Surface-to-Surface Radiation interface, when the **Expression** or **Tolerance** settings had been modified in the **View Factor Update** section of a model created in version 5.6 or earlier, these settings are reported in the new **View Factor** section.
- A modification of the numerical results within the solver tolerance range may be observed when computing surface-to-surface radiation models created in version 5.6 or earlier, due to the introduction in the Surface-to-Surface interface of the Jacobian contribution option set by default to Only local contributions to radiosity. When opening a model created in version 5.6 or earlier, a warning is displayed under the Surface-to-Surface Radiation interface. Note that the accuracy level remains the same because the convergence criteria and the solver tolerance are unchanged.
- The Local Thermal Nonequilibrium predefined multiphysics interface and the Local Thermal Nonequilibrium multiphysics coupling are now obsolete. When opening a model created in version 5.6 or earlier, this feature is not migrated. It should be manually replaced by the Porous Medium feature with the Porous medium type set to Local thermal nonequilibrium. Support for features marked as obsolete is not ensured in future versions; thus it is strongly recommended to transfer this feature to the new feature.
- In the Moisture Transport interface, the **Hygroscopic Porous Medium** feature not supporting the **Porous Material** functionality is now obsolete. When opening a model created in version 5.6 or earlier, this feature is not migrated. It should be manually replaced by the new **Hygroscopic Porous Medium** feature. Support for features marked as obsolete is not ensured in future versions; thus it is strongly recommended to transfer this feature to the new feature.

Backward Compatibility with Version 5.5

The Heat and Moisture multiphysics coupling feature is no longer applicable to domains where the Fluid (Heat Transfer interface) feature is active. Models built in version 5.5 and before, where the Heat and Moisture multiphysics feature and the Fluid feature with the Moist air option have been used, are updated when opened in version 5.6. The Fluid feature is then replaced by the Moist air feature so the multiphysics coupling is still active. In the unexpected case where the Heat and Moisture multiphysics feature and the Fluid feature with an option other than Moist air were combined, the multiphysics coupling is no longer active in version 5.6, which is likely to modify the model solution.

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

Liquid & Gas Properties Module

Improved Functionality in Version 6.1

- Evaluation of thermodynamic properties is now faster. Models where a significant part of the solution time is spent evaluating functions from the **Thermodynamics** functionality can now be solved in 30% less time.
- The filter search field, used when adding species to a thermodynamic system, has been improved. Filtered species can now be entered one at a time, from the top, using the Enter key. Also, it is no longer necessary to update the filter each time a species is added to the **Selected species** list.

New Model in Version 6.1

CO2 STORAGE IN A GEOLOGIC FORMATION

This example simulates the underground storage of CO2 in a part of the Johansen formation off the coast of Norway. The CO2 is injected using an injection well at a rate of 15 kg/s over a period of 25 years, after which the well is closed. The model is used to compute the spreading of CO2 over the simulation domain during the injection phase and during the 25 years directly following the shut down of the injection well. The pressure- and temperature-dependent density and viscosity of supercritical CO2 are computed using the Peng–Robinson equation of state and the Brokaw viscosity model with high-pressure correction.

MEMS Module

New Functionality in Version 6.1

NEW MULTIPHYSICS INTERFACE: PYROELECTRICITY

The new Pyroelectricity multiphysics interface combines a Heat Transfer in Solids interface and an Electrostatics interface, coupled by the new **Pyroelectricity** multiphysics coupling. Both the direct pyroelectric effect and the inverse electrocaloric effect can be modeled.

NEW MULTIPHYSICS INTERFACE: PIEZOELECTRICITY AND PYROELECTRICITY

The new Piezoelectricity and Pyroelectricity multiphysics interface combines a Heat Transfer in Solids interface, an Electrostatics interface, and a Solid Mechanics interface. Three multiphysics couplings are also used in these multiphysics interfaces:

Pyroelectricity, **Thermal Expansion**, and **Piezoelectric Effect**. This provides a full coupling between pyroelectric effects, piezoelectricity, and thermal expansion of the material.

SOLID MECHANICS INTERFACE IN ID AND ID AXISYMMETRY

For details, see Solid Mechanics interface in 1D and 1D axisymmetry in the release notes for the Structural Mechanics Module.

TESTING OF MATERIAL MODELS

For details, see Testing of Material Models in the release notes for the Structural Mechanics Module.

MATERIALS ON BOUNDARIES OF SOLIDS

For details, see Materials on Boundaries of Solids in the release notes for the Structural Mechanics Module.

MULTIPHYSICS INTERFACES FOR THIN-FILM DAMPING

For details, see Multiphysics Interfaces for Thin-Film Damping in the release notes for the Structural Mechanics Module.

MAGNETOSTRICTION INTERFACE CHANGES

For details, see Magnetostriction Interface Changes in the release notes for the Structural Mechanics Module.

DEAD LOADS IN LINEARIZED BUCKLING ANALYSIS

For details, see Dead Loads in Linearized Buckling Analysis in the release notes for the Structural Mechanics Module.

CONTINUITY USING NITSCHE'S METHOD

For details, see Continuity Using Nitsche's Method in the release notes for the Structural Mechanics Module.

COMPONENT MODE SYNTHESIS IMPROVEMENTS

For details, see Component Mode Synthesis Improvements in the release notes for the Structural Mechanics Module.

NEW SEARCH METHOD FOR CONTACT ANALYSIS

For details, see New Search Method for Contact Analysis in the release notes for the Structural Mechanics Module.

CONTACT USING NITSCHE'S METHOD

For details, see Contact Using Nitsche's Method in the release notes for the Structural Mechanics Module.

NEW FORMULATION FOR CONTACT EQUATIONS

For details, see New Formulation For Contact Equations in the release notes for the Structural Mechanics Module.

IMPROVED SUPPORT FOR SELF-CONTACT

For details, see Improved Support for Self-Contact in the release notes for the Structural Mechanics Module.

POSSIBLE TO TRIGGER A CUTBACK IN THE SOLVER FOR ALL CONTACT METHODS

For details, see Possible to Trigger a Cutback in the Solver for All Contact Methods in the release notes for the Structural Mechanics Module.

BASE EXCITATION

For details, see Base Excitation in the release notes for the Structural Mechanics Module.

GRAVITY AND LINEARLY ACCELERATED FRAME

For details, see Gravity and Linearly Accelerated Frame in the release notes for the Structural Mechanics Module.

LOADS GIVEN AS RESULTANT

For details, see Loads Given as Resultant in the release notes for the Structural Mechanics Module.

RESULTS IN LOCAL DIRECTIONS

For details, see Results in Local Directions in the release notes for the Structural Mechanics Module.

NEW INPUT OPTIONS FOR NONISOTROPIC ELASTIC MATERIALS

For details, see New Input Options for Nonisotropic Elastic Materials in the release notes for the Structural Mechanics Module.

MULTIPLICATIVE FORMULATION FOR PIEZOELECTRICITY

For details, see Multiplicative Formulation for Piezoelectricity in the release notes for the Structural Mechanics Module.

CELL PERIODICITY CAN CREATE MATERIAL DATA ON COMPLIANCE FORM

For details, see Cell Periodicity can Create Material Data on Compliance Form in the release notes for the Structural Mechanics Module.

EXTENDED BUILT-IN MATERIAL

For details, see Extended Built-In Material in the release notes for the Structural Mechanics Module.

LIBRARY OF PARTS FOR HOMOGENIZATION OF MICROSTRUCTURES

For details, see Library of Parts for Homogenization of Microstructures in the release notes for the Structural Mechanics Module.

RELEASED DEGREES OF FREEDOM IN RIGID CONNECTOR

For details, see Released Degrees of Freedom in Rigid Connector in the release notes for the Structural Mechanics Module.

SINGULARITY SUPPRESSION FOR TWO-POINT RIGID CONNECTOR

For details, see Singularity Suppression for Two-Point Rigid Connector in the release notes for the Structural Mechanics Module.

NEW FORMULATION OF RIGID MOTION SUPPRESSION

For details, see New Formulation of Rigid Motion Suppression in the release notes for the Structural Mechanics Module.

IMPROVED PERFORMANCE FOR RANDOM VIBRATION EVALUATION

For details, see Improved Performance for Random Vibration Evaluation in the release notes for the Structural Mechanics Module.

RIGID DOMAIN RENAMED TO RIGID MATERIAL

For details, see Rigid Domain Renamed to Rigid Material in the release notes for the Structural Mechanics Module.

REDUCED NUMBER OF DEFAULT PLOTS

For details, see Reduced Number of Default Plots in the release notes for the Structural Mechanics Module.

PREVIEW PLOT FOR RAYLEIGH DAMPING

For details, see Preview Plot for Rayleigh Damping in the release notes for the Structural Mechanics Module.

NEW OPTION FOR GENERATION OF MATERIAL DATA IN CELL PERIODICITY

For details, see New Option for Generation of Material Data in Cell Periodicity 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.

RESTRUCTURED REDUCED-ORDER MODEL STUDIES

For details, see Restructured Reduced-Order Model Studies in the release notes for the Structural Mechanics Module.

LIMITATIONS OF SELECTIONS ON AXIS OF ROTATION

For details, see Limitations of Selections on Axis of Rotation in the release notes for the Structural Mechanics Module.

REMOVED SECTION IN PHYSICS INTERFACE SETTINGS

For details, see Removed Section in Physics Interface Settings in the release notes for the Structural Mechanics Module.

New Models in Version 6.1

CAPACITIVE MICROMACHINED ULTRASONIC TRANSDUCER

This model demonstrates the operation of a capacitive micromachined ultrasonic transducer (CMUT) for converting ultrasound into electrical signal in high-resolution imaging applications. The transducer is connected to an external circuit and a Harmonic Perturbation boundary load represents the ultrasonic energy. The model also analyzes force-displacement characteristics of the transducer.

ELECTROSTATIC CHUCK

This model demonstrates the operation of an electrostatic chuck used to secure a wafer on a temperature-controlled platform during wafer processing. The model computes the pressure-dependent cooling of the wafer using **Electromechanical Forces**, **Fluid– Structure Interaction**, **Nonisothermal Flow**, and **Thermal Expansion** couplings.

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

This tutorial demonstrates how to build complex 3D geometry more efficiently using the layer-by-layer method that resembles actual MEMS fabrication. This model creates the biased resonator by importing layouts from a GDS file and by using operations available in the CAD Import Module and the Design Module. The model solves for the eigenmodes of the structure.

PRESTRESSED MICROMIRROR VIBRATIONS: THERMOVISCOUS-THERMOELASTICITY COUPLING

This model analyzes the operation of a prestressed micromirror in air. The model includes the effects of thermal losses and thermoviscous damping on the vibration response. This model demonstrates the use of the **Thermoviscous Acoustics– Thermoelasticity Boundary** multiphysics coupling.

PYROELECTRIC DETECTOR

This model demonstrates the operation of a lithium niobate (LiNbO_3) pyroelectric detector used for measuring laser energy. The detector is connected to an external circuit and a pulsed heat flux represents the laser energy. It computes the temperature

evolution in the material and the pyroelectric current and is set up using the Piezoelectricity and Pyroelectricity interface.

Metal Processing Module

New Functionality in Version 6.1

ALPHA-BETA PHASE TRANSFORMATION INTERFACE

A new physics interface, Alpha-Beta Phase Transformation, has been added to simulate phase transformations in α - β titanium alloys.

NEW PHASE TRANSFORMATION MODEL

A **Hyperbolic rate** phase transformation model has been added to the phase transformation interfaces.

ID AND ID AXISYMMETRIC FORMULATIONS

The physics interfaces and multiphysics couplings are now available also for 1D and 1D axisymmetric components.

TRANSFORMATION TEMPERATURE SMOOTHING

A **Transformation temperature smoothing** option is available in conjunction with the **Define temperature limits** option. This can be used to make activation and deactivation of phase transformations less abrupt.

PHASE-DEPENDENT ELECTROMAGNETIC PROPERTIES

Electromagnetic material properties can now be defined per phase, and corresponding compound material properties can be computed. This is useful in the context of induction heating.

PARAMETERIZED TTT DIAGRAMS

The phase transformation model formulations **Parameterized TTT diagram** and **Parameterized TTT diagram, fixed exponent** have been added. They are used to characterize diffusive phase transformations using simplified, parameterized descriptions of time-temperature-transformation (TTT) diagram curves.

MISCELLANEOUS

- The effect of the **Initial phase fraction** can optionally be included in the rate expression for the **Johnson-Mehl-Avrami-Kolmogorov (JMAK)** phase transformation model.
- When **Compute transformation times** is used, a transformation diagram is automatically generated. This can be used to compute continuous cooling transformation (CCT) and TTT diagrams.

New Models in Version 6.1

WELDING OF A TITANIUM PLATE

This model simulates arc welding of a titanium plate made from an alpha-beta titanium alloy. During a weld pass, the material experiences a thermal transient, including both heating and cooling. Phase transformations are modeled, and the evolution of the phase composition over time is computed.

Microfluidics Module

New Functionality in Version 6.1

MULTIPHASE MATERIALS FOR MULTIPHASE FLOW COUPLINGS

The **Two-Phase Flow, Level Set**, **Two-Phase Flow, Phase Field**, and **Three-Phase Flow, Phase Field** multiphysics couplings have a new option to obtain the effective material properties from a **Multiphase Material** with built-in mixing rules. This is especially effective when coupling these multiphysics interfaces with other physics, such as heat transfer or electrostatics, since the multiphase material will use appropriate mixing rules for nonfluid material properties. In older versions, this would require you writing user-defined expressions based on the volume fraction of each fluid phase to compute the effective material properties used in each physics interface.

INELASTIC NON-NEWTONIAN MATERIAL PROPERTY GROUPS

Dedicated material property groups have been added for all available inelastic non-Newtonian models. The material property group contains all necessary material parameters and the apparent viscosity expression. It picks up the shear rate from the fluid flow interface to define the dynamic viscosity in the **Basic (def)** material model by means of a synchronization rule. Thus, an inelastic non-Newtonian model can be selected directly by adding the corresponding **Material Properties** group as a subnode to a material node.

IMPROVEMENTS TO THE LEVEL SET INTERFACE

In the Level Set interface, a new **Porous Medium** feature now supports the functionality to update the definition of the porosity in the **Materials** node. The new **Level Set in Porous Media** interface includes this node by default.

Updated Applications in Version 6.1

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. Applications using Two-Phase Flow, Level Set, Two-Phase Flow, Phase Field, or Three-Phase Flow, Phase Field interfaces now include a **Multiphase Material** to compute the effective material properties from volume fractions and the material properties of each phase.

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.

Backward Compatibility with Version 5.5 and Earlier

TWO-PHASE FLOW

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

Molecular Flow Module

New Functionality in Version 6.1

SYMMETRY

The new global **Symmetry** node extends the capability of the **Plane Symmetry** node. In 2D, you can specify either a single symmetry plane or two orthogonal planes. In 3D, you can now specify either one, two, or three symmetry planes as long as all of the planes are mutually orthogonal.

With this node, you can also apply sector symmetry to your models. This can be done in two different ways: by selecting the boundaries in the start plane and end plane of a single sector (or unit cell) of the symmetric geometry or by selecting the start plane and a reflection plane (where the model geometry is assumed to be one half of a single sector).

Updated Applications in Version 6.1

DIFFERENTIAL PUMPING

This model has been updated to use the new Symmetry feature.

MOLECULAR FLOW THROUGH AN RF COUPLER

This model has been updated to use the new Symmetry feature.

Backward Compatibility with Version 6.0 and Earlier

PLANE SYMMETRY

The **Plane Symmetry** node is still available in version 6.1. Any models created with a previous version using the **Plane Symmetry** nodes will generate identical results when computed in version 6.1. However, a warning message will be displayed stating, "This feature is obsolete and will be removed in future versions. Consider using the new global **Symmetry** feature instead. Do not attempt to use both of these features together." Consequently, replacing the old **Plane Symmetry** with the new **Symmetry** feature is recommended.

Multibody Dynamics Module

New Functionality in Version 6.1

PHYSICS INTERFACE FOR WIRES

For details, see Physics Interface for Wires in the release notes for the Structural Mechanics Module.

CONTINUITY USING NITSCHE'S METHOD

For details, see Continuity Using Nitsche's Method in the release notes for the Structural Mechanics Module.

COMPONENT MODE SYNTHESIS IMPROVEMENTS

For details, see Component Mode Synthesis Improvements in the release notes for the Structural Mechanics Module.

NEW SEARCH METHOD FOR CONTACT ANALYSIS

For details, see New Search Method for Contact Analysis in the release notes for the Structural Mechanics Module.

CONTACT USING NITSCHE'S METHOD

For details, see Contact Using Nitsche's Method in the release notes for the Structural Mechanics Module.

NEW FORMULATION FOR CONTACT EQUATIONS

For details, see New Formulation For Contact Equations in the release notes for the Structural Mechanics Module.

IMPROVED SUPPORT FOR SELF-CONTACT

For details, see Improved Support for Self-Contact in the release notes for the Structural Mechanics Module.

POSSIBLE TO TRIGGER A CUTBACK IN THE SOLVER FOR ALL CONTACT METHODS

For details, see Possible to Trigger a Cutback in the Solver for All Contact Methods in the release notes for the Structural Mechanics Module.

BASE EXCITATION

For details, see Base Excitation in the release notes for the Structural Mechanics Module.

GRAVITY AND LINEARLY ACCELERATED FRAME

For details, see Gravity and Linearly Accelerated Frame in the release notes for the Structural Mechanics Module.

LOADS GIVEN AS RESULTANT

For details, see Loads Given as Resultant in the release notes for the Structural Mechanics Module.

RESULTS IN LOCAL DIRECTIONS

For details, see Results in Local Directions in the release notes for the Structural Mechanics Module.

NEW INPUT OPTIONS FOR NONISOTROPIC ELASTIC MATERIALS

For details, see New Input Options for Nonisotropic Elastic Materials in the release notes for the Structural Mechanics Module.

RELEASED DEGREES OF FREEDOM IN RIGID CONNECTOR

For details, see Released Degrees of Freedom in Rigid Connector in the release notes for the Structural Mechanics Module.

SINGULARITY SUPPRESSION FOR TWO-POINT RIGID CONNECTOR

For details, see Singularity Suppression for Two-Point Rigid Connector in the release notes for the Structural Mechanics Module.

NEW FORMULATION OF RIGID MOTION SUPPRESSION

For details, see New Formulation of Rigid Motion Suppression in the release notes for the Structural Mechanics Module.

RIGID DOMAIN RENAMED TO RIGID MATERIAL

For details, see Rigid Domain Renamed to Rigid Material in the release notes for the Structural Mechanics Module.

REDUCED NUMBER OF DEFAULT PLOTS

For details, see Reduced Number of Default Plots in the release notes for the Structural Mechanics Module.

PREVIEW PLOT FOR RAYLEIGH DAMPING

For details, see Preview Plot for Rayleigh Damping 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.

REMOVED SECTION IN PHYSICS INTERFACE SETTINGS

For details, see Removed Section in Physics Interface Settings in the release notes for the Structural Mechanics Module.
Nonlinear Structural Materials Module

New Functionality in Version 6.1

VISCOPLASTICITY IMPROVEMENTS

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

- A new framework to handle large viscoplastic strains has been implemented. It is based on the multiplicative decomposition of deformation gradients.
- The new formulation for large strains makes it possible to add **Creep** and **Viscoplasticity** nodes under the **Hyperelastic Material** node in the Solid Mechanics interface.
- Two new viscoplastic models have been introduced: the **Bingham** model and the **Peric** model. It is also now possible to have a **User defined** viscoplastic model.
- A new general framework provides significant improvements in computational speed and memory usage. For larger models, a speedup by a factor of 10 or more can be expected.
- The **Chaboche** and **Perzyna** models have been updated. The main change is with respect to the time integration. You can now select the type of scheme that is used. In the **Time Stepping** section, you can set **Method** to **Automatic**, **Backward Euler**, or **Domain ODEs**. The latter method provides backward compatibility with versions prior to 6.0 and is usually significantly slower for large models. This section is not present in the Layered Shell interface or in the layered material models in the Membrane and Shell interfaces.

As an effect of these changes, there are some potential backward compatibility issues:

- API code that references the old field names in order to modify solver settings will fail.
- Manual variable scaling of the viscoplastic degrees of freedom will be lost, even when **Domain ODEs** is selected.
- The UI and some API names for the Viscoplasticity node have been updated.

HYPERELASTICITY IMPROVEMENTS

The functionality in the Hyperelastic Material node has been subject to major changes:

- A new framework to handle the compressibility for all hyperelastic materials has been implemented. Five new volumetric strain energy variables are now available. As a result, all hyperelastic material models now have a compressible option.
- It is now possible to add **Phase Field** damage to **Hyperelastic Material** nodes in the Solid Mechanics interface.
- The stress variables are restructured in the **Hyperelastic Material** node and its subnodes in the Solid Mechanics, Shell, Layered Shell, and Membrane interfaces. As a result, large strain viscoelasticity now uses the elastic isochoric stress as the source for the relaxation equations.
- The incompressible or nearly incompressible hyperelastic materials now generate correct postprocessing results when inelastic stresses are added.
- The new formulation for large strains makes it possible to add **Creep** and **Viscoplasticity** nodes under the **Hyperelastic Material** node in the Solid Mechanics interface.
- For incompressible hyperelastic materials, the volumetric damping is set to zero for isotropic loss factor damping and viscous damping.
- The **User defined** hyperelastic strain energy density can be expressed as a function of the elastic deformation gradient.
- For the user-defined hyperelastic material, there is a new **Advanced** section in which variables for the equivalent stiffness needed for contact problems can be defined.

ENHANCEMENTS TO DAMAGE

- Some user inputs in the **Damage** node can now be directly imported from the **Materials** node.
- The **Phase field** damage is added to the **Hyperelastic Material** node in the Solid Mechanics interface.
- The default solver suggestions for Phase field damage have been improved.
- It is now possible to combine the **Damage** node with the **Plasticity**, **Soil Plasticity**, and **Viscoplasticity** nodes.
- For **Scalar** damage, the expression for the damaged stress variable is corrected when used in combination with the mixed formulation.

ENHANCEMENTS TO WRINKLING

- It is now possible to model wrinkling of rubber or biological tissue by adding the **Wrinkling** node under the **Hyperelastic Material** node in the Membrane interface.
- The **Plasticity**, **Viscoplasticity**, **Creep**, and **Inelastic Strain Rate** nodes are now overridden by the **Wrinkling** node in the Membrane interface.

ENHANCEMENTS TO PLASTICITY

- The **Plasticity**, **Porous Plasticity**, and **Shape Memory Alloy** nodes now have an **Advanced** section where it is possible to control the number of iterations and relative tolerance used by the plasticity algorithm.
- The UI and some API names for the **Plasticity** node have been updated.

ADDITIONAL FUNCTIONALITY FOR THIN LAYER

In the Solid Mechanics interface, the new **Thin Layer** node can add **Hyperelastic** and **Nonlinear Elastic** material models. Other inelastic effects can also be added to material models under the **Thin Layer** node by adding, for example, **Damage**, **Creep**, or **Plasticity** subnodes.

ADIABATIC HEATING

A new framework to handle adiabatic heating in solids, shells, and membranes is implemented. Adiabatic heating is important when rapid deformations dissipate energy, which in turn increases the temperature in the object. To incorporate this effect, you can add an **Adiabatic Heating** subnode to the Solid Mechanics, Layered Shell, Shell, or Membrane interfaces.

NEW BUILT-IN MATERIAL

A new material, **Rubber**, has been added to the **Built-in** folder in the Material Library. It contains material data for a generic rubber material that can be used in several of the material models under the **Hyperelastic Material** node.

Also, the existing **Structural Steel** material has been augmented with many new nonlinear material properties to be used with, for example, plasticity or creep.

New Models in Version 6.1

WRINKLING OF A CYLINDRICAL MEMBRANE WITH VARYING THICKNESS

In this example, wrinkling is studied in a cylindrical membrane of nonuniform thickness under axial and pressure loading. The membrane is modeled as an

incompressible Mooney–Rivlin material. During axial stretching, certain portions of the membrane undergo wrinkling; however, the wrinkling disappears when the internal pressure is increased.

INFLATION OF A SQUARE HYPERELASTIC AIRBAG

In this example, a square airbag made of a hyperelastic compressible neo-Hookean material is inflated using pressurized air. Compressive stresses trigger wrinkling in some regions of the thin airbag.

The wrinkling behavior is modeled using tension field theory, which provides a correct description of the stress distribution and the wrinkling pattern in the inflated membrane.

CONTACT ANALYSIS OF A RUBBER BOOT SEAL

In this example, the deformation of a rubber boot seal connected to a rigid pipe is studied. As the pipe rotates and the seal deforms, the flanges of the seal self-intersect, and the seal also comes into contact with the pipe.

INTERFACE MODELS WITH A THIN LAYER FEATURE

This model demonstrates alternative implementations used for describing a thin layer as well as the impact of the choice on the continuity of the displacement and stress fields. The model shows how a perfect interface can be obtained by asymptotically changing the material parameters.

SUBMODELING WITH PLASTICITY

This tutorial model demonstrates how to incorporate a localized nonlinearity into a submodel.

In this example, an initial elastic analysis reveals that a small region of a structure has stresses above the yield limit. To improve the results, an elastoplastic material model is added to a submodel.

The global model is analyzed using linear elasticity in this case, but this is not a necessary assumption.

Optimization Module

New Functionality in Version 6.1

GRADIENT-BASED OPTIMIZATION FOR EIGENVALUE PROBLEMS

Gradient-based optimization is now supported for eigenvalue problems when the objective (or constraints) depends explicitly on the eigenvalues. Related to this, the **Polynomial Boundary** feature has been extended to 3D, and a **Polynomial Shell** feature has been added. This combination of new functionality enables gradient-based shape optimization of shells in terms of the lowest eigenfrequency.

MILLING CONSTRAINTS FOR TOPOLOGY OPTIMIZATION

For topology optimization, the **Density Model** feature has been extended with support for milling constraints. The resulting topologies can often be used as inspiration for geometries subject to casting constraints.

CONTINUITY FOR FREE SHAPE AND POLYNOMIAL BOUNDARY FEATURES

The Free Shape Boundary, Free Shape Shell, 2D Polynomial Boundary, and Polynomial Shell features have been extended with settings for continuity so that it is easier to preserve slope continuity during shape optimization.

The **Transformation** feature has been extended with support for free rotation and rotation around an axis confined to a plane.

The **Fixed Boundary** feature used to be a hidden default boundary condition in shape optimization, but it is now visible in the user interface.

ADDITIONAL NEW FUNCTIONALITY

The **Control Function** feature has been extended to support piecewise polynomials. The slope is continuous between the polynomials. This is also the case for periodic functions.

The default solver suggestion for optimization has been changed to use an iterative solver, which reduces memory consumption for large 3D shape optimization problems.

MAXIMIZING THE EIGENFREQUENCY OF A SHELL

This model demonstrates how to use the **Polynomial Shell** feature to increase the lowest eigenfrequency of a shell by deforming its shape.

BRACKET — STRESS OPTIMIZATION WITH FATIGUE EVALUATION

This example demonstrates shape optimization together with a structural fatigue evaluation and minimizes the approximate maximum stress of a bracket. The mass and stiffness are constrained with respect to the initial values. The fatigue life is improved after optimization as compared to the initial geometry.

TOPOLOGY OPTIMIZATION OF A STEP THRUST BEARING

Topology optimization is used to design a step thrust bearing capable of supporting a large load. Designs with different numbers of grooves can be identified by modifying the initial design. The result is verified with a body-fitted mesh.

SHAPE OPTIMIZATION OF A STEP THRUST BEARING

Shape optimization is used to design a step thrust bearing capable of supporting a large load. **Control Function** features are used to move the steps in the azimuthal direction.

TOPOLOGY OPTIMIZATION OF A BEAM WITH MILLING CONSTRAINTS

The mass of an aluminum beam is minimized through topology optimization subject to a displacement constraint and a distributed load. This previously available model has been modified to demonstrate the use of the new milling constraints with milling from the x- and y-axes.

EXPORTING AND IMPORTING A TOPOLOGY-OPTIMIZED HOOK

In this topology optimization of a hook, the simulation finds the optimal material distribution when the structural part is subjected to two load cases. This previously available model has been updated to use the new milling constraints available in the **Density Model** feature.

SHAPE OPTIMIZATION OF A TUNING FORK

Extending the Tuning Fork model, this example shows how to set up an **Optimization** study to determine the prong length at which the fork vibrates at the standard concert pitch, 440 Hz. This is achieved using the **Deformed Geometry** interface so that gradient-based optimization can be utilized.

TOPOLOGY OPTIMIZATION AND VERIFICATION OF AN ACOUSTIC MODE IN A 2D ROOM

This model has been updated to use the new milling functionality of the **Density Model** feature.

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 6.1

IMPROVED PSEUDORANDOM NUMBER GENERATION

The Particle Tracing Module includes a variety of features that rely on pseudorandom number generation (PRNG), such as the following:

- Monte Carlo modeling of collisions between model particles and molecules of a background gas
- · Brownian motion of small particles in a fluid
- Motion of particles in a turbulent fluid
- Boundary conditions that cause particles to be diffusively or isotropically reflected
- · Secondary particle emission at a specified probability
- · Conditional particle-wall interactions

For such features, the calls to pseudorandom number generators have been extensively reviewed and revised. The new expressions are much less likely to incur unwanted correlations between random numbers that should be completely uncorrelated. This includes random forces acting on different particles, different components of certain random forces, and different physics features or different instances of the same feature.

IMPROVED SAMPLING FROM THERMAL VELOCITY DISTRIBUTIONS

The Particle Tracing Module includes several features that sample the particle speed from a distribution based on temperature when releasing or reflecting particles at a surface. This includes the **Thermionic Emission** particle release feature for the Charged Particle Tracing interface, the **Thermal** velocity distribution type for the **Inlet** node, and the **Thermal Re-Emission** boundary condition. The way in which values of the particle speed are sampled from a distribution based on temperature is now more accurate. The improvement should be most obvious when computing statistics for particles at the extreme low and high ends of the distribution when the sample size is large.

THERMAL VELOCITY DISTRIBUTION AVAILABLE FOR THE SECONDARY EMISSION

A new **Thermal** velocity distribution type is now available for the **Secondary Emission** attribute of **Wall** and **Axial Symmetry** features. Accordingly, you can now sample

secondary particles based on thermal velocity distribution when active particles hit a wall.

New Applications in Version 6.1

MULTIPACTOR SATURATION

The multipactor effect is a resonance phenomenon in which an avalanche of secondary electrons can be released within an RF cavity. This tutorial is a bidirectionally coupled, time-dependent model in which the space charge density of the emitted secondary electrons and the effect of this space charge density on the electric potential in the cavity are computed, along with the electron trajectories. At sufficiently high space charge density, saturation occurs, meaning that the electrons are spread out by their mutual electrostatic repulsion, slowing down some electrons before they can hit the cavity walls and causing the total population of electrons to reach a state of dynamic equilibrium.

CASCADE IMPACTOR

This example simulates particles in a range of sizes that move through and get collected in different levels of a cascade impactor. A cascade impactor is a particle separation device consisting of multiple levels of collecting plates connected through progressively finer nozzles.

Important Fixes in Version 6.1

RELEASE FEATURE LIST IN PARTICLE COUNTER FEATURE

In version 6.0, when a release feature selected in the **Release feature** list of the **Particle Counter** node was disabled, an error occurred. This has been fixed in version 6.1. The selection automatically switches to **All** now, without any error message.

MAGNETOPHORETIC FORCE

The documentation for the **Magnetophoretic Force** node has been revised where an easier way to evaluate derivatives of the magnetic field components is mentioned. This method is useful when magnetic field components are available from other physics interface but the derivatives of these components are not readily available.

IMPROVED PERFORMANCE FOR FORCE CALCULATION

Any force that applies to all domains can now be evaluated slightly faster. For such a case, in the **Domain Selection**, choose **All domains** rather than selecting **Manual** and then

selecting all domains in the geometry. The performance improvement is only achieved when **All domains** is selected in any feature calculating force.

DUPLICATING RELEASE FROM DATA FILE NODE

In version 6.0, the embedded or imported file was duplicated when the **Release from Data File** node was duplicated. When the original **Release from Data File** node was used to import another file, there was an error. Also, the duplicated node in some cases completely overrode the original node without displaying any error message. This has been fixed now. When you duplicate this node, the imported file will not be duplicated; you will need to import the file into every duplicated node.

LINE PLOTS FASTER IN PARTICLE TRAJECTORIES PLOT

In version 6.1, the default of **Maximum number of extra time steps rendered** is changed to **All** in the **Particle Trajectories** plot. Plotting **Line**, **Ribbon**, or **Tube** is now much faster than in version 6.0.

Backward Compatibility with Version 6.0 and Earlier

DUPLICATED RELEASE FROM DATA FILE NODE

If a model created using version 6.0 or earlier contains duplicated **Release from Data File** nodes, it is recommended to delete these nodes, create new nodes, and import the files again, even if the same file is used in each node.

Pipe Flow Module

New Functionality in Version 6.1

PIPE WALL HEAT TRANSFER

A new **Pipe Wall Heat Transfer** multiphysics coupling has been added to connect domains modeled using the Heat Transfer interfaces and edges modeled using the Heat Transfer in Pipes or Nonisothermal Pipe Flow interfaces.

CORRECTION FOR PIPE BENDS

In the Pipe Mechanics interface, you can now specify correction factors for stiffness and stress evaluation in pipe bends. To do this, you add the **Bend** subnode under **Pipe Cross Section**.

STRESS EVALUATION USING REDUCED PIPE THICKNESS

When specifying a **Pipe Cross Section** in the Pipe Mechanics interface, you can now enter a reduced pipe thickness for the stress evaluation. This can be used when you are, for example, taking corrosion allowance into account.

LIBRARY OF PIPE GEOMETRIES

In the **Part Libraries**, the content of the **Pipes** folder in the **COMSOL Multiphysics** branch has been augmented. The folder now contains parameterized geometries for the following common piping parts: straight pipe, bend, reducer, and T-junction. These geometries can be used for detailed analysis, using the fluid flow or structural mechanics interfaces. Such a part can be inserted into a pipe system modeled using the Pipe Flow or Pipe Mechanics interface, through the **Pipe Connection** or **Structure-Pipe Connection** multiphysics couplings.

Because the new parts are more general in scope, the old straight_pipe and bent_pipe parts have been removed.

Updated Models in Version 6.1

MOLD COOLING

This model has been updated to use the Pipe Wall Heat Transfer multiphysics coupling.

GROUND HEAT RECOVERY

This model has been updated to use Pipe Wall Heat Transfer multiphysics coupling.

Plasma Module

New Functionality in Version 6.1

INDUCTIVELY COUPLED PLASMA WITH RF BIAS INTERFACE

The new Inductively Coupled Plasma with RF Bias interface couples the Plasma, Time Periodic and Magnetic Fields interfaces and is used to model inductively coupled plasma reactors with a periodic RF bias (also known as ICP or CCP reactors).

FREQUENCY-TIME PERIODIC STUDY

The **Frequency-Time Periodic** study is used to compute the periodic steady-state solution for models using the Plasma, Time Periodic interface together with the electromagnetic fields in the frequency domain. This study is available for the Plasma Inductive Coupled Plasma with RF Bias interface.

ADD-IN TO ADD A PLASMA CHEMISTRY FROM A FILE

The new **Plasma Chemistry** add-in makes it possible to import a file that automatically creates a complete plasma chemistry. In the Model Builder, features like **Electron impact reactions**, **Species** and others are automatically created from instructions you can specify in a file. The following aspects of a plasma chemistry can be specified in the file:

- · Species properties including thermodynamic parameters
- Electron impact reactions from cross section and rate constants
- · Heavy species reactions
- Surface reactions

New models that use the Plasma Chemistry add-in are the following:

- Model of an Argon/Chlorine Inductively Coupled Plasma Reactor with RF Bias
- Model of an Argon/Oxygen Inductively Coupled Plasma Reactor
- Model of an Argon/Oxygen Capacitively Coupled Plasma Reactor

COUPLING THE PLASMA, TIME PERIODIC INTERFACE WITH HEAT TRANSFER AND FLUID FLOW

It is now possible to coupled the Plasma, Time Periodic interface with the Heat Transfer in Fluid and Laminar Flow interfaces. In the **Model Inputs** section of the **Plasma Model** node, the **Temperature**, **Absolute pressure**, and **Velocity** fields can be linked to the Heat Transfer in Fluid and Laminar Flow interfaces. In the Heat Transfer in Fluid and Laminar Flow interfaces, the **Fluid Properties** and **Fluid** nodes can receive period-averaged quantities computed in the plasma model, like the period-averaged density, thermal conductivity, and dynamic viscosity.

The **Heat Source** feature now includes the option to receive the period-average heat source for the gas computed in the Plasma, Time Periodic interface.

The new Model of an Argon/Chlorine Inductively Coupled Plasma Reactor with RF Bias model demonstrates the couplings mentioned above.

INFLOW AND OUTFLOW FEATURES FOR THE PLASMA, TIME PERIODIC INTERFACE

The **Inflow** and **Outflow** features in the Plasma, Time Periodic interface make it easy to set boundary conditions for many species related to fluid inlets and outlets.

The new Model of an Argon/Chlorine Inductively Coupled Plasma Reactor with RF Bias and Model of an Argon/Oxygen Capacitively Coupled Plasma Reactor models use these features in the context of complex plasma chemistries.

NEW EQUILIBRIUM DISCHARGE INTERFACES

The **Equilibrium Discharges** interfaces have been replaced by three new interfaces: Equilibrium Discharges, Out-of-Plane Currents; Equilibrium Discharges, In-Plane Currents; and Equilibrium Discharges. The interfaces look mostly identical as before, with the exception that the new Magnetic and Electric Fields interface is used in the Equilibrium Discharges, In-Plane Currents interface. The choice of which interface to use is now based on the plane for which the currents are to be computed.

The use of the Magnetic and Electric Fields interface increases numerical stability and significantly reduces computational time, as can be seen in the *Plasma Pulsed Arc* model.

INCONSISTENT STABILIZATION

It is now possible to add isotropic diffusion for ions in the **Inconsistent Stabilization** section of the Plasma, Time Periodic interface. This option is available when the **Heavy** species selection is set to **Base geometry**.

For the Plasma interface, the isotropic diffusion for ions and electrons in the **Inconsistent Stabilization** section is extended for when the **Finite element**, **log formulation** (linear shape function) and **Finite element**, **log formulation** (quadratic shape function) are used.

To enable the **Inconsistent Stabilization** section, click the **Show More Options** button and select **Stabilization** in the **Show More Options** dialog box.

New and Updated Applications in Version 6.1

MODEL OF AN ARGON/CHLORINE INDUCTIVELY COUPLED PLASMA REACTOR WITH RF BIAS

This tutorial model shows how to prepare a model for an ICP or CCP reactor using the new Inductively Coupled Plasma with RF Bias interface. A plasma chemistry for argon and chlorine is used, and strategies to model electronegative discharges are discussed in the documentation. This model uses the new Plasma Chemistry add-in, which automatically creates plasma chemistries from a file.

MODEL OF AN ARGON/OXYGEN INDUCTIVELY COUPLED PLASMA REACTOR

This model solves for an ICP reactor with a mixture of argon and oxygen. The plasma model is coupled with fluid flow and gas heating. Strategies to model electronegative discharges are discussed in the documentation. This model uses the new add-in Plasma Chemistry that automatically creates plasma chemistries from a file.

MODEL OF AN ARGON/OXYGEN CAPACITIVELY COUPLED PLASMA REACTOR

This model solves for an CCP reactor with a mixture of argon/oxygen. Strategies to model complex chemistries and electronegative discharges in the Plasma, Time Periodic interface are discussed. This model uses the new Plasma Chemistry add-in, which automatically creates plasma chemistries from a file.

DRY AIR BOLTZMANN ANALYSIS

This model solves the Boltzmann equation in the two-term approximation for a mixture of nitrogen and oxygen that represents dry air. Electron transport coefficients and source terms are computed by suitable integration of the electron energy distribution function over electron impact cross sections. The documentation shows how to prepare the electron energy distribution function to export so that it can be used in the Plasma interface. It is discussed what the data plasma fluid models need and how this can be computed with the Boltzmann Equation, Two-Term Approximation interface.

PLASMA PULSE ARC AND PLASMA ARC MODELS

The Plasma Pulsed Arc and Plasma Arc models use the new Magnetic and Electric Fields interface that combines the Magnetic Fields and Electric Currents interfaces.

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 version 6.1.

Backward Compatibility with Version 5.5 and Earlier

There are new versions of the following interfaces: Plasma; Drift Diffusion; Heavy Species Transport; Inductively Coupled Plasma; Microwave Plasma; and Boltzmann Equation, Two-Term Approximation. When opening models created with version 5.5 and earlier, a warning sign appears saying, "This physics interface is obsolete and will be removed in a future version. A new version of the physics interface is available from the Add Physics window." These models can still be changed and will run as before, but it is encouraged to recreate the models using version 6.1.

Porous Media Flow Module

New Functionality in Version 6.1

IMPROVED HANDLING OF FRACTURES

In the Darcy's Law and Fracture Flow interfaces, the following improvements are available:

- Fractures are now defined using phase-specific properties in the **Fracture** node. The **Fluid** subnode describes the fluid properties, and the **Fracture Material** subnode accounts for the fracture properties, such as porosity or transmissivity.
- It is now possible to model Darcian or Non-Darcian flow in fractures.
- The fractures can be highly conductive, like open channels, or can be represented by thin barriers.
- Fractures now also support the ideal gas and the linearized storage model.

FRACTURES FOR TRANSPORT OF DILUTED SPECIES

A new **Fracture** node is now available in both the Transport of Diluted Species in Porous Media and the Transport of Diluted Species in Fractures interfaces. The new node is consistent with other fracture nodes for porous media transport, and it includes two subnodes, **Fluid** and **Fracture Material**. The **Fluid** subnode describes the fluid, velocity, and diffusion properties, and the **Fracture Material** subnode accounts for the fracture properties.

IMPROVEMENTS TO THE LEVEL SET INTERFACE

In the Level Set interface, a new **Porous Medium** feature now supports the functionality to update the definition of the porosity in the **Materials** node. This feature is included by default in the new Level Set in Porous Media interface includes as well as in the Brinkman Equations, Two-Phase Flow, Level Set multiphysics interface.

STORAGE MODEL FOR UNSATURATED POROUS MEDIA

An option for the **Storage model** in the **Unsaturated Porous Medium** node has been added for the Darcy's Law interface. It is now possible to define storage as a function of the liquid content.

IMPROVEMENTS TO HYGROSCOPIC POROUS MEDIUM

- The **Moisture Flow** multiphysics coupling has been updated so that the evaporation rate variable computed by the Moisture Transport interface is accounted for in the mass balance computed by the Brinkman Equations interface.
- The **Open Boundary** and **Inflow** boundary conditions are now applicable to exterior boundaries adjacent to domains where a **Hygroscopic Porous Medium** feature is active.

USER-DEFINED PHASE TRANSITION

A **User defined** phase transition function in the **Phase Change Material** subnode is now available for the Heat Transfer in Porous Media interface. This option makes it possible to use an accurate phase change description from measured data.

THIN BARRIER MULTIPHYSICS COUPLING

The new **Thin Barrier** node for the Multiphase Flow in Porous Media multiphysics interface makes it possible to add thin layers that act as a resistance for the flow fields of all phases without meshing through the layer's thickness.

CLIMATE DATA: ASHRAE 2021

ASHRAE Weather Data 2021 has been integrated into COMSOL Multiphysics. ASHRAE 2021 contains additional locations; there are more than 8500 stations in the 2021 edition. You can select any of them in the **Ambient Properties** node. The database provides typical climate data for the temperature, humidity, precipitation, and solar radiation.

NEW PREDEFINED PLOTS

With the introduction of the new **Add Predefined Plots** feature, the number of default plots that are produced has been reduced. Several plots that were default plots are now instead optional and can be added from the **Add Predefined Plots** window. There are also several new useful predefined plots.

New Models in Version 6.1

PHASE CHANGE IN A SEMI-INFINITE SOIL COLUMN — LUNARDINI Solution

In this new model, a heat-conduction problem with phase change in a porous material is solved, and the results are compared with the analytical solution, also known as the Lunardini solution. This is the first test case from the InterFrost project.

RESIN TRANSFER MOLDING OF A WIND TURBINE BLADE

This example shows the modeling of a resin transfer molding (RTM) process for a wind turbine blade using the Two-Phase Flow, Level Set, Brinkman Equations interface.

Resin is injected into a preform consisting of different composites with different anisotropic permeabilities.

Polymer Flow Module

New Functionality in Version 6.1

MULTIPHASE MATERIALS FOR MULTIPHASE FLOW COUPLINGS

The **Two-Phase Flow, Level Set**; **Two-Phase Flow, Phase Field**; and **Three-Phase Flow, Phase Field** multiphysics couplings have a new option to obtain the effective material properties from a **Multiphase Material** with built-in mixing rules. This option is especially effective when coupling these multiphysics interfaces with other physics, such as heat transfer or electrostatics, since the multiphase material will use appropriate mixing rules for nonfluid material properties. In older versions, this would require you writing user-defined expressions based on the volume fraction of each fluid phase to compute the effective material properties used in each physics interface.

INELASTIC NON-NEWTONIAN MATERIAL PROPERTY GROUPS

Dedicated material property groups have been added for all available inelastic non-Newtonian models. The material property group contains all necessary material parameters and the apparent viscosity expression. It picks up the shear rate from the fluid flow interface to define the dynamic viscosity in the **Basic (def)** material model by means of a synchronization rule. Thus, an inelastic non-Newtonian model can be selected directly by adding the corresponding **Material Properties** group as a subnode to a material node.

CURING REACTION INTERFACE

The new default feature, No Flux, has been added to the physics interface.

NONISOTHERMAL VISCOELASTIC FLOWS

The **Fluid Properties** node in the Viscoelastic Flow interface now includes the option to specify the temperature dependence of the material properties using a set of predefined thermal functions.

The new **Nonisothermal Flow** multiphysics coupling can be used to couple the Viscoelastic Flow and Heat Transfer in Fluids interfaces. The heat source, corresponding to heating due to irreversible losses, can be accounted for by selecting the **Include irreversible losses** check box.

A new Nonisothermal Flow, Viscoelastic Flow multiphysics interface has been added for convenient setup of nonisothermal viscoelastic flow models. It combines the Viscoelastic Flow and Heat Transfer in Fluids physics interfaces using a **Nonisothermal Flow** multiphysics coupling node.

LICENSE UPDATE FOR PARAMETER ESTIMATION

The **Parameter Estimation** study step, and the BOBYQA, Levenberg–Marquardt, and IPOPT optimization solvers are now available with a Polymer Flow Module license.

IMPROVEMENTS TO THE LEVEL SET INTERFACE

In the Level Set interface, a new **Porous Medium** feature supports the functionality to update the definition of the porosity in the **Porous Matrix** node. This node is included by default in the new Level Set in Porous Media interface as well as in the Brinkman Equations interface and the **Two-Phase Flow, Level Set** multiphysics coupling node.

New Models and Updated Models in Version 6.1

2D NON-NEWTONIAN SLOT-DIE COATING

The model has been substantially reworked to use the new Multiphase Materials functionality. It also now demonstrates how to use the Least-Squares Fit function to estimate material parameters from measured data.

RESIN TRANSFER MOLDING OF A WIND TURBINE BLADE

This example shows the modeling of a resin transfer molding (RTM) process for a wind turbine blade using the Two-Phase Flow, Level Set, Brinkman Equations interface. Resin is injected into a preform consisting of different composites with different anisotropic permeabilities.

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 6.1

FLUENCE RATE CALCULATION

A new domain accumulator feature for computing fluence rate, **Fluence Rate Calculation**, is now available in version 6.1. Fluence rate is defined as the amount of radiation that would hit a small spherical detector at any location in the simulation domain, divided by the cross-sectional area of such a detector.

OPTICAL MATERIAL LIBRARY UPDATES

Fused silica and i-line glass materials from Ohara Corporation have been added to the Optical material library. Compared to other types of glass, fused silica and i-line glasses generally have greater transmittance at ultraviolet (UV) and near-UV wavelengths.

IMPROVED PSEUDORANDOM NUMBER GENERATION

The Ray Optics Module includes a variety of features that rely on pseudorandom number generation (PRNG), such as the following:

- Monte Carlo modeling of light extinction in particle-laden media
- Diffuse and isotropic scattering
- · Surface-normal perturbations for modeling rough boundaries
- · Conditional ray-boundary interactions

For such features, the calls to pseudorandom number generators have been extensively reviewed and revised. The new expressions are much less likely to incur unwanted correlations between random numbers that should be completely uncorrelated. This includes random boundary conditions acting on different rays, as well as unwanted correlations between different randomly generated vector components.

OPTION TO ONLY STORE ACCUMULATED VARIABLES IN SOLUTION

Depending on your application, the accumulated variables (such as the deposited ray power or fluence rate within a domain) might be more valuable information than the position and direction of individual rays. To reduce file size, you now have the option to only retain the accumulated variables in the solution while discarding the degrees of freedom associated with the rays.

ANNULAR ULTRAVIOLET REACTOR

This tutorial model demonstrates how to compute the volumetric fluence rate in a UV reactor. The geometry is the annular fluid surrounding a cylindrical lamp. The effect of reflection at the reactor walls on the radial fluence rate distribution is considered.

ANNULAR ULTRAVIOLET REACTOR WITH PARTICLE TRACING

In this example, a simple UV water purification reactor is modeled using a combination of ray tracing, computational fluid dynamics (CFD), and Lagrangian particle tracking. First, the volumetric fluence rate is accumulated along rays released from the surface of the UV lamp using the Geometrical Optics interface. Then, the fluid velocity in the reactor is solved for. Finally, particles are traced along the fluid velocity streamlines while the fluence rate along these particle trajectories is integrated using an **Auxiliary Dependent Variable**. This example requires the CFD Module and the Particle Tracing Module.

RAY OPTIC MODELING OF A FRESNEL LENS

This tutorial model shows how a Fresnel lens can be designed from a plano-convex lens as a way to reduce thickness. Here, the collimation property of a Fresnel lens is modeled.

PETZVAL LENS STOP ANALYSIS WITH SURFACE-TO-SURFACE RADIATION

This tutorial model has been updated to improve the accuracy of the calculation of surface-to-surface radiation. This improvement is done by covering a gap between the vacuum window and the thermal shroud window using a vacuum chamber.

Important Fixes in Version 6.1

REFRACTIVE INDEX VALUE AND GAUSSIAN BEAM FEATURE

In version 6.0, when the imaginary part of the refractive index of a material was nonzero, an error appeared in the **Gaussian Beam** node. This error has been fixed so that the node now only takes into account the real part of the refractive index value.

CORRECTED INITIALIZATION OF SEARCH DIRECTION OF RAYS

In version 6.0, when the initial position of a beam created using either the **Gaussian Beam** or **Solar Radiation** feature fell exactly on the surface or boundary of the geometry, the direction of search for other boundaries and surfaces did not always align with the beam propagation direction. This has been now been fixed.

DUPLICATING RELEASE FROM DATA FILE NODE

In version 6.0, the embedded or imported file was duplicated when the **Release from Data File** node was duplicated. When the original **Release from Data File** node was used to import another file, an error occurred. Also, the duplicated node in some cases completely overrode the original node without producing an error. This has been corrected; now, when you duplicate this node, the imported file will not be duplicated. You will need to import the file into every duplicated nodes.

LINE PLOTS FASTER IN RAY TRAJECTORIES PLOT

In version 6.1, the default of the **Maximum number of extra time steps rendered** setting is changed to **All** in the Ray Trajectories plot. Plotting using the **Line**, **Ribbon**, or **Tube** styles is now much faster than in version 6.0.

Backward Compatibility with Version 6.0 and Earlier

DUPLICATED RELEASE FROM DATA FILE NODE

If a model created using version 6.0 or earlier contains duplicated **Release from Data File** nodes, it is recommended to delete these nodes, create new nodes, and import the files again, even if same file is used in each node.

RF Module

New Functionality in Version 6.1

CONDUCTIVE DOMAINS

When conductive domains contain many boundaries, it is often cumbersome to apply the boundary condition to all of them individually. The **Perfect Electric Conductor** and **Impedance Boundary Condition** domain features can be directly applied to the conductive domain without having to locate all of the boundaries or manually remove the interior domain.

ELECTROSTATIC DISCHARGE AND LIGHTNING PULSES

The **Lumped Port** and **Edge Current** features in the Electromagnetic Waves, Transient interface support predefined and parameterized temporal pulse functions to describe electrostatic discharge (ESD) and lightning that may cause detrimental effects to electronic components. For convenience of inspection, the pulse shapes can be instantly plotted before running the simulation to ensure the choice of function parameters is suitable.

QUICK MESH REFINEMENT ON FAR-FIELD CALCULATION BOUNDARIES

In the physics-controlled mesh settings of the Electromagnetic Waves, Frequency Domain interface, the **Add far-field boundary layers** option is provided. When selected, a boundary layer mesh with a thickness of 1/40 of the default maximum mesh size is created in the far-field calculation boundaries adjacent to the selection of scattering boundary conditions or perfectly matched layers. This boundary layer mesh helps obtaining more accurate far-field analysis results such as total radiated power and radar cross section (RCS).

FOUR-PORT NETWORK

The Electromagnetic Waves, Frequency Domain interface now includes the **Four-Port Network** boundary condition, which characterizes the response of a four-port network component using S-parameters. You can import a Touchstone file to describe the physical behavior and response of a four-port device or system through four-port boundaries without addressing a complicated geometry.

DIELECTRIC SCATTERING WITH THE ELECTROMAGNETIC WAVES, BOUNDARY ELEMENTS INTERFACE

It is now possible, in the **Scattered field Formulation** for the Electromagnetic Waves, Boundary Elements interface to solve problems involving multiple dielectric scatterers embedded in an infinite domain.

LUMPED PORT IN THE ELECTROMAGNETIC WAVES, BOUNDARY ELEMENTS INTERFACE

The **Lumped Port** feature is available for the Electromagnetic Waves, Boundary Elements interface to excite or terminate metallic structures such as dipole antennas.

SYMMETRY PLANE

The **Symmetry Plane** feature simplifies the definition of perfect electric conductor (PEC) and perfect magnetic conductor (PMC) symmetry planes. This feature is used instead of the **Perfect Electric Conductor** and **Perfect Magnetic Conductor** boundary conditions when reducing the model size from symmetry considerations.

Furthermore, the information about the type and location of the **Symmetry Plane** features is used when calculating far fields and when defining analytical **Port** mode fields.

WEAK FORMULATION PORT OPTION

When expanding the electric field on a port boundary, the new weak port formulation adds a scalar dependent variable for the expansion coefficient (the S-parameter) and then solves for the S-parameters and the tangential electric field on the boundary using only a weak expression. Since no constraints are used, this formulation completely removes the constraint elimination step when solving.

This new port formulation replaces the other constraint-free port formulation that was introduced in version 6.0.

LAYERED IMPEDANCE BOUNDARY CONDITION

The Layered Impedance Boundary Condition is an extension of the Impedance Boundary Condition that makes it possible to model a sequence of geometrically thin layers on top of a substrate. It is used on exterior boundaries where the field is known to penetrate only a short distance outside the boundary. In brief, this feature combines a Layered Transition Boundary Condition with an Impedance Boundary Condition.

This new feature is available for the Electromagnetic Waves, Frequency Domain interface.

LINEARLY POLARIZED PLANE WAVE BACKGROUND FIELD IN 2D AXISYMMETRY

For 2D axisymmetry, there is a new background field, the **Linearly Polarized Plane Wave** type. Here, the plane wave is expanded in a series of Bessel functions. This functionality makes it possible to very efficiently model the scattering of plane waves from rotationally symmetric objects.

FEM-BEM MULTIPHYSICS COUPLING

To simplify the setup of models using both the Electromagnetic Waves, Frequency Domain and Electromagnetic Waves, Boundary Elements interfaces, the Electromagnetic Waves, FEM-BEM multiphysics interface can be selected in the Model Wizard.

This multiphysics interface adds the two physics interfaces and the **Electric Field Coupling** multiphysics coupling feature to the new model.

COVARIANT FORMULATION IN 2D AXISYMMETRY

In the 2D axisymmetric formulation, it is beneficial to formulate the out-of-plane dependent variable as

$$\Psi = rE_{\varphi},$$

referred to as the covariant formulation. Here, Ψ is the dependent variable, and the out-of-plane electric field component is calculated as

$$E_{\rm m} = \Psi/r$$
.

The covariant formulation now has better performance in terms of numerical stability and accuracy. In contrast with previous versions, eigenfrequency simulations can now return a different number of eigenfrequencies. However, the returned solutions have better accuracy, and there are many fewer spurious solutions returned.

This formulation is used for all study types, except **Mode Analysis** and **Boundary Mode Analysis**.

SKIN DEPTH CALCULATOR

The **Skin Depth Calculator** can be used to compute skin depth before performing a full model computation. This functionality is useful when setting up the physics and boundary conditions.

The calculator is available in the settings window of the Impedance Boundary Condition, Transition Boundary Condition, Layered Impedance Boundary Condition and Layered Transition Boundary Condition features.

PLOTTING OF ANALYTIC PORT MODE FIELDS BEFORE COMPUTATION

For **Rectangular**, **Circular**, and **Coaxial Port** types, it is now possible to plot and inspect the mode field before computing the full solution. This is useful for correctly defining the other boundary conditions used in the model.

UPWIND FLUX OPTION FOR THE ELECTROMAGNETIC WAVES, TIME EXPLICIT INTERFACE

The **Flux type** parameter in the **Wave Equations** node for the Electromagnetic Waves, Time Explicit interface now also includes the **Upwind flux** option. This option can sometimes be useful to improve S-parameter calculations that could have low accuracy due to overdissipation around perfect electric conductor (PEC) edges, when using the default **Lax-Friedrichs** flux parameters.

PERIODIC PORT REFERENCE POINT

The **Periodic Port Reference Point** subnode has been simplified to only allow the selection of a single reference point.

NEW DEFAULT COLOR TABLE WHEN ADDING PLOTS

When adding a new 2D or 3D plot, the created plot will use the RainbowLight color table by default.

New and Updated Models in Version 6.1

MODELING OF A DIFFERENTIAL MICROSTRIP LINE

A differential line is composed of two transmission lines excited by two out-of-phase signals. This configuration is known to be useful to enhance the signal-to-noise ratio. The model example shows how to set up differential and single-ended microstrip lines using TEM-type ports. The impact on computed S-parameters due to a fictitious noise source is demonstrated for each microstrip line configuration, respectively.

ELECTROSTATIC DISCHARGE (ESD) TEST OF A PCB

This example demonstrates how an ESD event causes a logic error in the microchip on a PCB board. The ESD current is based on the extended human body model (HBM) and is predefined in the **Lumped Port** feature.

LIGHTNING-INDUCED VOLTAGE OF A WIRE IN AN AIRPLANE

This model demonstrates how to compute the induced voltage of a wire loop inside an airplane under different electromagnetic shielding conditions. The wing of the airplane is struck by a lightning current pulse defined in the **Edge Current** feature.

LIGHTNING-INDUCED VOLTAGE OF AN OVERHEAD LINE OVER LOSSY GROUND

This model shows how to calculate the lightning-induced voltage of an overhead line over lossy ground. Lightning channels with different inclinations are considered. The effect of soil conductivity is also shown.

CLOAKING OF A CYLINDRICAL SCATTERER WITH GRAPHENE

In this model, a cloaking method is introduced using an electrically tuned monolayer of graphene. We will show that when a cylindrical dielectric scatterer is covered in graphene, the scattering cross section is greatly reduced at the designated frequency, making it electromagnetically invisible. This model aims to demonstrate the use of the **Linearly polarized plane wave Background wave type** option in a 2D axisymmetric model.

MICROWAVE OVEN

This model has been updated to use the new Symmetry Plane feature.

DIPOLE ANTENNA

This model has been updated to use the domain **Perfect Electric Conductor** feature.

DIPOLE ANTENNA WITH A QUARTER-WAVE COAXIAL BALUN

This model has been updated to use the domain **Perfect Electric Conductor** feature.

CAR WINDSHIELD ANTENNA EFFECT ON A CABLE HARNESS

This model has been updated to use the domain **Perfect Electric Conductor** feature.

DOUBLE-RIDGED HORN ANTENNA

This model has been updated to use the domain **Perfect Electric Conductor** feature.

SIMULATING ANTENNA CROSSTALK ON AN AIRPLANE'S FUSELAGE

This model has been updated with the new airplane geometry.

MICROSTRIP PATCH ANTENNA

This model has been updated to use the Array Factor dataset.

MODELING OF A PHASED ARRAY ANTENNA

This model has been updated to use the Array Factor dataset.

Backward Compatibility with Version 6.0 and Earlier

COVARIANT FORMULATION

In 2D axisymmetry, for all studies except **Mode Analysis** and **Boundary Mode Analysis**, the covariant formulation is used (see Covariant Formulation in 2D Axisymmetry). Thereby, Ephi is not a dependent variable. To get the correct out-of-plane electric field component, the variable emw.Ephi should be used. In this name, it is assumed that the dependent variable has been given the name E and that the tag for the physics interface is emw.

NEW DEFAULT VALUE FOR THE RELATIVE PERMITTIVITY, HIGH FREQUENCY PARAMETER

The default value for the **Relative permittivity**, **high frequency** parameter, **epsilonInf**, has changed from **From material** to **User defined**.

Rotordynamics Module

New Functionality in Version 6.1

MODELING OF ROTOR RUB

Contact between a rotor and its housing can be modeled using the new **Rotor Rub** node in the Beam Rotor interface. Optionally, friction between the parts can be included.

CHANGED UI FOR HYDRODYNAMIC BEARING

The settings for the Hydrodynamic Bearing interface have been changed to be more logical. The **Cavitation** section has been removed, and cavitation is now selected as an option in the **Fluid type** list.

CRITICAL SPEEDS IN CAMPBELL DIAGRAMS

The Campbell diagrams have been augmented with markers, showing the critical speeds of the rotor.

RESULTS IN LOCAL DIRECTIONS

For details, see Results in Local Directions in the release notes for the Structural Mechanics Module.

NEW INPUT OPTIONS FOR NONISOTROPIC ELASTIC MATERIALS

For details, see Results in Local Directions in the release notes for the Structural Mechanics Module.

RIGID DOMAIN RENAMED TO RIGID MATERIAL

For details, see Rigid Domain Renamed to Rigid Material in the release notes for the Structural Mechanics Module.

REDUCED NUMBER OF DEFAULT PLOTS

For details, see Reduced Number of Default Plots in the release notes for the Structural Mechanics Module.

PREVIEW PLOT FOR RAYLEIGH DAMPING

For details, see Preview Plot for Rayleigh Damping in the release notes for the Structural Mechanics Module.

REMOVED SECTION IN PHYSICS INTERFACE SETTINGS

For details, see Removed Section in Physics Interface Settings in the release notes for the Structural Mechanics Module.

New Models in Version 6.1

TOPOLOGY OPTIMIZATION OF A STEP THRUST BEARING

In this tutorial model, a step thrust bearing is topology optimized to maximize the bearing capacity. A step thrust bearing consists of a stepped bearing surface on which the end of the shaft rotates. The entire assembly is submerged in a lubricant.

The model shows that initializing the optimization with a nonuniform design can result in local minima with a specific number of steps.

SHAPE OPTIMIZATION OF A STEP THRUST BEARING

In this tutorial model, a step thrust bearing is shape optimized to maximize the bearing capacity. A step thrust bearing consists of a stepped bearing surface on which the end of the shaft rotates. The entire assembly is submerged in a lubricant.

The resulting design shows the optimal shape of the steps. The model also includes a parametric sweep over the number of steps to identify the optimal number.

Structural Mechanics Module

New Functionality in Version 6.1

SOLID MECHANICS INTERFACE IN ID AND ID AXISYMMETRY

The Solid Mechanics interface is now also available for components with 1D and 1D axisymmetric geometries. This is mainly interesting for multiphysics cases. Several multiphysics interfaces and multiphysics couplings are available in 1D and 1D axisymmetry. Some features and material models that are inherently multiaxial are not available in the 1D versions of the interface.

TESTING OF MATERIAL MODELS

A new feature, **Test Material**, has been added to the Solid Mechanics interface. With this feature, it is possible to set up single-element tests for various scenarios like uniaxial or biaxial tension. The purpose is to test and verify the behavior of a material model for a given set of material properties. For material models with an explicit time dependence, the test study can be time dependent.

MATERIALS ON BOUNDARIES OF SOLIDS

In the Solid Mechanics interface, a new feature called **Thin Layer** has been introduced. You can use it to add material models on interior and exterior boundaries. Most domain material models that would also be relevant for a thin layer are available. With the Composite Materials Module, a thin layer can itself be multilayered. There are different formulations of the thin layer materials, depending on the assumed kinematics.

PHYSICS INTERFACE FOR WIRES

A new physics interface dedicated to the analysis of wires and cables has been added: The Wire interface is available with the Structural Mechanics Module and Multibody Dynamics Module.

MULTIPHYSICS INTERFACES FOR THIN-FILM DAMPING

Two new multiphysics interfaces for modeling thin-film damping have been added: Solid Thin-Film Damping and Shell Thin-Film Damping. They combine a Thin-Film Flow interface with a Solid Mechanics or Shell interface, respectively. The couplings are made through the new **Structure Thin-Film Flow Interaction** and **Shell Thin-Film Flow Interaction** multiphysics couplings. As an effect of this more general approach, the **Thin-Film Damping** feature in the Solid Mechanics interface has been retired. It will still appear if you open an old model that contains such a feature, but you can no longer add it. Consider replacing it using the new modeling paradigm, since it will only be available for a few more releases and is no longer actively maintained.

MAGNETOSTRICTION INTERFACE CHANGES

The Magnetostriction multiphysics interface has been split into two different interfaces: Piezomagnetism and Nonlinear Magnetostriction.

DEAD LOADS IN LINEARIZED BUCKLING ANALYSIS

In a linearized buckling study, you can now separate the loads into those that are assumed to be independent of the load factor (dead loads) and those that are proportional to the load factor (live loads). To mark a certain load as being a dead load, you go to the **Linear Buckling** section in the settings for that load feature. This section is only shown when **Advanced Physics Options** is enabled. There are also special settings in the **Linear Buckling** study step to control the different load cases.

THICKNESS OF SHELLS AND MEMBRANES CAN CHANGE OVER TIME

In the Shell and Membrane interfaces, the **Thickness and Offset** node has been augmented by a subnode called **Thickness Change**. There, it is possible to enter a rate of thickness change. The purpose is to model phenomena like wear, corrosion, and electrodeposition.

WEAR MODELING IN SHELL AND MEMBRANE INTERFACES

A new **Wear** subnode has been added to the **Contact** node in the Shell and Membrane interfaces. With this feature you can model thickness reduction and corresponding changes in contact force distribution caused by wear. The functionality is similar to what was already available in the Solid Mechanics interface.

NEW DISCRETIZATION SETTING FOR WEAR

When the offset-based wear formulation is used, you can now select the type of shape function that is used to describe the offset field. This is done in the new **Discretization** section in the **Wear** node.

CONTINUITY USING NITSCHE'S METHOD

In the **Continuity** node in the Solid Mechanics and Multibody Dynamics interfaces, you can now select between three different methods for enforcing the continuity conditions. In addition to using pointwise or weak constraints, a new formulation

based on Nitsche's method is also available. The main advantage with the new method is that there is no need for constraint elimination, since it only adds terms to the stiffness matrix.

COMPONENT MODE SYNTHESIS IMPROVEMENTS

- Component mode synthesis (CMS) is now also available for the Shell interface.
- There are several improvements to the usability of the CMS functionality with respect to how different features in the physics interfaces are treated.
- The **Component** subnode under **Reduced Flexible Components** has been renamed to **Component Definition**.

NEW SEARCH METHOD FOR CONTACT ANALYSIS

A new search method for finding the distance between contacting boundaries has been implemented. It is significantly faster than the previous methods, in particular for large 3D models. The new method, **Hierarchical**, is now the default search method in the settings for a **Contact Pair**. The old search methods have been renamed to better reflect their properties. The performance has been improved for these methods as well.

CONTACT USING NITSCHE'S METHOD

For contact between solid objects, it is now possible to select **Nitsche** as **Contact method**. This method does not suffer from the introduction of an arbitrary stiffness, as in the penalty method, and still no extra degrees of freedom are required. The Nitsche method is robust but can be more costly because it produces a nonsymmetric stiffness matrix.

NEW FORMULATION FOR CONTACT EQUATIONS

The formulation for the contact equations has been improved for all contact methods. In particular, a reduced Jacobian is now used by default. You can control the assembly of the Jacobian through the **Jacobian contribution** control in the **Advanced** section of the **Contact** settings. Select **Legacy** to revert to the version 6.0 behavior.

IMPROVED SUPPORT FOR SELF-CONTACT

If you make the same selection for the source and destination in a **Contact Pair** node, a formulation of the contact conditions that is fully symmetric between source and destination boundaries is used.

IMPROVED FORMULATION FOR CONTACT IN SHELL AND MEMBRANE

The contact mapping is now done from the actual top and bottom surfaces of the structural elements, enabling a correct area scale for thick, highly curved shells.

POSSIBLE TO TRIGGER A CUTBACK IN THE SOLVER FOR ALL CONTACT METHODS

The **Trigger Cutback** section is now available for all contact methods. By setting an appropriate cutback criterion, you can force the solver to go back and start with a smaller parameter or time step, rather than spending many iterations on a solution that is far from convergence.

BASE EXCITATION

A new node at the global level, **Base Excitation**, has been added to the structural mechanics interfaces. The purpose is to handle the case where all support points experience a common acceleration history. Base excitation can be used for any dynamic study type, but it is most important for the modal-based procedures where it is not possible to directly prescribe a nonzero acceleration of displacement.

GRAVITY AND LINEARLY ACCELERATED FRAME

The **Gravity** node has been changed to be a global feature in all structural mechanics interfaces. The reason is that gravity is almost always applied to the entire structure. A new node, **Linearly Accelerated Frame**, makes it possible to prescribe nonuniform inertial loads.

LOADS GIVEN AS RESULTANT

For a **Boundary Load** or **Point Load**, it is now possible to give the total load as a force and moment resultant. To use such a load, select **Resultant** from the **Load type** list. This option is available in the Solid Mechanics, Multibody Dynamics, and Layered Shell interfaces.

For **Point Load**, there is now also the possibility to set **Load type** to either **Force per point** or **Total force**.

The difference between a total force and a resultant is that in the former case, the given value is simply divided by the number of selected points (or boundary area) before it is evenly distributed. A resultant implies a certain spatial distribution of the loads.

RESULTS IN LOCAL DIRECTIONS

By adding a **Local System Results** node in a Solid Mechanics, Multibody Dynamics, or Solid Rotor interface, you can get access to a number of result variables expressed in a
given coordinate system. This includes displacements, velocities, accelerations, stresses, strains, and material properties.

EVALUATION OF WELD STRESSES IN THE SHELL INTERFACE

When two boundaries with shell elements are joined using the **Edge to Edge** connection in the Shell interface, it is possible to define the connection as a weld and to evaluate weld-specific stress results. Three types of welds can be defined: single-sided fillet weld, double-sided fillet weld, and butt weld.

MOMENT AND SHEAR FORCE DIAGRAMS FOR BEAMS

In the Beam interface, a capability to draw moment and shear force diagrams has been included. Such section force diagrams can be added from the **Add Predefined Plot** window. The section force diagrams take the load distribution into account, so that even with a very coarse mesh, the diagrams are more or less exact.

A set of new section force variables, called *distributed section forces* has been added to facilitate this. The variable names are the same as for the standard section forces but appended by _d, for example, beam.My1_d.

STANDARD CROSS SECTIONS FOR TRUSS ELEMENTS

In the Truss interface, the **Cross-Section Data** node has been augmented by an option for defining the element cross section by geometrical properties. The available cross sections are the same as in the Beam interface, that is, Rectangle, Box, Circular, Pipe, H-profile, U-profile, T-profile, C-profile, and Hat.

SELECTION OF SIDE FOR SHELL CONNECTIONS

In the **Edge to Edge, Edge to Boundary**, and **Boundary to Boundary** connections in the Shell interface, it is now possible to select a through-thickness location (top, bottom, or midsurface) as the reference for the connection.

NEW INPUT OPTIONS FOR NONISOTROPIC ELASTIC MATERIALS

In the **Linear Elastic Material** and **Layered Linear Elastic Material** nodes, there are new input options for material properties:

A new class of material symmetries has been added, called Crystal. It facilitates the input of elasticity properties for seven different crystal systems, such as Cubic (3 constants) and Hexagonal (5 constants). This option requires the Structural Mechanics Module or the MEMS Module.

- For an orthotropic material, it is possible to select the special case **Transversely** isotropic, in which case five material constants are required.
- For an anisotropic material, you can now choose whether to enter an elasticity matrix or a compliance matrix.

MULTIPLICATIVE FORMULATION FOR PIEZOELECTRICITY

For the **Piezoelectric Material**, a new formulation has been implemented in which the inelastic (piezoelectric) strains under geometric nonlinearity are removed from the total strains through multiplicative decomposition. The formulation is controlled by the **Use multiplicative formulation** check box. When selected, any study will be forced to be geometrically nonlinear.

CELL PERIODICITY CAN CREATE MATERIAL DATA ON COMPLIANCE FORM

When you select **Average stress** as **Periodicity Type** in **Cell Periodicity**, it is now possible to create an average compliance matrix. This is an effect of the **Linear Elastic Material** setting now allowing input to be included on the compliance form.

EXTENDED BUILT-IN MATERIAL

The built-in **Structural Steel** material has been augmented so that material properties are temperature dependent. There is also material data for a wide range of nonlinear material models.

LIBRARY OF PARTS FOR HOMOGENIZATION OF MICROSTRUCTURES

In the Part Library, a new folder called **Representative Volume Elements** has been added to the **COMSOL Multiphysics** branch. It contains a number of parameterized geometries for common microstructures, like fiber and particulate composites. These geometries can, for example, be used to compute effective material properties using the representative volume element (RVE) method. The **Cell Periodicity** node in the Solid Mechanics interface is designed for this.

RELEASED DEGREES OF FREEDOM IN RIGID CONNECTOR

It is now possible to specify that there is no connection in a certain direction for a rigid connector. The direction can be specified in a local coordinate system, so that it, for example, can be the radial direction from a certain point. You specify this behavior in the new **Released Degrees of Freedom** in the settings for the **Rigid Connector** node.

SINGULARITY SUPPRESSION FOR TWO-POINT RIGID CONNECTOR

When a rigid connector has a selection that consists of only two points, it is now possible to add a spring type condition to suppress the rotational singularity that could

occur. The setting for this is in the **Advanced** section in the settings for the **Rigid Connector**. The **Add rotational stiffness for two-point selection** check box is shown when the **Include consistency checks** check box is cleared.

LIMITED DISPLACEMENT IN TRUSS INTERFACE

In the **Prescribed Displacement** node in the Truss interface, there is a new option, **Limited Displacement**. This type of boundary condition limits the displacement in a certain direction to a given maximum value. The **Prescribed Displacement** node has been redesigned in order to accommodate this.

CORRECTION FOR PIPE BENDS

In the Pipe Mechanics interface, you can now specify correction factors for stiffness and stress evaluation in pipe bends. To do so, you add the **Bend** subnode under **Pipe Cross Section**.

STRESS EVALUATION USING REDUCED PIPE THICKNESS

When specifying a **Pipe Cross Section** in the Pipe Mechanics interface, you can now enter a reduced pipe thickness for the stress evaluation. This can be used when, for example, taking corrosion allowance into account.

LIBRARY OF PIPE GEOMETRIES

In the Part Library, the content of the **Pipes** folder in the **COMSOL Multiphysics** branch has been augmented. The folder now contains parameterized geometries for the following common piping parts: straight pipe, bend, reducer, and T-junction. These geometries can be used for detailed analysis using the Solid Mechanics or Shell interface. Such a part can be inserted into a pipe system modeled using the Pipe Mechanics interface, through the **Structure-Pipe Connection** multiphysics coupling.

Because the new parts are more general in scope, the old straight_pipe and bent pipe parts have been removed.

NEW FORMULATION OF RIGID MOTION SUPPRESSION

The rigid motion suppression constraints are now applied in an average sense rather than at arbitrary points. The advantage is that the visual impression of the deformation shape is more intuitive.

IMPROVED PERFORMANCE FOR RANDOM VIBRATION EVALUATION

The operators used for the evaluation of random vibration results have been optimized and now exhibit significantly better performance.

RIGID DOMAIN RENAMED TO RIGID MATERIAL

The **Rigid Domain** material model has been renamed to **Rigid Material** in all physics interfaces. The purpose is to emphasize that this is a material model with the same status as other material models.

REDUCED NUMBER OF DEFAULT PLOTS

With the introduction of the new concept of *predefined plots*, the number of default plots that are always produced by the structural mechanics interfaces has been reduced significantly. Several plots that were default plots are now instead optional and can be added from **Add Predefined Plot** window. There are also several new useful predefined plots.

PREVIEW PLOT FOR RAYLEIGH DAMPING

In the **Damping**, **Layered Damping**, and **Mechanical Damping** nodes, it is now possible to create a preview plot that shows the damping as a function of frequency when data for Rayleigh damping has been given.

PREVIEW PLOT FOR THICKNESS AND OFFSET

In the **Thickness and Offset** nodes in the Shell and Membrane interfaces, it is now possible to create a preview plot that shows how the given inputs are interpreted.

NEW OPTION FOR GENERATION OF MATERIAL DATA IN CELL PERIODICITY

When creating material data from a **Cell Periodicity** node, you can now choose between the two actions **Create Material by Reference** and **Create Material by Value**. The new second option makes it possible to store material data in a file that can be used in other models.

GROUPING OF ODE VARIABLES IN SOLVER

In each **Rigid Connector** and **Attachment** node, a set of ordinary differential equation (ODE) degrees of freedom are created, representing the translation and rotation. Up to version 6.0, each 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. From version 6.1, the default is that such variables are grouped together instead.

You can control whether or not to use the grouping. This can be done in two places. The default behavior can be controlled from the **Advanced Settings** section in the settings for the physics interface. This section is visible if **Advanced Physics Options** is active.

The default behavior can, however, be overridden for each individual **Rigid Connector** or **Attachment** node, 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 two lines similar to

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

There are subtle differences in how the solver handles time stepping and convergence depending on whether dependent variables are grouped or not. With grouping, the ODE variables will be assigned less weight when compared with, for example, the displacement field, as long as no other settings are changed.

CHANGED DEFAULT FOLDLINE ANGLE

The default foldline angle in the Shell interface has been reduced to 0.001 rad in order to reduce the risk of artificial stiffening.

NEW OPTION WHEN CONNECTING SHELLS AND SOLIDS

When connecting a shell or membrane boundary to a solid domain boundary, using the **Solid-Thin Structure Connection** multiphysics coupling, you can now select for which physics interface the constraints are generated. If the **Connection type** is **Shared boundaries** or **Parallel boundaries**, you can make this selection in the new **Advanced** section.

RESTRUCTURED REDUCED-ORDER MODEL STUDIES

When one of the three reduced-order model studies (Frequency Domain, Modal Reduced-Order Model; Time Dependent, Modal Reduced-Order Model; or Frequency Domain, AWE Reduced-Order Model) is selected, the generated studies have a structure that differs from previous releases. One study (Frequency Domain or Time Dependent) is no longer needed. Instead, subnodes with the same names are created under the Model Reduction study step.

NODAL VERSION OF SHELL NORMAL CONSTRAINTS

In the **Advanced Settings** section in the Shell interface, you can now select whether the constraints on the rotation around the normal are added as nodal or elemental constraints.

MULTIPLICATIVE STRAIN DECOMPOSITION IN SHELL AND PLATE INTERFACES

When modeling using geometric nonlinearity, the inelastic strains are now removed by multiplicative decomposition in the Shell and Plate interfaces.

LIMITATIONS OF SELECTIONS ON AXIS OF ROTATION

For 2D axisymmetric components, it is no longer possible to select a line on the axis of rotation for a number of features. For the Shell and Membrane interfaces, such lines can no longer be part of the physics selection. For Solid Mechanics, a number of boundary conditions that are not relevant on the axis of rotation can no longer include it in the selection.

REMOVED SECTION IN PHYSICS INTERFACE SETTINGS

The **Reference Point for Moment Computation** section has been removed from the settings for all structural mechanics interfaces. The values of the properties are now always set in the feature under **Results** where they are used.

MISCELLANEOUS

- When using a strain-based mixed formulation, the elastic volumetric strain is now used in the weak contribution, rather than the total volumetric strain.
- In the Layered Linear Elastic Material, the degree of freedom (DOF) for the transverse strain is always active when a mixed formulation is used.
- In the Linear Elastic Material in the Shell interface in 2D axisymmetry, a discontinuous Lagrange shape function is now used to represent the transverse strain DOF.

New and Updated Models in Version 6.1

NONLINEAR HARMONIC RESPONSE

This tutorial model shows how to evaluate the harmonic response of a structure with a moderately nonlinear behavior. A nonlinear problem can only be solved using timedomain analysis. A way to speed up the computation is to use a linearized frequency response analysis to provide good initial conditions.

BRACKET — REDUCED-ORDER MODELING

This tutorial model illustrates how to use a reduced-order model to compute the response of a structure subjected to three simultaneous loads that have independent time histories. Reduced-order models provide an efficient way to analyze problems within linear structural dynamics.

SELF-CONTACT OF A LOADED SPRING

This example shows how to set up self-contact for a coil spring. As the spring is compressed by a vertical force applied to one of its ends, it comes into contact with itself and starts to rotate.

TORSION OF AN ISOTROPIC COSSERAT ELASTIC CYLINDER

This example demonstrates how to extend the built-in linear elastic material model to a Cosserat material through the addition of microrotation degrees of freedom. A cylindrical bar under pure torsion is analyzed, and the effect of the Cosserat lengthscale parameter on the response is studied.

MICROMECHANICAL MODEL OF A PARTICULATE COMPOSITE

In this example, the homogenized elastic and viscoelastic properties of a particulate composite are computed based on the individual properties of elastic particles embedded in a viscoelastic matrix.

Periodic boundary conditions are applied to a unit cell of the particulate composite using the **Cell Periodicity** feature in the Structural Mechanics Module.

The transient viscoelastic response of the composite is used to determine the homogenized material parameters using curve fitting optimization.

LINEAR BUCKLING ANALYSIS OF A TRUSS TOWER WITH DEAD LOADS

This example shows how to compute the critical buckling load in the presence of dead loads. A truss tower is supported by pretensioned guys. The pretension load and gravity on the truss are considered dead loads, while the vertical load applied at the top of the tower is considered the live load. The guys are modeled using the Wire interface.

VIBRATING STRING

This model has been updated so that the new Wire interface is used instead of the Truss interface.

HANGING CABLE

This model has been updated so that the new Wire interface is used instead of the Truss interface.

HEATING CIRCUIT

This model has been updated so that a **Thin Layer** in the Solid Mechanics interface is used to model the circuit, rather than a separate Membrane interface.

Subsurface Flow Module

New Functionality in Version 6.1

CLIMATE DATA: ASHRAE 2021

ASHRAE Weather Data 2021 has been integrated into COMSOL Multiphysics. ASHRAE 2021 contains additional locations; there are more than 8500 stations in the 2021 edition. You can select any of them in the **Ambient Properties** node. The database provides typical climate data for the temperature, humidity, precipitation, and solar radiation.

IMPROVED HANDLING OF FRACTURES

In the Darcy's Law and Fracture Flow interfaces, the following improvements are available:

- Fractures are now defined using phase-specific properties in the **Fracture** node. The **Fluid** subnode describes the fluid properties, and the **Fracture Material** subnode accounts for the fracture properties, such as porosity or transmissivity.
- It is now possible to model Darcian or non-Darcian flow in fractures.
- The fractures can be highly conductive, like open channels, or can be represented by thin barriers.
- Fractures now also support the ideal gas and the linearized storage model.

FRACTURES FOR TRANSPORT OF DILUTED SPECIES

A new **Fracture** node is now available in the Transport of Diluted Species in Porous Media and Transport of Diluted Species in Fractures interfaces. The new node is consistent with other fracture nodes for porous media transport, and it includes two subnodes, **Fluid** and **Fracture Material**. The **Fluid** subnode describes the fluid, velocity, and diffusion properties, and the **Fracture Material** subnode accounts for the fracture properties.

STORAGE MODEL FOR UNSATURATED POROUS MEDIA

A new option for the **Storage model** setting in the **Unsaturated Porous Medium** node is added. Now, it is possible to define the storage as a function of liquid content.

USER-DEFINED PHASE TRANSITION

A **User defined** phase transition function in the **Phase Change Material** subnode is now available for the Heat Transfer in Porous Media interface. This option enables you to use an accurate phase change description from measured data.

THIN BARRIER MULTIPHYSICS COUPLING

The new **Thin Barrier** node for the Multiphase Flow in Porous Media multiphysics interface makes it possible to add thin layers that act as a resistance for the flow fields of all phases without meshing through the layer's thickness.

NEW PREDEFINED PLOTS

With the introduction of the new **Add Predefined Plots** feature, the number of default plots that are produced has been reduced. Several plots that were default plots are now instead optional, and can be added from the **Add Predefined Plots** window. There are also several new useful predefined plots.

New Models in Version 6.1

PHASE CHANGE IN A SEMI-INFINITE SOIL COLUMN — LUNARDINI Solution

In this example, a heat-conduction problem with phase change in a porous material is solved, and the results are compared with the analytical solution, also known as the Lunardini solution. This is the first test case from the InterFrost project.

CO2 STORAGE IN A GEOLOGIC FORMATION

This example simulates the underground storage of CO_2 in a part of the Johansen Formation off the coast of Norway. The CO_2 was injected using an injection well at a rate of 15 kg/s over a period of 25 years, after which the well was closed. The model is used to compute the spreading of CO_2 over the simulation domain during the injection phase and during the 25 years directly following the shutdown of the injection well.

The model uses the new **Reinterpolate interpolation data on computational mesh** functionality for **Interpolation** functions, leading to a significant reduction of the computation time when evaluating porosity and permeability over large geometries.

Uncertainty Quantification Module

New Functionality in Version 6.1

INVERSE UNCERTAINTY QUANTIFICATION STUDY TYPE

The **Inverse Uncertainty Quantification** study propagates information of experimental data backward to gain more information on the calibration parameters. It computes the posterior distribution of the calibration parameters that best reflects their distribution based on the experimental data and the prior knowledge of the calibration parameters. The Markov chain Monte Carlo (MCMC) method is used to produce samples from the posterior distribution of the calibration parameters.

MORE INPUT PARAMETER DATA GENERATE SCHEMES

Settings for input parameters, surrogate-based Monte Carlo analysis parameters, and surrogate model verification parameters now support using data columns from results tables as the data source. They also support combining specified value and data columns with analytic probability distributions as the data source. The settings for surrogate-based Monte Carlo analysis parameters support using nominal values as the data source.

Wave Optics Module

New Functionality in Version 6.1

DIELECTRIC SCATTERING WITH THE ELECTROMAGNETIC WAVES, BOUNDARY ELEMENTS INTERFACE

It is now possible, in the **Scattered field Formulation** for the Electromagnetic Waves, Boundary Elements interface, to solve problems involving multiple dielectric scatterers embedded in an infinite domain.

WEAK FORMULATION PORT OPTION

When expanding the electric field on a port boundary, the new weak port formulation adds a scalar dependent variable for the expansion coefficient (the S-parameter) and then solves for the S-parameters and the tangential electric field on the boundary using only a weak expression. Since no constraints are used, this formulation completely removes the constraint elimination step when solving.

This new port formulation replaces the other constraint-free port formulation that was introduced in version 6.0.

LAYERED IMPEDANCE BOUNDARY CONDITION

The Layered Impedance Boundary Condition is an extension of the Impedance Boundary Condition that makes it possible to model a sequence of geometrically thin layers on top of a substrate. It is used on exterior boundaries where the field is known to penetrate only a short distance outside the boundary. In brief, this feature combines a Layered Transition Boundary Condition with an Impedance Boundary Condition.

This new feature is available for the Electromagnetic Waves, Frequency Domain interface.

LAYERED TRANSITION BOUNDARY CONDITION

The **Layered Transition Boundary Condition** node that was added in version 6.0 to the Electromagnetic Waves, Frequency Domain interface has now also been added to the Electromagnetic Waves, Beam Envelopes interface.

The **Layered Transition Boundary Condition** node has also been updated to include all material models that are available for the **Transition Boundary Condition** node, which simplifies the definition of material parameters for the boundary condition.

LINEARLY POLARIZED PLANE WAVE BACKGROUND FIELD IN 2D AXISYMMETRY

For 2D axisymmetry, there is a new background field, the **Linearly Polarized Plane Wave** type. Here, the plane wave is expanded into a series of Bessel functions. This functionality makes it possible to very efficiently model the scattering of plane waves from rotationally symmetric objects.

FEM-BEM MULTIPHYSICS COUPLING

To simplify the setup of models using both the Electromagnetic Waves, Frequency Domain and Electromagnetic Waves, Boundary Elements interfaces, the Electromagnetic Waves, FEM–BEM multiphysics interface can be selected in the Model Wizard.

This multiphysics interface adds the two physics interfaces and the **Electric Field Coupling** multiphysics coupling feature to the new model.

SYMMETRY PLANE

The **Symmetry Plane** feature simplifies the definition of perfect electric conductor (PEC) and perfect magnetic conductor (PMC) symmetry planes. This feature is used instead of the **Perfect Electric Conductor** and **Perfect Magnetic Conductor** boundary conditions when reducing the model size from symmetry considerations.

Furthermore, the information about the type and location of the **Symmetry Plane** features are used when calculating far fields and when defining analytical **Port** mode fields.

COVARIANT FORMULATION IN 2D AXISYMMETRY

In the 2D axisymmetric formulation, it is beneficial to formulate the out-of-plane dependent variable as

$$\Psi = rE_{\omega}$$
,

referred to as the covariant formulation. Here, Ψ is the dependent variable, and the out-of-plane electric field component is calculated as

$$E_{\phi} = \Psi/r$$
.

The covariant formulation now has better performance in terms of numerical stability and accuracy. In contrast with previous versions, eigenfrequency simulations can now return a different number of eigenfrequencies. However, the returned solutions have better accuracy, and there are many fewer spurious solutions. This formulation is used for all study types, except Mode Analysis and Boundary Mode Analysis.

SKIN DEPTH CALCULATOR

The **Skin Depth Calculator** feature can be used to compute skin depth before performing a full model computation. This functionality is useful when setting up the physics and boundary conditions.

The calculator is available in the Impedance Boundary Condition, Transition Boundary Condition, Layered Impedance Boundary Condition and Layered Transition Boundary Condition features.

PLOTTING OF ANALYTIC PORT MODE FIELDS BEFORE COMPUTATION

For **Rectangular**, **Circular**, and **Coaxial Port** types, it is now possible to plot and inspect the mode field before computing the full solution. This is useful for correctly defining the other boundary conditions used in the model.

NO INCIDENT FIELD OPTION FOR SCATTERING AND MATCHED BOUNDARY CONDITIONS

For the **Scattering Boundary Condition** and the **Matched Boundary Condition** for the Electromagnetic Waves, Beam Envelopes interface, there is now a new default option for the **Incident field** parameter — the **No incident field** value. This option represents the case when there are only outgoing waves at the particular boundary.

UPWIND FLUX OPTION FOR THE ELECTROMAGNETIC WAVES, TIME EXPLICIT INTERFACE

The **Flux type** parameter in the **Wave Equations** node for the Electromagnetic Waves, Time Explicit interface now also includes an **Upwind flux** value. This option can sometimes be useful to improve S-parameter calculations that could have low accuracy due to overdissipation around perfect electric conductor (PEC) edges, when using the default **Lax-Friedrichs** flux parameters.

PERIODIC PORT REFERENCE POINT

The **Periodic Port Reference Point** subnode has been simplified to only allow the selection of a single reference point.

NEW DEFAULT COLOR TABLE WHEN ADDING PLOTS

When adding a new 2D or 3D plot, the created plot will use the RainbowLight color table by default.

CLOAKING OF A CYLINDRICAL SCATTERER WITH GRAPHENE

In this model, a cloaking method is introduced using an electrically tuned monolayer of graphene. The model shows that when a cylindrical dielectric scatterer is covered in graphene, the scattering cross section is greatly reduced at the designated frequency, making it electromagnetically invisible. This model aims to demonstrate the use of the **Background wave type** option **Linearly polarized plane wave** in a 2D axisymmetric model.

OPTICAL YAGI-UDA ANTENNA

This model demonstrates the use of the Electromagnetic Waves, Boundary Elements interface to model dielectric scattering from an optical Yagi–Uda antenna. The antenna is driven by an electrical point dipole, which is implemented through the background field. The field distribution around the antenna and the far-field radiation pattern are evaluated, showing high directivity.

OPTICAL RING RESONATOR NOTCH FILTER 3D

This model calculates the spectral properties of a 3D optical ring resonator. The model demonstrates how to use parts from the Wave Optics Part Library and how to set up the **Field Continuity** boundary condition at boundaries, where there is a jump in the predefined phase approximation.

GRAPHENE METAMATERIAL PERFECT ABSORBER

Graphene, carbon atoms arranged in a two-dimensional hexagonal lattice, has sparked tremendous research and application interests since its experimental discovery about two decades ago. Besides being ultrathin, this material exhibits a plethora of interesting properties, including high electrical and thermal conductivities, high elasticity, high mechanical strength, and so on. Among various applications, a promising field is graphene-based electro-optical devices, such as photodetectors, photodiodes, and metamaterials. An additional desirable trait of graphene is that its optical response can be actively controlled by changing its Fermi energy via electrical gating. This model first demonstrates how to compute the optical conductivity of graphene using the Kubo formula. The computed conductivity is then used to model a graphene-based THz metamaterial absorber.

ENHANCED COATING FOR A MICROELECTROMECHANICAL MIRROR

This example demonstrates how to optimize the thickness of a microelectromechanical (MEMS) mirror coating material for maximum reflectivity. To reduce the simulation

time, a **Layered Impedance Boundary Condition** is used to model the thin coating material on top of the metallic mirror.

TAPERED WAVEGUIDE

A tapered optical waveguide structure is used for matching two waveguides that have different geometric cross sections or different material parameters. The tapered waveguide has one geometric cross section at the input plane and another cross section at the output plane. In between these planes, the core width changes linearly with the propagation distance.

This model demonstrates how to define a user-defined phase and wave vector for the Electromagnetic Waves, Beam Envelopes interface when used with perfectly matched layers (PMLs). The PMLs are used for absorbing the radiation that is not matching the port mode fields.

FRESNEL LENS

This model has been updated to use the new **Spatial FFT** dataset for performing a Fourier transformation in the postprocessing.

OPTICALLY ANISOTROPIC WAVEGUIDE

This model has been updated to use a coordinate transformation when rotating the optical axis to achieve longitudinal anisotropy.

OPTICAL SCATTERING OFF A GOLD NANOSPHERE

This model has been updated to use the new Symmetry Plane feature.

SELF-FOCUSING

This model has been updated to use the new Symmetry Plane feature.

FABRY-PEROT RESONATOR

This model has been updated to use the new Symmetry Plane feature.

Backward Compatibility with Version 6.0 and Earlier

COVARIANT FORMULATION

In 2D axisymmetry, for all studies except **Mode Analysis** and **Boundary Mode Analysis**, the covariant formulation is used (see Covariant Formulation in 2D Axisymmetry). Thereby, Ephi is not a dependent variable. To get the correct out-of-plane electric field component, the variable ewfd.Ephi should be used. Here, it is assumed that the

dependent variable has been given the name E and that the tag for the physics interface is ewfd.

NEW DEFAULT VALUE FOR THE INCIDENT-FIELD PARAMETER

For the **Scattering** and **Matched Boundary Condition** features for the Electromagnetic Waves, Beam Envelopes interface, when setting the **Incident electric field amplitude** parameter E0i, using the Java API, the IncidentField parameter must also be set to EField to make the electric field excitation effective. This is required as the new default value for the IncidentField parameter is NoIncidentField.

NEW DEFAULT VALUE FOR THE RELATIVE PERMITTIVITY, HIGH FREQUENCY PARAMETER

The default value for the **Relative permittivity**, **high frequency** parameter, epsilonInf, has changed from **From material** to **User defined**.

Material Library

New and Updated Material Data in Version 6.1

In version 6.1, the Material Library contains 12,763 materials and 107,005 material property datasets. The following improvements and additions have been made:

- Added data for: Nitronic 32, Nitronic 33, several tool steels, Cu-based alloys, commercial purity Ti, 4340, Inconel 600, 304, 304L, 314, 316, 316 H, 316 Ti, 416, 446, 15-5 PH, and several Hastelloys.
- Added data for several ECTFE, polypropylenes, PPS and PPE/PP, PPE/PS, and PPO/PP polymers blends.
- Added data for LiCoO2, Li22Si5, Li13Si4, Li7Si3, and Li12Si7.
- Added data for the following polymers: polycarbonates, polyamides, PPE, ULTEM, Xenoy, NORYL, Amodel, Meldin, Torlon, Ixef, PEEK, and KetaSpire.
- Added data for: Haynes 282, 20Cb-3, K63017, 21Cr-6Ni-9Mn, 420, 431, 201, 13-8 Mo PH, MP35N, BioDur 108, Custom 450, Custom 455, Temperature compensator 32 (type 1), Chrome core 8-FM, Chrome core 8-XP, Hiperco 27 alloy, Hiperco 50A alloy, Hiperco 50 alloy, Hiperco 50 HS alloy, Hiperco 15 alloy, and Manganin alloy 13.
- Added data for LiSi.
- Added data for several engineering plastics.
- Several epoxies and silicones have been added.
- Several tool steels and Fe-Cr alloys have been added.
- Added several circuit boards and antenna substrates.
- Some diallyl (ortho-, iso-) phthalates (DAP & DAIP) have been added.
- Data has been added for: Inconel 702, Nimonic 901, 15Cr15Ni1MoTi, liquid H and He, and some Mylars.
- Added data for PCBs from Rogers and TACONIC.
- Added properties for several Corning, Schott, Ohara, Incoloy 865, and FORMGLAS glasses.
- Added creep data for: S20910, AM-355, 15-7 Mo PH, 201LN, 422, Ti-140A, C-130AM, HM21 and HK31.

- Added strength data for: 304, 316, 316LN, Ti-155A, 52100, 440C, T1, T5, M1, M2, M10, Al 1000, Al 2024, Al 6013, 1010 steel, 201L, 2304, 403 and 403Nb.
- Added stress-strain and SN data for Ti Grade 4.
- Additional data has been added for the thermal expansion for several tool steels. In most cases, the data changed by less than 3%. The following had changes of 5% to 10% near 20 °C and less at higher temperatures: D2, H12, M4, and O1.
- A new reference is used for the elastic modulus of M2. The new values are similar at 20 °C but are 30% higher at 600°C.
- Added new references for H11 and H11 mod tool steels. For H11, the thermal data changed by up to 10% from previous values. For H11 mod the elastic properties changed by up to 5%, mainly at elevated temperatures.
- The data for the UTS, YS, and % elongation for stainless steels 420 and 440C were incorrect. This data was removed from the database. New yield strength data for 420 has been added.
- The electrical resistivities for Pyrex, Vycor, and window glass were too high by a factor of 100. It has been corrected.
- The specific heat and heat capacity for the following materials were extended down to the 10 K to 50 K range: Al 7075, Inconel X-750, BeO, Cr2O3, CuO, Cu2O, Fe2O3, Fe3O4, MnO, MnO2, Mn2O3, NiO, NbO2, Nb2O3, SiO2 (cristobalite), SiO2 (tridymite), SnO, SnO2, TiO2 (anatase), TiO2 (rutile), and ZnO. The room temperature values changed by 5% or less due to the new data.
- The stress-based fatigue data for the following four materials was in error: API-X100 pipeline steel, ATI 332Mo, ATI 334Mo, AZ31B-H24. The number of reversals was used instead of the number of cycles, and the cycles in the database were too high by a factor of 2. They have been corrected.
- The strain-based fatigue data for the following 14 materials was in error: 617, P92, GH413, API-X100, 34CrNiMo6, AF1410, SAE J435 Grade 0050A, Alloy 709 (NF 709), Ductile Ni-Resist D-5S, SAE J435 Grade 0030, 8630, 4140, and Inconel 617 (weld). The number of reversals was used instead of the number of cycles, and the cycles in the database were too high by a factor of 2. They have been corrected.
- The thermal conductivity for Schott Borofloat 33 was too high by a factor of approximately 20. It has been corrected.
- Added additional references for the electrical resistivity for Al and 310S stainless. The values changed by 5% or less due to the new data.

- Added additional references for the specific heat and heat capacity for: Al, Ti, Sn, Pb, and Constantan. The room temperature values changed by 5% or less due to the new data.
- The relative permittivity of sapphire was incorrect and has been fixed.
- The elastic modulus of 7049 Al was incorrect and has been fixed.
- Replaced the data for the thermal conductivity for Cu-based alloys C24000 and C51100 with a new reference. The values changed by up to 10% and 25%, respectively.
- Replaced the data for the thermal expansion for cobalt with a new reference. The values changed by up to 20%.
- The data for the thermal expansion for S20103, S20153, and S21800 has been replaced. The original reference had the materials mislabeled.
- The heat capacity and specific heat data for Li22Si5, Li13Si4, Li7Si3, and Li12Si7 were incorrect. The labeling in the reference was incorrect, and the data given in the reference was normalized to the number of atoms in a molecule. These have been fixed.

LiveLink[™] for Excel[®]

New Functionality in Version 6.1

The libraries for reading and writing Excel files have been updated.

LiveLink[™] for MATLAB[®]

New Functionality in Version 6.1

SYSTEM REQUIREMENTS

LiveLinkTM for MATLAB[®] is now again supported on all platforms, including macOS on Apple silicon processors like M1. MacOS is still supported on computers running on x64 chips, similar to the support for Linux[®] and Windows[®].

NEW FUNCTIONS FOR DATASETS

The mphdataset and mphdatasetinfo functions are new functions for plotting datasets and accessing information about datasets. These functions are very useful when working with datasets that are driven using changing parameters or with models that have varying geometries.

NEW FUNCTIONS FOR MODELS STORED IN MODEL MANAGER

The following new functions are introduced for working with models that are stored in databases using Model Manager:

- The mmmodelinfo function returns information about such models and can additionally be used to get information about models that are stored in an MPH-file.
- The mmsave function is used to store model as drafts or regular models.
- The mmgetbranch function can be used to obtain a batch variable that is used when saving a model in Model Manager that has not been previous stored in Model Manager.

Other functions such as mphtags, mphopen, and mphnavigator have been updated to support models loaded from Model Manager.

OTHER IMPROVEMENTS

• The mphrun function makes it possible to call the run() method on model nodes that support it and that support showing the progress bar window while solving. To do so, use the new argument 'run'. It is set to on by default, but you can turn it off in the rare event that you want to avoid recreating plot data even when the underlying model data or plot settings have changed. In most cases, users would want to leave this setting on.

- The mphsave function is used to save a model as an MPH-file or as source file in the Java[®], MATLAB[®], or VBA file formats. It now has support for setting the title and thumbnail directly when saving a model.
- It is now possible to open a COMSOL[®] model directly from the **Current Folder** browser in MATLAB[®]. Just right-click the COMSOL[®] MPH-file and select **Open**; the model will then be loaded into the MATLAB[®] workspace in the same way as when using mphopen. If you want to open a model in the COMSOL[®] user interface, instead right-click the MPH-file and select **Open Outside MATLAB**.

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 6.1

- The COMSOL Cosimulation block has been rewritten with focus on performance and stability. This means that simulation speed has improved substantially. It should now be easier to get models to simulate well when doing cosimulation with less or no manual adjustment of the solver settings or scaling of variables. Note that models that do require manual scaling (for example, most structural mechanics models) will also require manual scaling when doing cosimulation together with Simulink[®].
- If a Simulink[®] diagram is using more than one COMSOL Cosimulation block, then additional hidden COMSOL Multiphysics[®] server processes are started that handle the simulation of the additional blocks. This behavior may lead to a slightly longer startup time but also leads to a faster simulation time overall.
- Outputs are defined using a global scalar expression. The expression now has to include the component scope where the variable is defined, even if the model contains only one component (for example, comp1.point1). It is good practice to create a probe with the desired output expression.
- Unit handling has been updated so that units are shown for inputs, outputs, and block parameters for values that do have units. It is the user's responsibility to make sure that inputs and block parameters are set using the expected units. Outputs from the block are now converted into the unit that is shown for each output.

General Backward Compatibility

Models that have been 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 6.1 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:

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("rhogs", "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("rhogs", "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 6.1.

Index

A CC/DC Module

 new and updated models in 41
 new functionality in 37

 Acoustics Module

 new functionality in 44

 Application Builder, new functionality in

 I3

B backward compatibility, general considerations 32
 Battery Design Module
 new and updated models in 56
 new functionality in 53

- c CAD Import Module new functionality in 57 CFD Module new functionality in 60 new models in 62 Chemical Reaction Engineering Module new functionality in 64 **Composite Materials Module** new functionality in 70 new models in 71 COMSOL API changes 169 COMSOL Compiler new functionality in 17 COMSOL Multiphysics, new functionality in 17 **Corrosion Module** new and updated models in 74 new functionality in 72 D Design Module new functionality in 57 E ECAD Import Module
- new functionality in 75 Electrochemistry Module

new functionality in 76 Electrodeposition Module new functionality in 78

- F Fatigue Module new functionality in 80
 Fuel Cell & Electrolyzer Module new and updated models in 84
 new functionality in 81
- G general new functionality 10 Geomechanics Module new functionality in 85 geometry and mesh, new functionality for 21
- Heat Transfer Module
 backward compatibility 90–92, 104, 129
 new applications in 89
 new functionality in 86
- Liquid & Gas Properties Module
 new functionality in 94
 LiveLink for Excel
 new functionality in 165
 LiveLink for MATLAB
 backward compatibility 167
 new functionality in 165–166
 LiveLink for Simulink
 new functionality in 168
 LiveLink™ products for CAD
 new functionality in 57
 M Material Library
 new and updated material data in 162

MEMS Module new functionality in 95 new models in 99 Metal Processing Module new functionality in 101 new models in 102 Microfluidics Module new functionality in 103 updated applications in 103 Molecular Flow Module backward compatibility 105 new functionality in 105 Multibody Dynamics Module new functionality in 106

- N Nonlinear Structural Materials Module new functionality in 109 new models in 111
- operators, functions, and definitions, new and updated 25
 Optimization Module backward compatibility 115
 new functionality in 113
 new models in 114
- Particle Tracing Module Р backward compatibility 118 new applications in 105, 117 new functionality in 116 Pipe Flow Module new functionality in 119 new models in 119 Plasma Module backward compatibility 124 new applications in 123 new functionality in 121 **Polymer Flow Module** new functionality in 128 new models in 129 Porous Media Flow Module new functionality in 125
- **R** Ray Optics Module

backward compatibility 132 new applications in 131 new functionality in 130 results and visualization, new functionality in 27 RF Module new and updated apps and models in 136 new functionality in 133 Rotordynamics Module new functionality in 139 new models in 140

- S Structural Mechanics Module

 new functionality in 141
 new models in 150
 studies and solvers, new functionality in
 26
 Subsurface Flow Module
 new functionality in 153
 new models in 154
- U Uncertainty Quantification Module new functionality in 155
- Wave Optics Module new functionality in 156 new models in 159