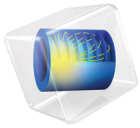

COMSOL Multiphysics

Release Notes

5.5

COMSOL

COMSOL Multiphysics Release Notes

Version: COMSOL 5.5

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries, contact Technical Support, or search for an address and phone number. You can also visit the Worldwide Sales Offices page at www.comsol.com/contact/offices for address and contact information.

If you need to contact Support, an online request form is located at the COMSOL Access page at www.comsol.com/support/case. Other useful links include:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/community
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM010001

Contents

Chapter 1: Release Notes

1

Release Notes

COMSOL Multiphysics® version 5.5 includes the new Metal Processing Module and Porous Media Flow Module; extended and improved versions of the Application Builder, COMSOL Compiler™, COMSOL Server™, and COMSOL Multiphysics®; as well as updates and improvements for all COMSOL Multiphysics® add-on products. These *Release Notes* provide information regarding new functionality in version 5.5 for all COMSOL® software products.

New Products

Metal Processing Module

The Metal Processing Module is a new add-on to COMSOL Multiphysics. Using this module, you can study how metallurgical phase transformations change the microstructure of a metallic material during a heating or cooling process. An example is the quenching of automotive steel transmission components, where the resulting microstructure is tailored to meet specific demands on strength and durability. Other examples include the study of phase transformations that occur during additive manufacturing of metal components and phase transformations in the heat-affected zone during welding. By combining the Metal Processing Module with the Heat Transfer Module, you get enhanced heat transfer functionality. Similarly, by combining the Metal Processing Module with the Structural Mechanics Module and its add-on modules, you get enhanced functionality for modeling boundary conditions, loads, and mechanical material behavior.

Porous Media Flow Module

The Porous Media Flow Module extends the COMSOL Multiphysics modeling environment to the quantitative investigation of mass, momentum, and energy transport in porous media. It is designed for researchers, engineers, teachers, and students, and it suits both single-physics and multiphysics modeling.

By combining the Porous Media Flow Module with the Heat Transfer Module, you get enhanced heat transfer functionality and the ability to model moisture transport with turbulent flow. Similarly, by combining the Porous Media Flow Module with the Chemical Reaction Engineering Module, you get enhanced functionality for modeling reversible, irreversible, and equilibrium reaction kinetics in free and porous media flow.

COMSOL Multiphysics

General New Functionality

NEW FLEXNET VERSION

The FlexNet license server has been upgraded from version 11.15.1 in COMSOL version 5.4 to version 1.16.2 in COMSOL version 5.5, so to run COMSOL version 5.5, you need to upgrade the license server.

UPGRADED JAVA VERSION

COMSOL 5.5 includes the AdoptOpenJDK 8 runtime update 222.

SUPPORT FOR IDLE TIMEOUT OF LICENSES

The license administrator can now specify an individual timeout value for each license feature using the keywords TIMEOUT and TIMEOUTALL in the LMCOMSOL.opt options file.

INSTALLER IMPROVEMENTS

- You can now choose to install COMSOL products with previews of the applications and example models in the Application Libraries for selected products or for all products.
- It is possible to use another Java® 8 runtime that you have licensed and installed.
- The look and feel of the installer has been improved.

New Functionality in the Application Builder

ADD-INS

There is now support for creating add-ins. Add-ins, or add-in programs, are extensions to the COMSOL Multiphysics software and make it possible to share methods and settings forms between models. You create them like applications using the tools in the Application Builder, and then define the add-in using the new **Add-in Definition**, **Form Definition**, and **Method Definition** nodes in the Application Builder. The add-in then becomes available from the **Add-ins** menu in the **Developer** toolbar. There is also an **Add-in Libraries** window that contains example add-ins and where you can add user-defined add-in libraries.

FILE DECLARATION AS THE TARGET

Command-line arguments for applications can now use a file declaration as the target. The argument value has to be a file on the client file system. Using a file declaration as the target is not supported when running the web client.

BRINGING ASSOCIATED EDITORS TO THE TOP

Selecting a **Form** or **Method** in the Application Builder tree brings its editor to the top if it is open. This functionality can be turned on and off using the **Link with Editor** button found in the Application Builder window's toolbar.

CHANGED SHORTCUT KEYS

The shortcuts for moving from the Model Builder to the Application Builder and vice versa are now Ctrl+Shift+A and Ctrl+Shift+M, respectively (instead of Ctrl+Alt+A and Ctrl+Alt+M).

IMPROVEMENTS TO THE FORM EDITOR

- You can now specify the line thickness and line color for **Line** form objects.
- There is a new **Add New Form** button in the **Panes** section header in the **Form Collection** object's Settings window. Clicking it creates a new form and uses it in the Form Collection. The **Add Choice List** buttons, found in the Settings windows for **Combo Box** and other form objects relying on choice lists, have also been moved to the section header.
- The **RGBA** option for the **Graphics** form object background color has been replaced with **Custom**, where a custom color can be selected from a color palette. The color setting is located in the **Appearance** section of the Settings window for the **Graphics** form object. Existing RGBA values are converted to their Custom counterparts.
- The **Graphics** form object now supports more detailed control over which standard toolbar groups to display.
- The background color of the **Graphics** form object's toolbar is now controlled by a **Background color** setting, which is located in the **Toolbar** section of the Settings window for the **Graphics** form object.
- The **Edit Custom Information Card** dialog box has been redesigned to have the same appearance as the **Edit Custom Toolbar Item** dialog box. In addition, it is now possible to add custom icons through the **Edit Custom Information Card** dialog box.
- The **Button** and **Toggle Button** form objects' style has been separated into size and style. The previous Small, Large, and Flat styles have been replaced by Small and

Large sizes and Flat and Raised styles. The **Style** list is available when **Size** is set to **Large**.

- The *onLoad* and *onClose* form events are now run when a form is referenced in a **Form Reference**, **Form Collection**, or **Card Stack** form object. Previously, the events were only run for referenced forms when switching between forms in a **Form Collection**.

IMPROVEMENTS TO THE METHOD EDITOR

- You can now run methods directly in the Application Builder by right-clicking the **Method** node and choosing **Run**. This functionality is not available for form methods and methods with arguments.
- The keyboard command for the **Check Syntax** action in the **Method** ribbon has been changed to F9 from F8, used in earlier versions. The **Compile Application** action found in the toolbar in the **Compiler** node's Settings window now has F8 as its keyboard command. F8 is also the keyboard command for the **Create Add-in** action found in the toolbar in the **Add-in Definition** node's Settings window.

IMPROVEMENTS AND CHANGES FOR THE APPLICATION LANGUAGE

These methods have been added or changed:

- For add-ins, the following new static methods are now available to retrieve the entity of a specific type that is considered the working entity in the COMSOL Multiphysics desktop window:

  Use `ModelNode getCurrentComponent();` to get the current component.

  Use `MeshSequence getCurrentMesh();` to get the current mesh sequence.

  Use `Physics getCurrentPhysics();` to get the current physics interface.

  Use `ResultFeature getCurrentPlotGroup();` to get the current plot group.

  Use `Study getCurrentStudy();` to get the current study.

  The `selectNode` method has been complemented by a method to retrieve the selected Model Builder tree node:

  ```
  ModelEntity getCurrentNode();
  ```

- Conversion methods for float types to double types have been added:

  ```
  double toDouble(float value)
  ```

  ```
  double[] toDouble(float[] value)
  ```

  ```
  double[][] toDouble(float[][] value)
  ```

- The new function `clearDebugLog()` clears the **Debug Log** windows in the Model Builder and the Application Builder.
- The utility method `String getClientFilePath(String)` has been changed to only return the filename part of the path when called from an application running in a web browser.
- The expanded state of sections in form collections can now be controlled using

  ```
  app.form("form1").formObject("formcollection1").
  expanded("form2", false);
  ```

- The expanded method is only supported by form collections that use sections. The first argument is the tag of the form that is represented by the section. The second argument determines if the sections should be expanded or collapsed.

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.

New Functionality in COMSOL Compiler™

- You can now download and install the COMSOL Runtime™ separately when needed. The files for the compiled applications can then be much smaller because they no longer include the runtime. The runtime will be installed in

  ```
  C:\Program Files\COMSOL\COMSOL55\Runtime
  ```

  When compiling an application, you can choose if the runtime should be downloaded or embedded in the compiled application.
- COMSOL Compiler now supports file larger than 4 GB on Windows® by using an external payload file with a .dat extension that the COMSOL Compiler creates automatically.
- The support for icons has been extended. You can now specify icon images for each individual platform selected for compilation. The supported formats for the icons are:
  - For Windows®: ico, png, and bmp.
  - For Linux®: png and bmp.
  - For macOS: icns, png, and bmp.
- The Settings window for the **Compiler** node now supports compiling applications that include custom physics interfaces created using the Physics Builder. You can choose to include Physics Builder development files, an external archive folder, or

an external Java archive (JAR file), if you have enabled the Physics Builder in the **Preferences** dialog box.

New Functionality in COMSOL Server™

The default COMSOL Server appearance has been updated, and the icons for COSMOL Server and COMSOL Client are now the same as for COMSOL Multiphysics.

New and Improved General Functionality in COMSOL Multiphysics

RIGHT-CLICK TO DEFINE SELECTIONS AND ADD PHYSICS FEATURES

It is now possible to right-click in the Graphics window to create a selection of geometric entities that acts as a *preselection*. You can then, if a physics interface node is selected in the Model Builder, right-click and add any applicable physics feature to that physics interface. While in this preselection mode, where the **Activate Selection** button is turned off in the physics feature's **Settings** window, you can select additional geometric entities and add them to the selection for the added physics feature or to another physics feature that you add from the context menu (or the **Physics** toolbar).

CHANGES TO PHYSICAL CONSTANTS

The values of the following physical constants have been updated according to new International System of Units definitions:

- Avogadro constant, from 6.022140857e23 1/mol to 6.02214076e23 1/mol.
- Boltzmann constant, from 1.38064852e-23 J/K to 1.380649e-23 J/K.
- Elementary charge, from 1.6021766208e-19 C to 1.602176634e-19 C.
- Planck's constant, from 6.626070040e-34 J·s to 6.62607015e-34 J·s.
- Permittivity of vacuum from 8.854187817e-12 F/m to `1/mu0_const/c_const/c_const`.
- Permeability of vacuum from `4*pi*1e-7` H/m to `2*alpha_const*h_const/c_const/e_const/e_const`.

SHOW MORE OPTIONS

The **Show More Options** settings for activating some advanced options have been moved from a menu to a dialog box (still accessed from the Model Builder toolbar) and split into more categories. Some categories have also been added. In addition to controlling

content in the Model Builder tree, now the ribbon content can also be updated in some cases.

SELECT BOX AND ZOOM BOX

It is now easier to use the **Select Box**, **Deselect Box**, and **Zoom Box** buttons multiple times in a row by keeping them enabled after the initial usage using the **Keep Enabled After Use** option. The **Keep Enabled After Use** option, when selected, keeps the corresponding selection mode active until you click the button again or right-click in the Graphics window. For **Select Box** and **Deselect Box**, there are also the new options **Entity Intersects** and **Entity Inside** so that you can choose whether geometric entities that the box intersects should be selected or deselected (**Entity Intersects**) or ignored unless they are fully inside the box (**Entity Inside**) as in earlier versions. The latter behavior is the default setting.

IMPROVED COPY-PASTE FUNCTIONALITY BETWEEN MODELS

The Model Builder nodes that you can copy-paste within models and between COMSOL sessions have been extended to also include:

- Functions
- Materials
- Under the **Definitions** branch, the nodes that relate to moving mesh, deformed geometry, optimization, and shared properties.
- Nodes under the **Multiphysics** branch.
- Plot groups and Plots
- Derived values
- Export nodes

Pasting to another model always does so using a best-effort principle, ignoring invalid settings. In a plot group, for example, the available settings depend a lot on the available datasets and solutions.

GRAPHICS WINDOW TOOLBAR IMPROVEMENTS

You can now control how to display buttons in the **Graphics** window toolbar using the **Graphics Toolbars** page in the **Preferences** dialog box. You can configure it to display toolbar items that you frequently use at the top level and place less common items in drop-down menus. There are four options for the buttons: **Normal**, **Compact**, **Wide**, and **Hidden**.

IMPROVED COMPARISON RESULTS TOOLS

In the **Comparison Results** window, the following new tools are available for comparisons of applications and models:

- In the **Comparison Results** window's toolbar, there are new buttons for creating a new comparison and for showing the next or previous difference.
- There is now an option for only including active settings.
- You can double-click nodes in the tree (or right-click and choose **Go to Source**) to move to the corresponding node in the Model Builder or Application Builder tree for the local file. When applicable, you can choose **Go to Remote Source** to move to the corresponding node in the Model Builder or Application Builder tree for the remote file.

IMPROVED SEARCH FUNCTIONALITY IN THE APPLICATION LIBRARIES WINDOW

The search functionality for finding applications and models of interest in the Application Libraries window has been improved with the following functionality:

- Disabled nodes are now ignored.
- You can search keywords for type, tag, and label using the prefixes `@type`, `@tag`, and `@label`, respectively.

IMPROVED ORGANIZATION OF THE TOOLS UNDER DEFINITIONS

The tools under the **Global Definitions** and **Definitions** nodes have been expanded and reorganized. Some changes compared to version 5.4:

- The **Component Couplings** menu has been renamed **Nonlocal Couplings**.
- The **Model Input** and **Ambient Properties** options are now available under **Shared Properties**.
- Under **Global Definitions**, **Common Model Inputs** has been renamed **Default Model Inputs**.

MISCELLANEOUS IMPROVEMENTS AND CHANGES

- A new **Calculated values** option in the **Excel Save** and **Excel Load** dialog boxes for export and import of table data. This option determines if the **Value** column is included in the exported data.
- It is now possible to move variables from one **Variables** node to another **Variables** node.

- You can now copy, paste, and duplicate the **Hide for Geometry** and **Hide for Physics** nodes and move them up and down in the model tree.
- In the root node's **Properties** window, there is a new **Search path** field, where you can specify a search path for external files.
- The **Developer** node under **Global Definitions** has been removed. Its children now appear directly under **Global Definitions**.
- There is a new preference setting: **Check for recovery files at launch**, which is selected by default. This setting controls if the **Recovery Files** window will appear at launch when there are any recovery files.
- You can now switch between model components using the **Component** menu in the **Model** section of the **Home** toolbar.

New Functionality in the Physics Interfaces

- The Nonisothermal Flow interface is now available in COMSOL Multiphysics for laminar flow.
- The Stabilized Convection-Diffusion Equation interface includes new interior boundary conditions with Dirichlet and flux options.
- The default feature in the Global ODEs and DAEs interface is now removable.
- The Laminar Flow interface has new default values for density, 1000 kg/m^3, and dynamic viscosity, 10^{-3} Pa·s, when the **User defined** option is selected.
- In tensor definitions for physical properties, **Anisotropic** has been changed to **Full** for the full 3-by-3 definition of an electrical conductivity, for example.
- For the PDE, Boundary Elements interface, the behavior for antisymmetry and zero charge at infinity has been improved.
- The behavior for pair selection in the physics interfaces has been improved. You can now add pairs using an **Add** dialog box.

New Geometry and Mesh Functionality

GEOMETRY FUNCTIONALITY

- For 2D geometry modeling, a new Sketch mode is available for interactively creating geometries in the Graphics window for a 2D geometry or part, or in a work plane for a 3D model. On the Sketch toolbar, you can choose to create the following 2D geometric objects: Circular Arc, Interpolation Curve, Quadratic and Cubic Bézier curves, Rectangle, Square, Circle, and Ellipse. If you have a license for the Design

Module, the Sketch toolbar also contains buttons for creating geometric constraints and dimensions. The snapping has also been improved and extended to include snapping to existing lines and circles and the possibility to snap lines and circles that you draw to existing points, lines, and circles.

- The Part Library for COMSOL Multiphysics has been extended and now includes the five Platonic solids (cube, tetrahedron, octahedron, dodecahedron, and icosahedron), a propeller with constant pitch, and a random flat surface.
- There are now more ways to rotate 3D geometry objects: In the **Rotate** feature, you can choose to specify an axis or rotation or to specify Euler angles. This option is also available in **Work Plane** features defined using the **Transformed** plane type and in the **Part Instance** nodes in the **Position and Orientation of Output** section. For the **Rotate** feature, there is also a new **Angle** field when you have selected to specify the axis of rotation, where you can add a vector of angles for creating multiple rotated objects.
- The geometry measurement information has been extended with the dimensions of the bounding box and the average coordinates (when you have selected two points).
- For 1D geometries, you can now specify intervals using interval lengths in addition to specifying coordinates in the Settings window for **Interval** nodes.
- The **Move** and **Copy** features now include a **Specify** list in the **Displacement** section with the options **Displacement vector** and **Positions**, which is a new option to specify the move using a vertex or coordinates for the old position and vertices or coordinates for the vertices to move to.
- If your license includes the CAD Import Module or any of the LiveLink™ products for CAD, you can export the geometry on the IGES and STEP formats.
- The geometry code and serialization have been improved and made more efficient.

MESHING FUNCTIONALITY

- For import of meshes on the STL and VRML formats, there are now separate **STL** and **VRML** file format options to choose from as the source in the **Import** node for mesh import. Also, for STL file import, there is now a **Create selection** check box. If you select it, a boundary selection of every `solid` section in the file is created during import.
- Mesh element shape optimization, which can result in curved elements inside the domain, is now available and is active by default. Interior elements are curved to improve the quality and avoid inverted higher-order elements. The new **Avoid inverted elements by curving interior domain elements** check box is available in the Settings windows for the main **Component** nodes and is selected by default.

- New **Create Vertices** and **Create Edges** mesh features are available in 2D and 3D for empty geometries. Additionally, **Create Faces** and **Create Domains** are available for adding faces and domains on imported 3D meshes. There is also a **Fill Holes** feature for filling holes in imported surface meshes. You find these tools on the **Create Entities** menu on the **Mesh** toolbar and when right-clicking the main **Mesh** node.
- You can now import and export 3D surface meshes on the PLY format (the Polygon File Format or Stanford Triangle Format) and the 3MF 3D printing format. You can choose to export and import to and from `*.ply` files as a PLY binary file or PLY text file and to and from `*.3mf` files as a 3MF file. You can also import and export meshes on the Sectionwise format.
- When exporting meshes on the COMSOL Multiphysics binary and text formats, a new **Export as second-order elements** check box is available. It is cleared by default, which means that the exported mesh contains linear elements.
- The **Adapt** node has been enhanced with the following new functionality:
  - For the **General modification** adaptation method, a **Maximum coarsening factor** setting is now available under the **Allow coarsening** check box.
  - For the **Regular refinement** and **Longest edge refinement** adaptation methods, a **Maximum number of refinements** setting is now available.
  - Under **External changes**, there is a **Reevaluate with Updated Model** button and an **Update when parameter is changed** list for selecting the parameter to notice any changes so that it affects the adaptation if it is used in a mesh size expression, for example.
- Free quad meshes for planar faces and 2D domains with concave corners have been improved using a new scheme in version 5.5. This new functionality is enabled by default in the Free Quad and Swept operations. It can be disabled by selecting **Legacy version 5.4** under **Tessellation** in a **Free Quad** node or by selecting **Quadrilateral (legacy version 5.4)** from the **Face meshing method** list under **Sweep Method** in a **Swept** node.
- Logical expressions for mesh functionality now support the built-in variables `h` (the local mesh size) and `qual*` (mesh quality measures).

New and Updated Operators, Functions, and Definitions

- New built-in mathematical functions:
  - `binomial(n,k)` — binomial coefficients.
  - `factorial(n)` — factorial of nonnegative integer.
  - `legendre(l,x)` and `legendre(l,m,x)` — Legendre polynomials and associated Legendre functions.
  - `zernike(n,m,r,phi)` — Zernike polynomials.
  - `sphericaly` and `sphericalyr` — spherical harmonic and real spherical harmonic functions, respectively.
- New variable utilities under Definitions:
  - **Vector Transform**: for transforming a vector of component expressions from one coordinate system to another.
  - **Matrix Transform**: for transforming rows or columns of a matrix of component expressions from one coordinate system to another.
  - **State Variables**: for defining dependent variables that are updated using an update expression before or after each completed solver step.
  - **Expression Operator**: for defining an operator that evaluates to arbitrary parameterized expressions on specified selections. You can also add **Operator Contribution** subnodes to an **Expression Operator** node.
- A new operator, `stddev`, is available for computing the standard deviation, using an integration or summation operator as its first input argument.
- New operators for time derivatives in different frames: *`frame`*`.dt(`*`<expr>`*`)`, where *`frame`* is the type of frame: `spatial`, `material`, `geometry`, or `mesh`.
- New units:
  - The *ounce* as `oz` and `ounce`, equal to 1/16 pound (0.028349523 kg).
  - The *fluid ounce* as `fl_oz` and `fluid_ounce`, equal to 1/160 imperial gallon ($2.8413063 \cdot 10^{-5}$ m^3)
  - The *rayleigh* as `rayl` and `rayleigh` (SI version, equal to 1 Pa·s/m) and `rayl_cgs` and `rayleigh_cgs` (CGS version, equal to 10 Pa·s/m), used for measures of specific acoustic impedance.
- A new **Combined System** coordinate system, which can be used in 1D, 2D, and 3D when you want to refer to different coordinate systems in different domains, is now available.

- 3D **Rotated System** coordinate systems are now also available in 2D so that you define general rotations using 3D Euler angle inputs.
- In the **Material Browser**, you can now reload material libraries by right-clicking a material library node and choose **Reload Selected**.
- For external material functions, there are new reserved return values: the return value –1 is now interpreted as a normal exit with a warning message; the return value –2 is interpreted as a normal exit with a log message. Both values previously indicated an unspecified error condition. Also, you can now add two new electromagnetic options: general H(B) relation and the General B(H) relation.

IMPROVEMENTS FOR MOVING MESHES AND DEFORMED GEOMETRIES

- Similar functionality for defining deformed geometries is now available in the **Deformed Geometry** submenu under **Definitions**, as were previously available for moving meshes.
- A new **Rotating Boundary** feature is available under both **Moving Mesh** and **Deformed Geometry** for defining boundaries of a geometry that is rotating. You can specify the rotation type and the rotation axis.

IMPROVEMENTS FOR LAYERED MATERIALS

- The layers in the **Layered Material Link**, **Layered Material Stack**, and **Layered Material Stack Link** nodes can be transformed as Symmetric, Antisymmetric, and Repeated.
- The Single Layer Material node has been deprecated in version 5.5. Instead, all standard boundary materials can be switched to a single layer material in one of the following ways:
  - By adding a **Shell** property group into the standard material node. You need to enter a thickness to let this material become a single-layer material.
  - Select **Single Layer Material** from the **Material** node's **Layers** submenu. Doing so will add a blank material together with a **Shell** property group as a subnode and set a default value of `1e-4[m]` for the thickness.
  - Assign a value to the requested thickness in the **Material Contents** table.

- Changes for the **Layered Material** node:
  - The default selection of the interface material has been changed from **None** to **From layer**, which means that the interface material properties will be computed from the adjacent layers' material properties by default.
  - A standard **Material** node in the global component can turn into a layered material by adding a **Shell** property group.
  - The layer thickness of a layered material can be a number or a parameter.
- For the **Layered Material Link** and **Layered Material Stack** nodes, the default selection of the nonlayered material has been changed from **None** to **Same as layered material**. It means that the nonlayered material properties are computed as an average value of the layers' material properties.
- Improvements for layer thickness variation:
  - The layer thickness can be scaled with a factor that can depend on geometry variables. The scale property is available in the settings for **Layered Material Link**, **Layered Material Stack**, and **Layered Material Stack Link** nodes.
  - The thickness of a single-layer material can be a number, parameter, or value.
  - When a layer is defined with a scaling factor, it will be seen in the preview windows with a darker color than a nonscaled layer.

New Functionality in Studies and Solvers

STUDIES AND STUDY STEP FUNCTIONALITY

- For adaptive mesh refinement, it is now possible to choose on which geometric level the mesh adaptation will be performed, so that you can do adaptive mesh refinement on surfaces, for example. You specify the geometric entity level in a new **Geometric Entity Selection for Adaptation** section in the main study step's Settings window. In that section, you can also select for which domains or surfaces, for example, to perform the mesh adaptation (that is, you can do adaptive mesh refinement in a subset of the geometry).
- It is now possible to add a number of global goal-oriented quantities to make the mesh adaptation terminate when those quantities are stable to a requested accuracy. These goal-oriented quantities could, for example, be the S-parameters for an RF simulation. The goal-oriented termination can be used for any error estimation method supported by the adaptation and error estimates algorithm. Choose **Manual** from the **Goal-oriented termination** list under **Adaptation and Error Estimates** in the

study step's Settings window to enter user-defined goal-oriented quantities and their tolerances.

- For time-dependent adaptive mesh refinement, the **General modification** method and **Rebuild mesh** methods are now available in the **Adaptive Mesh Refinement** node's Settings window. The general modification method can resolve sharp fronts with fewer mesh elements in total compared to the previous methods.
- Select the new **Plot the location of undefined values** check box in the **Settings** window for the main **Study** node to create a separate window that contains a plot that shows the location of any undefined values, such as Inf or NaN, which can help you locate a problematic part of the model.
- All model tree nodes that can appear in the **Modify model configuration for study step** tree now include a **Disable in All Studies** option in their context menus if there is at least one study step where the **Modify model configuration for study step** check box is selected; this option simplifies disabling a model tree node in several study steps.
- In the **Optimization** study step's Settings window, a new **Enforce design constraints strictly** check box is available in the **Constraints** section for derivative-free optimization solvers that support the penalty constraint handling method (Nelder–Mead, Coordinate search, and Monte Carlo) and for COBYLA. If selected, the solver evaluates all design constraints before the forward problem is run. If infeasible constraints are found, the forward problem is not run, and the optimization solver proceeds to the next iteration.

SOLVER FUNCTIONALITY

- A new Domain Decomposition (Schur) solver is now available to provide domain decomposition using an exact Schur complement and an algebraic hybrid direct-iterative solver. The Domain Decomposition solver in earlier versions of COMSOL Multiphysics is now available as Domain Decomposition (Schwarz).
- It is now possible to stop iterative improvements if the residual is not reduced by using the new **Error ratio bound** setting in the solver's Settings window's **Error** section. By default, it is set to 0.5 (valid values are between 0 and 1; a lower value means that the iterations terminate more quickly). When the **Check error estimate** setting is set to **Automatic**, a single warning of *Iterative refinement triggered* appears in the **Log** window if the iterative refinement is triggered.
- An **Upper Limit** node is now available under the **Segregated** solver node to specify an upper limit for field values in a similar way as for the already available **Lower Limit** node.

- The MUMPS solver has been upgraded to version 5.2.1 and now supports blocked low-rank factorization, both when computing the LU factors and when storing them. This is an approximate but accurate LU-factorization method that can potentially save memory in your models. You activate it by selecting the **Block low rank factorization** check box in the Settings window for the MUMPS direct solver.
- For the Generalized Alpha and BDF methods, when Backward Euler is selected for the consistent initialization, there is a new **Rescale after initialization** check box. Select it, for example, in fluid-flow models that can suffer from a bad initial scaling due to a zero initial value for the velocity, to provide a smooth start and avoid using too small time steps.
- You can now get a log and plot of errors per field and state during the solution for applicable multiphysics models if you select **Detailed** from the **Solver log** list in the **Advanced** node's Settings window.
- A new **Lower element order first (any)** setting is available for the algebraic multigrid (AMG) and smoothed aggregation AMG (SAAMG) solvers. It makes it possible to use the combination of first using the geometric multigrid (GMG) solver with lower order until order 1 is reached and then using the AMG or SAAMG solver to generate the coarser levels in a multigrid approach. The advantage with the new setting is that you can control the total number of multigrid levels in one setting and that you do not have to repeat the pre- and postsmoother settings in two places.
- For controlling oscillations around discontinuities and for stabilizing the computations of typically nonlinear conservation laws, a WENO (weighted essentially nonoscillatory) limiter has been added to the discontinuous Galerkin method. WENO is available for the Wave Form PDE interface and for the Compressible Euler Equations interface in the CFD Module. In those physics interfaces, select **WENO** from the **Limiter** list to activate the WENO limiter in the main interface nodes' Settings window.
- You can now specify a maximum value for the scaling of eigenvectors in the Settings window for the **Eigenvalue Solver** node by choosing **Maximum** from the **Scaling of eigenvectors** list and then entering a value in the **Maximum absolute value** field. The peak value is then normalized to that value. You can use this setting to keep eigenmodes small.

CLUSTER AND BATCH FUNCTIONALITY

- For saving solutions on a cluster, you can now right-click a **Solution** node and choose **Store Solution Using Distributed Storage** to store the solution using a distributed storage on clusters, which can improve performance using parallel I/O.

- There is a new **Keep solutions before and after stop condition** check box under the **Stop condition** field in the **Parametric Sweep** node's Settings window under **Job Configurations**. Using this option, you can store solutions for the parametric solutions both before and after the stop condition has been satisfied.
- A new `-batchlogout` option for batch runs is available to direct the log to standard out when storing on file.
- A new `-norun` option for batch runs is available for not running the model, which can be useful if you, for example, just want to run `-clearsolution` or `-clearmesh` for a model that already includes a solution or mesh and then save it, without a solution or mesh, without computing the model first.

New and Improved Results and Visualization Functionality

NEW AND IMPROVED DATASET AND EVALUATION FUNCTIONALITY

- Using the new **Filter** dataset, you can filter another dataset's geometry with respect to an expression. It is also possible to create a mesh part from this dataset by right-clicking it and selecting **Create Mesh Part**.
- The **Image** export feature has been improved:
  - The previous 1D Image, 2D Image, and 3D Image export features have been merged to a single **Image** feature.
  - The **Image** export feature can now export images of many more things than plot groups, such as geometries, meshes, and selections for physics features.
  - You can create custom presets that specify the image size as well as what to include in the exported image.
  - The export target can now also be the clipboard and PowerPoint® (on Windows® only) in addition to the file target.
- Evaluation groups now support all evaluation features.
- It is now possible to disable the closing surfaces for incomplete revolutions in Revolution 2D and Revolution 1D datasets, by clearing the **Add end caps if the revolution is not full** check box.
- In **Evaluation Group** nodes, you can now choose **Difference** and **General** from the **Type** list in the **Transformation** section. The difference computes the difference of all evaluations (`evaluation1-evaluation2-evaluation3`, and so on). The **General** option provides the possibility to enter a general expression in an **Expression** field.

Such an expression can include global parameters and mathematical and physical constants, for example.

- New **Global Evaluation Sweep** nodes are available under **Derived Values** and **Evaluation Group** nodes. You can use them to vary a small number of model parameters as an evaluation sweep during postprocessing. A global evaluation sweep can be useful for postprocessing of reduced models, for example.
- It is now possible to do transposed evaluation in evaluation groups by selecting the **Transpose** check box in the **Transformation** section of the Settings window for an **Evaluation Group**.
- When exporting plot data for 1D plots with multiple curves, it is now possible to export in a compact format where there is one column containing x-coordinates and one or more columns containing, for example, y-coordinates. To do so, select **Append as columns** from the **If multiple curves** list.
- In numerical evaluations and plots that can take multiple expressions, the **Add Expression** and **Replace Expression** lists now contain an **All expressions in this group** option for some groups of related expressions, such as vector components, so that you can add all of them at once.
- In the **Mirror 2D** and **Mirror 3D** datasets, you can now choose to remove elements on the symmetry axis or symmetry plane, respectively. You can use this functionality to get a plot that only shows relevant edges in the mirror dataset.
- It is now possible to evaluate maximum and minimum values in Gauss points and Lagrange points for maximum and minimum datasets, numerical evaluation features, and plots.
- The **Shell** dataset is now available also for 2D axisymmetric models.
- It is now possible to make interface and layer selections in the **Layered Material** dataset.
- You can now control the precision when exporting tables and evaluation groups to Excel®.
- The export of data to Excel is now also significantly faster than in earlier versions.

NEW AND IMPROVED GRAPHICS AND PLOT FUNCTIONALITY

- Selections are now available in the settings for 2D and 3D Plot Group nodes. By default, the entire geometry is selected, but you can choose to only use the plot group in some part of the geometry.
- You can now export images to PowerPoint® in the **Image Snapshot** window. With the new add-in for PowerPoint, which can be installed together with COMSOL

Multiphysics, you can work with linked images between COMSOL and PowerPoint. The installer registers the COMSOL Interface add-in with PowerPoint. When the interface add-in is loaded, it provides the following additions to the PowerPoint user interface:

- The COMSOL ribbon tab, where you can insert and update images.
- The COMSOL tab in the File Backstage View, where you can manage the source references for the linked images.
- The COMSOL Image tools tab, where you can change image settings.
- The COMSOL contextual menu, where you can quickly access the most used commands.

- In the **Expression** sections in the **Settings** windows for plot nodes, you can now use **Plot Next Expression** and **Plot Previous Expression** buttons to step though all predefined expressions in a group of expressions in the **Replace Expression** tree.
- You can now control how numbers are formatted in the title and the parameter indicator. In the **Title** section for the plot group nodes, there is a new **Number format** list with the following options: **Default**, **Automatic**, **Engineering**, **Scientific**, **Stopwatch**, and **Scientific stopwatch** with associated settings for controlling the number format.
- Streamline plots now support interactive spheres (points) as well as interactive arrows. It is also possible to generate animations of the interactive spheres and of arrows that move with the field.
- The color scale in plots can now be defined as the gradient between two colors.
- You can now export 3D surface plots on the PLY format (the Polygon File Format or Stanford Triangle Format) and the 3MF 3D printing format. You can choose to export to `*.ply` files as a PLY binary file or PLY text file and to `*.3mf` files as a 3MF file.
- For 3D image export in the glTF™ format, there is now an option to not include line segments. Lines segments are not supported by all glTF viewers.
- In 1D plots, a new **Manual** option is available for placing the legend box freely in a position that you provide as x- and y-coordinates.
- In plot titles, you can now evaluate expressions in manual parameter indicators.
- In **Filter** subnodes to plot nodes, a **Use derivatives** check box is now available when the **Element nodes to fulfill expressions** setting is **Smooth**. Select it to use cubic interpolation in, for example, postprocessing of topology optimization models.
- Arrow plots in 2D and 3D can now use double arrows to, for example, illustrate moments in solid mechanics.

- For **Arrow Volume**, **Coordinate System Volume**, and **Principal Stress Volume** plots in 3D, a new **Placement** list provides the options to place the arrows or symbols in a **Grid**, **Gauss points**, or **Mesh nodes**. For **Arrow Surface**, **Coordinate System Surface**, and **Principal Stress Surface** plots in 2D, you can choose **Grid** or **Mesh nodes**. For similar plots on lower dimensions (such as an **Arrow Line** plot in 2D), the position settings are available in a separate **Arrow Positioning** or **Positioning** section.
- Histogram plots include a new normalization setting. Select **Sum of values** to divide the absolute count of each bin by the sum of the counts over all bins. Select **Integral** to divide the absolute count of each bin by the sum of the counts over all bins, each multiplied by the width of the bin. The new **Sum of values** option is equivalent to the **Integral** option in previous versions.
- When you define the number of contour or isosurface levels in **Contour** and **Isosurface** datasets and in **Contour**, **Isosurface**, and **Directivity** plots, a new **Round the levels** check box is available and selected by default to provide a rounded uniform step between the levels.
- You can now add **Deformation** subnodes to **Annotation** plot nodes so that annotations can be moved corresponding to a deformed surface plot, for example.
- **Deformation** subnodes are also available for 2D and 3D **Radiation Pattern** plots.
- **Directivity** plots are now available in 1D plot groups.
- Support for plotting matrix variables has been added to the following plot types: **Coordinate System Surface** in 3D and **Coordinate System Line** in 2D and 3D (choose **Matrix variable** from the **Source** list).
- A time resolution setting is now available in a **Quality** section in Global and Point Graph 1D plots referring to a time-dependent solution.
- The **Reference direction** settings are now available also for 1D **Radiation Pattern** plots of 2D models.
- There is now an option to show the axial symmetry line in 2D plots that refer to a dataset with a 2D axisymmetric geometry.
- Whirl plots can now handle multiple rotors.
- It is now possible to plot ellipses without arrows in the trajectory plots.
- You can now display the interfaces between layers in **Through Thickness** plots. You can also choose a length unit for the thickness coordinate.
- It is now possible to evaluate at multiple locations in **Layered Material Slice** plots. The **Location definition** list contains two new options: **Interfaces** and **Layer midplanes**. In addition, the **Local z-coordinate** field now accepts an array of values. In a new **Layout**

section, you can define the displacements as **Linear** or **Rectangular** and select an orientation of the displacements. You can also select a **Show description** check box to add an annotation (the *z*-coordinate, layer, or interface numbering, depending on the through-thickness location). Using this functionality, you can create a plot that required several **Layered Material Slice** and **Annotation** plots in previous versions. **Layered Material Slice** plots are now also available in 2D models and can, in 3D **Layered Material Slice** plots, evaluate in Revolution 2D datasets that refer to 2D axisymmetric solutions containing layered materials. There are also location input and location definition settings in numerical evaluation nodes such as **Volume Average**, **Volume Integration**, **Volume Maximum**, and **Volume Minimum** for 3D models that contain a layered material.

- You can now add a **Filter** subnode to **Through Thickness** plots.

New Java® API Methods

LOADING AND SAVING FILES FOR PARAMETRIC SWEEPS

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

IMAGE OBJECT FOR MESH FEATURES

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

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

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

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

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

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

ONLY PLOT WHEN REQUESTED

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

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

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

```
boolean isOnlyPlotWhenRequested();
```

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

General Backward Compatibility Considerations

COMSOL Multiphysics version 5.5 can open MPH-files saved from COMSOL Multiphysics versions 4.0–5.4.

COMSOL Multiphysics version 5.5 can run Java® files saved from COMSOL Multiphysics versions 4.0–5.4. However, the Java® files may need to be modified in accordance with information following in this document and may need to be recompiled with the `comsol compile` command in version 5.5.

Backward Compatibility with Version 5.4

GEOMETRY REPRESENTATION ARRAYS

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

INTERPOLATION CURVES

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

SCALING 2D GEOMETRY OBJECTS

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

FREE QUAD MESHING

Models saved in versions 5.3 to 5.4 with the free quad tessellation method set to **Automatic** (in a **Free Quad** feature) or face meshing method set to **Quadrilateral** (in a **Swept** feature), will, when opened in version 5.5, have the tessellation method or face meshing method set to **Legacy version 5.4** or **Quadrilateral (legacy version 5.4)**, respectively.

PARTICLE TRAJECTORIES STUDY STEP

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

IMAGE EXPORT PROPERTIES

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

Backward Compatibility with Version 5.3a

ADAPTATION AND ERROR ESTIMATES

The property `rmethod` has been renamed `meshadaptmethod`, with additional possible values. The old property (`rmethod`) and its values can still be used.

MASS PROPERTIES AND MASS CONTRIBUTIONS

In the **Density** section, the **Integration frame** list in previous versions is now called **Density input frame**.

In the API, the corresponding property name has been changed from `integrationFrame` to `densityFrame`.

NO FLUX FORMULATION

The formulation of the No Flux feature has been changed in the Transport of Diluted Species interface in version 5.4 so that the flux relative to the convective flux is set to

zero. The new formulation is used when opening a model from a previous session. In the previous formulation, the total flux was set to zero.

LEGACY FLAG -DCS.LEFTRIGHTSELECTION HAS BEEN REMOVED

The legacy flag `-Dcs.leftrightselection` for selections of geometric entities using left-right click has been removed in version 5.4.

AMD ACML LIBRARIES

The AMD ACML library is no longer supported by AMD. It has been replaced by standard BLAS and LAPACK as an alternative to Intel MKL. On Linux, the BLIS library is also available. Intel MKL is still the default BLAS library.

STRICTER SOLUTIONINFO VALIDATION

The `SolutionInfo` validation now ensures that `SolutionInfo.isValid()` returns true before using other `SolutionInfo` methods, unless stated otherwise in the documentation of the methods.

Backward Compatibility with Version 5.3

PDE, BOUNDARY ELEMENTS INTERFACE

The PDE, Boundary Elements interface has been reimplemented and improved. Backward compatibility is provided for MPH files but not for the Java API.

STABILIZATION IN SOME PHYSICS INTERFACES

Model files created in version 5.3 and earlier retain the old stabilization formulation. The new formulation is obtained by replacing the physics interface with a new one. The improved stabilization breaks Model Java-file backward compatibility for time-dependent models that contain one of the affected physics interfaces. Please contact COMSOL Support to learn how to retain backward compatibility for your specific Model Java-file.

STORE SOLUTION ON DISK

The **Store solution on disk** option has been removed.

FUNCTIONS IN GEOMETRY FEATURES

The button **Rebuild with Updated Functions** has been removed from the following geometry features: **Parametric Curve**, **Parametric Surface**, and **Sweep**. Instead, use the **Refresh** or **Clear Functions** button in the function features.

In the API, the method `importData()` is obsolete for the geometry features `ParametricCurve`, `ParametricSurface`, and `Sweep`. Instead, use the `refresh()` method in the function features.

PARTICIPATION FACTOR

Support for participation factors in the eigenvalue solver has been replaced by a new and improved **Participation Factors** feature, which is available under **Definitions>Variable Utilities** in a model component for computing participation factors in structural simulations.

INTERPOLATION CURVES

For interpolation curves, in the case when relative tolerance is zero (which is the default), the algorithm in 5.3a has changed so that the shape of the curve becomes somewhat different (while still interpolating the given points).

ITERATIVE SOLVERS

For the iterative solvers, the error handling mechanism has been improved to contribute to a general robustness of the computation. The improved mechanism is enabled per default and can give the following effects for nonlinear iterations:

- Increased total number of linear iterations
- Failure with the error "Divergence of the linear iterations", if the preconditioner is extremely ill-conditioned.

UNITS IN THE SIZE EXPRESSION NODE

The size expression is now interpreted in the model component's unit system rather than in the geometry's unit system. Also, the spatial coordinates (x, y, z, and so on) are now always in the component's unit system. In version 5.3, when evaluating on a grid, the spatial coordinates were in the geometry's unit system.

STRESS/STRAIN PLOTS

Principal stress/strain plots now evaluate the stresses and strains in the centers of mesh elements instead of in the mesh nodes.

Backward Compatibility with Version 5.2a

FRAME DEFINITIONS

In version 5.3a, all frames are always defined. If you open models from version 5.2a or earlier, there will be a **Permanently Define All Frames** button under **Frames** in the General

section of the **Component** nodes. When you click the button, all frames are defined, and the button disappears.

FRAME CONTROL

In previous versions of COMSOL Multiphysics, it was possible to apply multiple frame-controlling physics on the same selection. For example, it was possible to add two Solid Mechanics interfaces (with the **Include geometric nonlinearity** option enabled) on the same selection and solve. In this case, an "override rule" was applied between the controlling physics, effectively meaning that the last physics in the tree had control of the frame deformation for the overlapping domains. This situation could cause unexpected results.

In version 5.3, multiple physics interfaces controlling the same frame are not allowed on the same selection and will cause an error when trying to solve. However, it is still possible to use multiple frame-controlling physics with overlapping selection, but you now have to explicitly disable frame control on all but one of the physics. This gives you better control on what is going on. The functionality to disable frame control for physics has been incorporated in the **Settings** window for study steps to allow choosing different physics to control the frame in different study steps.

Frame-Scoped Variables

Certain variables pertaining to frame deformation are now defined with "frame prefix". For example, the relative element volume is now called `spatial.relVol`. Previously, these variables were added by the physics interface controlling the frame deformation so they used a "physics prefix" (for example, `ale.relVol`). For backward compatibility of old models, the physics interfaces still define the old variables, but they are aliases of the new frame variables.

API SYNTAX FOR ERROR HANDLING

The old syntax for an error

```
feature("ftag").feature("prob1").feature("error1")
```

is no longer supported. Instead, use the following documented syntax:

```
feature("ftag").problem("error1")
```

FREE QUAD MESHING

Models saved in version 5.2a with the free quad tessellation method set to **Automatic** (in a **Free Quad** feature) or face meshing method set to **Quadrilateral** (in a **Swept** feature), will, when opened in 5.3, have the tessellation method or face meshing method set to **Legacy version 5.2a** or **Quadrilateral (legacy version 5.2a)**, respectively.

Backward Compatibility with Version 5.2

FREE QUAD MESH

The new free quad meshing algorithm is used in new models, but for models created in earlier versions, the legacy algorithm is used instead. The default for the new `method` property is therefore `auto` in new models, and `legacy52` in migrated models.

GEOMINFO CHECK() METHOD

The return value of the `check()` method in GeomInfo has been removed. Now, `check()` issues an error if the geometry is invalid.

THE CONST PROPERTY IN SOLVERS

The `const` property available for many solvers has been removed in version 5.2a and replaced with the `cname` and `clist` properties, which are string arrays for the constant names and corresponding constant values, respectively. For the **Dependent Variables** node, the new property `initparametersmethod`, which can be set to `auto` (the default) or `manual`, has been added to control automatic synchronization of values of parameters to use for initial expressions. The **Dependent Variables** node now also includes the `cname` and `clist` properties.

Backward Compatibility with Version 5.1

CREATING SELECTIONS

In version 5.2, the `selresult` property replaced the `createselection` property. `createselection` is still supported for backward compatibility.

SELECTIONS IN PART INSTANCES

For backward compatibility for selections in part instances, there is a **Keep noncontributing selections** check box, cleared by default, in the **Selection Settings** section of the Settings window for **Part Instance** nodes. If you select the **Keep noncontributing selections** check box, the **Keep column** is disabled, and the selection is kept if the **Contribute to** value is **None**. In the COMSOL API, the default is an active Keep noncontributing selections setting.

MESH PARTS

For backward compatibility regarding STL/VRML import, and for COMSOL API compatibility, the old user interface for mesh import (with a specified filename and import properties) and the corresponding properties are available in the COMSOL

API but not in the COMSOL Desktop, unless it is the active option. This means that the old user interface can only be reached if the filename is set in combination with the STL import type (the API type `stlvrml`), which cannot be done from the new user interface.

DORMAND-PRINCE 5 TIME-STEPPING SOLVER

The Dormand-Prince 5 Runge-Kutta solver in 5.1 does not use field norm scaling in version 5.1 when estimating errors. This means that the errors of, for instance, an ODE might become very small when solved together with a field with many DOFs, and the time steps taken might then be too large. In version 5.2a, the Runge-Kutta solver uses field norm scaling, which means that old models using the Dormand-Prince 5 might need tighter tolerances to produce results in earlier versions that are similar to those in 5.2a.

Backward Compatibility with Version 5.0

MESH IMPORT

The method used to automatically partition the boundary of imported meshes in 3D has been improved. If you have an existing model, you can work with it without being affected. However, if you click the **Import** button of the **Import** feature in the meshing sequence, the file is read again and the new partitioning method is used. The exception is if the **Import** feature had **Boundary partitioning** set to **Manual**. In this case, the modified parameter values are preserved under the **Feature detection** setting, which uses the same algorithm as in version 5.0.

COMSOL tries to map the old selection on boundaries to the new boundaries, but it is not always possible to do accurately when new faces have appeared or old faces have disappeared. You may have to manually review and update boundary, edge, and point selections after reimporting the mesh.

If you have a Java® or MATLAB® program that imports meshes, the number of geometric entities may have changed compared to older versions.

SECURITY SETTINGS

In version 5.2a, the **Allow external process and libraries** check box on the **Security** page in the **Preferences** dialog box is cleared by default to not allow applications to start external processes on the computer. The default setting in 5.0 is set to allow such external processes.

DISPLAY OF MAX/MIN MARKERS IN PLOTS

The display of max/min markers in plots is now off by default to make plots fit better when using a small graphics canvas. In previous versions, the display of max/min markers in plots was on by default.

MERGED MPH-FILE FORMAT

The MPHAPP file and MPH file formats have been merged since version 5.1, and all application files use the `.mph` file suffix. You can still open MPHAPP files created in version 5.0.

AC/DC Module

New Functionality in Version 5.5

ELECTRIC CURRENTS IN SHELLS AND LAYERED SHELLS

The new AC/DC interfaces, Electric Currents in Shells and Electric Currents in Layered Shells, are an evolution of the Electric Currents, Layered Shell interface released in version 5.4 and the older Electric Currents, Shell interface. They provide improved usability and robustness. Modeling of both nonlayered shells, as well as layered shells has become more streamlined, and integration with other physics has been improved. These new AC/DC interfaces only differ in the default settings and can be used interchangeably, so in reality, it is one interface with two entry points.

When the interface is used for layered shell modeling, the equations are solved in a product space spanned by the shell's boundary selection in 3D and an extra dimension that points in the shell's normal direction. This accounts for both tangential and normal electric fields in the shell. In this state, the interface can model conductors as well as dielectrics. It supports Stationary, Frequency Domain, and Time Dependent studies. Together with the MEMS Module or Structural Mechanics Module and the add-on Composite Materials Module, this allows for modeling piezoelectric materials in layered shells. For modeling layered shells, the interface replaces the Electric Currents, Layered Shell interface released in version 5.4.

When used for nonlayered shells, only tangential electric fields are accounted for. It then replaces the old Electric Currents, Shell interface, except for piezoresistive modeling. The old Electric Currents, Shell interface has been renamed Electric Currents, Single Layer Shell and is available in the Model Wizard and Add Physics dialog under: **AC/DC>Electromagnetics and Mechanics>Piezoresistivity**.

For backward compatibility, models created in the old physics interfaces will open using legacy versions of the same old physics interfaces.

MULTIPHYSICS COUPLING FOR ELECTRO-ACOUSTIC TRANSDUCERS

The new Lorentz Coupling feature supports a two-way coupling between the Magnetic Fields physics interface and the Solid Mechanics interface. The Lorentz force is determined by computing the cross product of J and B in the volume of the domain. This force is then applied on the mechanics side as a volumetric force. At the same

time, the velocity is taken from Solid Mechanics and applied in the Magnetic Fields interface as a Lorentz velocity term.

The feature is intended for conductive, nonmagnetizable domains (typically, copper coils). Together with the Acoustic-Structure Boundary multiphysics coupling, this allows for modeling electro-acoustic transducers. It is available in 2D and 3D for Time Dependent, Frequency Domain (Perturbation), and Eigenfrequency analysis and requires the AC/DC Module together with one of these modules: Structural Mechanics, Acoustics, or MEMS.

NEW AC/DC MODEL WIZARD TREE

The physics interfaces in the AC/DC branch of the Model Wizard tree have been reorganized to allow for easier navigation. Important multiphysics interfaces that involve the AC/DC Module have been included as well.

EASIER MODELING OF PERMANENT MAGNETS

About 50 sintered NdFeB grades (adhering to the Chinese standard) have been added to the AC/DC material library. The materials include N, M, H, SH, UH, EH, and TH grades, and are characterized by typical values for the remanent magnetic flux density norm and the recoil permeability.

The remanent flux density constitutive relation has been updated to support these new materials. The magnitude of the remanent flux density is provided by the material, while the direction is specified in the physics settings. This allows for quick and easy modeling of devices with permanent magnets. By combining the materials together with a Material Switch feature, you can sweep over different grades to investigate the corresponding performance. A number of models in the AC/DC Module Application Library have been updated to use the new materials.

MATERIAL MODEL IMPROVEMENTS

The user interface and section naming of the material models have been improved and made more consistent across all interfaces within the AC/DC Module, see also previous section.

IMPROVED B-H CURVES IN NONLINEAR MAGNETIC MATERIALS

A new utility application, the B-H Curve Checker, is available for investigating the quality, smoothness, and physical correctness of nonlinear magnetic B-H curves. Imported data can be corrected and exported for use in a numerical model. The methods in the application have been applied to the built-in materials out of which 35

were optimized for increased performance and stability. The list of corrected materials follows.

AC/DC

- Soft Iron (Without Losses), B-H curve and Effective B-H curve
- Soft Iron (With Losses), B-H curve and Effective B-H curve
- Nonlinear Permanent Magnet, B-H curve.

Nonlinear Magnetic

- Silicon Steel NGO 35JN200
- Silicon Steel NGO 35PN210
- Silicon Steel NGO 35PN230
- Silicon Steel NGO 35PN250
- Silicon Steel NGO 50PN1300
- Silicon Steel NGO 50PN600
- Silicon Steel NGO 50PN700
- Silicon Steel NGO 50PN800
- Silicon Steel NGO M-22
- Silicon Steel GO 3%
- Silicon Steel GO 3413
- Silicon Steel GO 3423
- Silicon Steel GO Silectron 4 mil cross
- Silicon Steel GO Silectron 4 mil rolling
- Metglas Nano Finemet 50 Hz NoFieldAnnealed
- Cobalt Steel Vacoflux 50
- Nickel Steel 4750
- Nickel Steel Monimax Nonoriented
- Nickel Steel Mumetal 80% Ni
- Nickel Steel Square 50
- Nickel Steel Superperm 49
- Low Carbon Steel 50H470
- Low Carbon Steel Magnetite
- Low Carbon Steel Soft Iron

- Low Carbon Steel Vacofer S1 Pure Iron
- Alloy Powder Core Hiflux 125 mu
- Alloy Powder Core Hiflux 160mu
- Alloy Powder Core Koolmu 125 mu
- Alloy Powder Core Koolmu 40 mu
- Alloy Powder Core Koolmu 75 mu
- Alloy Powder Core Koolmu 90 mu
- Alloy Powder Core MPP 60 mu

Note that materials added to a model prior to version 5.5 are not affected unless reloaded from the material library.

MAGNETOSTRICTIVE MATERIAL WITH HYSTERESIS

The nonlinear magnetostrictive material model has been extended to include the Jiles–Atherton model for magnetic hysteresis. The model is suitable for investigating the hysteretic loss effects in, for example, power transformers and electric machines. The model parameters are related to microscopic physical effects in magnetic materials, and they can also be estimated based on experimental data. Magnetostrictive modeling is supported by the AC/DC Module together with one of these modules: the Structural Mechanics Module, MEMS Module, or Acoustics Module.

PARAMETRIC HYSTERESIS

The Jiles–Atherton model for magnetic hysteresis has been extended to support parametric stationary studies in addition to the previously available time-dependent analysis. Ferromagnetic hysteresis is for low-to-moderate frequencies rate-independent and can be analyzed using a parametric stationary study, for example, when studying magnetization and demagnetization.

SMALL IMPROVEMENTS

A number of smaller improvements and fixes have been implemented.

- The Effective Nonlinear Magnetic Curves Calculator application will now generate a curve where the saturation region has a differential relative permeability equal to unity.
- The Transition Boundary Condition now allows for adding explicit surface current density contributions on the up and down sides via a subfeature.
- Support for spatially dependent conductivity has been added to the domain Coil feature when operated in single conductor mode.

- The Gauge fixing feature has added support for using a magnetostatic approximation (with Coulomb gauge) in nonconducting domains for Frequency Domain and Time Dependent studies.
- The solver suggestions for nonlinear time-dependent models have been improved (for the Magnetic Fields interface, together with Time Dependent studies).
- The accurate coil voltage calculation is now available for the Time Dependent study and has been made available also for boundary coils (Frequency Domain and Time Dependent studies).
- The method to calculate the coil length for a circular coil has been improved with volumetric averaging for computing the coil length. It is no longer necessary to select a set of edges with the correct average length; only the edge directions matter.
- The Contact Impedance feature has been corrected with respect to frequency domain losses.

UPDATES TO THE AC/DC APPLICATION LIBRARY

In addition to the new B-H Curve Checker application and the updated Effective Nonlinear Magnetic Curves Calculator application covered in previous sections, the following entries have been updated or are entirely new:

- E-Core Transformer
- Electrodynamics of a Power Switch
- Electromagnetic Forces on Parallel Current-Carrying Wires
- Frequency Domain Study of Three-Phase Motor
- Heating Circuit
- Induction Heating of a Steel Billet
- Inductor in an Amplifier Circuit
- Magnet Falling Through Copper Tube
- Magnetic Damping of Vibrating Conducting Solids (New Lorentz Coupling feature)
- Magnetic Signature of a Submarine
- Pacemaker Electrode
- Permanent Magnet
- Permanent Magnet Motor in 3D
- Piezoelectricity in a Layered Shell (New)
- Planar PCB Coil

- Small-Signal Analysis of an Inductor
- Solid Multilayer Shell Comparison
- Topology Optimization of a Magnetic Circuit
- Touchscreen Simulator

Backward Compatibility with Version 5.4 and Earlier

REMANENT FLUX DENSITY MATERIAL PROPERTY GROUP

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

REFACTORING OF MODEL INPUTS

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

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

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

Backward Compatibility with Version 5.2 and Earlier

COIL FEATURES

Multi-Turn Coil features in old models will be mapped to the new **Coil** feature using the Homogenized Multi-Turn Conductor model. **Single-Turn Coil** features in old models will import as is with a warning that the feature is obsolete and will be removed in future versions.

The **User Defined Coil Geometry** subnode, which is available for 3D **Coil** nodes with the User-Defined Coil type, is now a domain feature. You can now select coil input and

output boundaries using the new **Input** and **Output** subnodes. The selection is set up correctly when opening old models.

REMOVED THE OLD INFINITE ELEMENTS FEATURES

The old **Infinite Elements** feature, which are obsolete since version 4.2, have now been discontinued and will be automatically removed when opening old models.

Backward Compatibility with Version 5.0 and Earlier

COIL CURRENT CALCULATION

When models saved in previous versions are opened in version 5.1, the Coil Current Calculation study steps are migrated to Coil Geometry Analysis. Old solutions stored in the model can still be postprocessed, and the solver sequence will be regenerated automatically (with a Stationary solver) the first time the study is solved.

The Coil Geometry Analysis study step (formerly Coil Current Calculation) will now solve for all of the coils in the active interfaces. To solve only for specific coils (specified using the `CoilName` property), set the `SpecifyCoil` property to `1`.

The default solver sequence generated by the Coil Geometry Analysis is different in version 5.1. Code that accesses specific solver features in the generated solver sequence may need to be reviewed.

OTHER COIL IMPROVEMENTS

Harmonic Perturbation subnodes under coil features (**Single-Turn Coil**, **Multi-Turn Coil**) are now global features, so the call to the create method should use the appropriate space dimension (`-1`):

```
model.physics("mf").feature("stcd1").create("hp1",
   "CoilHarmonicPerturbation", -1);
```

Global features have no selections, so code that accesses the selection of the Harmonic Perturbation features may need to be reviewed.

Some of the improvements in the 3D **Multi-Turn Coil** features may require a review of existing code that uses the COMSOL API.

Subnodes required to set up the coil features are now added automatically. Existing code that uses the coil features may need to be updated.

The parameters `eCoil` and `length` have been moved from the **Multi-Turn Coil** features (boundary and domain) to the new subfeature `UserDefinedCoilGeometry`.

OTHER IMPROVEMENTS

New functionality introduced in version 5.1 is disabled by default when opening models created in previous versions:

- New boundary conditions for **Gauge Fixing** features
- Accurate coil voltage calculation

This functionality can be enabled using the appropriate inputs in the Settings window. Refer to the documentation for the individual features for more details.

Acoustics Module

New Functionality in Version 5.5

NEW ELASTIC WAVES, TIME EXPLICIT PHYSICS INTERFACE

The new **Elastic Waves, Time Explicit** physics interface is based on the discontinuous Galerkin time explicit method and enables efficient multicore computations of elastic wave propagation in solids. The interface supports realistic material data such as anisotropy and damping. The interface is suited for modeling ultrasound propagation in solids like transducers and sensors, or in nondestructive testing (NDT) applications. The interface is not limited to ultrasound simulation but applies to any acoustically large system that involves the transient propagation over many wavelengths. This also includes seismic wave propagation in soil and rock.

Acoustic structure interaction (ASI) is modeled by combining the new elastic waves interface with the **Pressure Acoustics, Time Explicit** interface and the **Acoustic-Structure Boundary, Time Explicit** multiphysics coupling. For geometry assemblies, the **Pair Acoustic-Structure Boundary, Time Explicit** can be used.

MULTIPHYSICS FOR ACOUSTIC-STRUCTURE INTERACTION WITH THE TIME EXPLICIT INTERFACES

For modeling large acoustic structure interaction (ASI) problems using the time explicit method, the new **Acoustic-Structure Boundary, Time Explicit** multiphysics coupling has been introduced. The new feature couples the **Pressure Acoustics, Time Explicit** and the new **Elastic Waves, Time Explicit** physics interfaces. The **Acoustic-Structure Interaction, Time Explicit** multiphysics interface adds the two physics and the multiphysics coupling. For taking full advantage of the time explicit formulation, the use of nonconforming meshes is essential when coupling domains with different properties. This is achieved by using the new **Pair Acoustic-Structure Boundary, Time Explicit** multiphysics coupling that handles geometric assemblies. The use of nonconforming meshes is a natural extension and use of the properties of the discontinuous elements.

MATERIAL DISCONTINUITY, PAIR CONDITIONS, AND ATTENUATION FOR THE ACOUSTIC TIME EXPLICIT INTERFACES

The existing interfaces of the Acoustics Module that are based on the discontinuous Galerkin (dG) time explicit method now all have the option to include dissipation.

Dissipation plays an important role when modeling high-frequency applications like ultrasound imaging and flowmeters. The new option exists for the **Pressure Acoustics, Time Explicit** interface and the **Convected Wave Equation, Time Explicit** interface.

In **Pressure Acoustics, Time Explicit**, the **Fluid model** list now includes:

- Linear elastic
- Viscous
- Thermally conducting
- Thermally conducting and viscous
- General dissipation
- Ideal gas

In **Convected Wave Equation, Time Explicit**, the **Fluid model** list now includes:

- Linear elastic
- General dissipation

The **Pressure Acoustics, Time Explicit** interface has been improved with the addition of a **Material Discontinuity** (interior) boundary condition and a **Continuity** pair feature. Both are used to handle jumps in material properties for either a union with a conforming mesh or an assembly using a nonconforming mesh, respectively.

PORTS FOR THERMOVISCOUS ACOUSTICS

A new **Port** boundary condition has been added to the **Thermoviscous Acoustics, Frequency Domain** interface. It is used to excite and absorb acoustic waves that enter or leave waveguide structures in microacoustics applications. The condition is essential to provide the full acoustic description at a waveguide inlet/outlet in a mathematically consistent manner, including the viscous and thermal boundary layers. The port conditions provide a near-perfect, nonreflecting radiation condition for waveguides. In many cases, using the new **Port** condition provides superior ease of use and accuracy compared to an Impedance condition or a perfectly matched layer (PML) configuration. When working with small acoustic subsystems, two **Port** conditions are used and combined to automatically compute the scattering matrix, transfer matrix, and impedance matrix relating the inlet to the outlet. These are all simplified lumped representations of the subsystem typically used for full system analysis.

UPDATES FOR THE PORT FEATURE IN PRESSURE ACOUSTICS

The **Port** condition is now also available in 2D for the **Pressure Acoustics, Frequency Domain** interface. The condition has a **User defined** option and a **Slit** option to define

the mode shapes. In general, when a port sweep is performed and two ports are used, one at the inlet and one at the outlet, the transfer matrix and the impedance matrix of the system is automatically computed. New transmission loss variables are also set up, systematically relating two or more ports together. The port sweep functionality now also works when an inner sweep is performed over the port number.

BACKGROUND FLUID FLOW COUPLING AND MAPPING STUDY FOR AEROACOUSTICS

A new **Background Fluid Flow Coupling** multiphysics coupling and a dedicated **Mapping** study now automate and simplify the coupling of a CFD model and a convected acoustic model, including the linearized Navier-Stokes, linearized Euler, and Convected Wave Equation physics. The multiphysics coupling and mapping ensure that the computed CFD solution is correctly mapped from the fluid flow mesh to the acoustics mesh while also taking care of different discretization orders. The mapping and interpolation is essential in order not to introduce numerical noise into the acoustics model, where the reactive terms are especially important to treat correctly.

ANISOTROPIC MATERIAL FEATURE IN PRESSURE ACOUSTICS

The new **Anisotropic Acoustics** feature in Pressure Acoustics now makes it possible to define fluids with an effective anisotropic density and a scale effective bulk modulus. This makes it possible to set up homogenized material properties for metamaterials or for defining effective fluid properties of porous and fibrous materials that have anisotropic structures. The effective density can be defined as having an **Isotropic**, **Diagonal**, or **Symmetric** structure.

MULTIPHYSICS COUPLING FOR MODELING ELECTROACOUSTIC TRANSDUCERS (LORENTZ COUPLING)

The **Lorentz Coupling** feature supports a two-way coupling between the **Magnetic Fields** physics interface and the **Solid Mechanics** interface. The Lorentz force is determined by computing the cross product of J and B in the volume of the domain. This force is then applied on the mechanics side as a volumetric force. At the same time, the velocity is taken from **Solid Mechanics** and applied in the **Magnetic Fields** interface, as a Lorentz velocity term. The feature automatically handles the frames and moving mesh effects.

The feature is intended for conductive, nonmagnetizable domains (typically, copper coils). Together with the Acoustic-Structure Boundary multiphysics coupling, this allows for modeling electro-acoustic transducers. It is available in 2D, 2D axisymmetry, and 3D for Time Dependent, Frequency Domain (Perturbation), and Eigenfrequency

analysis. The **Lorentz Coupling** feature requires the AC/DC Module but it is highly relevant for modeling dynamic loudspeakers.

ACOUSTIC-PIPE ACOUSTIC CONNECTION MULTIPHYSICS COUPLING

The new **Acoustic-Pipe Acoustic Connection** multiphysics coupling allows coupling a **Pressure Acoustics** interface to the **Pipe Acoustics** interface in both frequency- and time-domain simulations. The coupling is defined between a point in the pipe interface and a boundary in the pressure acoustics interface.

ACOUSTIC-STRUCTURE COUPLINGS FOR LAYERED SHELLS

The multiphysics couplings between acoustics and structures have been extended to support the **Layered Shell** physics interface. This allows modeling of vibroacoustic problems involving composite materials and other layered structures. The Layered Shell interface is now supported for the following multiphysics couplings:

- Acoustic-Structure Boundary
- Thermoviscous Acoustic-Structure Boundary
- Aeroacoustic-Structure Boundary
- Porous-Structure Boundary

NEW ACOUSTOPHORETIC RADIATION FORCE

A new and updated **Acoustophoretic Radiation Force** feature (replacing the Acoustoforetic Force feature) includes formulations of the radiation force that acts on acoustically small solid particles and liquid drops; that is, particles and drops studied in the Rayleigh limit. New formulations of the force expressions can be selected for very small drops and particles that are comparable to or smaller than the viscous and thermal acoustic boundary layers (the thermal and viscous penetration depth). The feature can be combined with both **Pressure Acoustics** and **Thermoviscous Acoustics** to model particle sorting and other acoustofluidic applications. This feature requires the Particle Tracing Module.

IMPROVEMENTS TO ITERATIVE SOLVER SUGGESTIONS IN ACOUSTICS

The default generated solver suggestions for models that include one or more interfaces from the Acoustics Module and multiphysics couplings have been further improved. The handling for multiple acoustics interfaces and multiphysics couplings now includes the following functionality:

- Common iterative solver suggestions are now set up by the Acoustic BEM-FEM Boundary, the Acoustic-Structure Boundary, the Thermoviscous Acoustic-

Structure Boundary, the Acoustic-Thermoviscous Acoustic Boundary, the Aeroacoustic-Structure Boundary, the Piezoelectric Coupling, and the Solid-Shell Connection.

- Lagrange multiplier variables are handled correctly by the Vanka preconditioner when necessary.

Other default and solver suggestion improvements include:

- An iterative solver suggestion for the Compressible Potential Flow interface.
- New Stationary-Frequency and Stationary-Transient solver configuration when coupling Compressible Potential Flow and Linearized Potential Flow in a convected acoustics simulation.
- A second iterative solver suggestion is now added for models that couples Pressure Acoustics to Solid Mechanics with the Acoustic-Structure Boundary multiphysics coupling.
- A better default solver has been added when using the Linearized Euler interfaces.

NEWS SOLVERS FOR LARGE ACOUSTIC PROBLEMS

Two new specialized iterative solver methods have been introduced for frequency-domain simulations modeled with the **Pressure Acoustics, Frequency Domain** interface; that is, large scalar Helmholtz problems (when measured on the number of wavelengths) based on the finite element method. The new solvers can be used to analyze enclosed volumes such as that of a car cabin interior as well as other acoustics simulations. The domain decomposition (DD) method supports the use of absorbing boundary conditions for the domain boundaries. This is important for cluster computation using DD for Helmholtz problems. Secondly, the complex shifted Laplacian technique can be used for both the normal multigrid preconditioner as well as the DD method. The multigrid alternative is the best option for large problems when not using a cluster. With version 5.5, you can solve significantly larger problems in acoustics than before. For example, a car cabin interior acoustics model can now be solved up to 7 kHz (solving 83.5 million DOFs using 105 GB of RAM), whereas it would only converge up to about 3 kHz in earlier versions. This corresponds to solving a one-order-of-magnitude larger problem.

RESTRUCTURE OF THE MODEL WIZARD TREE AND APPLICATION LIBRARY

With the introduction of the new **Elastic Waves, Time Explicit** interface, the physics interface locations in the Model Wizard tree have been updated with two new branches: Elastic Waves and Pipe Acoustics.

To get a better overview of the existing and many new tutorial models, the Application Library has also been updated with following new categories:

- Elastic Waves
- Tutorials, Pressure Acoustics
- Tutorials, Pipe Acoustics
- Tutorials, Thermoviscous Acoustics.

IMPORTANT ENHANCEMENTS IN THE ACOUSTICS MODULE

- In the **Exterior Field Calculation** feature, the infinite symmetry and antisymmetry planes can now be moved (offset) by assigning the location of the symmetry plane.
- The **Octave Band** plots now also have the option to use 1/6 octaves.
- The reference direction can now be set in 1D Radiation Pattern plots of 2D models
- The **Directivity** plot now comes with a true logarithmic axis. The plot has been moved to 1D plot groups.
- **Exclude Edges** and **Exclude Points** option have been added for all constraint-type boundary conditions (Dirichlet conditions) in Thermoviscous Acoustics, Linearized Navier-Stokes, and Linearized Euler interfaces. This option allows to handle over-constrained problems and simplify certain combinations of boundary conditions. This new options is available when the **View Advanced Physics** option is selected.
- The Rayleigh unit, rayl, used for the specific acoustic impedance is available in both the SI unit and the CGS unit version. They are named `[rayl]` and `[rayls_cgs]`, respectively.
- Surface stress variables now exist on both exterior and interior boundaries in the Thermoviscous Acoustics and Linearized Navier-Stokes physics interfaces.
- A new **Absorption Coefficient** option in the Impedance boundary condition in Pressure Acoustics. This new option simplifies the input of certain measured surface impedance data, and it is especially useful in the higher frequency ranges where the impedance phase becomes less important for accurate simulation results.
- The **Characteristic specific impedance** condition in Pressure Acoustics now works for waves propagating at a given angle toward the boundary.

NEWS IN RAY ACOUSTICS

Several enhancements and new features have been introduced in the The Ray Acoustics interface.

- When you release rays from a grid of points using the **Release from Grid** feature, you can now preview the initial ray positions in the Graphics window. In the **Initial Coordinates** section of the Settings window, click the **Preview Initial Coordinates** button to view the initial ray coordinates as a grid of points. Click the **Preview Initial Extents** button to view the spatial extents of the initial coordinates as a bounding box. These buttons allow you to check the initial ray positions before running a study.
- When you release rays in a cone, a new type of **Conical distribution** is available, **Hexapolar**. For the hexapolar cone type, rays are released at uniformly distributed angles from the cone axis, with each ring having six more rays than the previous one.
- You can now select **Isotropic scattering** as the wall condition when rays hit surfaces in the geometry. Like the **Diffuse scattering** condition, the **Isotropic scattering** condition causes rays to be reflected with randomly sampled directions around the surface normal. However, whereas the **Diffuse scattering** condition uses a probability distribution based on the cosine law, the **Isotropic scattering** condition follows a probability distribution that gives equal flux across any differential solid angle in the hemisphere.

New and Updated Model Examples in 5.5

- The *Sound Transmission Loss through a Concrete Wall* model presents a practical and efficient method to compute the sound transmission loss (STL) through a building component.
- The *Headphone on Artificial Ear* model shows how to model the coupling of a circumaural headphone to a generic artificial ear. The model uses the **Poroelastic Waves** physics interface to model the foam. The built-in **Interior Perforated Plate** model is used to represent perforates and meshes in the headphone casing.
- The *OW Microspeaker: Simulation and Correlation to Measurements* model analyzes the electromagnetic, mechanical, and acoustical characteristics of the OWS-1943T-8CP (discontinued) speaker. Beside certain details, the geometry, material properties, and measurements are copyrighted by Ole Wolff. Starting from the geometry of the speaker, an axisymmetric electromagnetic model is used to characterize the frequency-dependent response of the voice coil and

electromagnetic circuit. This response is included in a 3D model where the vibroacoustic response of the speaker is analyzed and compared with measurements.

- The *Wax Guard Acoustics: Transfer Matrix Computation* tutorial is set up to analyze the acoustic properties of a wax guard. A wax guard is a small perforated mesh used to protect the receiver (the miniature loudspeaker in a hearing aid) used for hearing aids. Because of the very small dimensions of the structure, the thermal and viscous boundary layer losses need to be included in detail, and therefore the Thermoviscous Acoustics, Frequency Domain interface is used. The transfer matrix (or two-port) of the wax guard is computed using the **Port Sweep** functionality and the **Port** boundary condition.
- The *Head and Torso HRTF Computation* tutorial model shows how to import a 3D scanned geometry of a human head and torso and compute the head-related transfer function (HRTF). The scan is imported as an STL file and converted into a COMSOL geometry. The HRTF is computed using the reciprocity principle and the BEM method.
- The *Shape Optimization of an Acoustic Demultiplexer* model shows how shape optimization can be used to design an acoustic demultiplexer. A demultiplexer is a data distributing device; in this case, it will distribute acoustic energy.
- In the *Ground Motion After Seismic Event: Scattering off a Small Mountain* tutorial, the propagation of elastic waves in the ground after a seismic event is simulated using a 2D model and the **Elastic Waves, Time Explicit** interface.
- In the *Isotropic-Anisotropic Sample: Elastic Wave Propagation* tutorial, a test sample consists on one side of an isotropic material and on the other side of a heterogeneous anisotropic material (a transverse anisotropic zinc crystal). Elastic waves in the sample are excited by a point-like force. The model is solved with the **Elastic Waves, Time Explicit** physics interface.
- The *Acoustics of a Pipe System with 3D Bend and Junction* tutorial shows how to model the propagation of acoustic waves in large pipe systems by coupling the Pipe Acoustics interface to the Pressure Acoustics interface using the **Acoustic-Pipe Acoustics Coupling** multiphysics feature. The tutorial is set up in both the time and frequency domain.
- The *Spherical Scatterer: BEM Benchmark* tutorial is a classical BEM benchmark model. A spherical scatterer is placed in a plane wave background field. When the sphere is modeled as sound hard, the problem has an analytical solution. The model compares the results using the **Pressure Acoustics, Boundary Elements** interface with the analytical solution for several frequencies. The results show very good agreement. The model results do not show any irregular modes.

- The *Acoustic Cloaking* model has been updated to use the new **Anisotropic Acoustics** feature to model the metamaterial used to cloak the sphere.
- The *Optimizing the Shape of a Horn* model has been updated to use the new built-in shape optimization functionality.

Backward Compatibility with Version 5.4

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

Backward Compatibility with Version 5.3a

For the Linearized Navier-Stokes interfaces, the override rules for boundary conditions have changed. To obtain the old behavior of a model, disable the boundary conditions that should not contribute to the solution.

Backward Compatibility with Version 5.3

For the Linearized Euler, Transient interface, initial conditions are only used for the dependent variables. The initial conditions for the time derivatives have therefore been removed. For a model with fields rho, u, and p, remove the Java® lines setting drho/dt, du/dt, and dp/dt (these parameters change names when the field names change).

Backward Compatibility with Version 5.2

The default settings have changed for the Incident Pressure Field in the Pressure Acoustics interfaces. Add the following line to obtain the old behavior of this feature when `ipf1` is a subfeature to pwr1:

```
model.physics("acpr").feature("pwr1").feature("ipf1").
set("c", "acpr.c_c");
```

Backward Compatibility with Version 5.1

- Plane Wave Radiation has been removed from the Linearized Potential Flow, Frequency Domain and Linearized Potential Flow, Transient interfaces in 1D axisymmetry.

- A new Lagrange multiplier variable (`ta.lm_slip`) has been added to the slip conditions in the Thermoacoustics, Frequency Domain interface. If you run a model with a segregated solver, you will need to add this variable to the group containing the velocity degree of freedom. You can also regenerate the default solver. In the API, you need to add `comp1_ta_lm_slip` to the declaration of the segregated step; for example, adding:

  ```
  model.sol("sol1").feature("s1").feature("se1").feature("ss1").
  set("segvar", new String[]{"comp1_p", "comp1_u", "comp1_T",
  "comp1_ta_lm_slip"});
  ```

- The Waveguide end impedance option has been removed from 2D, 1D axisymmetric, and 1D as it had no physical meaning in these space dimensions.

Batteries & Fuel Cells Module

New Functionality in Version 5.5

- A new charge balance model, “Single-ion conductor”, has been added to the Lithium-Ion Battery interface. A new Model Wizard entry, “Lithium-Ion Battery, Single-Ion Conductor”, which uses the Single-Ion Conductor charge balance model by default, has been added to the Model Wizard.
- New Thin Electrolyte Layer node available in the Lithium-Ion Battery interface. The node is available when the Single-Ion conductor charge balance model has been selected.
- Added support for including convective contribution, with a specified velocity, to the electrolyte concentration mass balance equation in the Lithium-Ion Battery and Battery with Binary Electrolyte interfaces.
- An overhaul has been done for the lithium insertion and lithium metal kinetics in the Lithium-Ion Battery and Single Particle Battery interfaces.
- In the **Particle Intercalation** node, the **Number of elements** setting under P**article Discretization** now supports input from Parameters.
- The Batteries & Fuel Cells material library has been updated with individual literature references for each material property.
- The Lumped Battery interface now includes three new operation modes: Potentiostatic, Power, and Circuit Voltage Source.
- The Lumped Battery interface now includes a Short Circuit node.
- New **User defined** and **From definitions** options are now available in the **Cell Equilibrium Potential** node in the Lumped Battery interface and Battery Equivalent Circuit Model Wizard entry (the **Battery Open Circuit Voltage** node in the Electric Circuit interface) for defining the equilibrium potential and the temperature derivative of the potential.
- An RC circuit element has been added to the Electrical Circuit interface. The RC element has also been added to the Battery Equivalent Circuit Model Wizard entry.
- The **Thermodynamics** node has been added to the Batteries & Fuel Cells Module license.
- The **Mass flow** boundary condition option in the Inlet feature in the Single Phase Flow interface has been added to the Batteries & Fuel Cells Module license.

- A new **Mass flow rates** option has been added to the **Inflow** node of the Transport of Concentrated Species interface.
- Partial pressure variables have been added to the Transport of Concentrated Species interface, when **Ideal gas** is selected as the mixture density. This facilitates setting up fuel cell models, especially when using the Nernst Equation option for the equilibrium potential.
- A new **Nernst Equation** option is now available for defining the equilibrium potential in the **Electrode Reactions** and **Porous Electrode Reaction** nodes. In the Tertiary Current Distribution, Nernst-Planck interface, the concentration dependence of the equilibrium potential for the Nernst Equation option is defined automatically based on the reaction stoichiometry.
- The exchange current density for the **Butler-Volmer** and **Linearized Butler-Volmer** kinetics options in the **Electrode Reaction** and **Porous Electrode Reaction** nodes can now be made automatically concentration dependent. The option is available if the **Nernst-Equation** option is used when defining the equilibrium potential. In the Tertiary Current Distribution, Nernst-Planck interface, the options **Mass action law** and **Lumped multistep** are available for the exchange current density type.
- The Tertiary Current Distribution, Nernst-Planck interfaces have changed default settings to use the **Nernst equation** option for the equilibrium potential, **Butler-Volmer** for the kinetics expression type, and **Mass action law** for the exchange current density type.
- The Electroanalysis (elan) interface has been replaced with the new Electroanalysis (tcd) entry in the Model Wizard. The new Electroanalysis (tcd) interface provides seamless switching between the various charge conservation model options supported by the Tertiary Current Distribution, Nernst-Planck interface.
- The Chemistry interface, added to the Electrochemistry Module in version 5.5, now supports Electrode Reactions and Electrode Reaction Groups. The Chemistry interface allows for defining multiple species and electrode reactions, as well as ordinary chemical reactions. Also, mixture and thermodynamic properties, such as equilibrium potentials, can be calculated automatically by the Chemistry interface. Variables defined by these features can then be used in any physics interface.
- The Non-Faradaic Reactions nodes now also allow to set reaction rates for the bulk species in the electrolyte.
- The Common model input has been set as the new default for Temperature model input in all electrochemistry interfaces.

- The Cyclic Voltammetry boundary condition at the Electrode Surface node and Cyclic Voltammetry study step is now available for all Electrochemistry interfaces.
- The Thin Electrode Surface node is now also available in the Primary Current Distribution interface.
- New and updated default plots have been added in the Transport of Diluted Species, Tertiary Current Distribution, Nernst Planck, and Darcy's Law interfaces.
- A new mass inflow boundary condition has been added to Darcy's Law.
- The extended Inlet/Outlet boundary condition in the Darcy's Law interface now includes Velocity (Inlet/Outlet), Pressure (Inlet/Outlet) and Mass flow (Inlet) options.
- The Darcy's Law interface now supports Frequency Domain studies (with licenses for the Batteries & Fuel Cells Module, Corrosion Module, Electrochemistry Module, and Electrodeposition Module).
- More accurate definitions of the internal cH and pH variables (in basic electrolytes) and the cOH and pOH variables (in acid electrolytes) in the Tertiary Current Distribution, Nernst-Planck (water-based charge balance option) and Electrophoretic Transport interfaces have been implemented.
- The implementation of the Donnan boundary conditions used by the **Ion-Exchange Membrane** node has been improved.
- A new boundary condition formulation option has been added in the **Film Resistance** section of the **Electrode Surface**, **Internal Electrode Surface**, and **Thin Electrode Surface** nodes in the Primary Current Distribution interface.
- The default stoichiometric number for Dissolving-Depositing species has been changed to 0 (from 1).

New and Updated Applications and Models in Version 5.5

- The *Parameter Estimation of a Time-Dependent Lumped Battery Model* has been updated to include a third study for load cycle prediction.
- A new *Lithium-Ion Battery with Single-Ion Conducting Solid Electrolyte* has been added, which demonstrates the Lithium-Ion Battery, Single-Ion Conductor interface.
- A new *Solid Oxide Electrolyzer* model has been added, which demonstrates some of the new functionalities in the Thermodynamics and Chemistry nodes.

- A new *Aging Analysis of a Lumped Battery Model* has been added, which demonstrates the Capacity Loss feature and some of the new operation modes in the Lumped Battery interface.
- A new *Thermal Distribution in a Pack of Cylindrical Batteries* has been added, demonstrating thermal management modeling using the Lumped Battery interface.
- The following models have been updated to use the new Electroanalysis (tcd) interface: impedance_spectroscopy, microdisk_voltammetry, and cyclic_voltammetry_1d.
- The impedance_spectroscopy, microdisk_voltammetry, cyclic_voltammetry_1d, ht_pem, pem_mea_1d, fuel_cell_cathode, sofc_unit_cell, orange_battery, v_flow_battery, pb_flow_battery, and zn_ago_battery_1d models have been updated to use the new Nernst equation option and concentration-dependent exchange current density in Butler-Volmer kinetics.

Backward Compatibility with Version 5.4

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

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

New Functionality in Version 5.5

CAD IMPORT MODULE GEOMETRY KERNEL UPGRADE

The CAD Import Module, the Design Module, and the LiveLink™ products for CAD utilize the Parasolid® geometry kernel from Siemens PLM for solid modeling operations, geometry repair, and defeaturing. (Without these products, a COMSOL-native geometry modeling kernel is used.) The CAD Import Module released with COMSOL 5.5 includes an upgraded version of the Parasolid kernel. As a result, a number of stability issues have been fixed, which makes the import of CAD models and solid operations more robust.

- The CAD file import functionality included with these products has been extended to support new versions for some of the supported file formats (see under Read from File, CAD on www.comsol.com/products/specifications/cad).
- The CAD file import functionality now supports associative geometry import to retain physics and other settings on the geometry after the file is imported again. The import reads information in the CAD files to identify the geometric entities in the file. This information is usually available when importing CAD files saved in the native format of the software where it was created.
- The CAD file import functionality now generates selections based on the material and layer assignments of the geometric objects when this is available in the CAD file. Selections are also generated for assigned colors to geometric entities. The selections are available in the appropriate input selection lists of features in the model tree.
- You can use the new Delete Holes defeaturing tool to search for and delete cylindrical holes smaller then a specified radius.
- Support has been added to export geometry to the STEP and IGES formats.

New Functionality in the Design Module in Version 5.5

The Design Module adds support for geometric constraints and dimensions in 2D geometry sequences, including in geometry parts and work planes in 3D.

New Functionality in LiveLink™ for AutoCAD® in Version 5.5

- The LiveLink interface now keeps track of the layer state of the synchronized CAD file, and the same layer state is used for subsequent synchronizations.
- Layer selections, which are generated based on the layer name that the geometric objects belong to in the CAD file, are now also available as object selections in the appropriate input selection lists of geometry features following the LiveLink™ feature in a geometry sequence.
- The LiveLink™ interface now supports AutoCAD® 2020.

New Functionality in LiveLink™ for Inventor® in Version 5.5

- The LiveLink interface now synchronizes the material properties for the parts in the CAD files. The Material nodes are created in the component of the LiveLink feature in the model, and their selection is set to the selection corresponding to the material.
- The LiveLink™ interface now supports Inventor® 2020.

New Functionality in LiveLink™ for PTC® Creo® Parametric™ in Version 5.5

- The LiveLink interface now synchronizes the material properties for the parts in the CAD files. Based on the material data, Material nodes are created in the component of the LiveLink feature, and their selection is set to the selection corresponding to the material.
- The LiveLink™ interface now supports PTC® Creo® Parametric™ 6.0.
- Support for PTC® Creo® Parametric™ 2.0 is no longer available by the LiveLink™ interface. File import of PTC® Creo® Parametric™ 2.0 continues to be supported.
- Support for PTC® Creo® Parametric™ 3.0 by the LiveLink™ interface will be removed in the next major release of COMSOL Multiphysics®. File import of PTC® Creo® Parametric™ 3.0 will continue to be supported.

New Functionality in LiveLink™ for Revit® in Version 5.5

- The LiveLink interface now synchronizes the material properties for elements in the Revit® project. Based on the material data, Material nodes are created in the

component of the LiveLink feature, and their selection is set to the selection corresponding to the material.

- The LiveLink™ interface now supports Revit® 2020.

LiveLink™ for Solid Edge®

NEW FUNCTIONALITY IN VERSION 5.5

- The LiveLink interface now synchronizes the material properties for the parts in the CAD files. Based on the material data, Material nodes are created in the component of the LiveLink feature, and their selection is set to the selection corresponding to the material.
- Expanding on the functionality that synchronizes selections based on material assignments to the CAD design in Solid Edge®, the LiveLink™ interface now adds support for user-defined selections. In the added COMSOL Selections interface in Solid Edge®, you can define selections that are synchronized to the COMSOL Multiphysics® model. You can choose to synchronize selections for assembly features, bodies, faces, or points, which become selections in the model when the design is synchronized with the COMSOL Multiphysics® model.
- The LiveLink™ interface now supports Solid Edge® 2020.

BACKWARD COMPATIBILITY WITH VERSION 5.0

The default value for the `keepfree` property of the `LiveLinkSolidEdge` function is now set to `on`. Previously, the default was set to `off`.

LiveLink™ for SOLIDWORKS®

NEW FUNCTIONALITY IN VERSION 5.5

- The LiveLink interface now synchronizes the material properties for the parts in the CAD files. Based on the material data, Material nodes are created in the component of the LiveLink feature, and their selection is set to the selection corresponding to the material.

BACKWARD COMPATIBILITY WITH VERSION 5.3A

The new `fillholes` property has been added for the `Import` function. The following applies:

- The combination of the `fillholes` property set to `on` and the `knit` property set to `solid` is equivalent to the `knit` property set to `solid` in previous versions.
- The combination of the `fillholes` property set to `off` and the `knit` property set to `surface` is equivalent to the `knit` property set to `surface` in version 5.3a.
- The combination of the `fillholes` property set to `on` and the `knit` property set to `surface` is equivalent to the `knit` property set to `surface` in version 5.3 and previous versions.

The new `fillholes` property has been added for the `Knit` function. The `Knit` function with the `fillholes` property set to `on` corresponds to the `Knit` function in previous versions.

BACKWARD COMPATIBILITY WITH VERSION 5.2

Geometry Repair Tolerances

The geometry repair tolerance can be automatic, relative, or absolute in version 5.2a. In previous versions, the repair tolerance was always a relative tolerance.

- If you have not set the relative repair tolerance `repairtol`, you will get the new default repair tolerance type: `auto`. This will result in a valid geometry in almost all cases. In extreme cases, the result can have a different topology than in 5.2.
- In rare cases, if you have set the `repairtol` tolerance, version 5.2a can fail when building the geometry. If so, set `repairtoltype` to `auto` (select **Automatic** from the **Repair tolerance** list) to get the 5.2 behavior.

BACKWARD COMPATIBILITY WITH VERSION 5.0

The default value for the `keepfree` property of the `LiveLinkSOLIDWORKS` function is now set to `on`. Previously, the default was set to `off`.

CFD Module

New Functionality in Version 5.5

- The user interface for the **Fluid Properties** node in the Laminar Flow and Creeping Flow interfaces has been redesigned. It now contains a new section, **Constitutive relations**, with two options, **Newtonian** and **Inelastic non-Newtonian**. The first option contains settings for the dynamic viscosity, whereas the second options contains the predefined inelastic non-Newtonian models.
- A warning is given by the solver when a model lacks a **Pressure Point Constraint** node and the pressure level is not set in any other boundary condition node.
- Better control over constraints in the Wall boundary feature. Whereas in previous versions, it was possible to override the default constraints by weak constraints, it is now also possible to choose between pointwise, discontinuous Galerkin, and mixed constraints.
- The new physics interface, Viscoelastic Flow, for incompressible, laminar flow of fluids exhibiting both viscous and elastic behavior has been added under Single-Phase Flow in the Fluid Flow branch. The predefined constitutive models include Oldroyd-B, Giesekus, and FENE-P.
- New functionality in the Bubbly Flow interfaces includes a new Flow Continuity pair feature, a new Periodic Flow pair feature, and application of the Boussinesq approximation to model buoyancy effects in the low-gas-concentration limit.
- New functionality in the Euler-Euler Model interfaces includes an improved Wall boundary feature, which picks up the frame motion by default; a new Interior Wall feature; a new Flow Continuity pair feature; and a new Mass Transfer domain feature.
- The laminar Two- and Three-Phase Flow interfaces now support all inelastic non-Newtonian constitutive relations.
- The Two-Phase Flow, Phase Field interface now supports Marangoni effects. The Phase Field interface also has an improved Continuity pair feature for nonmatching meshes.
- A new physics area, Phase Transport Mixture Model, containing physics interfaces for simulating laminar and turbulent n-phase flow, has been added under Phase Transport in Multiphase Flow. All RANS turbulence models are supported. The physical properties of the various phases, and the slip velocity between the dispersed

phases and the continuous phase, can be specified in the Mixture Model multiphysics coupling node.

- In the Brinkman Equations and Free and Porous Media Flow interfaces, the Porous Matrix Properties section in the Fluid and Matrix Properties domain feature now contains a **Non-Darcian** option in the drop-down list for the **Permeability model**. This option also replaces the old Forchheimer Drag subfeature.
- In the Darcy's Law interface, the Matrix Properties section in the Fluid and Matrix Properties domain feature now contains a **Non-Darcian** option in the drop-down list for the **Permeability model**. This can be used to add a Forchheimer drag term to the momentum equation. The Darcy's Law interface also has a new setting, at the interface level, for including gravity effects.
- Large Eddy Simulation (LES) interfaces for nonisothermal flow are now available in the Model Wizard. The three new interfaces, LES RBVM, LES RBVMWV, and LES Smagorinsky, are all based on variational multiscale methods. Additional subgrid heat fluxes are added in the Nonisothermal Flow multiphysics coupling node in consistency with the chosen variational multiscale method (RBVM, RBVMWV, or Smagorinsky).
- The new physics interface, Compressible Euler Equations, has been added under High Mach Number Flow in the Fluid Flow branch. It is applicable to transient, isentropic flow of ideal gases at transonic and supersonic speeds.

New Model in Version 5.5.

The model *Unsteady 3D Flow Past a Cylinder*, under Verification Examples in the Application Library, compares the results for lift and drag coefficients with those found in literature. *Oil-Water Flow Through an Orifice - A Droplet Population Model,* under Multiphase Flow in the Application Library, considers turbulent flow and break-up of five populations (size containers) of droplet size. Two new fluid-structure interaction models are available with the Structural Mechanics Module: *Two-Phase Flow with Fluid-Structure Interaction* models the deformation of a thin plate by a lock-release gravity current, and *Bimetallic Strip in Air Flow* models the deformation of a bimetallic strip by thermal stresses, dynamic pressure, and convective cooling.

Backward Compatibility with Version 5.4 and Earlier

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

Old Java files need to replace the line

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

with

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

Forchheimer is the default. Also replace

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

with

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

Backward Compatibility with Version 5.3a

MIXTURE MODEL EQUATION FORMULATION

The Mixture Model equation formulation has been revised to increase the accuracy and facilitate the solution of a number of problem setups. As a consequence of the revision, old models can return slightly different results in the new version compared to previous versions.

EULER-EULER MODEL

The results from models using the Euler-Euler Model interfaces may differ in version 5.5 due to the more general formulation for the viscous stress in the momentum equation for the dispersed phase. The behavior in the old formulation may be reproduced by dividing the dispersed phase viscosity, μ_D, by the variable `ee.phidPos`.

NO FLUX FORMULATION

The formulation of the No Flux feature has been changed in the Transport of Diluted Species interface in version 5.4 so that the flux relative to the convective flux is set to zero. The new formulation is used when opening a model from a previous session. In the previous formulation, the total flux was set to zero.

FULLY DEVELOPED FLOW REPLACES LAMINAR INFLOW AND LAMINAR OUTFLOW

The **Laminar inflow** and **Laminar outflow** options on Inlets and Outlets, respectively, have been replaced with the **Fully developed flow** option. Models built in versions prior to 5.4 will contain warnings about obsolete functionality.

Backward Compatibility with Version 5.0

TRANSPORT OF CONCENTRATED SPECIES

See Chemical Reaction Engineering Module.

Chemical Reaction Engineering Module

New Functionality in Version 5.5

- A new Generate Material wizard is available for Thermodynamic System nodes.
- A new Generate Chemistry wizard is available for Thermodynamic System nodes.
- A new thermodynamic model, Water (IAPWS), for steam and water properties as defined by the International Association for Properties of Water and Steam has been added.
- New diffusivity models are available in Thermodynamics. Two new properties, Diffusion coefficient at infinite dilution and Maxwell-Stefan diffusivity, are available both for gas-phase and liquid-phase systems.
- New equilibrium calculations available in Thermodynamics. Equilibrium calculations at a specified density, or a specified volume, can now be created.
- A new single-phase property, the Helmholtz energy, is available in the Thermodynamic System nodes.
- Two new viscosity models for liquids, molar cubic mixing and mass cubic mixing, are available in the Thermodynamic System nodes.
- Updated default plots for all mass transport interfaces. Plots for all included species are now generated.
- Extended multiphase support in the Chemistry interface. Surface species and solid-phase species are now handled in separate lists in the Species Matching section.
- A new setting to include the convection, is available in the No Flux, Flux, and Symmetry boundary nodes for mass transfer.
  - The setting can be used to include the convective contribution in the prescribed flux.
  - Affected interfaces: Transport of Diluted Species, Transport of Diluted Species in Porous Media, Transport of Concentrated Species (only in the Flux node), Nernst-Planck Equations, and Electrophoretic Transport.
- New adsorption models for mass transport in porous media. The Toth and BET models are now available in the Adsorption node.

- Mass flow rates can now be prescribed, per species, in the Inflow feature in the Transport of Concentrated Species interface.
- The mass flow rate can now be prescribed in the Inflow feature in the Darcy's Law interface.

Backward Compatibility with Version 5.4 and Earlier

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

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

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

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

WALL FUNCTION VARIABLES OMITTED IN LAMINAR REACTING FLOW

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

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

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

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

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

Backward Compatibility with Version 5.3a and Earlier

NO FLUX FORMULATION

The formulation of the No Flux feature has been changed in the Transport of Diluted Species interface in version 5.4 so that the flux relative to the convective flux is set to zero. The new formulation is used when opening a model from a previous session. In the previous formulation, the total flux was set to zero.

Affected interfaces: Transport of Diluted Species, Transport of Diluted Species in Porous Media, Transport of Concentrated Species, Nernst-Planck Equations, and Electrophoretic Transport.

Backward Compatibility with Version 5.0 and Earlier

REACTION ENGINEERING AND CHEMISTRY INTERFACES

The API syntax for creating surface species has changed. Running Java® files from previous versions may fail due to the new default name indexes, including **ads** for surface species and **s** for solids in the Reaction Engineering and Chemistry interfaces. The surface species names and solid species names need to be changed accordingly in any Java® code:

- Change any species index `(s)` to `(ads)`
- Change any species index `(S)` to `(s)`

The following shows an example of the needed code change:

Version 5.0 and earlier:

```
model.physics("re").feature("rch1").set("formula", "A+B(S)=>C+D(s)");
model.physics("re").feature("spec1").set("specName", "As(s)");
model.physics("chem").feature("rch1").set("formula", "A+B(S)=>C+D(s)");
model.physics("chem").feature("spec1").set("specName", "As(s)");
```

Change this to

Version 5.2 and later:

```
model.physics("re").feature("rch1").set("formula", "A+B(s)=>C+D(ads)");
model.physics("re").feature("spec1").set("specName", "As(ads)");
model.physics("chem").feature("rch1").set("formula", "A+B(s)=>C+D(ads)");
model.physics("chem").feature("spec1").set("specName", "As(ads)");
```

TRANSPORT OF CONCENTRATED SPECIES INTERFACE

Updated Internal Variable Names

The internal variable names of vector and tensor components have changed in the *Transport of Concentrated Species* interface. In most cases, users will not be affected by this change. When opening a model file created in a previous version, the previous variable definitions remain. However, when updating or recomputing the solution, new variable names will be generated. This means that if any of these variable names have been used in a user-defined expression or plot, they need to be updated manually as described below.

In version 5.1, the spatial direction indicator string (`x`, `xy`, and so on) has been moved to the end of the variable name. This applies to all vector and tensor variables generated by the interface.

For example, the diffusive flux vector components have changed (in 5.0 and earlier) from

```
tcs.dfluxx_w1, tcs.dfluxy_w1, tcs.dfluxz_w1
```

to (in 5.2 and later)

```
tcs.dflux_w1x, tcs.dflux_w1y, tcs.dflux_w1z
```

and the components of the diffusion tensor, defined by the Fick's Law diffusion model, have been changed (in 5.0 and earlier) from

```
tcs.Dfxx_w1, tcs.Dfxy_w1, tcs.Dfxz_w1, tcs.Dfyx_w1, ...
```

to (in 5.2 and later)

```
tcs.Df_w1xx, tcs.Df_w1xy, tcs.Df_w1xz, tcs.Df_w1yx, ...
```

Java® API Syntax Changes

The API syntax for setting physics properties has been changed in the Transport of Concentrated Species interface. Now the property name corresponds to the name of the GUI section containing the property. In order to run Java® files from previous versions, apply the syntax changes as given in the examples below:

Version 5.0 and earlier:

```
model.physics("tcs").prop("DiffusionModel").setIndex("DiffusionModel",
"MaxwellStefan", 0);
model.physics("tcs").prop("DiffusionModel").setIndex("DiffusionModel",
"MixtureAveraged", 0);
model.physics("tcs").prop("DiffusionModel").setIndex("DiffusionModel",
"FicksLaw", 0);
model.physics("tcs").prop("Convection").setIndex("Convection", "0", 0);
```

```
model.physics("tcs").prop("Migration").setIndex("Migration", "0", 0);
model.physics("tcs").prop("Regularization").setIndex("Regularization", "1",
0);
```

For version 5.2 and later, change this to:

```
model.physics("tcs").prop("TransportMechanism").set("DiffusionModel",
"MaxwellStefan");
model.physics("tcs").prop("TransportMechanism").set("DiffusionModel",
"MixtureAveraged");
model.physics("tcs").prop("TransportMechanism").set("DiffusionModel",
"FicksLaw");
model.physics("tcs").prop("TransportMechanism").set("Convection", false);
model.physics("tcs").prop("TransportMechanism").set("Migration", false);
model.physics("tcs").prop("AdvancedSettings").set("Regularization", "0");
```

SURFACE REACTIONS INTERFACE

Updated Internal Variable Names

The same pattern for vector and tensor variable name changes, as described in the Transport of Concentrated Species Interface section above, holds true for the Surface Reactions interface in version 5.4.

Java® API Syntax Changes

The API syntax for setting the diffusion tensor of surface species, the molar mass and density of the bulk species, and initial values have been changed in the Surface Reactions interface.

Setting Diffusion Tensor

The surface species name needs to be included in the first argument of the set method. In order to run Model Java® files from previous versions, apply the syntax changes as shown in the example below:

Version 5.0 and earlier:

```
model.physics("sr").feature("sp1").set("D_0", new String[]{"Dcs1", "0", "0",
"0", "Dcs1", "0", "0", "0", "Dcs1"});
model.physics("sr").feature("sp1").set("D_1", new String[]{"Dcs2", "0", "0",
"0", "Dcs2", "0", "0", "0", "Dcs2"});
```

For version 5.2 and later, change this to:

```
model.physics("sr").feature("sp1").set("D_cs1", new String[]{"Dcs1", "0",
"0", "0", "Dcs1", "0", "0", "0", "Dcs1"});
model.physics("sr").feature("sp1").set("D_cs2", new String[]{"Dcs2", "0",
"0", "0", "Dcs2", "0", "0", "0", "Dcs2"});
```

Setting Molar Mass and Density

The bulk species name needs to be included in the first argument of the `setIndex` method. Apply the Java® API syntax changes as shown in the example below:

Version 5.0 and earlier:

```
model.physics("sr").feature("sp1").set("M_0", "Mcb1");
model.physics("sr").feature("sp1").set("M_1", "Mcb2");

model.physics("sr").feature("sp1").set("rho_0", "rhocb1");
model.physics("sr").feature("sp1").set("rho_1", "rhocb2");
```

For version 5.2 and later, change this to:

```
model.physics("sr").feature("sp1").setIndex("M_cb1", "Mcb1", 0);
model.physics("sr").feature("sp1").setIndex("M_cb2", "Mcb2", 0);

model.physics("sr").feature("sp1").setIndex("rho_cb1", "rhocb1", 0);
model.physics("sr").feature("sp1").setIndex("rho_cb2", "rhocb2", 0);
```

Setting Initial Values

Apply the Java® API syntax changes as shown in the example below:

Version 5.0 and earlier:

```
model.physics("sr").feature("init1").set("cs1", "cs10");
model.physics("sr").feature("init1").set("cs2", "cs20");
model.physics("sr").feature("init1").set("cb1", "cb10");
model.physics("sr").feature("init1").set("cb2", "cb20");
```

For version 5.2 and later, change this to:

```
model.physics("sr").feature("init1").setIndex("initcs", "cs10", 0);
model.physics("sr").feature("init1").setIndex("initcs", "cs20", 1);
model.physics("sr").feature("init1").setIndex("initcb", "cb10", 0);
model.physics("sr").feature("init1").setIndex("initcb", "cb20", 1);
```

Composite Materials Module

New Functionality in Version 5.5

DELAMINATION MODELING

A common failure mode in laminates is delamination. This can be modeled using the new **Delamination** node in the Layered Shell interface. There are several different displacement- and energy-based cohesive zone models available for describing the damage together with different traction separation laws. When two layers are in a delaminated state, either initially or after applying a load, a contact condition is used to avoid penetration between the layers.

NEW FAILURE CRITERIA

New failure criteria have been added for layered composites. They are:

- Zinoviev
- Hashin-Rotem
- Hashin
- Puck
- LaRC03

These criteria are available for the **Linear Elastic Material** in the Layered Shell interface and in the **Layered Linear Elastic Material** in the Shell interface.

MATERIAL ACTIVATION IN LAYERED SHELL INTERFACE

The addition of an **Activation** node to the **Linear Elastic Material** in the Layered Shell interface makes it possible to analyze the stress state in a composite laminate where certain layers are added or removed. An example is when you want to model the addition of material during processes such as additive manufacturing.

NEW MULTIPHYSICS INTERFACE FOR THERMAL STRESS ANALYSIS

A new multiphysics interface for thermal stress analysis, **Thermal Stress, Layered Shell** has been added. It couples the Heat Transfer in Shells interface with a Layered Shell interface. The connection is made using the **Layered Thermal Expansion** multiphysics coupling. In the previous version, the name of the corresponding coupling was **Thermal Expansion, Layered Shell**.

The **Layered Thermal Expansion** multiphysics coupling can also connect to a **Thin Layer** feature in the Heat Transfer in Solids interface.

The new interface is located in a group named **Thermal Stress** in the **Add Physics** dialog.

NEW MULTIPHYSICS COUPLINGS IN LAYERED SHELL INTERFACE

New multiphysics couplings have been enabled for the Layered Shell interface to provide couplings with fluid load and acoustics. The new multiphysics couplings for this interface are:

- **Acoustics-Structure Boundary**
- **Thermoviscous-Structure Boundary**
- **Aeroacoustic-Structure Boundary**
- **Porous-Structure Boundary**
- **Fluid-Structure Interaction**

COUPLINGS TO OTHER STRUCTURAL MECHANICS INTERFACES

Two new multiphysics couplings have been added for connecting the Layered Shell interface to other structural mechanics interfaces.

The **Layered Shell-Structure Cladding** multiphysics coupling can be used to model cases where the top or bottom surface of the layered shell is connected to a boundary in the Solid Mechanics, Shell, or Membrane interfaces.

The **Layered Shell-Structure Transition** multiphysics coupling makes it possible to connect edges in a Layered Shell interface either to edges in a Shell interface or to boundaries in a Solid Mechanics interface.

NEW LAYERED LINEAR ELASTIC MATERIAL IN THE MEMBRANE INTERFACE

A new material model, the **Layered Linear Elastic Material**, has been added to the Membrane interface to facilitate the modeling of multilayered thin films. It is available in 3D and 2D axisymmetry.

There are several material models that can be used together with the **Layered Linear Elastic Material**; for example, viscoelaticity and plasticity. The nonlinear material models require the Nonlinear Structural Materials Module.

For structural analysis in the Membrane interface, the ordering of the layers is not important, since this is an equivalent single layer theory and the bending stiffness is

assumed to be zero. The ordering may still be important in a multiphysics context, though.

LAYERED LINEAR ELASTIC MATERIAL IN 2D AXISYMMETRY

The **Layered Linear Elastic Material** in the Shell interface is now also available in 2D axisymmetry.

NONLINEAR MATERIAL MODELS IN THE LAYERED SHELL INTERFACE

Hyperelastic Material has been added as a new top-level material family in the Layered Shell interface. It contains the same hyperelastic material models (Neo-Hookean, Mooney-Rivlin, etc.) as in the Solid Mechanics interface. It is also possible to include viscoelasticity and Mullins effect.

In the **Linear Elastic Material** in the Layered Shell interface, it is now possible to add **Plasticity**. The same set of yield functions and hardening models as in the Solid Mechanics interface is available. The formulation assumes small plastic strains, which give a good approximation up to 10–20% strain.

Nonlinear materials require the Nonlinear Structural Materials Module.

NONLINEAR MATERIAL MODELS IN THE SHELL INTERFACE

The **Layered Linear Elastic Material** in the Shell interface now has a set of subnodes for defining new material models. These are:

- **Viscoelasticity**
- **Plasticity**
- **Creep**
- **Viscoplasticity**

Nonlinear materials require the Nonlinear Structural Materials Module.

PIEZOELECTRIC MATERIAL IN LAYERED SHELL INTERFACE

The addition of a piezoelectric material model to the Layered Shell interface makes it possible to model thin piezoelectric devices where a piezoelectric material is embedded in a composite laminate. A new multiphysics interface, Piezoelectricity, Layered Shell, has been added. It combines two physics interfaces, Layered Shell and Electric Currents in Layered Shells, and a Layered Piezoelectric Effect multiphysics coupling.

MIXED FORMULATION IN LAYERED SHELL INTERFACE

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

CONTACT MODELING IN LAYERED SHELL INTERFACE

It is now possible to model contact in the Layered Shell interface. In this release, it is however not possible to extend the contact model with friction, adhesion, or decohesion.

VARIABLE THICKNESS LAYERS IN LAYERED SHELLS

It is now possible to model one or more layers of a composite laminate with a thickness that is a function of the coordinates. This new functionality is available through the new **Scale** setting in the **Layered Material Link** and **Layered Material Stack** nodes. In a single-layer material, a thickness expression can be given directly in the **Thickness** property.

Variable thickness layers are supported in the Layered Shell interface as well as in the **Layered Linear Elastic Material** node in the Shell and Membrane interfaces.

REPETITION OF LAYUP DEFINITIONS

It is now possible to perform various transformation operations on the layup defined in **Layered Material** in order to avoid defining all layers in structures having symmetries or being repetitive. The transformations are:

- **Symmetric**
- **Antisymmetric**
- **Repeated**

These transformations are defined by the **Transform** option in the **Layered Material Link** and **Layered Material Stack** nodes.

SPECIALIZED INTERFACE SELECTIONS

In nodes in the Layered Shell interface that have interface selection, such as T**hin Elastic Layer**, a number of special layer selection shortcuts have been added:

- **Top interface**
- **Bottom interface**
- **Exterior interfaces**
- **All interfaces**

DISPLAY OF NORMAL DIRECTIONS IN LAYERED SHELL INTERFACE

When physics symbols are switched on in the Layered Shell interface and a material model node is selected in the Model Builder, a green arrow will indicate the positive normal direction (the stacking direction) for each boundary.

THIN ELASTIC LAYER WITH MASS

In the **Thin Elastic Layer** and **Thin Elastic Layer, Interface** nodes in the Layered Shell interface, it is now also possible to assign a mass distribution to the layer. This mass is taken into account both in dynamic analysis and in loads from gravity and rotating frames.

OUTPUT OF ABD MATRIX

The **Layered Linear Elastic Material** in the Shell interface now defines a matrix variable containing the complete 'ABD' (elastcity) and 'abd' (compliance) matrices. The variables are named `shell.ABD` and `shell.abd`, respectively. They can be used in, for example, a **Point Matrix Evaluation** node, and can be found under **Shell>Stiffness and flexibility** in the **Replace Expression** dialog.

USE ALL LAYERS OPTION ADDED TO SHELL INTERFACE

In the **Layered Linear Elastic Material** in the Shell interface, it is now possible to select several layered materials active on different boundaries through the new **Use All Layers** check box. In the previous version, this type of selection was only available in the Layered Shell interface.

SYNCHRONIZATION OF ELASTIC PROPERTIES

If isotropic material data (for example, E and ν) are present in a **Material**, then this material can also be used if the **Solid model** in a linear elastic material is set to **Orthotropic** or **Anisotropic**. Similarly, orthotropic material data can be used when **Solid model** is set to **Anisotropic**. This is particularly important in layered structures where some layers are anisotropic and other layers are isotropic, since it is now possible to use a common anisotropic material model in all layers in this situation.

NEW DEFAULT PLOTS

In the Layered Shell and Shell interfaces, new default plots showing the layup angles are generated. For the former interface, there is also a new default plot showing the shell as a 3D geometry.

RESULT EVALUATION IMPROVEMENTS

New settings have been added to the **Layered Material** dataset. It is now possible to select not only boundaries, but also layers and interfaces. It is also possible to define a through-thickness location when the dataset is referenced in a node under **Derived Values**.

In the **Layered Material Slice** plot, there are several improvements:

- In addition to plotting in layers, it is also possible to plot results on interfaces.
- You can define a layout so that an array of plots for several locations are generated in a single plot node.
- Layer names can be added to the plot automatically, without separate **Annotation** nodes.

In the **Through Thickness** plot, the locations of interfaces between layers can now be drawn automatically. It is also possible to plot quantities that are only defined in certain layers.

CONSISTENT NAMING OF STRESS VARIABLES

The same convention is now used for the naming of stress variables in the Layered Shell interface and the Layered Linear Elastic Material in the Shell and Membrane interfaces. The following convention is used:

- Second Piola-Kirchhoff stress, local coordinate system: `Sl`
- Second Piola-Kirchhoff stress, laminate coordinate system: `Slm`
- Second Piola-Kirchhoff stress, layer coordinate system: `Sll`

New Models in Version 5.5

PROGRESSIVE DELAMINATION IN A LAMINATED SHELL

Interfacial failure or delamination in a composite material is a common phenomenon. It can be simulated using a cohesive zone model (CZM). This example shows the implementation of a CZM with a bilinear traction-separation law in a layered shell. It is used to predict the mixed-mode softening onset and delamination propagation.

MIXED-MODE DELAMINATION OF A COMPOSITE LAMINATE

This is a benchmark model for delamination, where the results from the Layered Shell interface are compared with the same model analyzed in the Solid Mechanics interface and also with published results. A CZM with a bilinear traction-separation is used.

STRESS AND MODAL ANALYSIS OF A COMPOSITE WHEEL RIM

This example demonstrates the modeling of a wheel rim made of a laminated carbon-epoxy composite material. First, a stress analysis of a composite wheel rim is performed, in which the rim is subjected to an inflation pressure and a tire load. Second, a prestressed eigenfrequency analysis is performed for a range of operating speeds, and a Campbell diagram showing the variation of eigenfrequencies with rotation speed is generated.

PIEZOELECTRICITY IN A LAYERED SHELL

This is a tutorial example showing how to model piezoelectric applications using the layered shell functionality. Two cases of material orientation are studied. In the first case, the pole axis is normal to the shell surface, which results in a thickness change. In the second case, the pole axis is in the plane of the shell, which leads to bending.

CONNECTING LAYERED SHELLS WITH SOLIDS AND SHELLS

Layered shell elements, which are used for modeling composite shells, are often connected to solid and shell elements in cladding or side-by-side configurations to represent a realistic structure. For such applications, it becomes important to connect the layered shell element to other structural elements in an accurate and convenient manner. In this tutorial and verification problem, you will learn how to connect layered shell elements to solid and shell elements in different configurations. The results are also compared to a solid model.

Corrosion Module

New Functionality in Version 5.5

- A new Current Distribution, Pipe interface has been added. The interface is typically used to model internal pipeline corrosion.
- The Sacrificial Edge Anode feature now has an option to set zero capacity loss rate for cathodic current densities in the Secondary Current Distribution interface.
- A new Nernst Equation option is now available for defining the Equilibrium Potential in Electrode Reactions and Porous Electrode Reaction nodes. In the Tertiary Current Distribution, Nernst-Planck interface, the concentration dependence of the equilibrium potential for the Nernst Equation option is defined automatically based on the reaction stoichiometry.
- The exchange current density for the Butler-Volmer and Linearized Butler-Volmer kinetics option in Electrode Reaction and Porous Electrode reaction nodes can now be made automatically concentration dependent. The option is available if the Nernst Equation option is used when defining the Equilibrium potential. In the Tertiary Current Distribution, Nernst-Planck interface, the options Mass action law or Lumped multistep are available for the exchange current density type.
- The Tertiary Current Distribution, Nernst Planck interfaces have changed default settings to use the Nernst Equation for equilibrium potential, Butler-Volmer for kinetics expression type, and Mass action law for exchange current density type.
- The Electroanalysis (elan) interface has been replaced with the new Electroanalysis (tcd) entry in the Model Wizard. The new Electroanalysis (tcd) interface provides seamless switching between the various charge conservation model options supported by the Tertiary Current Distribution, Nernst-Planck interface.
- The Chemistry interface, added to the Electrochemistry License in version 5.5, now supports Electrode Reactions and Electrode Reaction Groups. The Chemistry interface allows for defining multiple species and electrode reactions, as well as ordinary chemical reactions. Also, mixture and thermodynamic properties, such as equilibrium potentials, can be calculated automatically by the Chemistry interface. Variables defined by these features can then be used in any physics interface.
- The Non-Faradaic Reactions nodes now also allow you to set reaction rates for the bulk species in the electrolyte.

- The Common model input has been set as the new default for Temperature model input in all electrochemistry interfaces.
- The Cyclic Voltammetry boundary condition at the Electrode Surface node and Cyclic Voltammetry study step are now available for all Electrochemistry interfaces.
- The Thin Electrode Surface node is now available also in the Primary Current Distribution interface.
- New and updated default plots have been added in the Transport of Diluted Species, Tertiary Current Distribution, Nernst Planck, and Darcy's Law interfaces.
- A new mass inflow boundary condition has been added to Darcy's Law.
- Extended Inlet/Outlet boundary condition in the Darcy's Law interface to include Velocity (Inlet/Outlet), Pressure (Inlet/Outlet), and Mass flow (Inlet) options.
- The Darcy's Law interface now supports frequency-domain studies (with licenses for the Batteries & Fuel Cells Module, Corrosion Module, Electrochemistry Module, and Electrodeposition Module).
- More accurate definitions of the internal cH and pH variables (in basic electrolytes), and the cOH and pOH (in acid electrolytes) in the Tertiary Current Distribution, Nernst-Planck (water-based charge balance option) and the Electrophoretic Transport interfaces have been implemented.
- The implementation of the Donnan boundary conditions used by the Ion-Exchange Membrane node has been improved.
- A new boundary condition formulation option has been added in the Film Resistance section of the Electrode Surface, Internal Electrode Surface and Thin Electrode Surface nodes in the Primary Current Distribution interface.
- Default stoichiometric number for Dissolving-Depositing species changed to 0 (from 1).

New and Updated Models in Version 5.5

- The new model *Internal Pipeline Corrosion Protection Using Resistor Controlled Cathodic Protection* has been added.
- The localized_corrosion_ls and under_deposit_corrosion models have been updated with new plots for the electrolyte potential.

- The following models have been updated to use the new Electroanalysis (tcd) interface: impedance_spectroscopy, microdisk_voltammetry, cyclic_voltammetry_1d, and co2_corrosion.
- The impedance_spectroscopy, microdisk_voltammetry, cyclic_voltammetry_1d, and orange_battery models have been updated to use the new Nernst equation and concentration-dependent exchange current density in Butler-Volmer kinetics.

Backward Compatibility with 5.4

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

ECAD Import Module

New Functionality in Version 5.5

- When importing copper layers from IPC-2581 and ODB++ files it is now possible to automatically exclude symbols located outside the outline of the board.
- The new button **Reload Layers** is now available in the settings for the **Import** feature node for the GDS-II, IPC-2581, and ODB++ formats. Use this button to re-read the layer information from the ECAD file if the file has changed on disk after the last time it was imported.
- Support for the ODB++(X) and NETEX-G formats is no longer available.

Backward Compatibility with Version 5.3a

The `elevation` property of the `Import` function is now also initialized when setting the `manualelevation` property to `off`. Previously, it was only initialized when setting the `filename` property, the `manualelevation` property is `off`, and the value of either the `height` or `importtype` properties is changed.

Backward Compatibility with Version 5.2a

- The default value for the `sellayer` property of the `Import` function is now `on`. Previously, the default was `off`.
- The default value for the `sellayershow` property of the `Import` function is now `all`. Previously, the default was `dom`.

Backward Compatibility with Version 5.0

- The default value for the `grouping` property of the `Import` function is now `layer`. Previously, the default was `all`.
- For ODB++® and ODB++(X) files (the `type` property is `odb`), new rules apply for the initialization of the `importlayer` property of the `Import` function. Now, only layers of the types *Metal* and *Dielectric* are initialized with the string `on` in the `importlayer` string array. Previously, this also included layers of the type *Drill*.

Electrochemistry Module

New Functionality in Version 5.5

- A new Nernst Equation option is now available for defining the Equilibrium Potential in Electrode Reactions and Porous Electrode Reaction nodes. In the Tertiary Current Distribution, Nernst-Planck interface, the concentration dependence of the equilibrium potential for the Nernst Equation option is defined automatically based on the reaction stoichiometry.
- The exchange current density for the Butler-Volmer and Linearized Butler-Volmer kinetics option in Electrode Reaction and Porous Electrode Reaction nodes can now be made automatically concentration dependent. The option is available if the Nernst Equation option is used when defining the equilibrium potential. In the Tertiary Current Distribution, Nernst-Planck interface, the options Mass action law and Lumped multistep are available for the exchange current density type.
- The Tertiary Current Distribution, Nernst-Planck interfaces have changed default settings to use the Nernst equation for equilibrium potential, Butler-Volmer for kinetics expression type, and Mass action law for exchange current density type.
- The Electroanalysis (elan) interface has been replaced with the new Electroanalysis (tcd) entry in the model wizard. The new Electroanalysis (tcd) interface provides seamless switching between the various charge conservation model options supported by the Tertiary Current Distribution, Nernst-Planck interface.
- The Chemistry interface, added to the Electrochemistry Module in version 5.5, now supports Electrode Reactions and Electrode Reaction Groups. The Chemistry interface allows for defining multiple species and electrode reactions, as well as ordinary chemical reactions. Also, mixture and thermodynamic properties, such as equilibrium potentials, can be calculated automatically by the Chemistry interface. Variables defined by these features can then be used in any physics interface.
- The Non-Faradaic Reactions nodes now also allow you to set reaction rates for the bulk species in the electrolyte.
- The Common model input has been set as the new default for Temperature model input in all electrochemistry interfaces.
- The Cyclic Voltammetry boundary condition at the Electrode Surface node and Cyclic Voltammetry study step are now available for all Electrochemistry interfaces.

- The Thin Electrode Surface node is now available also in the Primary Current Distribution interface.
- New and updated default plots have been added in the Transport of Diluted Species, Tertiary Current Distribution, Nernst Planck, and Darcy's Law interfaces.
- A new mass inflow boundary condition has been added to Darcy's Law.
- Extended Inlet/Outlet boundary condition in the Darcy's Law interface to include Velocity (Inlet/Outlet), Pressure (Inlet/Outlet), and Mass flow (Inlet) options.
- The Darcy's Law interface now supports frequency domain studies (with licenses for the Batteries & Fuel Cells Module, Corrosion Module, Electrochemistry Module, and Electrodeposition Module).
- More accurate definitions of the internal cH and pH variables (in basic electrolytes), and the cOH and pOH (in acid electrolytes) in the Tertiary Current Distribution, Nernst-Planck (water based charge balance option) and the Electrophoretic Transport interfaces have been implemented.
- The implementation of the Donnan boundary conditions used by the Ion-Exchange Membrane node has been improved.
- A new boundary condition formulation option has been added in the Film Resistance section of the Electrode Surface, Internal Electrode Surface and Thin Electrode Surface nodes in the Primary Current Distribution interface.

New and Updated Models in Version 5.5

- The following models have been updated to use the new Electroanalysis (tcd) interface: glucose_sensor, impedance_spectroscopy, microdisk_voltammetry, and cyclic_voltammetry_1d.
- The glucose_sensor, impedance_spectroscopy, microdisk_voltammetry, cyclic_voltammetry_1d, thin_layer_chrono_amperometry, orange_battery, tumor, and wire_electrode models have been updated to use the new Nernst equation and concentration-dependent exchange current density in Butler-Volmer kinetics.
- The wire_electrode model has been updated, making use of the Potential Coupling multiphysics feature.

Backward Compatibility with Version 5.4

- The Tertiary Current Distribution, Nernst Planck interfaces have changed default settings to use the Nernst equation for equilibrium potential, Butler-Volmer for

kinetics expression type, and Mass action law for exchange current density type. Java API backward compatibility may be affected.

Electrodeposition Module

New Functionality in Version 5.5

- A new Nernst Equation option is now available for defining the Equilibrium Potential in Electrode Reactions and Porous Electrode Reaction nodes. In the Tertiary Current Distribution, Nernst-Planck interface, the concentration dependence of the equilibrium potential for the Nernst Equation option is defined automatically based on the reaction stoichiometry.
- The exchange current density for the Butler-Volmer and Linearized Butler-Volmer kinetics option in the Electrode Reaction and Porous Electrode reaction nodes can now be made automatically concentration dependent. The option is available if the Nernst Equation option is used when defining the equilibrium potential. In the Tertiary Current Distribution, Nernst-Planck interface, the options Mass action law and Lumped multistep are available for the exchange current density type.
- The Tertiary Current Distribution, Nernst-Planck interfaces have changed default settings to use the Nernst equation for equilibrium potential, Butler-Volmer for kinetics expression type, and Mass action law for exchange current density type.
- The Electroanalysis (elan) interface has been replaced with the new Electroanalysis (tcd) entry in the Model Wizard. The new Electroanalysis (tcd) interface provides seamless switching between the various charge conservation model options supported by the Tertiary Current Distribution, Nernst-Planck interface.
- The Chemistry interface, added to the Electrodeposition Module in version 5.5, now supports Electrode Reactions and Electrode Reaction Groups. The Chemistry interface allows for defining multiple species and electrode reactions, as well as ordinary chemical reactions. Also, mixture and thermodynamic properties, such as equilibrium potentials, can be calculated automatically by the Chemistry interface. Variables defined by these features can then be used in any physics interface.
- The Non-Faradaic Reactions nodes now also allow you to set reaction rates for the bulk species in the electrolyte.
- The Common model input has been set as the new default for Temperature model input in all electrochemistry interfaces.
- The Cyclic Voltammetry boundary condition at the Electrode Surface node and Cyclic Voltammetry study step are now available for all Electrochemistry interfaces.

- The Thin Electrode Surface node is now available also in the Primary Current Distribution interface.
- New and updated default plots have been added in the Transport of Diluted Species, Tertiary Current Distribution, Nernst Planck, and Darcy's Law interfaces.
- A new mass inflow boundary condition has been added to Darcy's Law.
- Extended Inlet/Outlet boundary condition in the Darcy's Law interface to include Velocity (Inlet/Outlet), Pressure (Inlet/Outlet), and Mass flow (Inlet) options.
- The Darcy's Law interface now supports frequency-domain studies (with licenses for the Batteries & Fuel Cells Module, Corrosion Module, Electrochemistry Module, and Electrodeposition Module).
- More accurate definitions of the internal cH and pH variables (in basic electrolytes), and the cOH and pOH (in acid electrolytes) in the Tertiary Current Distribution, Nernst-Planck (water-based charge balance option) and the Electrophoretic Transport interfaces have been implemented.
- The implementation of the Donnan boundary conditions used by the Ion-Exchange Membrane node has been improved.
- A new boundary condition formulation option has been added in the Film Resistance section of the Electrode Surface, Internal Electrode Surface, and Thin Electrode Surface nodes in the Primary Current Distribution interface.
- Default stoichiometric number for Dissolving-Depositing species changed to 0 (from 1).

New and Updated Models in Version 5.5

- The new model *Electroplating of Multiple Components in a Rack* has been added.
- The new model *Two-phase flow Modeling of Copper Electrowinning using Bubbly Flow* has been added. (This model also requires the CFD Module to compute.)
- The *Copper Deposition in a Trench Using the Level Set Method* model has been updated to improve mass conservation.
- The following models have been updated to use the new Electroanalysis (tcd) interface: impedance_spectroscopy, microdisk_voltammetry, cu_electroless_deposition, and cyclic_voltammetry_1d.
- The impedance_spectroscopy, microdisk_voltammetry, cyclic_voltammetry_1d, orange_battery, cu_trench_deposition_ls, cu_trench_deposition, inductor_coil, microconnector_bump_3d, and fountain_flow models have been updated to use the

new Nernst equation and concentration-dependent exchange current density in Butler-Volmer kinetics.

Backward Compatibility with Version 5.4

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

Fatigue Module

New Functionality in Version 5.5

FATIGUE ANALYSIS IN SOLID ROTOR INTERFACE

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

MULTIPLE FATIGUE MODEL NODES IN ONE FATIGUE INTERFACE

The Fatigue interface now supports several fatigue models within the same interface. This means, for example, that you can evaluate several different fatigue criteria at the same time using a single Fatigue interface.

THE FATIGUE INTERFACE NO LONGER REQUIRES A DOMAIN SELECTION

The Fatigue interface is now an interface with global scope. In previous versions, it was a physics interface at the domain level. Thus, there is no longer any geometrical selection at the interface level.

The advantage with the new design is that you do not need to select a domain in order to do a fatigue evaluation on boundaries, edges, or points. This is particularly important when evaluating fatigue life in shell models, since such models no longer require the addition of superfluous domains.

Backward Compatibility with Version 5.3

Several of the fatigue laws for low-cycle fatigue are customarily expressed in terms of number of load reversals.

In previous versions, the distinction between *number of load reversals* to failure and *number of load cycles* to failure has not been clear, neither in the theory sections nor in result output. In version 5.3a, the result is always in terms of cycles, and the documentation has been clarified at a number of places.

The affected fatigue models are **Basquin**, **Coffin-Manson**, **Combined Basquin and Coffin-Manson**, **Morrow**, **Smith-Watson-Topper (SWT)**, **Wang-Brown**, and **Fatemi-Socie**.

Geomechanics Module

New Functionality in Version 5.5

CHANGES IN THE ELASTOPLASTIC SOIL FAMILY OF MODELS

The relationship between the void ratio and stress has been modified for the **Modified Cam-Clay**, **Modified Structured Cam-Clay**, and **Extended Barcelona Basic** models. When the option to specify Poisson's ratio (ν) is used, then the shear modulus (G) is derived from ν and the current secant bulk modulus. Also, the mean stress is now equal to reference pressure at zero strain.

For the **Modified Structured Cam-Clay** model, the angle of internal friction is no longer available as an input, unless the **Slope of critical state line** is set to **Match to Mohr-Coulomb criterion**.

For the **Extended Barcelona Basic** model, a new input for initial suction has been added. The effect is that the volumetric strain is now based on changes in suction, rather than on the total suction.

For the **Hardening Soil** model, the expressions for the cap yield functions and plastic potential have been modified. Also, the expressions for critical deviatoric stress and dilatancy angle have been updated.

NEW TENSION CUT-OFF CRITERION IN SOIL PLASTICITY

For the Mohr-Coulomb and Drucker-Prager models, you can now choose between two different tension cut-off criteria. Set **Tension Cut-off** to **Mean stress cut-off** to employ the new criterion, which is numerically more stable than the old **Principal stress cut-off** criterion.

IMPROVED PERFORMANCE FOR PLASTICITY

The speed of plasticity computations has been improved by up to 20% for several of the more common plasticity models.

New Models in Version 5.5

TRIAXIAL TEST WITH HARDENING SOIL MATERIAL MODEL

In this example, a triaxial test is simulated using the Hardening Soil material model. The test consists of two main stages; an initial isotropic compression, followed by axial

compressive/tensile loading. A hyperbolic stress-strain relation is recovered. It is also verified that the asymptotic value of the axial stress is equal to the analytical value of the failure stress.

ISOTROPIC COMPRESSION TEST FOR STRUCTURED CLAYS

This model simulates the isotropic compression of naturally structured and artificially structured clays using the Modified Structured Cam-Clay (MSCC) material model. The aim of the example is to reproduce the compression behavior given in a benchmark for four structured clays.

TRIAXIAL AND OEDOMETER TEST WITH MODIFIED CAM-CLAY MATERIAL MODEL

In this example, triaxial and oedometer tests are simulated using the Modified Cam-Clay material model. A nonlinear stress-strain relation is recovered with the constant Poisson's ratio formulation. The hardening and softening behavior is recovered for normally consolidated and highly overconsolidated soils.

Heat Transfer Module

New and Improved Functionality in Version 5.5

NEW LUMPED THERMAL SYSTEM INTERFACE

The Lumped Thermal System interface is now available for heat transfer modeling in discrete thermal systems. By analogy with electrical circuits, it provides a lumped element model that idealizes the domain and boundary conditions for heat transfer into components joined by a network of perfectly thermally conductive wires. It provides the following global features:

- Two-node components: **Conductive Thermal Resistor**, **Thermal Capacitor**, **Heat Rate Source**, **Convective Thermal Resistor**, **Radiative Thermal Resistor**, **Thermoelectric Module**, and **Heat Pipe**.
- One-node components: **Temperature**, **Heat Rate**, **Radiative Heat Rate**, **Thermal Mass**, and **External Terminal**.

The new **Lumped System Connector** feature, available under all Heat Transfer interfaces, can be used to connect an **External Terminal** node in the lumped thermal system to a finite element model in any dimension.

In addition, subsystems may be defined once with the **Subsystem Definition** component and then used several times in the circuit with the **Subsystem Instance** component.

This new set of functionality makes it possible to dramatically reduce the models' complexity by using global features not requiring any geometry nor mesh definition. The use of the new features is illustrated in the *Composite Thermal Barrier, Lumped Thermal System*, *Thermoelectric Cooler, Lumped Thermal System*, *Transient Conduction in a Wall, Lumped Thermal System*, and *Buried Cables Heating* tutorials.

EXTENDED CAPABILITIES FOR RADIATION IN PARTICIPATING MEDIA

The Radiation in Participating Media and Radiation in Absorbing-Scattering Media interfaces have a new **Wavelength dependence of radiative properties** list under the **Participating Media Settings** section. With the **Multiple spectral bands** option, an arbitrary number of spectral bands can now be set to model wavelength-dependent material properties, as for surface-to-surface radiation. The features of the two mentioned interfaces have been updated to improve usability when multiple spectral bands are used. It is possible to define material properties such as the absorption

coefficient, the scattering coefficient or the surface emissivity from a wavelength-dependent function or from a table, with one value per spectral band. The **Constant** option of the **Wavelength dependence of radiative properties** list reproduces the behavior of the features up to version 5.4.

In addition, the new **Radiative Source** feature allows you to add a source term in the radiative transfer equation by specifying either a **Power density**, a **Directional power density**, or an **Energy rate**. It is available in the Radiation in Participating Media and Radiation in Absorbing-Scattering Media interfaces.

Finally, the new **Initial Values** feature allows you to set a user-defined value for the initial radiative intensity (discrete ordinates method) or incident radiation (P1 Approximation). It is available in the Radiation in Participating Media and Radiation in Absorbing-Scattering Media interfaces. In the Radiation in Participating Media interface, the blackbody formulation can be used for the specification of the initial condition.

EXTENDED CAPABILITIES FOR SURFACE-TO-SURFACE RADIATION

It is now possible to specify the **Ambient emissivity** in the features modeling radiative surfaces (**Diffuse Surface** and **Diffuse Mirror** in the Surface-to-Surface Radiation interface). This is particularly interesting when it is combined with the **Multiple spectral bands** option of the Surface-to-Surface Radiation interface to model radiative cooling. Atmosphere can be modeled as a radiative body that emits at the ambient temperature, and the ambient emissivity can be used to take into account its high emissivity for some spectral bands, specially for clear-sky conditions.

In addition, the **Symmetry** and **External Radiation Source** features are now available when the **Ray shooting** radiation method is used. The **Symmetry** feature can be used to define up to three symmetry planes or to define sector symmetry in order to reduce the computational effort for the view factor computation. The **External Radiation Source** feature can be used to define radiation coming from an object that is not represented in the model geometry, like the sun. For time-dependent analysis, the external radiation source position or direction may vary during the simulation.

Finally, a new option is available to control the view factor update for time-dependent simulations, when the **Ray shooting** radiation method is used. The **View Factors Update** section is available with the **Advanced Physics Options** in the Surface-to-Surface Radiation interface. This allows to define a criterion based on a user-defined **Expression**. This may considerably reduce the computation time of time-dependent simulations.

OPEN BOUNDARY IMPROVEMENT

The **Open Boundary** feature, available on exterior boundaries of fluid-type domains, has a new **Inflow Condition** section to set the inflow from a Danckwerts condition or by using weak constraints on the temperature. The implementation is unchanged for outflow. The new **Flux (Danckwerts)** option applies an inflow of heat coming from a virtual domain with known upstream conditions. This flux is proportional to the temperature difference between the boundary and the upstream, and it is also proportional to the flow rate. It tends to prescribe the upstream temperature at the boundary for large flow rate, while the boundary temperature is influenced by the sources and sinks of the adjacent regions for small flow rates. It may improve the performance of natural convection computations by inducing a smoother numerical formulation.

EXTENDED CAPABILITIES FOR HEAT AND MOISTURE TRANSPORT

A new formulation is available in the Moisture Transport in Air interface to model convection and diffusion of vapor in air when the vapor content is high. In this case, the moist air density may vary significantly in space and time due to vapor concentration gradients, and the default **Diluted species** formulation should be replaced by the **Concentrated species** one. The new **Mixture type for moist air** list is available under the **Physical Model** section of the physics interface and allows you to choose between the two abovementioned formulations.

For both formulations, the equations are numerically solved by using the relative humidity (`mt.phi`) as the shape function. This information is now displayed under the **Dependent Variables** section. Note that the definition of the relative humidity in the Heat Transfer interface (`ht.phi`) is now consistent with the one of the Moisture Transport interface, with no bounding applied when the relative humidity goes above unity to allow for supersaturation conditions.

In addition, two new features are available to define flow conditions on the exterior boundaries of moist air domains:

- The **Inflow** feature allows to specify upstream moisture conditions to define the inflow of moisture that would be obtained if adding a virtual domain upstream of the inlet. A Dankwerts condition is used to estimate the flux across the boundary.
- The **Open Boundary** feature behaves identically to the **Inflow** feature for an incoming flow, and automatically switches to a zero diffusive flux condition (as with the **Outflow** feature) for an outgoing flow. It provides also an alternative way of specifying the inflow condition by using a **Discontinuous Galerkin** constraint on the temperature.

Moreover, the default solver settings have been updated to improve the numerical convergence for heat and moisture problems. A **Fully Coupled** node is set by default, containing both the temperature and relative humidity variables. Specific parameters (**Damping factor**, **Jacobian update**, **Maximum number of iterations**) are set for stationary and time-dependent studies. The use of a fully coupled approach strengthens the need of defining consistent initial and boundary conditions, and may require the use of a ramp approach to initialize the computation.

The **Relative tolerance** has been increased in the stationary and time-dependent steps, and a lower **Tolerance factor** is used in the **Fully Coupled** node to compensate. In particular, this releases the constraint on time steps for time-dependent studies.

Finally, the **Moist Air** feature is now available within all the Heat Transfer interfaces and can be found under the new **Specific Media** menu by right-clicking on the physics interface node.

EXTENDED CAPABILITIES FOR LAYERED MATERIAL FEATURES

The features and interfaces using the **Layered Material** functionality for the modeling of heat transfer in thin structures have been redesigned to improve the user experience in the model setting process. This process has been especially simplified when a single layered material is considered, while extended capabilities are available for more complex applications.

These updates apply to the **Thin Layer**, **Thin Film**, and **Fracture** features (and subfeatures) in the Heat Transfer interface, and to the Heat Transfer in Shells, Heat Transfer in Films, and Heat Transfer in Fractures interfaces (with all features), found under the **Heat Transfer>Thin Structures** branch.

It is now possible to select any boundary or edge from the interfaces and features, while only the geometric entities with a **Layered Material** defined were available for selection in version 5.4. A **Thickness** value can be defined in the **Material Contents** table of any material.

The **Shell Properties** section replaces the **Layer Selection** section in the interfaces and (boundary and edge) features.

By selecting the **Nonlayered shell** option in the **Shell type** list, it is possible to set a user-defined **Thickness** and to apply a classical material on the boundary, like for any other boundary feature. This simplified approach allows an optimized computation of heat transfer in a single-layer material with a lumped formulation. By default, the **Layer type** uses such a lumped formulation, with the **Thermally thin approximation** option, in the **Solid**, **Fluid**, and **Porous Medium** features of the Heat Transfer in Shells interfaces.

With the **Layered shell** option, you can model temperature variations through the thickness of the layered material, and it is possible to restrict the computation to a subset of layers. In addition, you can scale the layer thickness for a single-layer material. The scale can be defined from arbitrary expressions, in particular, from other physics quantities. For example, it enables you to define a film thickness from the films thickness computed by a Thin Film Flow interface.

For the features applying on interfaces between layers, predefined sets of interfaces are available in the **Interface Selection** section to ease the settings.

Finally, the definition of the coordinates system attached to a layered material has been improved to speed up computation, in particular for models with a very large number of boundaries.

NONISOTHERMAL FLOW MULTIPHYSICS COUPLING

The **Nonisothermal Flow** multiphysics coupling has new automatic settings in order to provide accurate physical results without the need of user action. In particular, the pressure work contribution to the energy equation is automatically added with the **Compressible flow** option only, and the **Include work done by pressure changes** check box has been removed. Changes in results are expected when computing models created in version 5.4 or earlier, in which the **Compressible flow** option was used without the **Include work done by pressure changes** check box selected.

Also, the **Include viscous dissipation** check box is now selected by default. This ensures energy conservation with the default settings of the multiphysics coupling node. In addition, it has been updated to give the possibility to couple a single heat transfer interface to multiple fluid flow interfaces using multiple Nonisothermal Flow Coupling features.

MISCELLANEOUS

- The **Heat transfer in alloys** and **Heat transfer in biological tissue** check boxes have been removed from the **Physical Model** section of the Heat Transfer interface. The **Biological Tissue** and **Shape Memory Alloy** features, activated by the abovementioned check boxes, can be found under the new **Specific Media** context menu when right-clicking on the physics interface node.
- The context menus of the Heat Transfer and Moisture Transport interfaces have been reorganized with the addition of the following submenus for domain and boundary features: **Specific Media**, **Flow Conditions**, **Thin Structures**, **Heat Sources**, and **Evaporation/Condensation**.

- The plot menus of the Heat Transfer interface have been reorganized with the addition of submenus for a better indexing of the predefined variables available for postprocessing.
- The **Ambient Thermal Properties** node, available under the **Definitions** node, has been moved to the **Shared Properties** subnode and renamed **Ambient Properties**. It contains a new input field for **Precipitation rate**.
- The `mt.delta` variable (vapor permeability of still air), required by the **Thin Moisture Barrier** feature, has been added to the Moisture Transport in Air interface.
- The definition of the fractional emissive power variable, used, for example, in the **Diffuse Surface** feature, has been improved.
- The default solver settings have been improved for models including the **Thermal Expansion** multiphysics coupling.
- A warning message has been added when the Heat Transfer with Surface-to-Surface Radiation coupling node is solved for and one of the linked physics interfaces is not solved for.
- The values of the normal diffusive heat fluxes at interior boundaries have been corrected in models with surface-to-surface radiation.

New Applications in Version 5.5

LED BULB COOLING

This example simulates the thermodynamical behavior of a LED bulb and evaluates the maximal temperature reached inside it while operating. It estimates the temperature of the system due to the heating of the LED chips, balanced by the cooling by a buoyancy-driven airflow both inside and outside the bulb and radiation to the ambient surrounding. The geometry of this 3D model includes a realistic representation of the LED bulb and the surrounding air domain.

COMPOSITE THERMAL BARRIER, LUMPED THERMAL SYSTEM

This new tutorial is a variant of the *Composite Thermal Barrier* tutorial, in which two ceramic thin layers with different thermal conductivities are sandwiched in a steel column. It shows how to connect two 3D finite element domains through a Lumped Thermal System for the modeling of heat transfer in the ceramic thin layers. The new lumped approach is validated against the FEM approach.

THERMOELECTRIC COOLER, LUMPED THERMAL SYSTEM

This new tutorial is a variant of the *Thermoelectric Cooler* tutorial, in which heat transfer is computed in a thermoelectric cooler. It shows how to use the Lumped Thermal System interface to estimate the cooling provided by the device over time, and validates the numerical results obtained with the Lumped Thermal System interface against those obtained with the finite element computation made on a 3D geometry.

TRANSIENT CONDUCTION IN A WALL, LUMPED THERMAL SYSTEM

This new tutorial performs a transient analysis of the temperature through a house wall. The wall is composed of different layers corresponding to the structure, insulation, and plaster. On the exterior and the interior boundaries, the wall is exposed to thermal radiation and convective cooling. The results obtained using the Lumped Thermal System interface are compared with results obtained with a FEM approach and data from the literature.

BURIED CABLES HEATING

This new tutorial computes the temperature distribution in three buried cables in a situation where the soil surface temperature is known. The temperature profile obtained with a thermal circuit approach for the cables is compared with a FEM approach. The thermal circuit properties are established using standard representation of buried cables under the Kennelly assumption (uniform soil surface temperature). The superposition principle is used to determine the mutual heating of cables in order to improve the equivalent thermal circuit accuracy.

SURFACE-TO-SURFACE RADIATION WITH DIFFUSE AND SPECULAR REFLECTION

This new tutorial shows how to use the Surface-to-Surface Radiation interface to simulate heat transfer with radiation between diffuse emitters and diffuse-and-specular reflectors. First, it focuses on the validation of the radiative heat flux computation with the ray shooting algorithm against an analytical solution. Then, the coupling with the Heat Transfer in Solids interface is illustrated.

HOW TO IMPROVE PERFORMANCE OF VIEW FACTOR COMPUTATION FOR SURFACE-TO-SURFACE RADIATION MODELING

This tutorial model has been updated to show how to use the features of the Heat Transfer Module to reduce the computation time and memory usage in view factor computations. Illustrative results of speedup and memory usage reduction are given on a benchmark model for the **Hemicube** and **Ray shooting** radiation methods.

HAMSTAD BENCHMARK 1: HEAT AND MOISTURE TRANSPORT IN AN INSULATED ROOF

This tutorial simulates coupled heat and moisture transport in an insulated roof structure made of two layers of different properties regarding capillary transport. This configuration brings a risk of internal condensation at the interface between the two layers. This 1D model allows to validate the numerical results obtained with COMSOL Multiphysics against the first benchmark defined in HAMSTAD-WP2 (modeling for the validation of numerical simulations for coupled heat and moisture transport in building materials).

HEAT AND MOISTURE TRANSPORT WITH MOLD GROWTH PREDICTION

This tutorial uses the temperature and relative humidity computed in the preceding tutorial as inputs for a mold prediction model. The VTT model for mold growth and decline prediction computes a mold index M by solving an ODE, in which the growth and decline rates depend on the time-dependent temperature and relative humidity in the building material.

Updated Applications in Version 5.5

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, the use of postprocessing features like extrusion datasets, streamlines with arrows, and temperature discontinuity plots has been extended in order to produce more explicit results plots.

In addition to these updates, some tutorials have been updated regarding physical modeling:

- The *Condensation Detection in an Electronic Device with Transport and Diffusion* tutorial now uses the Moisture Transport in Air interface for the computation of vapor concentration distribution, with the **Open Boundary** feature applied at the slits. In addition, the **Moist Surface** feature has been added on the interior walls of the device to account for the condensed water accumulated over time on these surfaces.
- The *Evaporative Cooling of Water* tutorial uses the new **Inflow** and **Open Boundary** features of the Moisture Transport in Air interface.
- The *Free Convection in a Water Glass* tutorial now uses an **Incompressible flow** formulation with the **Boussinesq approximation** provided with the **Nonisothermal Flow**

coupling node. In addition, the initial temperature has been updated to 278.15 K to be consistent with the one specified in the model description.

- In the *View Factor Computation* tutorial, the use of the **Symmetry for Surface-to-Surface Radiation** feature is discussed.
- In the *Thermal Analysis of a Turbine Stator Blade* tutorial, the temperature range used for the definition of the material properties has been updated to fit with the operating temperature range of the model.

In the *Heat Conduction in Slab* tutorial, the step function definition has been updated to improve accuracy of the computation.

Backward Compatibility with Version 5.4

- In the **Inflow** feature (Heat Transfer interface), the **Upstream absolute pressure** user input is now inactive, unless the new **Specify upstream absolute** pressure check box is selected. Scripts using the COMSOL API that used to fill this input field need to add an instruction for selecting the **Specify upstream absolute pressure** check box beforehand.
- The **Open Boundary** feature now comes with a new default formulation based on the Danckwerts inflow condition to specify the incoming temperature. The former default formulation is still available by choosing the **Discontinuous Galerkin constraints** option in the **Inflow Condition** section. To keep the former default formulation in scripts using the COMSOL API, you need to add the corresponding instruction to set the **Discontinuous Galerkin constraints** option in the **Inflow Condition** section.
- When the **Relative humidity** option is used in the **Moisture Content** boundary condition, the associated temperature now has to be defined. In version 5.4 and before, the moisture transport local temperature was used instead. In order to reproduce previous results, models created in version 5.4 or earlier will use the moisture transport interface temperature when opened in the GUI. The scripts using the COMSOL API will use the default temperature 293.15 K instead and may need to be updated to reproduce existing results.
- In the **Heat Flux, Interface** and **Heat Source, Interface** features, when the **Heat Rate** option is selected in the **Heat Flux** and **Boundary Heat Source** sections, respectively, the surface definition has been corrected in order to take into account several

interfaces in the **Interface Selection**. For example, if two interfaces are selected, the surface definition is now multiplied by two.

- The **Deposited Beam Power, Interface** and **Surface-to-Ambient Radiation, Interface** features are now applicable on the interior interfaces. In the scripts using the COMSOL API in which the **All interfaces** option was selected, the interior interfaces were selected but no contribution was added on these interfaces. Changes in the numerical results are expected now that the contribution is added also on the interior interfaces.

Backward Compatibility with Version 5.3a

- The Heat Transfer in Thin Shells (htsh), Heat Transfer in Thin Films (htsh), and Heat Transfer in Fractures (htsh) interfaces that do not support the **Layered Material** functionality are now obsolete. When opening a model created in version 5.3a or earlier in the GUI, these interfaces are not migrated. They should be manually replaced by the corresponding Heat Transfer in Shells (htlsh), Heat Transfer in Films (htlsh), and Heat Transfer in Fractures (htlsh) interfaces that support the **Layered Material** functionality. The **Thin Layer**, **Thin Film**, and **Fracture** features are replaced by the **Solid**, **Fluid**, and **Porous Medium** features in the new interfaces.
- The **Thin Layer**, **Thin Film**, and **Fracture** features (and their subfeatures) that do not support the **Layered Material** functionality are now obsolete. When opening a model created in version 5.3a or earlier in the COMSOL Desktop, these features are not migrated. They should be manually replaced by the corresponding features that support the **Layered Material** functionality. Note that the pair features for thin structures are no longer available for layered materials. The obsolete **Thin Layer**, **Thin Film**, and **Fracture** features remain available through the COMSOL API so that existing scripts remain compatible. However, it is recommended to update the scripts to use the new features as the obsolete features will be discontinued in future versions.
- The **Surface-to-surface radiation** check box has been removed from the **Physical Model** section of the Heat Transfer interface. As of Version 5.4, surface-to-surface radiation models should use the Surface-to-Surface Radiation physics interface coupled with a Heat Transfer domain physics interface via the Heat Transfer with Surface-to-Surface Radiation multiphysics coupling feature. The Heat Transfer interface does not support the **Diffuse Surface**, **Diffuse Mirror**, **Prescribed Radiosity**, **Opacity**, and **External Radiation Source** features anymore. Models built prior to v5.3a using surface-to-surface radiation features formerly available in the Heat Transfer interfaces will mark these features as obsolete. Support for features marked as

obsolete is not ensured in future versions, thus it is strongly recommended to transfer all these features to a Surface-to-Surface Radiation physics interface and use a Heat Transfer with Surface-to-Surface Radiation multiphysics coupling feature.

- The **Radiation in participating media** check box has been removed from the **Physical Model** section of the Heat Transfer interface. From Version 5.4, radiation in participating media models should use the Radiation in Participating Media physics interface coupled with a Heat Transfer physics interface via the **Heat Transfer with Radiation in Participating Media** multiphysics coupling feature. The Heat Transfer interface does not support the **Radiation in Participating Media**, **Opaque Surface**, **Incident Intensity**, and **Continuity on Interior Boundary** features anymore. Models built prior to v5.3a using radiation features formerly available in the Heat Transfer interfaces will mark these features as obsolete. Support for features marked as obsolete is not ensured in future versions, thus it is strongly recommended to transfer all these features to a Radiation in Participating Media physics interface and use a **Heat Transfer with Radiation in Participating Media** multiphysics coupling feature.
- The **Ambient Settings** section has been removed from the Heat Transfer interface. When opening a model created in version 5.3a or earlier in the COMSOL Desktop, in which some of the input fields of this section have been edited, an **Ambient Thermal Properties** node is automatically created under **Definitions>Shared Properties** with the same settings. If some ambient variables have been selected in features, the feature inputs are updated as well.
- The **Phase Change Material** feature has been replaced by the **Phase Change Material** subfeature. When opening a model created in version 5.3a or earlier in the COMSOL Desktop, a **Fluid** node with the **Phase Change Material** subfeature is automatically created in place of the obsolete feature. This node is labeled **Phase Change Material** in the model.
- The **Damaged Tissue** section has been removed from the **Biological Tissue** feature. When opening a model created in version 5.3a or earlier in the COMSOL Desktop, in which the **Include damage integral analysis** check box of this section has been selected, a **Thermal Damage** node is automatically added under **Biological Tissue** with the same settings. Note that for the **Temperature threshold** option, it is no longer possible to perform simultaneous hyperthermia and cryogenic analysis. The **Hyperthermia analysis** option is therefore selected by default when opening the model.
- The **Rosseland approximation** option, which was available in the **Radiation discretization method** list of the **Radiation in Participating Media** feature, has been

removed. When opening a model created in version 5.3a or earlier in the COMSOL Desktop, in which this option has been selected, an **Optically Thick Participating Medium** subnode is automatically created under the active domain node.

- The **Medium Properties** feature has been renamed **Absorbing Medium** in the Radiative Beam in Absorbing Medium interface, with identical settings and modeling. When opening a model created in version 5.3a or earlier in the COMSOL Desktop, an **Absorbing Medium** feature is automatically created in place of the obsolete feature. This node is still labeled **Medium Properties** in the model.
- The **Radiation in Participating Media** feature has been renamed **Participating Medium** in the Radiative Beam in Absorbing Medium interface. When opening a model created in version 5.3a or earlier in the COMSOL Desktop, a **Participating Medium** feature is automatically created in place of the obsolete feature. This node is still labeled **Radiation in Participating Media** in the model.
- The **Change Thickness** and **Change Cross Section** features have been renamed **Thickness** and **Cross Section**, respectively. When opening a model created in version 5.3a or earlier in the COMSOL Desktop, these features are not migrated. They should be manually replaced by the new features.

Backward Compatibility with Version 5.3

- The **Electromagnetic Heat Source** and **Boundary Electromagnetic Heat Source** coupling features are now obsolete. They have been merged into a single **Electromagnetic Heating** coupling feature. When opening a model created in version 5.3 or earlier in the COMSOL Desktop, the **Electromagnetic Heat Source** and **Boundary Electromagnetic Heat Source** features are automatically migrated to **Electromagnetic Heating** features with selections (respectively a domain and a boundary selection) that match the selections of the migrated features. The obsolete **Electromagnetic Heat Source** and **Boundary Electromagnetic Heat Source** features remain available through the COMSOL API so that existing scripts remain compatible. However, it is recommended to update the scripts to use the new **Electromagnetic Heating** feature as the obsolete feature will be discontinued in future versions. Note that temperature coupling is enforced in the 5.3a version of the **Electromagnetic Heating** coupling feature.
- The **Thermoelectric Effect** and **Boundary Thermoelectric Effect** coupling features have been merged into a new **Thermoelectric Effect** coupling feature. When opening a model created in version 5.3 or earlier in the COMSOL Desktop, the **Thermoelectric Effect** and **Boundary Thermoelectric Effect** features are automatically migrated to the new **Thermoelectric Effect** feature with selections (respectively a domain and a

boundary selection) that match the selections of the migrated features. The obsolete **Thermoelectric Effect** and **Boundary Thermoelectric Effect** features remain available through the COMSOL API so that existing scripts remain compatible. However, it is recommended to update the scripts to use the new **Thermoelectric Effect** feature as the obsolete feature will be discontinued in the future. Note that temperature coupling is enforced in the 5.3a version of the **Thermoelectric Effect** coupling feature.

- The **Inflow Heat Flux** boundary condition is now obsolete. It can be replaced by the new **Inflow** boundary condition and the **Boundary Heat Source** condition.
- The **Enable conversions between material and spatial frame** check box, displayed in the **Advanced Settings** section of the Heat Transfer interfaces when the model component supports moving mesh, has been removed. This means that conversion between material and spatial frames is always effective when these frames differ.
- The **Energy absorption** option has been renamed **Arrhenius kinetics** in the **Transformation model** list of the **Irreversible Transformation** and **Biological Tissue** nodes.
- The **Convective moisture flux** option has been renamed **Convective moisture flux, pressures difference** in the **Moisture Flux** node.
- The `rhoInt`, `CpInt`, and `gammaInt` variables, defined by the **Fluid** domain feature, have been removed.
- The `q_evap` variable, defined by the **Moist Surface** and **Wet Surface** boundary nodes, has been replaced by the `q_evaptot` variable.

The **General** option for **Thin film model** has been removed from the **Thin Film** fallback node.

Backward Compatibility with Version 5.2a

The obsolete features **Convective Heat Flux** and **Out-of-plane Convective Heat Flux** are removed in Version 5.3. These were marked as obsolete with a warning message since Version 5.0. These features will be removed from your model when opening in Version 5.3. Use instead the corresponding **Heat flux** and **Out-of-Plane Heat Flux** features, with the option **Convective heat flux**. The variables `ht.chflux`, `ht.chflux_u`, `ht.chflux_d`, and `ht.chflux_z` defined by these features have been also removed from equation view and postprocessing variables menu.

The new default option in **Thin Moisture Barrier** feature is **Vapor barrier material properties**, and the **Moisture barrier material properties** option has been removed. When loading a MPH file that was created before version 5.3 this latter option will be cleared,

and the corresponding user inputs used to define the **Moisture transfer coefficient** directly. However, for the API, it will be necessary to update the code with an existing option name.

In the **Equivalent Conductivity for Convection** section of the **Fluid** node, the new **Automatic** option for **Temperature difference** has become the default option. When loading a MPH file that was created before version 5.3, the **User defined** option will be set automatically with the previous value. However, for the API, it will be necessary to update the code to set the **User Defined** option.

Backward Compatibility with Version 5.2

Several enhancements or bug fixes may require an update of the existing applications or induce a change in the results. Here is the list of the changes:

- When **Solar position** is selected in **External Radiation Source**, the new option **Update time from solver** is selected by default. When loading an MPH file that was created before version 5.2a this option will be cleared. However, for the API, for time-dependent studies, it will be necessary to update the code to get the same behavior as in previous versions. In most cases, removing the *t* term in the Second field is the best option. Alternatively, clearing the **Update time from solver** option will ensure that the same behavior as before is obtained.
- The crosswind stabilization of the heat transfer and fluid flow interfaces has been updated. In general, the update reduces the smoothing induced by the stabilization compared to previous version. In addition, when you use the **Nonisothermal Flow** multiphysics coupling and the fluid is defined as incompressible, the uncoupled versions of the streamlines and crosswind stabilization are used.
- Since **Pair Thin Film** and **Pair Fracture** are now pair features, their selection needs to be redefined.
- The definition of the `ht.`*`feature`*`.fc(RH.T,pA)` function when **Moist air** fluid type is used in **Fluid** feature has been corrected. This change can affect the results in simulations where it is used.
- The frame support has been enhanced to support moving mesh combined with deformed geometry in the same component.

- The **Prescribed Radiosity** feature has been updated so that it no longer ignores the **Temperature** model input.
- In surface-to-surface boundary features, the ambient temperature definition has been fixed when the **Opacity Controlled** and **Defined ambient temperature on each side** options are used simultaneously.

Backward Compatibility with Version 5.1

CHANGES IN VARIABLE NAMES

The names of the heat capacity variables have changed in several features for consistency:

- In the **Bioheat** feature, `ht.cb` has become `ht.cp_b`.
- In the **Thin Layer** feature, `ht.cs` has become `ht.cp_s`.
- In the **Thin Rod** feature, `ht.cl` has become `ht.cp_l`.
- In the **Thin Layered Shell** feature, `htsh.cs1` has become `htsh.cp_s1`, and so on.
- In the **Heat Transfer in Porous Media** feature, `ht.c_pp` has become `ht.cp_p`, `ht.c_pp1` has become `ht.cp_p1`, and so on.
- In the **Immobile Fluids** feature, `ht.c_pg1` has become `ht.cp_g1`, and so on.
- In the **Fracture** feature, `ht.c_pfr` has become `ht.cp_fr`.

The names of the thermal resistance and absolute thermal resistance variables have changed in the **Isothermal Domain Interface** feature:

- `ht.idi1.Rs` has become `ht.idi1.R_t`
- `ht.idi1.Rsth` has become `ht.idi1.R_tth`

The name of the fluid viscosity variable has changed in the **Local Thermal Non-Equilibrium** multiphysics coupling: `ltne1.mu_f` has become `ltne1.mu`.

Backward Compatibility with Version 5.0

OBSOLETE FEATURES

A number of features are now obsolete because they have been merged or replaced by a more general features. However, when models from COMSOL 5.0 or before contain these features, some of the obsolete features are visible in version 5.2a with a warning sign notifying that the feature is now obsolete (see Table 1-1). Some features have

been completely removed (see Table 1-2). A model containing any of them can be opened in 5.2a, but the feature will be missing.

Updating the models is recommended. Use the new feature and remove the obsolete feature to make sure that the model will remain compatible with future versions of COMSOL Multiphysics.

The table below summarizes the changes:

TABLE 1-1: OBSOLETE FEATURES THAT ARE OBSOLETE BUT THAN CAN BE LOADED IN 5.1

OBSOLETE FEATURE	REPLACE IN 5.2A BY
Convective Heat Flux	**Heat Flux** with the **Convective heat flux** option selected
Surface-to-Ambient Radiation	**Diffuse Surface** with **Include surface-to-surface radiation** deselected
Surface-to-Surface Radiation	**Diffuse Surface** with **Include surface-to-surface radiation** selected
Highly Conductive Layer	**Thin Layer** with **Layer type** set to **Conductive**
Thin Thermally Resistive Layer	**Thin Layer** with **Layer type** set to **Resistive**

TABLE 1-2: OBSOLETE FEATURES THAT ARE NO LONGER AVAILABLE

OBSOLETE FEATURE	REPLACE IN 5.2A BY
Infinite Element	**Infinite Element** is no longer defined in the physics interface. Infinite elements are defined in the Definitions section instead.
Electrochemical Heat flux	**Boundary Heat Source**
Reaction Heat Flux	**Boundary Heat Source**
Heat Transfer in Participating Media	**Heat Transfer in Solids** contributing to **Radiation in Participating Media**
Porous Matrix	**Heat Transfer in Porous Media**

Note that it is not possible to add any of the obsolete features from the COMSOL Desktop in version 5.2a. Only the new features are available.

OBSOLETE POSTPROCESSING VARIABLES DELETED IN 5.1

List of deleted postprocessing variables: `ndflux_acc`, `ndflux_acc_u`, `ndflux_acc_d`, `ntflux_acc`, `ntflux_acc_u`, `ntflux_acc_d`, `nteflux_acc`, `nteflux_acc_u`, `nteflux_acc_d`, `qout`, `qout_u`, `qout_d`, `qin`, `qin_u`, `qin_d`, `qw`, `qw_u`, and `qw_d`.

CHANGES IN VARIABLE NAMES

The variable `ht.Tvar`, used for nonisothermal flow wall functions, becomes a fully scoped variable in the following features:

- **Heat Flux** (`ht.hf1.Tvar`), **Line Heat flux**, **Point Heat flux**, and **Out of plane heat flux**.
- **Heat Source** (`ht.hs1.Tvar`), **Boundary heat source**, **Line heat source**, and **Point heat source**.
- **Diffuse Surface** (`ht.ds1.Tvar`).

The fully scoped variables `Ptot`, `Pbtot`, `Pltot`, and `Pstot` become `P0`, `Pb`, `Pl`, and `Ps`, respectively, in the following features:

- **Heat Flux**
- **Inflow Heat Flux**
- **Line Heat Flux**
- **Heat Source**
- **Boundary Heat Source**
- **Line Heat Source**
- **Line Heat Source** (**Thin Rod** subfeature)
- **Line Heat Source on Axis**
- **Layer Heat Source** (**Thin Layer** subfeature)

In the **Thermal Contact** feature:

- `Qfric`, `QfricPow`, and `Qrate` become `Qb`, `Pb`, and `Qbpart`, respectively
- `Qfric0`, `Qrate0`, `Econtact0`, `hrad0`, `hgap0`, and `hconstr0` are removed and migrated to existing variables `Qb`, `Qbpart`, `Econtact`, `hrad`, `hgap`, and `hconstr`, respectively, in old models.

The **Change Effective Thickness** feature from the Heat Transfer in Thin Shells interface merges `htsh.de` into the existing variable `htsh.ds`.

The Heat Transfer in Thin Shells interface also merges the property `de` and variable `htsh.de` into existing property `ds` and variable `htsh.ds`. An API command such as `model.physics("htsh").prop("de").set("de", [...])` now does not have any effect.

Obsolete Features

The following features have been removed:

- **Optically Thick Participating Medium** (as a subfeature of **Shape Memory Alloy**)
- **Inflow Heat Flux** (replaced by **Inflow** and **Boundary Heat Source**)
- **Electromagnetic Heat Source** and **Electromagnetic Heat Source** (replaced by **Electromagnetic Heating**)
- **Boundary Thermoelectric Effect** (merged with **Thermoelectric Effect**)
- **Convective Heat Flux** (replaced by **Heat Flux** with **Convective heat flux option**)
- **Out-of-plane Convective Heat Flux** (replaced by **Out-of-plane Heat Flux** with **Convective heat flux option**)
- **Porous Matrix** (replaced by **Porous Medium**)
- **Infinite Element** (replaced by **Infinite Element Domain** declared in the **Definitions** node)
- **Electrochemical Heat Flux** (replaced by **Boundary Heat Source**)
- **Reaction Heat Flux** (replaced by **Boundary Heat Source**)
- **Heat Transfer in Participating Media** (replaced by **Heat Transfer with Radiation in Participating Media**)

MEMS Module

New Functionality in Version 5.5

RANDOM VIBRATION ANALYSIS

Functionality for random vibration analysis has been added. This type of analysis is intended for situations where the load on a structure is random, but has well-defined statistical properties that do not change with time. The excitation is described by its power spectrum density (PSD). If there are several sources of excitation, cross-correlation spectra are also needed. The output of the analysis is the PSD or RMS of any linear expression.

In the **Add Study** dialog, the new study type **Random Vibration (PSD)** is available. When selected, a set of three different studies is added to the model. Also, three nodes are created under **Global Definitions>Reduced-Order Modeling** for specifying the spectra.

STUDIES FOR REDUCED-ORDER MODELS

Three new studies for generation and simulation of reduced-order models (ROM) have been added to the **Add Study** dialog:

- **Time Dependent, Modal Reduced-Order Model**
- **Frequency Domain, Modal Reduced-Order Model**
- **Time Dependent, AWE Modal Reduced-Order Model**

When one of these studies is selected, a set of two or three different studies is added to the model. Also, two nodes are created under **Global Definitions>Reduced-Order Modeling** for providing the input to the ROM.

The old study **Modal Reduced-Order Model** has been removed from the **Add Study** dialog. It has been superseded by the more general **Time Dependent, Modal Reduced-Order Model**. It is, however, still possible to add a **Modal Reduced-Order Model** study step directly in the solver sequence if you want to access the old functionality.

RESPONSE SPECTRUM ANALYSIS

The response spectrum analysis capabilities that were introduced in version 5.4 are now also available with a MEMS Module license.

FLUID-STRUCTURE INTERACTION FOR TWO-PHASE FLOWS

In the **Add Physics** dialog, two new multiphysics interfaces for two-phase flows have been added: **Fluid-Solid Interaction, Two-Phase Flow, Phase Field** and **Fluid-Solid Interaction, Two-Phase Flow, Phase Field, Fixed Geometry**. These interfaces require also a license that contains the underlying two-phase flow interfaces; that is, the CFD Module or the Microfluidics Module.

ELASTIC WAVES, TIME EXPLICIT INTERFACE

The new **Elastic Waves, Time Explicit** interface makes it possible to analyze transient wave propagation problems in linear elastic materials under small deformations in a computationally efficient manner. In the **Add Physics** dialog, this interface is found under **Elastic Waves** in the **Acoustics** branch.

When the Acoustics Module is available, there is also a new multiphysics interface, **Acoustic-Solid Interaction, Time Explicit**, which combines an **Elastic Waves, Time Explicit** interface in solid domains with a **Pressure Acoustics, Time Explicit** interface in adjacent acoustic domains.

IMPROVEMENTS IN CONTACT MODELING

The contact modeling functionality has been extended and improved in many aspects.

- Contact can be modeled from a physics interface toward any meshed surface, even if that surface does not have any physics interface at all defined. This approach is particularly useful when modeling contact with a rigid shape.
- The more robust penalty method has replaced the augmented Lagrangian method as the default contact algorithm.
- The friction forces are now represented in a boundary coordinate system. As an effect, the number of friction degrees of freedom when using the augmented Lagrangian method is now two in 3D (previous version: three) and one in 2D (previous version: two).
- **Adhesion** and **Decohesion** are now two separate subnodes under **Contact**.
- When using Adhesion, the adhesive layer stiffness can now also be given using elastic material data by setting **Adhesive stiffness** to **Use material data**.
- In **Decohesion**, there are now new energy-based damage models. Also, you can now include a viscous regularization in time-dependent analysis in order to mitigate the inherent instable behavior when the material is losing stiffness.

- New default plots showing arrows for contact pressure and friction force are generated when a **Contact** node is present.
- The selection of pairs in the **Contact** node is more intuitive.

In a model created in a previous version, the old type of **Contact** node will still be kept. If you add another **Contact** node, it will be of the new type.

SOLID MECHANICS ON ROTATING DOMAINS

When modeling multiphysics problems in systems where there is a mixture of rotating and stationary domains, it is now possible to use the Solid Mechanics interface in a corotating configuration, so that only the deformations relative to the rotation are modeled. This is far more efficient than solving for the total displacements, including large global rotations. To this end, the **Rotating Frame** feature has been extended so that it can provide a superposition of the rigid body rotation and the relative displacements for controlling the spatial frame.

NEW DEFAULT SCALE FACTOR FOR EIGENMODES

In the **Eigenvalue Solver** node within the solver sequence, a new option for scaling eigenmodes has been added. This option is now used by all structural mechanics interfaces to ensure that eigenmode amplitudes are small relative to the size of the geometry. By doing this, it ensures that variables that are nonlinear by their nature will be evaluated using small perturbations to the original geometry.

MODAL PARTICIPATION FACTORS TABLE

If a **Participation Factors** node is present under **Definitions** when an eigenfrequency study is run, an evaluation group named **Participation Factors** is automatically generated. It contains a table with the translational and rotational participation factors for all computed eigenfrequencies.

NEW DEFAULT FOR TEMPERATURE-DEPENDENT PROPERTIES

When material properties have a temperature dependence, the default in the **Model Inputs** section is now to take the temperature from **Common model input**. In versions prior to 5.5, the default was **User defined.** In both cases, the default temperature is room temperature, as long as temperatures are not computed by another physics interface. With the new formulation, the temperature will automatically be picked up if defined under **Default Model Inputs** by a Heat Transfer in Solids interface, for example. The settings used in a model created in an earlier version are not affected.

THE TEMPERATURE COUPLING NODE IS OBSOLETE

The **Temperature Coupling** multiphysics node is no longer used for the transfer of temperatures from heat transfer interfaces to structural mechanics interfaces. Instead, the common model input mechanism is used for transferring the temperature. In models created prior to version 5.5, the **Temperature Coupling** node is still present and used. A warning message is however shown.

In all nodes where temperature-dependent properties are referenced, the default value for the temperature in the **Model Inputs** section is now **Common model input**.

SYNCHRONIZATION OF ELASTIC PROPERTIES

If isotropic material data (for example, E and ν) are present in a **Material**, then this material can also be used if the **Solid model** in a linear elastic material is set to **Orthotropic** or **Anisotropic**. Similarly, orthotropic material data can be used when **Solid model** is set to **Anisotropic**.

ENHANCEMENTS FOR CELL PERIODICITY

The **Cell Periodicity** node in the Solid Mechanics interface has a number of improvements:

- It is now available also in 2D. If the generalized plane strain formulation is used, then it is also possible to generate a global **Material** node, in the same manner as in the 3D case.
- One separate global **Material** node is created for each **Cell Periodicity** node. This makes it possible to evaluate several RVE cells in the same study.
- When using **Cell Periodicity**, a default evaluation group is generated for displaying the computed material properties. This group contains a **Global Matrix Evaluation** node where the elasticity or compliance matrix is shown.
- The **Create** button has been replaced by icons in a section toolbar and split into separate operations for study generation and material node generation.

MAGNETOSTRICTION WITH MAGNETIC HYSTERESIS

The **Magnetostrictive Material** has been extended to include the Jiles–Atherton model for magnetic hysteresis.

ENHANCEMENTS FOR EXTERNAL MATERIALS

When implementing your own material using **External Stress-Strain Relation** or **External Strain**, you can now pass back your own warning and log messages, which are displayed in the GUI.

Another improvement is that the limit for the number of parameters that can be passed to an external material has been increased significantly.

When using the **External Stress-Strain Relation** for a plane stress Solid Mechanics interface in 2D, additional degrees of freedom for the transverse shear strains can be activated by selecting the **Allow transverse shear strains** check box. This is needed for some type of anisotropies in the material model.

NEW SETTINGS FOR ROLLER IN SOLID MECHANICS

The settings in the **Roller Constraint** section in the **Roller** node have been changed for 2D and 2D axisymmetry in order to reflect the limitations and interpretations that the respective space dimensions impose.

WAVE SPEED COMPUTATION IN ELASTIC MATERIALS

For the **Linear Elastic Material** and **Piezoelectric Material**, it is now possible to compute wave speeds for elastic waves. To do that, add the **Wave Speeds** node, available under **Variables** in the context menu for the material models or under **Attributes** on the ribbon when the material model is selected.

The effect of this node is to make variables for phase and group velocities for elastic waves available for result presentation. The solution as such is not affected, and the variables will be available for any study type.

THERMAL STRESS MULTIPHYSICS INTERFACE RENAMED

The **Thermal Stress** multiphysics interface has been renamed to **Thermal Stress, Solid** and is now located inside a group named **Thermal Stress** in the **Add Physics** dialog. The reason is that several other similar multiphysics interfaces have been added. The interfaces that you actually see in the **Thermal Stress** group depend on the available licenses.

COROTATING LOADS ON RIGID DOMAIN

When applying loads in a **Rigid Domain** node in the Solid Mechanics interface through the **Applied Force** subnode, it is now possible to specify that the load direction follows the rotation of the body. The new **Direction** setting has two options: **Space-fixed direction** and **Body-fixed direction**.

PULL-IN AND PULL-OUT ANALYSIS OF A BIASED RESONATOR

An electrostatically actuated MEMS resonator is simulated. The device is driven by an AC + DC bias voltage applied across a parallel plate capacitor. In this example, the pull-in and pull-out voltages of the resonator are computed. This is done via a quasistatic analysis of the displacement-voltage trajectory of the full range of motion, from the initial relaxed state all the way to the pulled-in flattened state.

ISOTROPIC-ANISOTROPIC SAMPLE: ELASTIC WAVE PROPAGATION

In this 2D tutorial model, a test sample consists on one side of an isotropic material and on the other side of a heterogeneous anisotropic material (a transverse anisotropic zinc crystal). Elastic waves in the sample are excited by a point-like force. The model is solved with the **Elastic Waves, Time Explicit** physics interface.

GROUND MOTION AFTER SEISMIC EVENT: SCATTERING OFF A SMALL MOUNTAIN

In this tutorial, the propagation of elastic waves is simulated using a 2D model. The effect of the surface topology on the wave propagation is illustrated when an ideal half space is modified with the presence of a small mountain. The model is a variation of Lamb's problem. The propagation of the elastic waves is modeled using the **Elastic Waves, Time Explicit** physics interface and the model captures the propagation and scattering of pressure waves, shear waves, Rayleigh waves, and von Schmidt waves.

PIEZOELECTRICITY IN A LAYERED SHELL

This is a tutorial example showing how to model piezoelectric applications using the layered shell functionality. Two cases of material orientation are studied. In the first case, the pole axis is normal to the shell surface, which results in a thickness change. In the second case, the pole axis is in the plane of the shell, which leads to bending.

SHOCK RESPONSE OF A MOTHERBOARD

Electronic equipment often has to be certified to function after having been subjected to a specified shock load. In this example, the effect of a 50-g, 11-ms half-sine shock on a circuit board is investigated using response spectrum analysis. The results are compared with a time-domain analysis using mode superposition.

Backward Compatibility with Version 5.2

The friction slip velocity is no longer defined as a dependent variable. When running a model using the COMSOL API, you will need to remove the reference to this variable in the solver settings. The friction slip velocity is usually defined as `<comp>_<solid>_vslip_<pairname>`, where `<comp>` is the tag of the component, `<solid>` is the tag of the Solid Mechanics physics interface where the **Friction** node is defined, and `<pairname>` is the name of the contact pair.

Microfluidics Module

New Models in Version 5.5

New models are available showcasing shape and parameter optimization. These models require the Optimization Module. Specifically:

- The Shape Optimization of a Tesla Microvalve model makes use of the new shape optimization tools to optimize the flow through a microvalve.
- The Parameter Optimization of a Tesla Microvalve model make use of the parameter optimization to efficiently place obstructions in a microvalve to optimize the forward vs. reverse flow rate.

Backward Compatibility with Version 5.3a

NO FLUX FORMULATION

The formulation of the No Flux feature has been changed in the Transport of Diluted Species and Transport of Diluted Species in Porous Media interfaces in version 5.4 so that the flux relative to the convective flux is set to zero. The new formulation is used when opening a model from a previous session. In the previous formulation, the total flux was set to zero.

Mixer Module

New Functionality in Version 5.5

ROTATING MACHINERY, TWO-PHASE FLOW, LEVEL SET, AND PHASE FIELD INTERFACES

The two new physics areas, Rotating Machinery, Level Set and Rotating Machinery, Phase Field contain physics interfaces for tracking the interface between two immiscible fluids in geometries with one or more rotating parts. Both laminar and turbulent (RANS) flow is supported. Sliding mesh functionality is applied to enforce continuity of mass, momentum, and the level set or phase field function between rotating and stationary domains.

SUPPORT FOR FROZEN ROTOR STUDY STEP AND CONTINUITY PAIR FEATURE IN EULER-EULER MODEL AND BUBBLY FLOW INTERFACES

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

Molecular Flow Module

Backward Compatibility with Version 5.0 and Earlier

Due to the fact that multiple species are now supported, some variable names have changed. In general, variables will now have `_G` appended to their names, where `G` is the name specified for the **Incident molecular fluxes** in the **Dependent Variables** section. So, if a model is updated and resolved, the expressions used in results should be changed as follows:

- The number density should be changed from `fmf.N` to `fmf.N_G`.
- The pressure should be changed from `p` to `fmf.p_G`.
- The reconstructed number density should be changed from `fmf.N` to `fmf.ndr1.Nr_G`, where `ndr1` is the tag of the **Number Density Reconstruction** feature.
- Any other variables can be updated using the **Replace Expression** button available during results processing.

In addition, new variables exist for the total pressure, number density, and so on:

- The total number density, `fmf.ntot`.
- The total pressure, `fmf.ptot`.
- The total incident molecular flow, `fmf.Gtot`.

Multibody Dynamics Module

New Functionality in Version 5.5

CHANGE IN LICENSING STRUCTURE

From version 5.5, the Multibody Dynamics Module is a direct add-on to COMSOL Multiphysics. The module contains two physics interfaces: Multibody Dynamics and Lumped Mechanical Systems. In previous versions, the Multibody Dynamics Module also required the Structural Mechanics Module.

Thus, for basic multibody dynamics modeling, it is no longer necessary to have a Structural Mechanics Module license. More advanced modeling techniques may still indirectly require such a license, for example, when combining the Multibody Dynamics interface with the Shell or Beam interfaces. Also, nonlinear material models require also the combination of the Structural Mechanics Module (or MEMS Module) and the Nonlinear Structural Materials Module.

ROLLER CHAIN SPROCKET MODELING

Roller chain sprocket modeling functionality has been added. The new **Chain Drive** node in the Multibody Dynamics interface provides a framework for setting up such models. By selecting a chain geometry part in the chain drive node, all the relevant selections are picked up automatically. Sprockets and chain links can be defined as being rigid or elastic independently. Using the **Create Links and Joints** button, all the required **Rigid Domain**, **Attachment**, and **Hinge Joint** nodes are added automatically. There are two methods for modeling contact between sprocket and chain links; roller center based and mesh based. The first one is faster, but works only with rigid links, whereas the second one is more robust and uses a standard **Contact** node. It is also possible to model bushings and losses in the joints.

NEW GEOMETRY PARTS: ROLLER CHAIN SPROCKET

New geometry parts have been added for modeling roller chain sprocket assemblies. Parts are available for the creation of an individual component as a sprocket or a roller chain as well as a full roller chain sprocket assembly. In order to make the model setup fast and easy, the parts are highly parameterized; make use of selection color functionality; and provide selections such as links, joint boundaries, and contact boundaries to the physics interface.

AUTOMATIC RIGID DOMAIN CREATION FROM GEOMETRY OBJECTS

This functionality makes the model setup easier for large rigid multibody systems. A new section, **Automated Model Setup**, has been added in the Multibody Dynamics interface. This section provides buttons to automatically generate physics nodes based on geometric information. Using the **Create Rigid Domains** button, rigid domain nodes are generated for geometrically disconnected objects either on the entire physics selection or on a subset selection. It is also possible to add **Mass and Moment of Inertia** subnodes automatically.

AUTOMATIC GEARS CREATION FROM GEOMETRY PARTS

In the new section **Automated Model Setup** in the Multibody Dynamics interface, it is also possible to create gears from geometric information. Using the **Create Gears** button, different gear nodes are generated corresponding to each gear part present in the geometry. All the gear parameters (for instance, number of teeth, pitch diameter, pressure angle, gear center, and gear axis) are automatically entered in the settings for the auto-generated gear nodes.

IMPROVEMENTS IN CONTACT MODELING

The contact modeling functionality has been extended and improved in many aspects.

- Contact can be modeled from a physics interface toward any meshed surface, even if that surface does not have any physics interface at all defined. This approach is particularly useful when modeling contact with a rigid shape.
- The more robust penalty method has replaced the augmented Lagrangian method as the default contact algorithm.
- The friction forces are now represented in a boundary coordinate system. As an effect, the number of friction degrees of freedom when using the augmented Lagrangian method is now two in 3D (previous version: three) and one in 2D (previous version: two).
- **Adhesion** and **Decohesion** are now two separate subnodes under **Contact**.
- When using Adhesion, the adhesive layer stiffness can now also be given using elastic material data by setting **Adhesive stiffness** to **Use material data**.
- In **Decohesion**, there are now new energy-based damage models. Also, you can now include a viscous regularization in time-dependent analysis in order to mitigate the inherent instable behavior when the material is losing stiffness.

- New default plots showing arrows for contact pressure and friction force are generated when a **Contact** node is present.
- The selection of pairs in the **Contact** node is more intuitive.

In a model created in a previous version, the old type of **Contact** node will still be kept. If you add another **Contact** node, it will be of the new type.

NEW DEFAULT SCALE FACTOR FOR EIGENMODES

In the **Eigenvalue Solver** node within the solver sequence, a new option for scaling eigenmodes has been added. This option is now used by all structural mechanics interfaces to ensure that eigenmode amplitudes are small relative to the size of the geometry. By doing this, it ensures that variables that are nonlinear by their nature will be evaluated using small perturbations to the original geometry.

MODAL PARTICIPATION FACTORS TABLE

If a **Participation Factors** node is present under **Definitions** when an eigenfrequency study is run, an evaluation group named **Participation Factors** is automatically generated. It contains a table with the translational and rotational participation factors for all computed eigenfrequencies.

NEW DEFAULT FOR TEMPERATURE-DEPENDENT PROPERTIES

When material properties have a temperature dependence, the default in the **Model Inputs** section is now to take the temperature from **Common model input**. In versions prior to 5.5, the default was **User defined.** In both cases, the default temperature is room temperature, as long as temperatures are not computed by another physics interface. With the new formulation, the temperature will automatically be picked up if defined under **Default Model Inputs**; for example, by a Heat Transfer in Solids interface. The settings used in a model created in an earlier version are not affected.

THE TEMPERATURE COUPLING NODE IS OBSOLETE

The **Temperature Coupling** multiphysics node is no longer used for the transfer of temperatures from heat transfer interfaces to structural mechanics interfaces. Instead, the common model input mechanism is used for transferring the temperature. In models created prior to version 5.5, the **Temperature Coupling** node is still present and used. A warning message is however shown.

In all nodes where temperature-dependent properties are referenced, the default value for the temperature in the **Model Inputs** section is now **Common model input**.

COROTATING LOADS ON RIGID DOMAIN

When applying loads in a **Rigid Domain** node through the **Applied Force** subnode, it is now possible to specify that the load direction follows the rotation of the body. The new **Direction** setting has two options: **Space-fixed direction** and **Body-fixed direction**.

NEW SETTINGS FOR ROLLER IN 2D

The settings in the **Roller Constraint** section in the **Roller** node in 2D have been changed in order to reflect the limitations and interpretations that the space dimension impose.

New Models in Version 5.5

STRESS ANALYSIS OF A ROLLER CHAIN SPROCKET ASSEMBLY

This example demonstrates a stress analysis of an elastic chain sprocket assembly. The system consists of a roller chain wrapped around two sprockets in 3D. All the components of the system are assumed to be elastic bodies. An angular velocity is prescribed on the driver sprocket, and load torque is applied on the driven sprocket. A transient analysis is performed to understand the load path, contact forces, and stress distribution in various components of the chain drive.

DYNAMICS OF A ROLLER CHAIN SPROCKET ASSEMBLY

This example simulates the dynamics of a chain sprocket assembly. The system consists of a roller chain wrapped around two sprockets in 2D. The roller chain is constructed by assembling rigid roller plates and pin plates in such a way that the relative rotation between chain links is unrestricted. This chain is used to transmit rotational motion between the two sprockets. An angular velocity is prescribed on the driver sprocket and a load torque is applied on the driven sprocket. Using a transient study, the dynamics of the system are analyzed for two cases: when the driven shaft is loaded and free. Comparisons of chain link motion, contact forces, and other parameters are made.

SHAFT VIBRATION DUE TO GEAR RATTLE AND BEARING MISALIGNMENT

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

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

ELECTRODYNAMICS OF A POWER SWITCH — MULTIBODY VERSION

Electrical events, such as an overcurrent or overload, can seriously damage electrical circuits or power lines. To avoid expensive replacements of critical parts, electric switch circuit breakers can be installed. These mechanically interrupt the current flow or surge by moving a plunger as soon as a critical current is reached. In contrast to a fuse, which has to be replaced after it has been activated to protect the surrounding electrical components, a circuit breaker can be reset. The model simulates rigid body dynamics of the system under the influence of magnetic forces, induced currents, and different spring/constraint arrangements.

Nonlinear Structural Materials Module

New Functionality in Version 5.5

NONLINEAR MATERIAL MODELS IN THE SHELL INTERFACE

Three families of nonlinear material models have been added to the Shell interface:

- Plasticity
- Creep
- Viscoplasticity

These material models are added as subnodes to the **Layered Linear Elastic Material**. If the Composite Materials Module is available, the material models can also be used in multilayered shells, and the individual layers can have different material models.

NEW MATERIAL MODELS IN THE LAYERED SHELL INTERFACE

Hyperelastic Material has been added as a new top-level material family in the Layered Shell interface. It contains the same hyperelastic material models (such as Neo-Hookean and Mooney-Rivlin) as in the Solid Mechanics interface. It is also possible to include viscoelasticity and the Mullins effect.

In the **Linear Elastic Material** in the Layered Shell interface, it is now possible to add also **Plasticity**. The same set of yield functions and hardening models as in the Solid Mechanics interface are available. The formulation assumes small plastic strains, which give a good approximation up to 10–20% strain.

The Layered Shell interface requires the Composite Materials Module.

NONLINEAR MATERIAL MODELS IN THE MEMBRANE INTERFACE

In the Membrane interface, there is a new top-level material model, **Layered Linear Elastic Material**. To this node, you can add subnodes for three families of nonlinear materials:

- Plasticity
- Creep
- Viscoplasticity

If the Composite Materials Module is available, multilayered thin membranes can be modeled, and the individual layers can have different material models. This is the primary use of the **Layered Linear Elastic Material** and its subnodes.

Since previous versions, the nonlinear material models are available for the standard **Linear Elastic Material** in the Membrane interface.

SHAPE MEMORY ALLOY IMPROVEMENTS

In the Lagoudas material model for shape memory alloys, there are several enhancements:

- A stress-dependent maximum transformation strain has been introduced. This makes it possible to represent the physical observation that, at low stress levels, it is not possible to reach the maximum transformation strain. This effect is called two-way shape memory effect (TWSME).
- A stress hardening term has been added to represent the variation in the size of the transformation hysteresis loop with the stress level. The effect is that different slopes can be observed in phase diagrams.
- A new optional **Phase Transformation Direction** subnode has been added. In many cases, the direction of the transformation during the process is known. It is then possible to improve the convergence significantly by having the transformation direction prescribed rather than computed. In the previous version, there was a similar option, but it could only be applied to the structure as a whole, and not to individual domains.

JOHNSON-COOK MODEL FOR STRAIN-RATE-DEPENDENT PLASTICITY

The Johnson-Cook model is used for describing the effect of high strain rates on the plastic hardening. Effects of temperature softening can also be included. This hardening model is available as an isotropic hardening model in the **Plasticity** node and for the Chaboche and Perzyna models in the **Viscoplasticity** node.

IMPROVED PERFORMANCE FOR PLASTICITY

The speed of plasticity computations have been improved with up to 20% for several of the more common plasticity models.

TWISTING AND BENDING OF A METAL FRAME

A thin-walled metal frame with a central cutout is subjected to twisting and bending. The stresses around the cutout exceed the yield limit, and an elastoplastic analysis is performed in order to find the residual stresses and strains after unloading. The new possibility to model plasticity in the Shell interface is used.

PRESSURIZED ORTHOTROPIC CONTAINER — SHELL VERSION

In this example, plasticity in a thin structure is modeled using the Shell interface. It is a variant of the already existing example Pressurized Orthotropic Container, in which the same structure is studied using the Solid Mechanics interface.

A thin-walled container made of rolled steel is subjected to an internal overpressure. As an effect of the manufacturing method, the out-of-plane direction has a higher yield stress than the other two directions. Hill's orthotropic plasticity is used to model the difference in yield strength.

TENSILE TEST WITH STRAIN-RATE-DEPENDENT PLASTICITY

In this example, a tensile test is simulated at four different strain rates. The Johnson-Cook hardening law is used to model the strain-rate dependency of the plastic hardening.

The temperature distribution and thermal expansion caused by the heating generated by the plastic deformation are also computed. In a separate study, the influence of including a temperature dependency in the Johnson-Cook model is examined.

Optimization Module

New Functionality in Version 5.5

SHAPE OPTIMIZATION WITH BUILT-IN REGULARIZATION

It is now easier to set up shape optimization problems, because the process of adding control variables and using them for deforming geometry can be set up automatically using the **Free Shape Domain**, **Free Shape Boundary**, and **Polynomial Boundary** features. The features are available under **Definitions>Shape Optimization**. Shells can be optimized using the **Free Shape Shell** feature, and all of the features can be combined with the **Symmetry/Roller** to constrain points or edges to slide along boundaries.

TOPOLOGY OPTIMIZATION

The Filter dataset can be used to transfer the results of a topology optimization to a new component for a verification analysis. The process is more robust than exporting and importing a STL file, and setting up the new component is easier, because the selections from the old component can be recycled.

The **Density Model** feature introduced in version 5.4 fixes the material volume factor to 1 outside the selection of the **Density Model**. In version 5.5, the **Prescribed Density** feature can be used to specify a value, θ_{fix}, for the material volume factor, and if the **Density Model** uses Helmholtz filtering, the filter will be extended using θ_{fix} as the input to the filter.

STRICT ENFORCEMENT OF DESIGN CONSTRAINTS

Design constraints are constraints that can be evaluated without computing the dependent variables. These constraints can be used to prevent topology changes to the geometry, but unless the constraints are strictly enforced, the topology might change anyway. Strict enforcement of design constraints prevents evaluation of the geometry for infeasible design constraints. It is available for COBYLA as well as for coordinate search and Nelder-Mead, if they use the penalty constraint handling method.

CONFIDENCE INTERVALS FOR LEAST-SQUARES OBJECTIVE

The Levenberg-Marquardt solver can compute the confidence intervals of the control parameters. The interval is given in the parameter units in the log and in a table.

PARAMETRIC SWEEP OVER GRADIENT-BASED OPTIMIZATION

A parametric sweep can be added over a gradient-based optimization study step.

New and Updated Models in Version 5.5

- The *Shape Optimization of a Shell* model demonstrates how to perform shape optimization on shells using the new Free Shape Shell feature. The model uses geometric nonlinearity.
- The *Shape Optimization of a Wrench* model increases the stiffness of a wrench without increasing the mass. The model achieves this using shape optimization with the Free Shape Domain and Free Shape Boundary features.
- The *Shape Optimization of a Tesla Microvalve* model has an initial geometry that is inspired from the *Topology Optimization of a Microvalve* model. This model uses the Polynomial Boundary and Symmetry/Roller features to improve the pressure drop ratio (the diodicity), which is a critical performance metric for this type of device.
- The *Shape Optimization of an MBB Beam* model imports the optimization result from the T*opology Optimization of an MBB Beam* model and uses the Free Shape Boundary and Symmetry/Roller features to improve on the result.
- The *Optimization of an Extruded MBB Beam* model illustrates how to perform topology and shape optimization for problems that have to be modeled in 3D but where it also has to be possible to extrude the geometry. This model uses the Density Model feature for the topology optimization and equation-based modeling for the shape optimization. Extrusion operators are used to ensure the desired property for the geometry.
- The *Optimization Tutorial* application is a learning tool that illustrates the properties of the COMSOL Optimization solvers by making it possible for you to test them on various benchmark problems with two control variables. The optimization history is plotted in the optimization function landscape, and each problem comes with a small explanation.
- The *Maximizing the Buckling Load of a Diagonal Brace* model demonstrates how a linear buckling analysis can be combined with the optimization study step to increase the buckling load of a shell structure while constraining the mass of the structure.
- The *Parameter, Shape, and Topology Optimization of a Beam* model is a tutorial model that considers a structural mechanics problem and demonstrates how to use all three types of design optimization available with the Optimization Module.

- The *Impedance Tube Parameter Estimation with Data Generation* model illustrates how to model poroacoustics in an impedance tube. Noise is added to a generated frequency response so that the then noisy response can be used as synthetic data for parameter estimation of the poroacoustic properties.
- The *Shape Optimization of an Acoustic Demultiplexer* model demonstrates how to use shape optimization with the Free Shape Boundary feature to achieve demultiplexing of two acoustic frequency bands. The device has three ports, one input and two outputs. The objective is that the acoustic energy should go to different output ports depending on the frequency. Each frequency band is approximated with 5 frequencies, giving a total of 10 objective functions, which are combined in a min-max problem. This problem is solved using the MMA optimization method. The initially circular cavities are deformed such that the worst objective is better than 40 dB.

Backward Compatibility with Version 5.4

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

Particle Tracing Module

New Functionality in Version 5.5

GENERAL SPEEDUP

Some particle tracing models are significantly faster than in version 5.4 and earlier. There are two main factors in this speedup:

- When exerting forces on particles that depend on another field, such as the **Electric Force** or the **Drag Force**, the coupled field is excluded from Jacobian contributions whenever possible. In this case, the speedup is most significant when the coupled field has a large number of degrees of freedom, which may occur if the field is solved for on a very fine mesh.
- The default behavior of the **Time-Dependent Solver** for particle tracing is now to select **Minimal** from the **Jacobian Update** list. The previous behavior was **Once per time step**. Thus, the Jacobian is updated less often than before. It is not necessary to update the Jacobian at every time step because the handling of algebraic errors when solving for the particle trajectories has been made more robust in version 5.5.

VIRTUAL MASS AND PRESSURE GRADIENT FORCES

A built-in option to add the virtual mass (or added mass) and pressure gradient forces is now available in the Particle Tracing for Fluid Flow interface. In the settings for the **Drag Force**, select the check box **Include virtual mass and pressure gradient forces**.

ACOUSTOPHORETIC FORCE IMPROVEMENTS

New options are available for the **Acoustophoretic Force,** which has also been renamed to **Acoustophoretic Radiation Force**. The new settings include more accurate treatment of both solid particles and liquid drops, as well as thermodynamic loss models that take into account the viscous and thermal boundary layers around the particles.

Backward Compatibility

The old options to specify the **Particle bulk modulus** and **Particle compressibility** have been removed. Now, the speed of sound will always be specified.

NEW OPTION TO SPECIFY PARTICLE DIAMETER

You can now release polydisperse particles by either entering a distribution of particle masses or a list of particle diameters. Select the appropriate option from the **Particle size distribution** list, which replaces the **Compute particle mass** check box.

PREVIEW INITIAL COORDINATES IN RELEASE FROM GRID

When using the **Release from Grid** feature to release particles, you can view the initial particle positions before running the study. Click the **Preview Initial Coordinates** button to show the initial positions as points, or click the **Preview Initial Extents** button to show a bounding box containing the initial positions.

ISOTROPIC SCATTERING WALL CONDITION

You can now select **Isotropic scattering** from the **Wall condition** list in the settings for the **Wall** node. This applies random reflection to the particles that hit the wall, like the **Diffuse scattering** option, but does not follow the cosine law.

RELEASE SECONDARY PARTICLES WITH USER-DEFINED SPEED

When releasing secondary particles during a particle-wall interaction with the **Secondary Emission** subnode, you now have the option to initialize the particle speed so that it is equal to the incident particle speed, inversely proportional to the number of released secondary particles, or user defined. The **User defined** option is new in version 5.5.

BUILT-IN SPECIES FOR CHARGED PARTICLE TRACING

In the **Particle Properties** node for the Charged Particle Tracing interface, you can now choose from several built-in species, including electrons, protons, neutrons, alpha particles, and positrons. These choices, along with **User defined**, are available from the **Particle species** list.

Important Fixes in Version 5.5

DEFINITION OF THE KNUDSEN NUMBER FOR RAREFACTION MODELS

In the Particle Tracing for Fluid Flow interface, it is possible to apply corrections to the **Drag Force** for highly rarefied gases by first selecting the **Include rarefaction effects** check box. In version 5.4 and earlier, the rarefaction models **Basset**, **Epstein**, and **Phillips** used an inconsistent definition of the particle Knudsen number. After a literature review, it was determined that the original references for these models intended for the Knudsen number to be defined as the ratio of gas mean free path to particle radius, whereas some subsequent authors misinterpreted them to define the Knudsen number as the ratio of mean free path to diameter. Therefore, using these rarefaction models in version 5.5 will give a different result compared to version 5.4 and earlier.

MORE ROBUST DIFFUSE SCATTERING ON CURVED INTERIOR SURFACES

The **Diffuse scattering** wall condition now uses the normal and tangential directions computed on the boundary mesh element, rather than the directions computed on the underlying geometric entity, when reinitializing particle velocity. This reduces the likelihood that particles will fail to locate a subsequent wall interaction when they are reflected in a direction nearly tangent to the interior of a concave surface. In some cases, it may still be necessary to switch to a linear geometry shape order to prevent some particles from disappearing.

USER-DEFINED RAREFACTION MODEL

An error with the **User defined** rarefaction correction for the **Drag Force** in the Particle Tracing for Fluid Flow interface, in which a text field for the correction factor did not appear, has been fixed.

SPACE CHARGE LIMITED EMISSION WITH AUXILIARY DOFS

In version 5.4, using the **Space Charge Limited Emission** multiphysics coupling might give an error message if the coupled Charged Particle Tracing interface included additional degrees of freedom, such as out-of-plane position or collision counters. This has been fixed.

LABELING OF THE AZIMUTHAL AND POLAR ANGLES

In version 5.5, the labeling of azimuthal and polar angles in spherical coordinate systems has been made more consistent. Now, θ is used for the polar angle and ϕ is used for the azimuthal angle.

MASS DISTRIBUTION WITH COMPUTE MASS FLOW RATE OPTION

A n error in which the released particle mass could not be initialized in the Particle Tracing for Fluid Flow interface if the options **Specify mass flow rate** and **Compute particle mass** were used together has been fixed. As a side effect of this bug fix, the number of degrees of freedom solved for is expected to slightly increase under these circumstances, while the number of static degrees of freedom should decrease.

TANGENTIAL AND NORMAL VELOCITY COMPONENTS IN INLETS

Fixed an error in the **Inlet** feature for the Particle Tracing for Fluid Flow interface in which the initial velocity would be interpreted in global Cartesian coordinates even if the check box **Specify tangential and normal vector components** was selected.

VECTOR DISPLAY FOR THE PARTICLE BEAM FEATURE

The **Particle Beam** feature in the Charged Particle interface more consistently releases particles in the direction shown by the arrow in the Graphics window. Previously, particles would sometimes be released in the opposite direction if the **Particle Beam** selection included interior boundaries.

DEFAULT DATASET GENERATION

The generation of default **Particle** datasets has been made more robust for cases when the study contains multiple study steps.

NULL COLLISION METHOD STABILITY IMPROVEMENT

The Null collision method, which can be used with the **Collisions** feature in the Charged Particle Tracing interface, has been made more robust. It no longer reinitializes particle coordinates incorrectly when the **Store particle status data** or **Store particle release statistics** check boxes have been selected.

RELEASE STATISTICS WHEN USING THE NOZZLE FEATURE

Fixed an error in the calculation and storage of particle release statistics, like total number of particles released, when using the **Nozzle** feature in the Particle Tracing for Fluid Flow interface.

Backward Compatibility with Version 5.4

REMOVED THE PARTICLE TRAJECTORIES STUDY STEP

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

Backward Compatibility with Version 5.3 and Earlier

NOMENCLATURE CHANGES FOR MULTIPLE PARTICLE SPECIES

The **Override Properties** feature has been renamed to **Particle Properties** to emphasize that it can be used to release multiple species of particles in the same model. The list **Inherit Properties** in the settings for most particle release features, such as **Inlet** and **Release from Grid**, has been renamed to **Released particle properties**.

TURBULENT DISPERSION CHANGES

The turbulent dispersion models used by the **Drag Force** in the Particle Tracing for Fluid Flow interface have been modified and a deprecated turbulent dispersion model has been removed.

The option **Discrete random walk, variable time step** has been removed from the **Turbulent dispersion model** list. When an old model using this obsolete option is loaded in version 5.3a, the **Discrete random walk** option is now selected. However, **Discrete random walk** actually behaves more like the old **Discrete random walk, variable time step** model because it only samples the turbulent velocity perturbation at discrete time intervals based on the eddy lifetime or eddy crossing time in the flow.

CHANGES TO RELEASE FROM EDGE AND POINT

In the Particle Tracing for Fluid Flow interface, the **Release from Edge** and **Release from Point** features now allow the initial particle velocity to be selected from a list containing previously computed velocity fields, similar to the behavior of the **Release** and **Inlet** features. This makes the **Release from Edge** and **Release from Point** features easier to couple with another physics interface that models the fluid flow.

CHANGES TO PERIODIC ELECTRIC AND MAGNETIC FORCES

In models built in version 5.3 or earlier, if an **Electric Force** or **Magnetic Force** was present and the **Multiply force by phase angle** check box was selected, then in version 5.3a, **Time harmonic** will be selected from the new **Time dependence of field list**, which replaces this check box.

Backward Compatibility with Version 5.2a and Earlier

REFERENCE EDGE FEATURES HAVE BEEN REMOVED

The **Reference Edge** subnode, which was previously used to specify the orientation of the transverse beam directions for the **Particle Beam** feature in 3D, has been removed. Instead, the orientation of the transverse beam directions is specified using an extra selection field in the settings window for the **Particle Beam** feature itself. Similarly, the **Reference Point** subnode for specifying the beam center has been replaced by an extra selection in the settings window for the **Particle Beam** feature.

Backward Compatibility with Version 5.2

TURBULENT DISPERSION CHANGES

In the settings window for the **Drag Force** for the Particle Tracing for Fluid Flow interface, the **Turbulent dispersion** check box has been replaced by the **Turbulent dispersion model** list. If the check box is selected in a model created in version 5.2 or earlier, then after opening the model in version 5.2a, the option **Discrete random walk** will be selected from the list. If the check box is cleared, then **None** will be selected from the list.

PARTICLE DENSITY SPECIFICATIONS MERGED

In version 5.2, when specifying particle properties using the **Particle Properties** node or **Override Properties** node in the Particle Tracing for Fluid Flow interface, it is possible to specify two different values of the particle density, depending on whether the **Compute particle mass** check box is selected. In version 5.2a, these two values have been merged into a single parameter value; that is, the value or expression for the **Particle density** will be retained when selecting or clearing the **Compute particle mass** check box in the physics interface **Advanced Settings** section.

PARTICLE BEAM CHANGES

The names of several settings have been changed in the **Particle Beam** feature for the Charged Particle Tracing interface.

In the **Sampling from phase space distribution** list, **Uniform** has been renamed **KV**.

In the **Longitudinal velocity distribution** list, **Uniform** has been renamed **None** and **Gaussian** has been renamed **Normal**.

The parameters **Position refinement factor** and **Release distribution accuracy order** have been removed. Due to some changes in the algorithm for initializing particle positions, in version 5.2a, the distribution of particle positions no longer depends on the finite element mesh and may differ from the distribution in version 5.2.

Backward Compatibility with Version 5.1

RENAMED PHYSICS INTERFACE SETTINGS

The **Release type** list in the settings windows for the Charged Particle Tracing and Particle Tracing for Fluid Flow interfaces in earlier versions has been renamed **Particle release specification**. For the Charged Particle Tracing interface, the options **Static** and

Transient have been renamed to **Specify current** and **Specify release times** (the default), respectively. For the Particle Tracing for Fluid Flow interface, the options **Static** and **Transient** have been renamed to **Specify mass flow rate** and **Specify release times** (the default), respectively.

Backward Compatibility with Version 5.0 and Earlier

OLD NODES FOR PARTICLE-FIELD AND FLUID-PARTICLE INTERACTIONS ARE OBSOLETE

The old **Particle-Field Interaction** and **Fluid-Particle Interaction** nodes, which could be added directly to the Charged Particle Tracing and Particle Tracing for Fluid Flow interfaces, respectively, are now obsolete. They will be removed in a future version. Simulations of particle-field and fluid-particle interactions should instead use the dedicated multiphysics nodes.

ELASTIC COLLISION FORCE NODE HAS BEEN REPLACED

If a model created in version 5.0 or earlier containing the **Elastic Collision Force** node is opened in version 5.1 or later, this node will be replaced. If the **Collision model** in the old **Elastic Collision Force** node was set to **Monte Carlo**, a **Collisions** node with the **Elastic** subnode will be created. If instead the **Collision model** was set to **Friction**, the **Friction Force** subnode will be added.

Pipe Flow Module

New Functionality in Version 5.5

PIPE MECHANICS INTERFACE

The new Pipe Mechanics interface is intended for structural analysis of pipes. The interface is in many respects similar to the Beam interface in the Structural Mechanics Module, with important modifications necessary for the analysis of pipes. For instance:

- Stresses, strains, and axial deformations caused by internal overpressure are taken into account.
- The default material model is **Fluid and Pipe Materials**, reflecting that both the material in the pipe wall and the fluid are important.
- The **Pipe Cross Section** node has more and slightly different input parameters when compared to the corresponding **Cross Section Data** node in the Beam interface.
- A special type of load, **Fluid Load**, is included. There, you enter pressure and drag forces. These values can be taken directly from a Pipe Flow interface.
- A temperature gradient can be present through the pipe wall in the **Thermal Expansion** node. The value can be taken directly from a Heat Transfer in Pipes interface.

The Pipe Mechanics interface requires the Pipe Flow Module or the Structural Mechanics Module. It is documented in the Structural Mechanics User's Guide, which is installed together with the documentation for the Pipe Flow Module.

ACOUSTIC-PIPE ACOUSTIC CONNECTION MULTIPHYSICS COUPLING

The new multiphysics coupling allows to couple the Pressure Acoustics interface to the Pipe Acoustics interface. The coupling is between a point and a boundary.

T- AND Y-JUNCTION SELECTIONS IMPROVEMENT

The automatic search for T-junction and Y-junction points is now possible. This additional functionality significantly simplifies the modeling of large pipe networks.

New Models and Updated Models in Version 5.5

COUPLED ANALYSIS OF FLOW AND STRESS IN A PIPE

A new tutorial model has been added to demonstrate how to use the Pipe Mechanics interface. The flow in a pipe with a bend is computed using the Pipe Flow interface. The computed fluid flow load is used as input to a stress analysis in the Pipe Mechanics interface.

ACOUSTICS OF A PIPE SYSTEM WITH 3D BEND AND JUNCTION

This new tutorial shows how to model the propagation of acoustic waves in large pipe systems by coupling the Pipe Acoustics interface to the Pressure Acoustics interface. The tutorial is set up in both the time domain and the frequency domain.

PROBE TUBE MICROPHONE

This model has been updated to use the Acoustic-Pipe Acoustic Connection multiphysics coupling. This update significantly simplifies setting up the model.

Backward Compatibility with Version 5.3

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

Backward Compatibility with Version 5.1 and Earlier

REVISED FORMULATION OF THE PIPE FLOW INTERFACE

The Pipe Flow interface has been amended in version 5.2 with an inertial term. The stabilization and element shape functions have also been changed. Models created in previous versions will be opened using the old interface with a notification. For new models, only the new interface is available in the physics interface list.

Plasma Module

New Functionality in Version 5.5

SPACE-DEPENDENT MODELING OF NON-MAXWELLIAN DISCHARGES

It is now possible to compute the EEDF using a two-term approximation of the Boltzmann equation for 1D and 2D space dependent models.

NEW PHYSICS INTERFACE - CORONA DISCHARGE

A new **Corona Discharge** physics interface is available. This uses an approximate (non-self-consistent) method to quickly compute the charge distribution in a corona discharge.

NEW PHYSICS INTERFACE - ELECTRICAL BREAKDOWN DETECTION

A new **Electrical Breakdown Detection** physics interface uses an approximate method to estimate whether electrical breakdown will occur in an electrical system. The method integrates Townsend growth coefficients along electric field lines, and determines breakdown will occur if some threshold quantity is exceeded.

New and Updated Applications and Models in Version 5.5

NEW MODELS

- Electrostatic Precipitator
- Positive and Negative Corona Discharges
- Breakdown Between Spheres
- Positive Column with Space Dependent EEDF
- GEC Reference Cell with Space Dependent EEDF

Backward Compatibility with Version 5.3a and Earlier

Due to the way the detailed balance is computed for excitation collisions, models set up from scratch could provide different results from the same model in version 5.3a. Models are most likely to be affected if:

- The mole fraction of the metastable species is high.
- The electron density is very high (above around 1E20 $1/m^3$).

The new method for computing the detailed balance is more accurate and physically correct than the method used in version 5.3a and previous.

Backward Compatibility with Version 5.3 and Earlier

There have been several changes that could affect the solution obtained when running a model in version 5.3a versus 5.3 or earlier. This is due to the following changes and bug fixes:

- When electron impact reaction rate constants are computed from cross section data, due to a bug, the result will be different for the Maxwellian or Druyvesteyn electron energy distribution functions. For DC discharges, where Townsend coefficients are recommended, this should not make much of a difference. For inductively coupled or microwave discharges, the average electron density and plasma potential could change by as much as 5%, but typically around 2%. If significant differences in the solution are obtained, it is recommended that you contact `support@comsol.com`.
- Surface species are no longer available from the context menu. Instead, surface species will appear in the Model Builder when a surface reaction containing surface species is added. If the surface reaction is deleted, then the dependent surface species will also be deleted.
- The rate constant for a surface reaction that uses sticking coefficients has changed, and is now proportional to the molecular weight of the reactant instead of the mean molecular weight. In systems where the molecular weight is the same for all species, this change won't affect the results. In models containing species with very different molecular weights, the surface reaction rates may change according to the square root of the molecular weight of the reactant divided by the mean molecular weight. In such systems, it is recommended to activate the new **Mixture diffusion correction** property, which adds additional correction terms to the mass flux vector for the heavy species. Overall discharge properties should not be significantly affected by this change.
- The definition of surface site concentrations and surface reaction rate rates has been corrected for models that include a surface chemical mechanism where the surface species have different site occupancy numbers. This should only result in small changes to the solution, if any. The total surface site concentration is now a property on the physics interface level, rather than computed for each species on the feature level.
- The equation to compute binary diffusion coefficients from characteristic length parameters has changed. This may produce very slight changes in the diffusion

coefficients for the heavy species in the Plasma and Heavy Species Transport interfaces. Any change in the solution due to this should be a tiny fraction of a percent.

Backward Compatibility with Version 5.2 and Earlier

The old Inductively Coupled Plasma and Microwave Plasma interfaces can still be opened, edited, and solved. When such a model is opened, a warning is given indicating that the interface is obsolete and will eventually be removed.

Backward Compatibility with Version 5.1

Models using the Reduced Electric Fields study will interpret the unit in the edit field for the Reduced electric fields as Townsend. For example, when opening a model created in version 5.1, if the value for the Reduced electric fields was 10, this will now appear as `10[Td]` in the study settings.

Ray Optics Module

New Functionality in Version 5.5

NEW RAY RELEASE FEATURES FOR MULTISCALE MODELING

Two new ray release features, **Release from Electric Field** and **Release from Far-Field Radiation Pattern**, are now available. These features can be used for multiscale electromagnetics modeling.

To use these features, first solve for the electric field in the immediate vicinity of a radiation source using the Electromagnetics, Frequency Domain interface or the Electromagnetic Waves, Beam Envelopes interface. Typically, the modeling domain is only a few wavelengths or less in each direction. Then, use the electric field to initialize the intensity and polarization of the released rays, which can then propagate for many wavelengths without the need for a very fine mesh.

The **Release from Far-Field Radiation Pattern** feature sends rays in multiple directions with an intensity profile that matches the radiation pattern computed by the **Far-Field Domain feature**, allowing it to conveniently use antennas or waveguides as radiation sources in Ray Optics models.

SPOT DIAGRAM PLOT

The new **Spot Diagram** plot can be used to display the intersection points of rays with a surface, either a real physical boundary in the model or an intersecting plane. Often, the intersection points with the focal plane are used to report on the performance of an optical system.

The **Spot Diagram** has built-in options to filter or sort the rays shown to automatically locate a plane of minimal rms spot size and to insert text annotations into the plot.

The **Optical Aberration** plot has also been extended to include many of the same filtering, sorting, and automatic focusing capabilities of the **Spot Diagram** plot.

GRATING IMPROVEMENTS

The existing **Grating** feature has been improved, and a new **Cross Grating** feature is now available. Use the **Grating** for a substructure that is periodic in one direction and homogeneous in the orthogonal direction. Use the **Cross Grating** when the substructure is periodic in two different directions.

For the **Grating**, you can now specify either relative or absolute diffraction orders. Relative orders allow you to easily release the order closest to the direction controlled by a user-specified **Blaze angle**.

BETTER MULTITHREADING SUPPORT

The algorithm to model ray-boundary interactions has better multithreading support than in previous versions. For a sufficiently large number of rays, the computation time may be reduced on computers with multiple cores, especially if the rays are reflected many times and the media are homogeneous.

AIR MODEL IN THE VOID DOMAIN

There is now a built-in setting to treat the void region outside the selected geometry domains as air, where the refractive index is accurately computed as a function of pressure and temperature. The difference in refractive index between air and vacuum plays an important role in the design of optical systems.

PREVIEW INITIAL COORDINATES IN RELEASE FROM GRID

When using the **Release from Grid** feature to release rays, you can view the ray release positions before running the study. Click the **Preview Initial Coordinates** button to show the initial positions as points, or click the **Preview Initial Extents** button to show a bounding box containing the initial positions.

ISOTROPIC SCATTERING WALL CONDITION

You can now select **Isotropic scattering** from the **Wall condition** list in the settings for the **Wall** node. This applies random reflection to the rays that hit the wall, like the **Diffuse scattering** option, but does not follow the cosine law.

NEW RELEASE TYPE: HEXAPOLAR CONE

You can now release rays from a point so that their initial ray directions follow a hexapolar distribution.

NEW PARTS

The Part Libraries for the Ray Optics Module have been considerably extended in version 5.5.

Aspheres

New aspheric lens and mirror parts are now available:

- Aspheric even lens (or mirror)
- Aspheric odd lens (or mirror)

- Aspheric lens (or mirror) where the surface sag is defined using a Q-type polynomial basis to describe the deviation from a best fit sphere
- Aspheric lens (or mirror) where the surface sag is defined using a Q-type polynomial basis to describe the deviation from a best fit conic

The advantage of the Q-type polynomials is that all of their coefficients are of a similar order of magnitude, whereas in the even and odd aspheres, the coefficients can change by many orders of magnitude. Thus, the Q-type aspheres are arguably less susceptible to roundoff error when very-high-order terms are included.

Doublet and Triplet Lenses

Built-in spherical doublet and triplet lenses are included. The inner surfaces can either be air-spaced or in contact.

Polygonal Mirrors

The new polygonal mirror parts include the following:

- Conic polygonal mirror off axis
- Conic polygonal mirror on axis
- Spherical polygonal mirror

Polygonal mirrors are frequently used in larger telescopes where a single larger mirror is difficult or expensive to manufacture.

New Applications in Version 5.5

COMPACT CAMERA MODULE

Compact camera modules are widely used in electronic devices such as mobile phones and tablet computers. In order to reduce both the size and number of elements required, the optical design will typically incorporate several highly aspheric surfaces. This model demonstrates a five-element design using the Aspheric Even Lens 3D part from the Ray Optics Module Part Library.

GREGORY–MAKSUTOV TELESCOPE

The Gregory–Maksutov telescope is a simple catadioptric telescope comprising a spherical corrector lens and a spherical primary mirror. In this example, the corrector lens and mirror are formed using the Spherical Lens 3D and Spherical Mirror 3D parts, respectively, from the Ray Optics Part Library.

KECK TELESCOPE

The Keck Telescope is a 10-meter diameter telescope with a Ritchey–Chretien optical design. It is noted for being one of the first large optical telescopes to utilize a lightweight segmented primary mirror. This tutorial demonstrates how to use built-in polygonal mirrors from the COMSOL Part Library to construct a model of the Keck Telescope segmented primary mirror.

SCHMIDT-CASSEGRAIN TELESCOPE

The Schmidt–Cassegrain telescope demonstrated in this tutorial is a classic design using an aspheric corrector lens and two spherical mirrors. The aspheric corrector is created using the Aspheric Even Lens 3D part from the Ray Optics Part Library.

CROSS GRATING ÉCHELLE SPECTROGRAPH

This tutorial demonstrates the use of a Cross Grating feature in an échelle spectrograph. A cross grating is a periodic surface with two directions of periodicity. In this model, the cross grating is used in high order in one direction and in first order in the orthogonal ("cross") direction. By this means, a two-dimensional cross-dispersed spectrum can be produced with a single grating. A parametric sweep across several orders is made and the resulting ray and spot diagrams are generated.

NEWTONIAN TELESCOPE STRUCTURAL ANALYSIS

This tutorial demonstrates a structural analysis of a simple telescope. The deformation of the telescope structure under gravity is examined and the effect on image quality is demonstrated.

PETZVAL LENS STOP ANALYSIS WITH HYPERELASTICITY

This model demonstrates an integrated structural-thermal-optical performance (STOP) analysis of an optical system using a nonlinear material model. The Petzval Lens STOP analysis tutorial is used as the basis for this model. The lens supports are modeled using a hyperelastic material feature. The resulting displacement and stress fields within the optical system are shown together with nominal and best focus spot diagrams.

RAY RELEASE BASED ON A PLANE ELECTROMAGNETIC WAVE

This tutorial shows how to set up a ray release based on the incident electric field at a boundary. First, the Electromagnetic Waves, Frequency Domain interface is used to solve for the electric field of a plane wave. Then, rays are released with initial intensity and polarization matching that of the electric field at the releasing boundary.

RAY OPTICS WITH A DIPOLE ANTENNA SOURCE (3D)

In this tutorial model, the far-field radiation pattern of a dipole antenna is computed in a 3D model component. Then, in a separate 3D model component, a ray is released using the far-field radiation pattern to initialize the ray's intensity, polarization, and phase.

RAY OPTICS WITH A DIPOLE ANTENNA SOURCE (2D AXISYMMETRIC)

In this tutorial model, the far-field radiation pattern of a dipole antenna is computed in a 2D axisymmetric model component. Then, in a separate 3D model component, a ray is released using the far-field radiation pattern to initialize the ray's intensity, polarization, and phase.

Important Fixes in Version 5.5

MORE ROBUST DIFFUSE SCATTERING ON CURVED INTERIOR SURFACES

The **Diffuse scattering** wall condition now uses the normal and tangential directions computed on the boundary mesh element, rather than the directions computed on the underlying geometric entity, when reinitializing the ray direction. This reduces the likelihood that rays will fail to locate a subsequent wall interaction when they are reflected in a direction nearly tangent to the interior of a concave surface, like the inside of a tube. In some cases, it may still be necessary to switch to a linear geometry shape order to prevent some rays from disappearing.

LABELING OF THE AZIMUTHAL AND POLAR ANGLES

In version 5.5, the labeling of azimuthal and polar angles in spherical coordinate systems has been made more consistent. Now, θ is used for the polar angle and ϕ is used for the azimuthal angle.

DEFAULT DATASET GENERATION

The generation of default **Ray** datasets has been made more robust for cases when the study contains multiple study steps.

LINEAR POLARIZER WITH RAY POWER COMPUTATION

The **Linear Polarizer** boundary condition no longer gives an error when ray power is solved for in the Geometrical Optics interface.

MIRROR WITH RAY INTENSITY AND POWER

An error message that appeared when adding **Accumulator** subnodes to the **Mirror** boundary condition in the Geometrical Optics interface, while also solving for ray intensity and power, has been fixed.

Backward Compatibility with Version 5.4

HANDLING OF GRATING ORIENTATION DIRECTION

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

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

RENAMED RAY RELEASE FEATURES

The **Inlet** feature has been renamed to **Release from Boundary**.

The **Inlet on Axis** feature has been renamed to **Release from Symmetry Axis**.

RENAMED RAY STATISTICS

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

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

Backward Compatibility with Version 5.3 and Earlier

DIFFRACTION GRATING SETTINGS MOVED TO A SUBNODE

In version 5.3 and earlier, the **Grating** feature for the Geometrical Optics interface automatically released transmitted and reflected rays of diffraction order zero. In version 5.3a, the rays of diffraction order zero are controlled by a **Diffraction Order** subnode, as are all other diffraction orders that might be released.

NEW OPTIONS FOR SPECIFYING DIFFRACTION GRATING ORIENTATION

In the **Grating** feature for the Geometrical Optics interface, the options to control grating orientation in 3D have been reorganized and expanded. It is now possible to specify either the direction of the grating lines or the direction of periodicity in the grating surface. The input **Grating orientation specification** has been renamed **Direction of grating lines**.

NEW OPTIONS FOR SPECIFYING REFLECTANCE OR TRANSMITTANCE

In the Geometrical Optics interface, the settings to control **Thin dielectric films on boundary** in the settings for the **Material Discontinuity** feature have been renamed and rearranged.

The old options **Single layer, specified reflectance** and **Single layer, specified transmittance** have been renamed to **Specify reflectance** and **Specify transmittance**, respectively. For both of these options and for the **Anti-reflective coating** option, which has not been renamed, the **Treat as single layer dielectric film** check box is selected when opening the model in version 5.3a. Otherwise, this check box is cleared, which is the default behavior.

ADJUSTMENT TO THE RAY TERMINATION FEATURE

In the **Ray Termination** feature for the Geometrical Optics interface, the **Termination criteria** list has been renamed **Additional termination criteria**.

REMOVED CHECK BOXES FOR OBSOLETE FUNCTIONALITY

In the settings for the Geometrical Optics interface, the **Allow propagation outside selected domains** check box has been removed. This check box was flagged as obsolete in version 5.3, so clearing it would have created a Warning node.

In the Settings window for the **Release from Grid** feature, the check box **Suppress interaction with coinciding exterior boundaries** has also been removed.

Backward Compatibility with Version 5.2a and Earlier

REFERENCE EDGE FEATURES HAVE BEEN REMOVED

The **Reference Edge** subnode, which was previously used to specify the orientation of some optical components in 3D, has been removed. Instead, the orientation of these components is specified directly in the parent feature settings using an extra selection input. The following features are affected:

- **Linear Polarizer**

- **Linear Wave Retarder**
- **Mueller Matrix**
- **Grating**

Backward Compatibility with Version 5.2

ANISOTROPIC REFRACTIVE INDICES CAN NO LONGER BE SPECIFIED

In version 5.2 and earlier, it is possible to specify an anisotropic refractive index in the settings window for the **Medium Properties** node, despite the fact that the anisotropy of the refractive index is not taken into account when modeling ray propagation. In version 5.2a, only isotropic refractive indices can be specified in the settings window for the **Medium Properties** node. If an anisotropic refractive index was specified in an older version, then when opening the model in version 5.2a, all of the diagonal terms will be populated using the first diagonal value and all off-diagonal terms will be set to zero.

RENAMED INTENSITY COMPUTATION METHODS

The options in the **Intensity Computation** list in the settings window for the Geometrical Optics interface have been renamed.

- **Using principal curvatures** has been renamed to **Compute intensity**.
- **Using principal curvatures and ray power** has been renamed to **Compute intensity and power**.
- **Using curvature tensor** has been renamed to **Compute intensity in graded media**.
- **Using curvature tensor and ray power** has been renamed to **Compute intensity and power in graded media**.

RENAMED POLARIZATION SETTING

In the **Initial polarization type** list, available for most ray release features in the Geometrical Optics interface, the option **Partially coherent** has been renamed to **Partially polarized**.

CHANGES TO PERTURBATIONS FOR SOLAR RADIATION

The treatment of perturbations due to sunshape and surface roughness has been changed in version 5.2a. This change affects the **Illuminated Surface** and **Solar Radiation** features for the Geometrical Optics interface.

The **Use corrections for finite source diameter** check box has been replaced by the **Corrections for finite source diameter** list. If the check box was selected, **Create light cones at release points** will be selected from the list after opening the model in version 5.2a. If the check box was cleared, **None** will be selected from the list.

For the **Illuminated Surface**, the **Standard deviation of the angle of incidence** parameter has been renamed **Surface slope error** and is now used to define a Rayleigh distribution for perturbations to the surface normal, rather than the angle of incidence of radiation. To account for the different effect that the surface slope error has on the model, any value or expression entered for the **Standard deviation of the angle of incidence** in version 5.2 or earlier will be divided by 2, before being used to specify the **Surface slope error** in version 5.2a.

The **Number of rays in wave vector** space $N_{w,r}$ has been removed. The effect of surface roughness can no longer multiply the number of degrees of freedom created by a release feature at each release point.

ACTIVATION CONDITIONS ON OPTICAL COMPONENTS

The following boundary conditions require some form of ray intensity computation in order to have any effect on the solution:

- **Linear Polarizer**
- **Linear Wave Retarder**
- **Circular Wave Retarder**
- **Ideal Depolarizer**
- **Mueller Matrix**

If any of these features is present in a model created in version 5.2 or earlier, and **None** is selected from the **Intensity computation** list in the settings window for the Geometrical Optics interface, then the feature will be automatically disabled when the model is opened in version 5.2a. To enable these features, select any option other than **None** from the **Intensity Computation** list.

Backward Compatibility with Version 5.0

RENAMED SETTINGS

The Wall condition **Bounce** has been renamed to **Specular reflection**.

CHANGES TO FREQUENCY-DEPENDENT MATERIAL PROPERTIES

The **Frequency-dependent refractive indices** check box has been renamed to **Allow frequency distributions at release features**.

Expressions involving the ray frequency must always be included within the `noenv()` operator. Some expressions for user-defined refractive indices may fail to be evaluated properly in version 5.2a unless this operator is applied.

THIN DIELECTRIC FILM SETTINGS

In version 5.0, if the ray intensity is computed, the option to add **Thin Dielectric Film** nodes to a **Material Discontinuity** node is always available. In version 5.2a, the Thin Dielectric Film subnode can only be added if **Add layers to surface** or **Add layers to surface, repeating** is selected from the **Thin dielectric films on boundary** list in the **Material Discontinuity** settings window. If a model created in version 5.0 is opened in version 5.2a, and any **Thin Dielectric Film** nodes have been added, then **Add layers to surface** is selected from the **Thin dielectric films on boundary** list instead of the default **None**.

CHANGES TO THE CIRCULAR WAVE RETARDER

The **Retardance** of the **Circular Wave Retarder** now indicates the rotation angle of the polarization ellipse of a ray that passes through the device. In version 5.0, the retardance corresponds to half of this rotation angle. The behavior of the **Circular Wave Retarder** is now consistent with the documentation. If a model created in version 5.0 is opened in version 5.2a, the expression for the **Retardance** of any **Circular Wave Retarder** node is multiplied by 2.

OBSOLETE DEPOSITED RAY POWER NODE

The **Deposited Ray Power** node for absorbing domains is obsolete and will be removed in future versions. For simulations in which rays generate heat due to absorption, the **Ray Heat Source** multiphysics node should be used instead.

RF Module

New Functionality in Version 5.5

ADDITIONAL RF MATERIAL LIBRARY FOR DIELECTRIC

The RF material library includes more than 25 dielectric materials from the company Premix Group to assist in modeling millimeter-wave applications.

MIXED-MODE S-PARAMETERS

The mixed-mode S-parameters are calculated using a full S-parameter matrix of a four-port network that is composed of four single-ended lines. This describes the responses of a circuit with balanced ports excited and terminated by two types of modes: common and differential modes.

SPECIFIC ABSORPTION RATE (SAR)

A predefined postprocessing variable is available for the calculation of specific absorption rate (SAR); that is, the amount of exposure from radio frequency (RF) emission without going through manual definition of multiple variables.

TRANSVERSE ELECTROMAGNETIC (TEM) TYPE PORT

A new port type, transverse electromagnetic (TEM), in the Port boundary condition has been added to address the mode field of a port configured by user-defined electric potential and ground settings.

VIA TYPE LUMPED PORT

A new lumped port type, called Via, is useful to terminate or excite metallic boundaries of cylindrical shape, such as via holes.

PORT UTILITY

Cutoff Frequency Calculator

A cutoff frequency calculator is available from the rectangular and circular port type Settings windows. This helps to quickly identify the cutoff frequency for a given mode in an arbitrary-sized waveguide.

Coaxial Line Impedance Calculator

A coaxial line impedance calculator is available for the Lumped Port boundary condition. The impedance of an arbitrary coaxial geometry can be computed without going through a simulation.

ADDITIONAL 3D FAR-FIELD FUNCTIONS FROM 2D AXISYMMETRIC COMPONENT MODEL

Effective 3D antenna gain and realized gain functions for 2D axisymmetric models are added.

DEFAULT PROBE PLOT

The default S-parameter plot generated when running a frequency or parametric sweep is visualized and dynamically updated during the simulation.

COUPLING FROM FULL-WAVE TO RAY OPTICS SIMULATION

Two new features, Release from Electric Field and Release from Far-Field Radiation Pattern, have been added to the Geometrical Optics interface in the Ray Optics Module that either release rays based on a near field or far field computed from a full-wave simulation. The full-wave field can be produced by the Electromagnetic Waves, Frequency Domain interface in the RF Module.

GAUSSIAN BEAM INPUT OPTION FOR SCATTERING BOUNDARY CONDITION

The Scattering boundary condition has a new option for the Incident field list. Choosing the Gaussian beam option allows you to propagate a Gaussian beam into an arbitrary direction. The Gaussian beam is defined using the paraxial Gaussian beam formula.

POLARIZATION PLOTS AND JONES VECTOR VARIABLES

Periodic ports now create a new default Polarization plot. The Polarization plot depicts the polarization state for the different diffraction orders and is based on new postprocessing variables for Jones vectors elements. Also the base vectors, used for defining the Jones vectors, are available for plotting and evaluation.

EVANESCENT WAVES INCLUDED IN THE PLANE-WAVE EXPANSION FOR GAUSSIAN BEAM BACKGROUND FIELDS

When using the Plane wave expansion option for defining a Gaussian beam background field, evanescent waves can also be included in the expansion. This option

can be useful when simulating a tightly focused (spot radius smaller than the wavelength) Gaussian beam propagating away from the focus.

REFERENCE POINT SUBFEATURE ADDED TO AND UPDATED FOR THE SCATTERING CONDITION FEATURE

The Reference Point subfeature is now available for the Scattering Boundary Condition feature for the Electromagnetic Waves, Frequency Domain interface when an input field is active. The reference position is defined as the average position of the selected points. This feature is mainly useful when the domain material includes absorption or gain.

New and Models in Version 5.5

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

Users of consumer electronics with radiating devices are exposed to radio frequency (RF) emission. The amount of exposure is defined as the specific absorption rate (SAR). That is, the SAR value represents the radio frequency (RF) energy rate absorbed by a body. This model computes local SAR values over a simplified human head and brain mock-up when a microstrip patch antenna operating at the Wi-Fi frequency range is placed close to the human head.

MIXED-MODE S-PARAMETERS ANALYSIS

Mixed-mode S-parameters describe the responses of a circuit with balanced ports excited and terminated by two types of modes: common and differential modes. They are calculated using a full S-parameter matrix of a four-port network that is composed of four single-ended lines. This example analyzes two adjacent microstrip lines and computes the mixed-mode S-parameters.

MODELING OF MICROSTRIP LINES WITH VIAS

There are multiple ways to excite and terminate transmission lines using different types of port and lumped port features. In this example, transverse-electromagnetic (TEM) type ports and a via-type lumped port are used to simulate two adjacent microstrip lines. One via end is terminated as a metalized via while the other via end is probing an inflow signal. The computed S-parameters show the amount of crosstalk between the lines and the strength of the signal coupled through the cylindrical via.

Rotordynamics Module

New Functionality in Version 5.5

MULTI-SPOOL BEARING

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

SQUEEZE FILM DAMPER

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

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

DYNAMIC COEFFICIENTS CALCULATION FOR HYDRODYNAMIC BEARINGS

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

DEFAULT GEOMETRY PLOT FOR BEAM ROTOR INTERFACE

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

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

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

CHANGE ROTOR SPEED FEATURE IN BEAM ROTOR INTERFACE

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

FILM BOUNDARY CONDITION IN HYDRODYNAMIC BEARING

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

AUTOMATIC SETTINGS FOR CAMPBELL PLOT

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

FATIGUE ANALYSIS IN SOLID ROTOR INTERFACE

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

NEW POSTPROCESSING VARIABLES IN HYDRODYNAMIC BEARING INTERFACE

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

New Models in Version 5.5

CRITICAL SPEED OF A DUAL ROTOR SYSTEM

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

EVALUATION OF THE DYNAMIC COEFFICIENTS OF A PLAIN JOURNAL BEARING

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

DAMPING COEFFICIENTS OF A SQUEEZE FILM DAMPER

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

SHAFT VIBRATION DUE TO GEAR RATTLE AND BEARING MISALIGNMENT

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

aligned with the shafts, and in the second case, one of the bearings has a small angular misalignment. The results show larger torsional and axial vibrations in the shaft in the presence of misalignment.

Semiconductor Module

New Functionality in Version 5.5

DENSITY-GRADIENT FORMULATION

A new discretization option based on the density-gradient theory is introduced to include the effect of quantum confinement in the conventional drift-diffusion formulation. This provides an efficient alternative to other more sophisticated quantum mechanical methods, which are computationally expensive.

TRAP-ASSISTED HETEROINTERFACE RECOMBINATION

A new boundary condition is added for simulating the effects of interface traps at heterojunctions. It is straightforward to set up discrete and/or continuous energy level traps to account for their carrier capture and charging effects on heterointerfaces.

USER-DEFINED JUNCTION CURRENT

A new option for the heterojunction boundary condition allows the user to specify any arbitrary current density at the junction.

TRAP QUASI-FERMI LEVEL FORMULATION

A new formulation for explicit traps is introduced to use the trap quasi-Fermi level as the dependent variable to be solved for.

DOPING-DEPENDENT MATERIAL PROPERTIES

Acceptor concentration and donor concentration are now available under the **Transport** branch of the **Model Input** lists to facilitate the construction of doping-dependent material properties.

SOLID MECHANICS COUPLING

The variable for the time derivative of the displacement field is updated to allow mesh deformation for coupling to solid mechanics.

SCHRÖDINGER-POISSON EQUATION

The range of weights for the Schrödinger-Poisson multiphysics coupling is extended from 20 kT to 40 kT to cover cases with higher temperature ranges.

NEW BUILT-IN FUNCTIONS

New Fermi-Dirac functions `log_FD_half_inv_an` and `log_FD_half_an` are available to offer a full input argument range.

ACCURACY IMPROVEMENTS

The Caughey-Thomas Mobility Model formulation is updated to improve accuracy.

Default solver settings for time-dependent studies are updated for better accuracy.

Default scaling for quasi-Fermi level formulations are updated for better accuracy.

MODEL UPDATE

The mosfet_mobility tutorial model has been updated with the list of swept values optimized for the continuation parameter of the field-dependent mobility models. An unnecessary value is removed from the list to make the model solve faster and the file size smaller.

CORRECTIONS AND BACKWARD COMPATIBILITY

- An error in the time-dependent weak term for the quasi-Fermi level formulation has been fixed since 5.4 update 3.
- A missing effect of explicit surface trap charges on Schottky contacts is fixed.
- The boundary current density formula between two adjacent Semiconductor Material Model features is corrected for the finite volume discretization.
- The selection filter for the Zero Charge boundary condition is fixed to not override the Axial Symmetry boundary condition.

New Models in Version 5.5

DENSITY-GRADIENT AND SCHRÖDINGER-POISSON RESULTS FOR A SI INVERSION LAYER

This tutorial demonstrates the use of the density-gradient formulation to include the effect of quantum confinement in the device physics simulation of a silicon inversion layer. This formulation requires only a moderate increase of computational resources as compared to the conventional drift-diffusion equation. Thus it allows much speedier engineering investigation than other more sophisticated quantum mechanical methods. The result of the density-gradient theory is compared with the solution of the Schrödinger-Poisson equation. The computed electron density profiles from both

theories show the expected behavior of quantum confinement and both agree well with the published figures in the reference paper.

DENSITY-GRADIENT ANALYSIS OF AN INSB P-CHANNEL FET

This tutorial analyzes the DC characteristics of an InSb p-Channel FET, using the density-gradient theory to add the effect of quantum confinement to the conventional drift-diffusion formulation without a large increase of computational resources. The confinement effect is applied both in the quantum well channel and on the top insulator interface, which is close to the channel. The use of an anisotropic density-gradient effective mass matrix is demonstrated, as well as the technique to configure a general field-dependent mobility model. The hole density profile and the Id-Vg curve obtained from the 2D model compare well with the published figures in the reference paper.

3D DENSITY-GRADIENT SIMULATION OF A NANOWIRE MOSFET

This 3D model of a nanowire MOSFET employs the density-gradient theory to add the effect of quantum confinement to the conventional drift-diffusion formulation, without requiring excessively high computational costs. The oxide layer is simulated explicitly with geometric domains, and quantum confinement at the silicon-oxide interface is accounted for via a dedicated boundary condition. The density-gradient effective mass is anisotropic. Various selection utilities are used to simplify the assignment of physics settings and plot selections. The result matches well with the Id-Vg curves and electron density profiles published in the reference paper.

RADIATION EFFECTS IN A PIN DIODE

This tutorial performs steady-state and transient analyses of the response of a PIN diode to constant and pulsed radiation, respectively. The effect of radiation is modeled as a spatially uniform generation of electron-hole pairs within the device. At high dose rates, the separation of the generated charges causes the reduction of the interior electric field and prolonged storage of excess carriers. A quantitative prediction of this phenomenon is only possible with numerical simulation, since an analytical solution is unattainable. Several techniques for achieving convergence in the cases of high reverse bias, field-dependent mobility, and time-dependent studies are demonstrated. The computed carrier concentrations and electric field distribution agree well with the reference paper.

Structural Mechanics Module

New Functionality in Version 5.5

RANDOM VIBRATION ANALYSIS

Functionality for random vibration analysis has been added. This type of analysis is intended for situations where the load on a structure is random, but has well-defined statistical properties that do not change with time. The excitation is described by its power spectrum density (PSD). If there are several sources of excitation, also cross-correlation spectra are needed. The output of the analysis is the PSD or RMS of any linear expression.

In the **Add Study** dialog, the new study type **Random Vibration (PSD)** is available. When selected, a set of three different studies is added to the model. Also, three nodes are created under **Global Definitions->Reduced-Order Modeling** for specifying the spectra.

STUDIES FOR REDUCED-ORDER MODELS

Three new studies for generation and simulation of reduced-order models (ROM) have been added to the **Add Study** dialog:

- **Time Dependent, Modal Reduced-Order Model**
- **Frequency Domain, Modal Reduced-Order Model**
- **Time Dependent, AWE Modal Reduced-Order Model**

When one of these studies is selected, a set of two or three different studies is added to the model. Also, two nodes are created under **Global Definitions->Reduced-Order Modeling** for providing the input to the ROM.

The old study **Modal Reduced-Order Model** has been removed from the **Add Study** dialog. It has been superseded by the more general **Time Dependent, Modal Reduced-Order Model**. It is, however, still possible to add a **Modal Reduced-Order Model** study step directly in the solver sequence if you want to access the old functionality.

IMPROVEMENTS IN CONTACT MODELING

The contact modeling functionality has been extended and improved in many aspects.

- You can now model contact in all structural mechanics interfaces that are defined on domains or boundaries: Solid Mechanics, Shell, Layered Shell, Membrane, and

Multibody Dynamics. In the shell interfaces, thickness and offset are taken into account.

- Contact can occur between two different physics interfaces.
- Contact can be modeled from one physics interface toward any meshed surface, even if that surface does not have any physics interface at all defined. This approach is particularly useful when modeling contact with a rigid shape.
- The more robust penalty method has replaced the augmented Lagrangian method as the default contact algorithm.
- The friction forces are now represented in a boundary coordinate system. As an effect, the number of friction degrees of freedom when using the augmented Lagrangian method is now two in 3D (previous version: three) and one in 2D (previous version: two).
- **Adhesion** and **Decohesion** are now two separate subnodes under **Contact**.
- When using Adhesion, the adhesive layer stiffness can now also be given using elastic material data by setting **Adhesive stiffness** to **Use material data**.
- In **Decohesion**, there are now new energy-based damage models. Also, you can now include a viscous regularization in time-dependent analysis in order to mitigate the inherent instable behavior when the material is losing stiffness.
- New default plots showing arrows for contact pressure and friction force are generated when a **Contact** node is present.
- The selection of pairs in the **Contact** node is more intuitive.

In a model created in a previous version, the old type of **Contact** node will still be kept. If you add another **Contact** node, it will be of the new type.

NEW MULTIPHYSICS INTERFACES FOR THERMAL STRESS ANALYSIS

Three new multiphysics interfaces for thermal stress analysis have been added:

- The **Thermal Stress, Shell** interface couples a Heat Transfer in Shells interface with a Shell interface.
- The **Thermal Stress, Layered Shell** interface couples a Heat Transfer in Shells interface with a Layered Shell interface. This interface is only available with the Composite Materials Module.
- The **Thermal Stress, Membrane** interface couples a Heat Transfer in Shells interface with a Membrane interface.

In all three cases, the connection is made using the **Layered Thermal Expansion** multiphysics coupling. This means that the **Layered Linear Elastic Material** is used in the Shell and Membrane interfaces. The **Layered Thermal Expansion** multiphysics coupling can also connect to a **Thin Layer** feature in the Heat Transfer in Solids interface.

As an effect of this, the old **Thermal Stress** multiphysics interface has been renamed to **Thermal Stress, Solid**. All four thermal stress interfaces now reside in a group named **Thermal Stress** in the **Add Physics** dialog.

NEW MULTIPHYSICS INTERFACE FOR FLUID-STRUCTURE INTERACTION WITH HEAT TRANSFER

In some fluid-structure Interaction (FSI) problems, heat transfer between the fluid and the solid is important. Usually, this is also accompanied by thermally induced deformations or stresses in the solid. A new multiphysics interface, **Fluid-Solid Interaction, Conjugate Heat Transfer** has been added in order to facilitate a convenient setup of models combining these effects. It combines three physics interfaces, Heat Transfer in Solids and Fluids, Solid Mechanics, and Laminar Flow, together with a moving mesh and appropriate multiphysics couplings. As with all other FSI interfaces, the flow can be changed to turbulent if appropriate licenses are available.

FLUID-STRUCTURE INTERACTION FOR TWO-PHASE FLOWS

In the **Add Physics** dialog, two new multiphysics interfaces for two-phase flows have been added: **Fluid-Solid Interaction, Two-Phase Flow, Phase Field** and **Fluid-Solid Interaction, Two-Phase Flow, Phase Field, Fixed Geometry**. These interfaces require also a license that contains the underlying two-phase flow interfaces; that is, the CFD Module or the Microfluidics Module.

PIPE MECHANICS INTERFACE

The new Pipe Mechanics interface is intended for structural analysis of pipes. The interface is, in many respects, similar to the Beam interface, with important modifications necessary for the analysis of pipes. For instance:

- Stresses, strains, and axial deformations caused by internal overpressure are taken into account.
- The default material model is **Fluid and Pipe Materials**, reflecting that both the material in the pipe wall and the fluid are important.
- The **Pipe Cross Section** node has more and slightly different input parameters when compared to the corresponding **Cross Section Data** node in the Beam interface.

- A special type of load, **Fluid Load**, is included. There, you enter pressure and drag forces. These values can be taken directly from a Pipe Flow interface.
- A temperature gradient can be present through the pipe wall in the **Thermal Expansion** node. The value can be taken directly from a Heat Transfer in Pipes interface.

The Pipe Mechanics interface requires the Structural Mechanics Module or the Pipe Flow Module.

ELASTIC WAVES, TIME EXPLICIT INTERFACE

The new **Elastic Waves, Time Explicit** interface makes it possible to analyze transient wave propagation problems in linear elastic materials under small deformations in a computationally efficient manner. In the **Add Physics** dialog, this interface is found under **Elastic Waves** in the **Acoustics** branch.

When the Acoustics Module is available, there is also a new multiphysics interface, **Acoustic-Solid Interaction, Time Explicit**, which combines an **Elastic Waves, Time Explicit** interface in solid domains with a **Pressure Acoustics, Time Explicit** interface in adjacent acoustic domains.

NONRIGID JOINTS BETWEEN BEAMS

In the Beam interface, it is now possible to specify that two or more edges that meet in a point have disconnected degrees of freedom. This can, for example, be used for modeling internal hinges in a frame. To do this, you add a **Beam End Release** node and select the common point, as well as the degrees to be disconnected. If there are more than two beams attached to the point, you may also add one or more **Edge Group** subnodes to control the structure of the connection.

SOLID MECHANICS ON ROTATING DOMAINS

When modeling multiphysics problems in systems where there is a mixture of rotating and stationary domains, it is now possible to use the Solid Mechanics interface in a corotating configuration, so that only the deformations relative to the rotation are modeled. This is far more efficient than solving for the total displacements including large global rotations. To this end, the **Rotating Frame** feature has been extended so that it can provide a superposition of the rigid body rotation and the relative displacements for controlling the spatial frame.

DIRECT STIFFNESS INPUT FOR SHELLS AND PLATES

In the Shell and Plate interfaces, it is now possible to describe the elastic stiffness of a cross section directly in terms of membrane and bending stiffness, rather than by

material data and thickness. This facilitates modeling of homogenized complex structures, such as corrugated sheets.

EXTENSIONS OF LINEAR ELASTIC MATERIAL MODEL IN SHELL INTERFACE

In version 5.5, the **Layered Linear Elastic Material** is present without the Composite Materials Module license. This license is however still needed in order model multilayered shells. The ability of this material model to handle more general through-thickness stress distribution does, however, make it important also for single-layer shells.

Through the **Layered Linear Elastic Material**, several new materials models have been implemented. Viscoelasticity is present in the Structural Mechanics Module, and with the addition of the Nonlinear Structural Materials Module, there are also several nonlinear material models such as plasticity.

There are also several new multiphysics couplings implemented through the **Layered Linear Elastic Material**.

NEW LAYERED LINEAR ELASTIC MATERIAL IN THE MEMBRANE INTERFACE

A new material model, the **Layered Linear Elastic Material**, has been added to the Membrane interface. With the Composite Materials Module license, it can be used to model multilayered thin films.

There are several material models that can be used together with the **Layered Linear Elastic Material** — for example, viscoelasticity and plasticity. The nonlinear material models require the Nonlinear Structural Materials Module. Since these material models are already also available with the standard **Linear Elastic Material**, the main use is for multilayered membranes.

SHELL DATASET FOR 2D AXISYMMETRY

The Shell dataset, by which a shell can be displayed by its top and bottom surface, is now available also in 2D axisymmetry.

POINT SELECTIONS IN RIGID CONNECTOR FOR SHELL

The selection in a **Rigid Connector** in the Shell interface can now be a combination of edges and points. It is also possible to add a rigid connector that is connected only to a set of points by adding it at the point level.

NEW DEFAULT SCALE FACTOR FOR EIGENMODES

In the **Eigenvalue Solver** node within the solver sequence, a new option for scaling eigenmodes has been added. This option is now used by all structural mechanics interfaces to ensure that eigenmode amplitudes are small relative to the size of the geometry. By doing this, it ensures that variables that are nonlinear by their nature will be evaluated using small perturbations to the original geometry.

MODAL PARTICIPATION FACTORS TABLE

If a **Participation Factors** node is present under **Definitions** when an eigenfrequency study is run, an evaluation group named **Participation Factors** is automatically generated. It contains a table with the translational and rotational participation factors for all computed eigenfrequencies.

NEW DEFAULT FOR TEMPERATURE DEPENDENT PROPERTIES

When material properties have a temperature dependence, the default in the **Model Inputs** section is now to take the temperature from **Common model input**. In versions prior to 5.5, the default was **User defined.** In both cases, the default temperature is room temperature, as long as temperatures are not computed by another physics interface. With the new formulation, the temperature will automatically be picked up if defined under **Default Model Inputs**, for example, by a Heat Transfer in Solids interface. The settings used in a model created in an earlier version are not affected.

TEMPERATURE COUPLING NODE IS OBSOLETE

The **Temperature Coupling** multiphysics node is no longer used for the transfer of temperatures from heat transfer interfaces to structural mechanics interfaces. Instead, the common model input mechanism is used for transferring the temperature. In models created prior to version 5.5, the **Temperature Coupling** node is still present and used. A warning message is, however, shown.

In all nodes where temperature-dependent properties are referenced, the default value for the temperature in the **Model Inputs** section is now **Common model input**.

SYNCHRONIZATION OF ELASTIC PROPERTIES

If isotropic material data (for example, E and ν) are present in a **Material**, then this material can also be used if the **Solid model** in a linear elastic material is set to **Orthotropic** or **Anisotropic**. Similarly, orthotropic material data can be used when **Solid model** is set to **Anisotropic**.

ENHANCEMENTS FOR CELL PERIODICITY

The **Cell Periodicity** node in the Solid Mechanics interface has a number of improvements:

- It is now available also in 2D. If the generalized plane strain formulation is used, then it is also possible to generate a global **Material** node, in the same manner as in the 3D case.
- One separate global **Material** node is created for each **Cell Periodicity** node. This makes it possible to evaluate several RVE cells in the same study.
- When using **Cell Periodicity**, a default evaluation group is generated for displaying the computed material properties. This group contains a **Global Matrix Evaluation** node where the elasticity or compliance matrix is shown.
- The **Create** button has been replaced by icons in a section toolbar and split into separate operations for study generation and material node generation.

MAGNETOSTRICTION WITH MAGNETIC HYSTERESIS

The **Magnetostrictive Material** has been extended to include the Jiles–Atherton model for magnetic hysteresis.

USER-DEFINED MATERIAL FOR MEMBRANES

The **External Stress-Strain Relation** material model has been added also to the Membrane interface. With this material model, you can add your own material models, coded in C or other programming languages.

ENHANCEMENTS FOR EXTERNAL MATERIALS

When implementing your own material using **External Stress-Strain Relation** or **External Strain**, you can now pass back your own warning and log messages, which are displayed in the GUI.

Another improvement is that the limit for the number of parameters that can be passed to an external material has been increased significantly.

When using the **External Stress-Strain Relation** for a plane stress Solid Mechanics interface in 2D, additional degrees of freedom for the transverse shear strains can be activated by selecting the **Allow transverse shear strains** check box. This is needed for some type of anisotropies in the material model.

IMPROVED SYMMETRY AND ANTISYMMETRY FOR MEMBRANE

In the **Symmetry** and **Antisymmetry** nodes in the Membrane interface, a new **Face Defining the Local Orientations** section has been added. The purpose is to define a

unique orientation of the symmetry plane in cases where several boundaries are attached to the selected edge.

SUPPORT FOR GENERAL ANISOTROPY IN MEMBRANE INTERFACE

The anisotropic version of the **Linear Elastic Material** in the Membrane interface can now include also terms coupling transverse shear to in-plane behavior.

THIN ELASTIC LAYER WITH MASS

In the **Thin Elastic Layer** node in the Solid Mechanics interface, it is now also possible to assign a mass distribution to the layer. This mass is taken into account both in dynamic analysis and in loads from gravity and rotating frames.

UPDATED COUPLING BETWEEN SOLIDS AND THIN STRUCTURES

The **Solid-Shell Connection** multiphysics coupling has been renamed **Solid-Thin Structure Connection** in order to reflect that it now also supports couplings to the Membrane interface. There are also some minor changes to the GUI of the coupling.

COROTATING LOADS ON RIGID DOMAIN

When applying loads in a **Rigid Domain** node through the **Applied Force** subnode, it is now possible to specify that the load direction follows the rotation of the body. The new **Direction** setting has two options: **Space-fixed direction** and **Body-fixed direction**.

DEFAULT LOAD PLOTS

All structural mechanics interfaces will generate default vector plots of applied loads. By running **Get Initial Value**, you can get such plots generated for checking input data. During result evaluation, the load plots will display the loads corresponding to the current time or parameter value.

The load plots use standard plot types under **Results**. The options are chosen so that you can easily customize the plots to get a good visual impression.

As part of this project, new variables containing the loads are defined in all load features. These variables have names that are unified between all load features in all structural mechanics interfaces. Any old variables are still available for backward compatibility, with one exception: the variable `solid.Fp`, which was defined by the **Point Load** node. Any expression in an existing model containing, for example, `solid.Fpy`, must be replaced by `solid.F_Py`.

NEW SETTINGS FOR ROLLER IN SOLID MECHANICS

The settings in the **Roller Constraint** section in the **Roller** node have been changed for 2D and 2D axisymmetry in order to reflect the limitations and interpretations that the respective space dimensions impose.

THERMAL BENDING FOR LAYERED LINEAR ELASTIC MATERIAL IN THE SHELL INTERFACE

In the **Thermal Expansion** subnode under the **Layered Linear Elastic Material** in the Shell interface, it is now also possible to enter a through-thickness temperature difference or temperature gradient. This will generate bending thermal strains.

HYGROSCOPIC BENDING FOR LAYERED LINEAR ELASTIC MATERIAL IN THE SHELL INTERFACE

In the **Hygroscopic Swelling** subnode under the **Layered Linear Elastic Material** in the Shell interface, it is now also possible to enter a through-thickness concentration difference or concentration gradient. This will generate bending hygroscopic swelling strains.

LOCAL EDGE SYSTEM FOR BOUNDARY CONDITIONS IN THE MEMBRANE INTERFACE

For boundary conditions applied to edges in the Membrane interface, it is now possible use also a local edge system for defining the orientation. The following boundary conditions are affected: **Edge Load**, **Prescribed Displacement**, **Prescribed Velocity**, **Prescribed Acceleration**, **Spring Foundation**, and **Added Mass**.

IMPROVEMENTS FOR SHELL LOCAL SYSTEM IN 2D AXISYMMETRY

The **Shell Local System** node now has a **Coordinate System Selection** section also in 2D axisymmetry. Only boundary systems can be selected. The main purpose is to access the option to flip the normal orientation that exists in a boundary system.

The **Shell Local System** node has also been added to the **Rigid Domain** material model in 2D axisymmetry.

WAVE SPEED COMPUTATION IN ELASTIC MATERIALS

For the **Linear Elastic Material** and **Piezoelectric Material**, it is now possible to compute wave speeds for elastic waves. To do that, add the **Wave Speeds** node, available under **Variables** in the context menu for the material models or under **Attributes** on the ribbon when the material model is selected.

The effect of this node is to make variables for phase and group velocities for elastic waves available for result presentation. The solution as such is not affected, and the variables will be available for any study type.

DISPLAY OF NORMAL DIRECTIONS IN SHELL AND MEMBRANE INTERFACES

When physics symbols are switched on in the Shell or Membrane interface, and a material model node is selected in the Model Builder, a green arrow will indicate the positive normal direction for each boundary.

SAFETY MOVED IN CONTEXT MENUS

The **Safety** feature, which is available for many material models in various physics interfaces, has been moved in the context menu. When right-clicking on a material model, you will now find it under a subfolder called **Variables**.

DEFAULT PLOTS GENERATED BY SAFETY NODE

When you add a **Safety** node in one of the Shell, Layered Shell, or Membrane interfaces, a default plot with the failure index is generated. Such plots are placed in a group named **Failure Indices**. The label of these plots is derived from the label of the corresponding **Safety** node.

NEW GROUPING OF DEFAULT PLOTS IN BEAM INTERFACE

All default plots of section forces in the Beam interface are now placed inside a group named **Section Forces** in order to reduce the number of result nodes that are directly visible.

DEFAULT PLOTS FILTERED BY ACTIVATION STATUS

When an **Activation** node is present, the default surface plots will get a **Filter** subnode. The filter serves the purpose of suppressing the plotting of domains not currently activated.

BACKWARD COMPATIBILITY DISCONTINUED FOR PRE-4.4 BEAM INTERFACE

The **Backward Compatibility** section in the settings for the Beam interface has been removed. It is no longer possible to open models containing a Beam interface built in versions earlier than 4.4. If you need to convert such models, open them in any version between 4.4 and 5.4 and convert to the more recent formulation.

ERROR AND WARNINGS IN EXTERNAL MATERIALS

When programming your own constitutive relations using the external materials functionality, certain code segments might return errors or warnings. The general stress-strain relation, the general stress-deformation relation, the inelastic residual strain, and the inelastic residual deformation now include the possibility to add an argument to return detailed internal error messages and to display warnings in the solver log.

New Models in Version 5.5

EIGENMODES OF A VISCOELASTIC STRUCTURAL DAMPER

In this model, methods for performing eigenfrequency analysis of materials with frequency-dependent material properties are explored. In this situation, the eigenvalue problem becomes nonlinear. In the example, model methods are used for finding the eigenfrequencies one-by-one.

BIMETALLIC STRIP IN AIRFLOW

This is a conceptual model illustrating how to couple fluid-structure interaction, heat transfer, and thermal expansion. The new **Fluid-Solid Interaction, Conjugate Heat Transfer** multiphysics interface is used. A bimetallic strip in an air channel is heated so that it bends. After some time, an airflow with an inlet temperature that varies in time is introduced. As a result, there are changes in deformation caused both by the fluid pressure and by the convective cooling.

TWO-PHASE FLOW WITH FLUID-STRUCTURE INTERACTION

This example demonstrates techniques for modeling a fluid-structure interaction containing two fluid phases in COMSOL Multiphysics. It illustrates how a heavier fluid can induce movement in an obstacle using the arbitrary Lagrangian-Eulerian (ALE) technique. The **Fluid-Solid Interaction, Two-Phase Flow, Phase Field** is used.

COUPLED ANALYSIS OF FLOW AND STRESS IN A PIPE

In this tutorial model, the flow in a pipe with a bend is computed using the Pipe Flow interface. The computed fluid load is used as input to a stress analysis in the Pipe Mechanics interface. Gravity loads from the pipe and fluid are also taken into account.

Running the full example requires the Pipe Flow Module.

SENSITIVITY ANALYSIS OF A TRUSS TOWER

Sensitivity analysis is an efficient way of computing the gradient of an objective function with respect to many control variables. This example uses the pitch and yaw in the top of a truss tower as objective functions. It shows how to compute the sensitivity of these angles with respect to changes in the individual truss member diameters.

BRACKET — TOPOLOGY OPTIMIZATION

This example shows how to perform a topology optimization with several load and constraint cases. The density feature for topology optimization is used together with a parametric sweep where each optimized solution is used as input for a further improved optimization. The final optimized geometry is exported and then imported for a verification analysis in which the now void domain is excluded.

BRACKET — RANDOM VIBRATION ANALYSIS

This example shows how to perform a random vibration analysis of a structure using power spectral density (PSD). The computations are based on the modal reduced-order model (ROM).

RANDOM VIBRATION ANALYSIS OF A DEEP BEAM

In this example, forced random vibrations of a simply-supported deep beam are studied. The beam is loaded by a distributed force with a uniform power spectral density (PSD). The output PSD is computed for the displacement and bending stress response. The computed values are compared with analytical results.

BLOCK PRESSING ON ARCH

This conceptual example shows how to calculate critical points in models with contact. The model consists of a block modeled with the Solid Mechanics interface pressing on an arch modeled with the Shell interface. The contact problem is solved using the augmented Lagrangian method. The block is loaded with a displacement-controlled boundary load and is incrementally pressed toward the arch. During loading, there is a snap-through behavior of the arch but the solution remains stable.

INSTABILITY OF TWO CONTACTING ARCHES

This is a benchmark example where instability is studied in a model with contact. Two contacting arches are modeled in the Shell interface. The contact problem is solved using the penalty method. The upper arch is loaded with displacement-controlled edge loads and is pressed toward the lower arch. During loading, a snap-through behavior of the lower arch is observed with several critical points related to sideways instability.

In order to obtain a stable reference solution, the sideways movement of the mid edge of both arches is constrained.

ISOTROPIC-ANISOTROPIC SAMPLE: ELASTIC WAVE PROPAGATION

In this 2D tutorial model, a test sample consists on one side of an isotropic material and on the other side of a heterogeneous anisotropic material (a transverse anisotropic zinc crystal). Elastic waves in the sample are excited by a point-like force. The model is solved with the **Elastic Waves, Time Explicit** physics interface.

GROUND MOTION AFTER SEISMIC EVENT: SCATTERING OFF A SMALL MOUNTAIN

In this tutorial, the propagation of elastic waves in the ground after a seismic event is simulated using a 2D model. The effect of the ground surface topology on the wave propagation is illustrated when an ideal half space is modified with the presence of a small mountain. The model is a variation of Lamb's problem. The propagation of the elastic waves is modeled using the **Elastic Waves, Time Explicit** physics interface and the model captures the propagation and scattering of pressure waves, shear waves, Rayleigh waves, and von Schmidt waves.

Backward Compatibility with Version 5.2

The friction slip velocity is no longer defined as a dependent variable. When running a model using the COMSOL API, you will need to remove the reference to this variable in the solver settings. The friction slip velocity is usually defined as `<comp>_<solid>_vslip_<pairname>`, where `<comp>` is the tag of the component, `<solid>` is the tag of the Solid Mechanics physics interface where the **Friction** node is defined, and `<pairname>` is the name of the contact pair.

Subsurface Flow Module

New Functionality in Version 5.5

- Ambient properties, such as temperature, relative humidity, absolute pressure, wind velocity, and solar irradiance, can be defined in the Ambient Properties node. Beside the possibility to add user-defined meteorological data, two meteorological datasets are available that contain ambient data from several weather stations in the world. When transient studies are performed, the climate data is automatically synchronized. In addition, a new boundary condition is added to the Darcy's Law interface to account for the precipitation rate given in the Ambient Properties node.
- In the Brinkman Equations and Free and Porous Media Flow interfaces, the Porous Matrix Properties section in the Fluid and Matrix Properties domain feature now contains a **Non-Darcian** option in the drop-down list for the **Permeability model**. This option also replaces the old Forchheimer Drag subfeature.
- In the Darcy's Law interface, the Matrix Properties section in the Fluid and Matrix Properties domain feature now contains a **Non-Darcian** option in the drop-down list for the **Permeability model**. This can be used to add a Forchheimer drag term to the momentum equation. The Darcy's Law interface also has a new setting, at the interface level, for including gravity effects.
- The Inlet and Outlet boundary conditions in the Darcy's Law interface have been extended to include the possibility to specify the pressure and the mass flow in the Inlet node, and the pressure in the Outlet node.
- New adsorption models for mass transport in porous media. The Toth and BET models are now available in the **Adsorption** node.

NEW HEAT TRANSFER IN FRACTURES INTERFACE

The new **Heat Transfer in Fractures** interface models heat transfer in fractured porous media. Using this interface prevents representing the fracture thickness in the geometry. The interface provides a lumped model for cost-effective modeling, and dedicated settings to define the fluid and the material properties. The flow velocity in the fracture can be taken from the Fracture Flow interface.

NEW REACTING FLOW IN POROUS MEDIA INTERFACE

The **Reacting Flow in Porous Media, Transport of Diluted Species** multiphysics interface is used to study the flow and chemical composition of a gas or liquid moving through

the interstices of a porous medium. The multiphysics interface combines the **Brinkman Equations** and **Transport of Diluted Species in Porous Media** interfaces. The **Reacting Flow, Diluted Species** multiphysics coupling is added to couple fluid flow and mass transport. The chemical species are assumed to be solutes dissolved in a solvent of significantly higher concentration.

New Models in Version 5.5

FROZEN INCLUSION

This benchmark problem studies the melting process of an ice inclusion within a porous medium, and it demonstrates how to couple the Darcy's Law interface with the Heat Transfer in Porous Media interface including phase change.

Backward Compatibility with Version 5.4

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

Old Java files need to replace the line

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

with

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

Forchheimer is the default. Also replace

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

with

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

Backward Compatibility with Version 5.3a and Earlier

NO FLUX FORMULATION

The formulation of the No Flux feature has been changed in the Transport of Diluted Species and Transport of Diluted Species in Porous Media interfaces in version 5.4 so

that the flux relative to the convective flux is set to zero. The new formulation is used when opening a model from a previous session. In the previous formulation, the total flux was set to zero.

Wave Optics Module

New Functionality in Version 5.5

COUPLING FROM FULL-WAVE TO RAY OPTICS SIMULATION

Two new features, Release from Electric Field and Release from Far-Field Radiation Pattern, have been added to the Geometrical Optics interface in the Ray Optics Module that either release rays based on a near- or far-field computed from a full-wave simulation. The full-wave field can be produced by the Electromagnetic Waves, Frequency Domain and Electromagnetic Waves, Beam Envelopes interfaces in the Wave Optics Module or the Electromagnetic Waves, Frequency Domain interface in the RF Module.

GAUSSIAN BEAM INPUT OPTION FOR SCATTERING AND MATCHED BOUNDARY CONDITIONS

The Scattering and the Matched boundary conditions have a new option for the Incident field list. Choosing the Gaussian beam option allows you to propagate a Gaussian beam into an arbitrary direction. The Gaussian beam is defined using the paraxial Gaussian beam formula.

POLARIZATION PLOTS AND JONES VECTOR VARIABLES

Periodic ports now create a new default Polarization plot. The Polarization plot depicts the polarization state for the different diffraction orders and is based on new postprocessing variables for Jones vectors elements. Also, the base vectors, used for defining the Jones vectors, are available for plotting and evaluation.

EVANESCENT WAVES INCLUDED IN THE PLANE-WAVE EXPANSION FOR GAUSSIAN BEAM BACKGROUND FIELDS

When using the Plane wave expansion option for defining a Gaussian beam background field, evanescent waves can also be included in the expansion. This option can be useful when simulating a tightly focused (spot radius smaller than the wavelength) Gaussian beam propagating away from the focus.

SLIT PORTS ARE AVAILABLE ALSO FOR THE UNIDIRECTIONAL FORMULATION FOR THE ELECTROMAGNETIC WAVES, BEAM ENVELOPES INTERFACE

Slit ports are now available also for the Unidirectional formulation for the Electromagnetic Waves, Beam Envelopes interface. Slit ports can be useful if several modes are propagating, but only the reflectance or transmittance of one of the modes is of interest. Then, use a domain-backed slit port for the mode of interest and let the rest of the modes be absorbed by a perfectly matched layer (PML) in the domain behind the port.

REFERENCE POINT SUBFEATURE ADDED TO AND UPDATED FOR THE SCATTERING AND MATCHED BOUNDARY CONDITION FEATURES

The Reference Point subfeature is now available in the Scattering and Matched Boundary conditions for the Electromagnetic Waves, Frequency Domain and Electromagnetic Waves, Beam Envelopes interfaces, when an input field is active. The reference position is defined as the average position of the selected points. This feature is mainly useful when the domain material includes absorption or gain.

PORT UTILITY

Cutoff Frequency Calculator

A cutoff frequency calculator is available from the rectangular and circular port type Setting windows. This helps to quickly identify the cutoff frequency for a given mode in an arbitrary sized waveguide.

ADDITIONAL 3D FAR-FIELD FUNCTIONS FROM 2D AXISYMMETRIC COMPONENT MODEL

Effective 3D antenna gain and realized gain functions for 2D axisymmetric models are added.

PHYSICS-CONTROLLED MESHING FOR PAIR FEATURES

The physics-controlled meshing algorithm now produces copied meshes for pair features.

TOTAL INTERNAL REFLECTION

This model demonstrates that the bidirectional formulation of the Electromagnetic Waves, Beam Envelopes interface can be used for simulating a beam being reflected in two main directions as it propagates along a waveguide.

An almost collimated Gaussian beam is excited at the left boundary and exhibits total internal reflection (TIR) at the interface between the waveguide and the surrounding air. The beam exits the waveguide after propagating along the waveguide for 20 mm.

This type of light guide is useful for virtual reality (VR) simulation.

ORBITAL ANGULAR MOMENTUM BEAM

This model simulates a Laguerre–Gaussian beam with the Electromagnetic Waves, Beam Envelopes interface, using the unidirectional wave formulation. The input beam is a focusing Gaussian beam with a spiral phase distribution.

This phase distribution produces a Gaussian donut beam. The phase rotates around the optical axis as the beam propagates. The resulting beam is called a vortex beam.

FREQUENCY-SELECTIVE SURFACE, PERIODIC COMPLEMENTARY SPLIT RING RESONATOR

Frequency selective surfaces (FSS) are periodic structures with a bandpass or a bandstop frequency response. This example shows that only signals around the center frequency can pass through the periodic complimentary split ring resonator layer.

This model demonstrates the use of slit ports and shows how to use the Adaptive Frequency Sweep study step.

DIRECTIONAL COUPLER

This model has been extended to include the following three parts:

- Unidirectional propagation, showing basic modeling steps when using the Electromagnetic Waves, Beam Envelopes method. This part was available in 5.4.
- Bidirectional copropagation, showing that the mesh requirements can be reduced by propagating the two modes using the bidirectional formulation. This part was also available in 5.4.
- Approximation of a target input field using overlap integrals for calculating the exciting ports' input powers and mode phases. New in 5.5.

The model is also restructured to keep the physics interfaces, meshes, studies, and plot groups for the three different modeling parts.

FOCUSING LENS

This model has been updated to use the Transition boundary condition to represent an antireflection (AR) coating on the lens boundaries.

UPDATED MODELS

Application Library Paths for Models Using the New Gaussian Beam Input Option for the Scattering and Matched Boundary Conditions

- Wave_Optics_Module/Nonlinear_Optics/self_focusing
- Wave_Optics_Module/Optical_Scattering/beam_splitter
- Wave_Optics_Module/Optical_Scattering/brewster_interface

Application Library Paths for Models Using the New Polarization Plot

- Wave_Optics_Module/Gratings_and_Metamaterials/frequency_selective_surface_csrr
- Wave_Optics_Module/Gratings_and_Metamaterials/hexagonal_gratings
- Wave_Optics_Module/Verification_Examples/fresnel_equations

Material Library

New and Updated Material Data in Version 5.5

In version 5.5, the following improvements have been made:

- Surface tension data has been added to 115 datasets as `SurfF` for the surface tension coefficient.
- The thermal conductivity for 304 stainless steel above room temperature was revised. Data from a new reference was added. The new values are the same as the old values near 20°C but are 35% lower at 600°C.
- The thermal conductivity for 316 stainless steel above room temperature was revised. Data from a new reference was added. The new values are within 5% of the old values at all temperatures.
- Data was added for several Kynar grades — 460, 710, 720, 740, and 760 — and for Pebax.
- Data was added for E glass and S-2 glass fibers and for MgB2.
- Data was added for the C18150, ZEK100, CMSX-4, NARloy-Z, GRCop-84, Cu-4Cr-2Nb, Fe-22Mn, Fe-24Mn, API X80, 1144, and 4142 alloys.
- Data was added for Bi2Te3 (n- and p-type).
- The elastic properties for Ti-6Al-4V and Inconel 718 were refit. The largest change was in Poisson's ratio. For 718, the change was a few percent. For Ti-6Al-4V, the change was approximately 15% near room temperature and close to 0 at 930°C.
- Data was added for RAFM steel (9Cr-1W), 304L weld metal, CMSX-4, CMSX-10, Thermo-Span alloy, PWA 1485, SRR99, MAR M247, and Rene 94.
- Data was added for H13, Zn-48Sb, Modified 9Cr-1Mo, A316, 5115 steel, API X60 steel, Mg-4Li, CRONIDUR 30, and 409 wire.
- Data was added for FGH96, ZK60, AZ91, AZ91 Be, AM60, AMZ40, AJ62, Nitinol, CMSX-4, 25MnCrSiVB6, 50CrMo4 (4150), 100CrMnMoSi8, Ir, Rh, Pt alloys, Sn-8Zn-3Bi,ScF3, SrF2, LaNbO4, propylene glycol, Li7Sn3, Li13Sn5, Li7Sn2, Li17Sn4, and ZrB2-B4C-SiC.
- The reference "E.P. Papadakis, Materials Science and Engineering, v. 10, p. 195 (1972)" was used for the expansion of several materials. This reference has been replaced with new references for the affected materials: 317/317L, FS-85 (Nb-28Ta-10W-1Zr), Cb-752 (Nb-10W-2.5Zr), Inconel 700, and ZrSiO4.

- Data was added for Pt-10Rh, Pt-20Rh , P91, P92, 9 Cr - 1 Mo - V - Nb (T91), Ferritic ductile iron (3.3 C - 3.25 Si - 0.13 Mn - 0.002 Ce), Inconel 617, 38MnVS & 25MnCrVS, ZrN, P4M, HT7, 5754, Cu, W-5Ni-2Fe, Nb-10Ti-5Zr, Nb-10W-5Zr, Nb-5Mo-5V-1Zr, and Nb - 15 W - 5 Mo - 1 Zr.
- The reference for the thermal expansion of TZM was changed. The new values are 50% higher near room temperature and 10% higher at elevated temperatures.
- Data was added for Ti-1555A, Ti-150A, Ti-140A, C-130AM, AM60B and Inconel 617 welds.
- Data was added for the following acrylics: PLEXIGLAS G, PLEXIGLAS MC, Perspex-CQ, Acrylite OP-1, and Solacryl 2750. The expansion data for the PMMA was changed to be the average of these new acrylics.
- The reference for the vapor pressure of iron was corrected.
- The elastic properties of copper were extended from 300 K to 1330 K.
- Data was added for Invar, Ta, Nb-10W-5Zr, 304, and 1018, and for hydrocarbons and molten salts (used for energy storage).

LiveLink™ for MATLAB®

New Functionality in Version 5.5

LiveLink™ *for* MATLAB® now supports MATLAB® 2019a and 2019b.

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

USER INTERFACES

The user interface mphnavigator has been updated with new functionality for updating the properties and methods panes using a button on the toolbar.

There is a new tool for copying code to the clipboard that shows how to make a loop for the currently selected node in the model tree. For a work plane, the copied code may look like this:

```
tag1 = 'geom1';
tag2 = 'wp2';
tags = cell(model.geom(tag1).feature(tag2).geom.feature.tags);
for i=1:length(tags)
    obj = model.geom(tag1).feature(tag2).geom.feature(tags{i});
end
```

This feature saves a lot of manual typing. The copied code can then be tailored to a specific use and inserted in scripts and functions.

Mphmodellibrary, which is used to browse the Application Libraries, now also show any user-defined libraries. You can add user-defined libraries using the COMSOL Desktop.

PLOTTING

The function `mphplot` has been updated to support new plot types added to COMSOL 5.5.

Mphplot and other plotting function have improved support for plotting in windows created by the App Designer, which uses `uiaxes` for plotting instead of the regular axes objects in MATLAB figures and user interfaces created by GUIDE.

MPHGETADJ

An extra output has been added to `mphgetadj`:

```
[n, m] = mphgetadj(model,geomtag,returntype,adjtype,adjnumber)
```

returns the indices of all the adjacent geometry entities in N. M contains the indices of entities that connect the entities the best. The latter can be used to, for example, extract edge entities that connect a certain set of points.

MPHMEASURE

Mphmeasure is a new function. It measures entities in a geometry.

```
[m1,m2]=mphmeasure(model,geomtag,entity,…)
```

Based on the entity, you can measure quantities such as distances, areas, and volumes in a geometry.

MPHTABLE

This function now supports evaluation groups as well as tables.

Backward Compatibility with Version 5.4

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

Backward Compatibility with Version 5.2

The functions `mpheval`, `mphglobal`, and `mphinterp`, which extract numerical data, now support complex values by default; that is, the property '`complexout`' is '`on`' by default. If you want to have the previous behavior, set '`complexout`' to '`off`'.

Backward Compatibility with Version 5.0

- The `mphint2` command: For data sets other than Solution, Particle, Cut Line, Cut Plane, Time Integral, Time Average, Surface, and Line, the integration order now corresponds to the element refinement. The default value is set to 4 instead of 1. This change can affect the solution in cases where the problem is not resolved well enough.

The COMSOL API for Use with Java®

COMSOL 5.4 API Changes

CAD IMPORT MODULE, DESIGN MODULE, AND LIVELINK PRODUCTS FOR CAD CHANGES

The new `fillholes` property has been added for the `Import` function. The following applies:

- The combination of the `fillholes` property set to `on` and the `knit` property set to `solid` is equivalent to the `knit` property set to `solid` in previous versions.
- The combination of the `fillholes` property set to `off` and the `knit` property set to `surface` is equivalent to the `knit` property set to `surface` in version 5.3a.
- The combination of the `fillholes` property set to `on` and the `knit` property set to `surface` is equivalent to the `knit` property set to `surface` in version 5.3 and previous versions.

The new `fillholes` property has been added for the `Knit` function. The `Knit` function with the `fillholes` property set to `on` corresponds to the `Knit` function in previous versions.

ECAD IMPORT MODULE CHANGES

- The properties `selnetcontributetobnd`, `selnetcontributetodom`, `selnetkeepbnd`, `selnetkeepdom`, `selnetnamebnd`, `selnetnamedom`, `selnetshowbnd`, `selnetshowdom`, `selnettagbnd`, and `selnettagdom` have been added to the `Import` function. The properties apply to the IPC-2581 and ODB++ file types and enable working with the selections generated for the electrical nets found in the imported files.
- The `elevation` property of the `Import` function is now initialized when setting the `manualelevation` property to `off`. Previously, it was only initialized when setting the `filename` property and when the `manualelevation` property is `off` and the value of either the `height` or `importtype` properties is changed.

COMSOL 5.3a API Changes

PARTITION OBJECTS PROPERTIES

The `keep` property has been deprecated in version 5.3a. Instead, use the new properties `keepinput` and `keeptool`. If you set `keep` to `on`, both `keepinput` and `keeptool` are set to `on`. If you get the value of `keep`, it is `on` if `keepinput` and `keeptool` are `on`.

COMSOL 5.3 API Changes

NEW COMPONENT SYNTAX

The geometry component is now included in the code for features that appear in a component. For example,

```
model.geom("geom1")
```

in earlier versions is now instead

```
model.component("comp1").geom("geom1")
```

to indicate which component the geometry belongs to. To keep the old syntax in generated Model Java® files, clear the **Use component syntax** check box under **Code generation** on the **Methods** page in the **Preferences** window. All old scripts and Java® source code files will continue to work using the old syntax.

ECAD IMPORT MODULE CHANGES

- The default value for the `sellayer` property of the `Import` function is now `on`. Previously, the default was `off`.
- The default value for the `sellayershow` property of the `Import` function is now `all`. Previously, the default was `dom`.

COMSOL 5.2a API Changes

AC/DC MODULE CHANGES

New Coil Feature

In the Magnetic Fields, Magnetic and Electric Fields, and Rotating Machinery, Magnetic interfaces, a new unified **Coil** feature has replaced the **Multi-Turn Coil** feature. The functionality of the old **Multi-Turn Coil** feature can be replicated by using the new **Coil** feature with the Conductor model set to Homogenized Multi-Turn, but all of the

other parameters are unchanged. The only change required is to the create statements such as:

```
model.physics("mf").feature().create("mtcd1",
"MultiTurnCoilDomain", 3);
```

which should be modified to:

```
model.physics("mf").feature().create("mtcd1", "Coil", 3);
model.physics("mf").feature("mtcd1").set("ConductorModel",
"Multi");
```

User-Defined Coil Geometry Subfeature Selection

In 3D models, when using a user-defined geometry for a **Coil** feature, the **User Defined Coil Geometry** subfeature has been transformed from a boundary feature to a domain feature. New **Input** and **Output** subnodes are used to select the coil input or output boundaries instead. Any code setting the selection of this feature should be modified to set the selection on the **Input** subfeature instead.

Relative Permeability for Magnetic Shielding in Magnetic Fields, No Current

The relative permeability parameter `mur` in the **Magnetic Shielding** feature in the Magnetic Fields, No Currents interface is now a scalar quantity. Setting a tensor quantity will not cause exception, but only the first component of the tensor will be used as the value.

ACOUSTICS MODULE CHANGES

The default settings have changed for the Incident Pressure Field in the Pressure Acoustics interfaces. Add the following line to obtain the old behavior of this feature when `ipf1` is a subfeature to `pwr1`:

```
model.physics("acpr").feature("pwr1").feature("ipf1").
set("c", "acpr.c_c");
```

COMSOL 5.1 API Changes

ACOUSTICS MODULE CHANGES

The ratio of specific heats is now a material parameter. If the default value was used in a model, add the following line to the Java® file to obtain the old behavior:

```
model.physics("cpf").feature("cpf1").set("gamma_mat", "userdef");
```

ELECTROCHEMISTRY MODULES CHANGES

There are changes for the Tertiary Current Distribution, Nernst-Planck interface. The Java® API syntax for creating and accessing vectors and tensors has changed, as well as the syntax for setting physics properties. See the following examples:

- Specifying the diffusion tensor.

  Old syntax:

```
model.physics("tcdee").feature("ice1").set("D_0", new
String[]{"D1", "0", "0", "0", "D1", "0", "0", "0", "D1"});
model.physics("tcdee").feature("ice1").set("D_1", new
String[]{"D2", "0", "0", "0", "D2", "0", "0", "0", "D2"});
model.physics("tcdee").feature("ice1").set("D_2", new
String[]{"D3", "0", "0", "0", "D3", "0", "0", "0", "D3"});
```

  New syntax (includes the species name in the first argument of the set method):

```
model.physics("tcdee").feature("ice1").set("D_c1", new
String[]{"D1", "0", "0", "0", "D1", "0", "0", "0", "D1"});
model.physics("tcdee").feature("ice1").set("D_c2", new
String[]{"D2", "0", "0", "0", "D2", "0", "0", "0", "D2"});
model.physics("tcdee").feature("ice1").set("D_c3", new
String[]{"D3", "0", "0", "0", "D3", "0", "0", "0", "D3"});
```

- Accessing tensor components (in the definitions of other variables, for example).

  Old syntax:

```
Dxx_c1, Dxy_c1, Dxz_c1...
```

  New syntax (component index after the species name):

```
D_c1xx, D_c1xy, D_c1xz...
```

- Specifying the reaction rate vector.

  Old syntax:

```
model.physics("tcdee").feature("reac1").set("R_0", new
String[]{"R1"});
model.physics("tcdee").feature("reac1").set("R_1", new
String[]{"R2"});
model.physics("tcdee").feature("reac1").set("R_2", new
String[]{"R3"});
```

  New syntax (includes the species name in the first argument of the set method):

```
model.physics("tcdee").feature("reac1").set("R_c1", new
String[]{"R1"});
model.physics("tcdee").feature("reac1").set("R_c2", new
String[]{"R2"});
model.physics("tcdee").feature("reac1").set("R_c3", new
String[]{"R3"});
```

- Accessing vector components.

  Old syntax:

  ```
  tfluxx_c1, tfluxy_c1, tfluxz_c1
  ```

  New syntax (component index after the species name):

  ```
  tflux_c1x, tflux_c1y, tflux_c1z
  ```

- Specifying initial values.

  Old syntax:

  ```
  model.physics("tcdee").feature("init1").setIndex("c2", "c20", 0);
  model.physics("tcdee").feature("init1").setIndex("c3", "c30", 0);
  model.physics("tcdee").feature("init1").setIndex("V", "V0", 0);
  ```

  New syntax:

  ```
  model.physics("tcdee").feature("init1").setIndex("initc", "c20",
  1);
  model.physics("tcdee").feature("init1").setIndex("initc", "c30",
  2);
  model.physics("tcdee").feature("init1").setIndex("initphil",
  "V0");
  ```

- Selecting properties.

  Old syntax:

  ```
  model.physics("tcdee").prop("Convection").set("Convection", 1,
  "0");
  model.physics("tcdee").prop("ConvectiveTerm").setIndex("Convectiv
  eTerm", "noncons", 0);
  ```

  New syntax (the property name corresponds to the section in the user interface):

  ```
  model.physics("tcdee").prop("TransportMechanism").
  set("Convection", 1, "0");
  model.physics("tcdee").prop("AdvancedSettings").setIndex("Convect
  iveTerm", "noncons", 0);
  ```

For information about API backward compatibility for the Surface Reactions interface, see the Chemical Reaction Engineering Module release notes.

ECAD IMPORT MODULE CHANGES

- The default value for the `grouping` property of the `Import` function is now set to `layer`. Previously, the default was `all`.
- For ODB++® and ODB++(X) files (the `type` property is `odb`), new rules apply for the initialization of the `importlayer` property of the `Import` function. Now, only

layers of the types *Metal* and *Dielectric* are initialized with the string `on` in the `importlayer` string array. Previously this also included layers of type *Drill*.

LIVELINK™ FOR SOLID EDGE® CHANGES

The default value for the `keepfree` property of the `LiveLinkSolidEdge` function is now set to `on`. Previously, the default was set to `off`.

LIVELINK™ FOR SOLIDWORKS® CHANGES

The default value for the `keepfree` property of the `LiveLinkSOLIDWORKS` function is now set to `on`. Previously, the default was set to `off`.

COMSOL 5.0 API Changes

DOCUMENTATION

The *COMSOL Multiphysics Programming Reference Manual* replaces the *COMSOL API for Use with Java® Reference Manual*.

DEPRECATED METHODS

The following methods were deprecated in COMSOL 5.0:

TABLE 1-3: DEPRECATED METHODS AND THEIR REPLACEMENTS

DEPRECATED METHOD	NEW METHOD
`com.comsol.model.OptFeature.field()`	`OptFeature.comp()`
`com.comsol.model.OptFeature.field(String)`	`OptFeature.comp(String)`
`com.comsol.model.Group.identifier()`	`Group.paramName()`
`com.comsol.model.Material.identifier()`	`ModelEntity.tag()`
`com.comsol.model.MaterialModel.identifier()`	`ModelEntity.tag()`
`com.comsol.model.ModelNode.identifier()`	`ModelEntity.tag()`
`com.comsol.model.physics.Physics.identifier()`	Use `tag()` instead.
`com.comsol.model.physics.MultiphysicsCoupling.identifier()`	`ModelEntity.tag()`
`com.comsol.model.Group.identifier(String)`	`Group.paramName(String)`
`com.comsol.model.Material.identifier(String)`	`ModelEntity.tag(String)`
`com.comsol.model.MaterialModel.identifier(String)`	`ModelEntity.tag(String)`

TABLE 1-3: DEPRECATED METHODS AND THEIR REPLACEMENTS

DEPRECATED METHOD	NEW METHOD
`com.comsol.model.ModelNode.identifier(String)`	`ModelEntity.tag(String)`
`com.comsol.model.physics.Physics.identifier(String)`	Use `tag(string)` instead.
`com.comsol.model.physics.MultiphysicsCoupling.identifier(String)`	`ModelEntity.tag(String)`
`com.comsol.model.ModelEntity.name()`	`ModelEntity.label()`
`com.comsol.model.ModelEntity.name(String)`	`ModelEntity.label(String)`
`com.comsol.model.ParameterEntity.set(String, int, double)`	Use the 0-based version (`setIndex`) instead.
`com.comsol.model.ParameterEntity.set(String, int, double)`	Use the 0-based version (`setIndex`) instead.
`com.comsol.model.ParameterEntity.set(String, int, double[])`	Use the 0-based version (`setIndex`) instead.
`com.comsol.model.ParameterEntity.set(String, int, int)`	Use the 0-based version (`setIndex`) instead.
`com.comsol.model.ParameterEntity.set(String, int, int[])`	Use the 0-based version (`setIndex`) instead.
`com.comsol.model.ParameterEntity.set(String, int, int, double)`	Use the 0-based version (`setIndex`) instead.
`com.comsol.model.ParameterEntity.set(String, int, int, int)`	Use the 0-based version (`setIndex`) instead.
`com.comsol.model.ParameterEntity.set(String, int, int, String)`	Use the 0-based version (`setIndex`) instead.
`com.comsol.model.ParameterEntity.set(String, int, String)`	Use the 0-based version (`setIndex`) instead.
`com.comsol.model.ParameterEntity.set(String, int, String[]`	Use the 0-based version (`setIndex`) instead.

This concludes the release notes for COMSOL Multiphysics version 5.5.

Index