



COMSOL Multiphysics

Release Notes

5.0

 COMSOL

COMSOL Multiphysics Release Notes

© 1998–2014 COMSOL

Protected by U.S. Patents listed on www.comsol.com/patents, and U.S. Patents 7,519,518; 7,596,474; 7,623,991; 8,219,373; and 8,457,932. Patents pending.

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

Support for implementation of the ODB++[®] Format was provided by Mentor Graphics Corporation pursuant to the ODB++ Solutions Development Partnership General Terms and Conditions. ODB++ is a trademark of Mentor Graphics Corporation.

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

Version:

October 2014

COMSOL 5.0

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
- 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



Release Notes

COMSOL[®] version 5.0 introduces the Application Builder, a comprehensive set of tools for creating and deploying custom applications with custom user interfaces based on COMSOL models. Also new in version 5.0, the Design Module extends the COMSOL software's 3D CAD capabilities, and the Ray Optics Module provides tools for ray optics and for modeling of systems in which the electromagnetic wavelength is much smaller than the smallest geometric detail in the model. A new LiveLink[™] for Revit[®] makes it possible to import CAD data from Autodesk's Revit building design software. There are also many new functions and additions to the COMSOL product suite. These *Release Notes* provide information regarding new functionality in existing products.

COMSOL Multiphysics

New Products in Version 5.0

RAY OPTICS MODULE

The new Ray Optics Module provides tools for ray optics, also known as geometrical optics or ray tracing. Electromagnetic plane wave propagation can be approximated with rays for electrically large-enough objects (relative to wavelength). The Ray Optics Module is useful for simulations of, for example, optical systems such as lenses, image aberration calculations, and illuminations, also in combination with multiphysics simulations in COMSOL Multiphysics.

DESIGN MODULE

The new Design Module extends the CAD Import Module (and includes all its functionality) with the following useful features for 3D CAD modeling: 3D fillets and 3D chamfers, along with loft, midsurface, and thicken operations.

LIVELINK FOR REVIT

The new LiveLink™ for Revit® provides synchronization of building designs between the Autodesk Revit® Architecture building design software for Building Information Modeling (BIM) and COMSOL Multiphysics. The synchronization can be configured to automatically create the geometric objects of selected rooms from the Revit design. Geometric objects for room bounding elements (for example, walls and roofs) and for other elements (for example, doors, windows, or furniture) can also be transferred during synchronization. Synchronized elements are automatically added to selections that can be used when defining materials or boundary settings in the COMSOL model.

Major New Functionality in Version 5.0

APPLICATION BUILDER FOR CREATING AND DEPLOYING APPLICATIONS

The Application Builder is included in the Windows® version of COMSOL Multiphysics. A COMSOL application is a COMSOL Multiphysics model with a user interface, which you can extend using the Form Editor. With the Method Editor you can also add code to extend the functionality of the application. You can deploy applications to run as web clients in a web browser, for example. See the *Introduction*

to *Application Builder* and *Application Builder Reference Manual* for more information.

General New Functionality

IMPROVED HANDLING OF MATERIALS

There is a new Material branch under the Global branch in the Model Builder, which makes it possible to define materials once and then use these in several model components, using a Material Link node. You can also add a Switch node, under which you can add materials that you want to sweep over in a Material Sweep study step.

PIECEWISE FUNCTIONS

For piecewise functions, a **Smooth at end points** check box is available when you apply smoothing so that you can activate smoothing at the end points (start and end) of the piecewise function.

TRANSPORT OF DILUTED SPECIES

The name (default prefix) for the Transport of Diluted Species is now more intuitive: tds (previously: chds).

New Geometry and Mesh Functionality

CREATE GEOMETRY FROM MESH

A new **Create Geometry from Mesh** button is available on the Mesh toolbar. You can use it to create a new model component, with a geometry import feature that imports the mesh as geometry. You can then add more geometry features as usual.

GENERAL MESH COPY OPERATION

A new **Copy** feature makes it possible to copy a mesh from another component (an imported mesh, for example) and insert that copy in the meshing sequence where you add the Copy node. The mesh copy makes it possible to, for example, add geometry, typically defining a surrounding domain, to an imported mesh while still keeping the imported mesh on the original geometry. The Copy feature is available in 2D and 3D meshing sequences based on a geometry.

GEOMETRY SUBSEQUENCE IMPROVEMENTS

Linked Subsequences

You can store a number of subsequences in a Model MPH-file, making up a library of such subsequences. From a model, you can link to one or more subsequences from the library using **Global>Geometry>Link Subsequences**. The chosen subsequences are imported to the model, and appear as **Linked Subsequence** nodes under **Global>Geometry**. Use the **Relink** button to import the subsequence again after changing the library. A linked subsequence can call another (linked or not) subsequence, even if they are defined in different Model MPH-files.

Use of Global Parameters

You can now use global parameters within a subsequence.

Improved Associativity

There is now full associativity for the resulting objects of a Subsequence Call. If you change the Subsequence Call feature so that it calls another subsequence, you get no associativity because there is no associative connection between different subsequences. In that case, using selections defined within the subsequence can be helpful.

PARTITION EDGES VIRTUAL GEOMETRY OPERATION

The **Partition Edges** operation partitions selected edges at specified positions in a 3D geometry. You can specify the positions either by entering arc length-based parameters or by selecting existing vertices whose orthogonal projections on the edges specify the positions.

LOCAL COORDINATE VARIABLES

Local (barycentric) coordinates ξ_i in each mesh element are available as **xi1**, **xi2**, and **xi3**.

New Functionality in Studies and Solvers

MATERIAL AND FUNCTION SWEEPS

New **Material Sweep** and **Function Sweep** study steps are parametric sweeps over different materials and functions, which you add under Switch nodes, which you can add under Definitions and Materials.

NODAL CONSTRAINT METHOD

A new *nodal constraint method* is available as an alternative to the elemental constraint method available before. The nodal method has advantages such as:

- Multipoint constraints are assembled together before the constraint handling takes place. This makes the process of eliminating the constraints a simpler problem because, in contrast to the elemental method, most of the constraints are linearly independent. This makes the sparse null-space function useful in more situations.
- The orthonormal null-space function can work on smaller blocks because there are fewer constraints.

Also, new **Explicit-Orthonormal** and **Explicit-Sparse** null-space functions are available. These methods eliminate a subset of the nodal constraints (explicit constraints). The remaining constraints are handled by the orthonormal or the sparse method. The **Automatic** null-space function uses the explicit methods if there are explicit constraints in the model. Most continuity and periodicity constraints are explicit when the nodal method is used.

The new method does not affect weak constraints or flux conditions.

FFT AND IFFT STUDY STEPS AND SOLVER

New **Frequency to Time FFT** and **Time to Frequency FFT** study steps includes an FFT and IFFT (inverse FFT/FFT) solver that computes a discrete Fourier transformation for time-dependent or frequency-dependent input solutions. The input solution can have real or complex-valued data. A window function can be used to restrict the input data.

STUDY REFERENCES AND SEQUENCES

Use the new **Study Reference** node to refer to another study in the model. You can use a study reference node to combine and nest several studies. A **Sequence** node (under **Job Configurations**) is generated automatically by a Study Reference when it is used without any Parametric Sweep study step. It then runs the referenced studies in a sequence.

RESET SOLVER TO DEFAULT

A new option resets the solver nodes in all attached solver configurations under the **Solver Configurations** node to the default solvers you get when you select **Show Default Solver** and that the study uses if you have not made any changes to the solver settings. Using **Reset Solver to Default** is useful if you have tried various solver settings and want to return to the default solvers without having to create a new solver configuration.

PREVIOUS SOLUTION OPTIONS FOR TIME-DEPENDENT SOLVER

A **Previous Solution** subnode to the **Time Dependent Solver** node handles field variables that have to be accessed at a previous parameter value or time. In time-dependent studies, it can be useful for time-dependent contact problems with friction, for example.

EIGENVALUE SEARCH REGION

A new eigenvalue search method that searches for an approximate number of eigenvalues within a specified region is now available. You can specify the approximate and maximum number of eigenvalues and the smallest and largest real and imaginary parts for the search region.

ADDITIONAL NEW AND IMPROVED FUNCTIONALITY

- Parametric sweeps and frequency sweeps now support units for the parameters.
- Compute buttons and corresponding keyboard shortcuts are now available for all study step nodes. Units, however, are not stored or loaded from file.
- Anderson acceleration is available for accelerating the nonlinear solver and provide more efficient time stepping for fluid dynamics models.
- Improved data set and solver sequence naming for easier identification of which solver configuration's data a data set refers to.
- An option to continue the solution process is now available from the Progress window.
- Improved usability and simplified settings for the domain decomposition solver.
- An SAI (sparse approximate inverse) preconditioner is available as preconditioner, pre- and postsmoother, and coarse grid solver. A blocked version, which searches for and exploits repeated sparsity patterns, is active by default.
- A new automatic option for computing the in-core memory for the MUMPS and PARDISO direct solvers is now the default instead of specifying the in-core memory directly.
- A matrix-free format is available in addition to the sparse and full matrix formats for the solvers. This option assembles the matrix vector product when needed. This can reduce the memory usage significantly for solvers that support the following formats:
 - The **Iterative** solver without preconditioner
 - The **Krylov Preconditioner** without preconditioner

- The **Derived Value** subnode in **Job Configurations>Parametric** has a new **Update table** list for controlling the behavior of table updates. It replaces the **Clear previous** check box in version 4.4. It is now possible to display and export several numerical values in a parametric sweep. This list adds some convenient options for the table updates:
 - **Clear initial table** (selected when the **Clear previous** check box was selected in 4.4 models)
 - **Append data to table** (selected when the **Clear previous** check box was cleared in 4.4 models)
 - **Clear table for every new parameter value**: the table is cleared for every parameter in the sweep.
- Using probes to track and plot quantities while solving is now more efficient and adds less computational overhead than in earlier versions.
- You can now get outputs from probes and plots while solving also for a stationary solver using settings in the new **Results While Solving** section in the Stationary Solver node's Settings window (except when added as a subnode to a Optimization solver node).
- Support for the SLURM job scheduler in Cluster Computing study steps.

New and Improved Results and Visualization Functionality

- A new *Spectrum* color table includes violet at the small-wavelength end of the visible spectrum (which the Rainbow color table does not). It also includes richer shades of green to more closely replicate the human perception of visible light. You can use it with the Ray Optics Module, for example, to accurately visualize polychromatic light.
- You can now edit table headers. When a table is imported from file, the headers are taken from the last comment row preceding the data.
- Preprocessing of table graph and table surface plots: The x , y , and data values can be scaled and translated to better match the values of other data sets, for example.
- Better control of the view scaling (aspect ratio). You can choose no view scaling, automatic view scaling, and manual scaling, where you can manually set the view scale factors in the x , y , and z directions.
- In tables, you can now select a column and right-click to copy that column (with or without its header) to the clipboard. You can also select and copy the contents of a single table cell.
- Contour lines can be plotted as tubes.

- Support for integration and average with axial symmetry taken into account is available in probes and evaluation data sets.
- In plots and results evaluation, the **Insert Expression** and **Replace Expression** menus contain submenus under **Built-In** for all available **Mathematical functions**, **Operators**, and **Physical constants**.

General Backward Compatibility Considerations

COMSOL 5.0 is backward compatible with COMSOL versions 4.0–4.4. To open models created using COMSOL version 3.5a in version 5.0, follow these steps:

- 1 Open and then save the COMSOL 3.5a mode using any of the COMSOL versions 4.0–4.4.
- 2 Open the model saved in versions 4.0–4.4 in COMSOL 5.0.

Backward Compatibility With Version 4.4

COMSOL SERVER

The COMSOL Server from version 4.4 is now called the COMSOL Multiphysics Server.

MESH IMPORT

An edge in an imported mesh that has a common start and end vertex, or that lacks start and end vertices, is now split into two edges with distinct start and end vertices. For meshes where this happens, the numbering of all geometric entities may change when the mesh is rebuilt.

For Model MPH-files created in earlier versions, selection are automatically updated with the new entity numbers.

LOCAL COORDINATE VALUES

For model components created in version 5.0 and onward, the variable names `xi1`, `xi2`, and `xi3` are reserved for the predefined local coordinate variables. To get the previous behavior (that local coordinate variables do not exist), you can use the API method `model.modelNode(<tag>).defineLocalCoord(false)`.

TRANSPORT OF DILUTED SPECIES

Running Model Java-files from previous versions may fail due to the new default Name for Transport of Diluted Species. This can be avoided by adding a command that

specifies the identifier in accordance with the Name of the interface. For example, when creating an interface using the old Name (chds):

```
model.physics().create("chds", "DilutedSpecies", "geom1",  
    new String[][]{{"c"}});
```

Add the following line to specify the identifier accordingly:

```
model.physics("chds").identifier("chds");
```

The Java API syntax for creating and accessing vectors and tensors in the Transport of Diluted Species interface has changed as well as the syntax for settings physics properties. This does not affect Model MPH-files. See the backward compatibility notes for the Chemical Reaction Engineering Module for additional information that also applies to the Transport of Diluted Species interface.

DOCUMENTATION

The *COMSOL Multiphysics Programming Reference Manual* replaces the *COMSOL API for use with Java[®] Reference Manual*.

Backward Compatibility With Version 4.3b

MODEL NODES ARE NOW COMPONENT NODES

The Model nodes (as they were called in previous versions), which contained separate model components in a model file, are called **Component** nodes in version 5.0.

CHANGES TO PARAMETRIC SWEEPS

Old models that use stationary parametric sweeps are loaded with the **Reuse solution for previous step** list set to **Yes**. The **Run continuation for** list is set to the parameter used, unless the continuation algorithm would not have been used for this model in previous versions (for example, if multiple parameters are used or if the parameter list is not monotonous).

CHANGES TO THE PHYSICS SELECTION IN STUDY SETTINGS

The names of the states of a physics in the physics tree, which you can modify under **Physics and Variables Selection** in the study steps' settings windows, have changed:

- **Provide Degrees of Freedom** is now called **Disable in Solvers**.
- **Disable** is now called **Disable in Model**.

REVISED FORMULATION FOR LAMINAR INFLOW AND LAMINAR OUTFLOW

The formulations of the laminar inflow and laminar outflow conditions have been corrected. The modified formulation gives a more accurate mass flux. Some models may now produce a slightly different flow field.

Laminar inflow and laminar outflow are available in the fluid flow physics interfaces in the following modules:

- Batteries and Fuel Cells Module
- CFD Module
- Corrosion Module
- Electrochemistry Module
- Electrodeposition Module
- Heat Transfer Module
- Microfluidics Module
- Plasma Module
- Subsurface Flow Module

NEW DEFAULT FOR GRAPHICS OPTIMIZATION

Under **Graphics and Plot Windows** in the **Preferences** dialog box, the default settings in the **Optimize for** list is now **Quality** instead of **Performance**.

VELOCITY/ACCELERATION INTEGRATION VARIABLE

For the Solid Mechanics interface (and all related multiphysics interfaces) and the Truss interface, a help variable u_0 (velocity integration variable) is used in **Prescribed Velocity** and **Prescribed Acceleration** features for Time Dependent study types. This variable computes the displacement for each point where the condition is prescribed. It is changed to use the full feature scope in order to avoid collisions in cases where several such features exist within the same model component. As a result, when opening and running any old model that uses such features together with a segregated solver, an error message appears, stating that not all dependent variables occur in at least one of the segregated solver steps. The relevant action is to manually add the velocity integration variable to the segregated step containing the corresponding displacement field. Alternatively, you can regenerate any affected solver sequence.

MATH LIBRARIES ON AMD PROCESSORS

MKL is now the default math library on AMD processors. Switching to the ACML math library for AMD processors might improve performance in some cases.

ERROR ESTIMATION CHANGES IN THE SOLVERS

The **Automatic** method for **Check error estimate** for direct linear solvers and for **Validate error estimate** has changed. For nonlinear problems and for time-dependent problems the underlying nonlinear solver does not accept terminating for a linear solution step that does not fulfill the error estimate (unless the step size is very small). The motivation for this change is that the old method can lead to premature termination of the nonlinear solution process, which in some cases introduces a large error for the computed solution.

This change can cause the solvers to take more nonlinear steps for stationary problems and more time steps for time-dependent problems, and it may also lead to convergence problems. For such cases, use the **No** method to obtain the old behavior. However, doing so can hide numerical problems and potentially lead to large numerical errors.

TERMINATION CRITERION FOR STATIONARY SOLVERS

The default termination criterion for stationary solvers has changed. In 4.3a the settings corresponded to **Solution**; now the default is **Solution or residual**. This change in default termination criterion might affect models created in earlier versions of COMSOL if you regenerate the solver sequence in 5.0.

BACKWARD EULER INITIALIZATION TIME STEP

A new setting in the **Advanced** section of the settings window for the **Time-Dependent Solver**, called **Fraction of initial step for Backward Euler**, provides an option for entering a dimensionless quantity that determines the size of the time step for the backward Euler method (in terms of the initial step). This value can improve the accuracy of the initialization step but can also affect the start-up of some models. The default value is 0.001 (this differs from earlier versions, which used a value of 1). When opening models created in version 4.3a or earlier, the value for this fraction is set to 1 to maintain compatibility with those versions.

LOADING EXTERNAL PHYSICS BUILDER JAR FILES

External physics builder JAR archives compiled with earlier versions of COMSOL include a manifest file that contains a reference to the CDO library, which is removed. To load such JAR files in version 5.0, you must first do one of the following:

- Delete the META-INF/MANIFEST.MF file in the archive source on the file system, and then recompile the JAR archive.
- Manually remove the line with `org.eclipse.emf.cdo` in the META-INF/MANIFEST.MF file in the JAR archive. You can do so directly in a file archive manager such as 7-Zip or similar.

HIGHLIGHTING GEOMETRY OBJECTS WHEN DRAWING IN 2D

When you have drawn one geometry object on top of another object, toggling of the highlighting of these objects occurs when you click several times. It is important that you click without moving the cursor in a position where the objects overlap in order to toggle. For example, if you draw a circle (C1) and then draw another smaller circle (C2) inside of C1, then a first click on C2 may highlight C1, while a second click, without moving the cursor, highlights C2. In previous versions, you would move the cursor after the first click in order to highlight C2.

Backward Compatibility With Version 4.3

NEW TERMINOLOGY FOR CONSTRAINT TYPES

The following constraint types have new names in version 5.0:

- *Bidirectional, symmetric* is now *Apply reaction terms on: All physics (symmetric)*.
- *Unidirectional* is now *Apply reaction terms on: Individual dependent variables*.

WEAK CONSTRAINTS UPDATE FOR THE LAMINAR FLOW INTERFACE

The weak constraint formulations for the following boundary conditions has been updated:

- Symmetry
- The Slip option in the Wall feature

These boundary conditions are now formulated using the same set of Lagrange multipliers as all the other boundary conditions in the Laminar Flow interface. The Lagrange multiplier `un_lm` has been removed.

Models saved in version 4.3 will include `un_1m` until the model is re-solved. In some cases, occurrences of `un_1m` in the solver sequence must be replaced manually. This is the case if `un_1m` was the only Lagrange multiplier component in a segregated group or the only Lagrange multiplier component of a Vanka smoother. Alternatively, you can generate a new automatic solver sequence. Models saved in versions earlier than 4.3 must either be re-solved in version 5.0 for postprocessing, or opened and re-saved in version 4.3 before being opened in version 5.0.

AUTOMATICALLY CREATED IDENTITY/CONTACT PAIRS

The pairs created by the Form Assembly feature in the geometry can differ from their form in 4.3 for certain 3D and 2D geometries. The pairs should now be the same for the CAD and COMSOL representations.

New Model in Version 5.0

ANISOTROPIC HEAT TRANSFER THROUGH WOVEN CARBON FIBERS

Carbon-fiber-reinforced polymers contains woven carbon fibers that have a thermal conductivity along the fiber axis is much higher than perpendicular to it. This tutorial model shows how to use the curvilinear coordinates interface to compute the local fiber orientation and to use it to define anisotropic thermal conductivity of fibers.

AC/DC Module

New Functionality in Version 5.0

AUTOMATED MESHING

Meshing of periodic structures and infinite element domains is now optionally automated through the physics-controlled auto mesh suggestion:

- Copy mesh is applied for periodic conditions
- Swept (3D) or mapped (2D) meshing is applied for infinite elements

NEW OPTIONS FOR ELECTRIC CURRENTS

The electric currents interface is enhanced with the following options:

- Dielectric loss and loss tangent material models
- Power terminal excitation

ENHANCEMENTS FOR MAGNETIC MODELING

- Stabilization for 3D Multi-Turn Coil fixes issues with inaccurate induced electric field, induced voltage, and coil impedance.
- The new floating potential condition for the electric potential in the 3D Single-Turn Coil allows for more flexible excitations.
- Terminals and Floating Potential now include the possibility of specifying initial values for the global degrees of freedom, which can be useful in time-dependent and nonlinear models.
- Domain and boundary conditions in Rotating Machinery, Magnetic are improved. The thin-layer boundary conditions (shielding, for example) now fully support the mixed formulation.
- Usability of the Magnetic and Electric Fields interface, in particular of the electric boundary conditions, is enhanced.

Acoustics Module

New Functionality in Version 5.0

NEW MULTIPHYSICS COUPLINGS AND MULTIPHYSICS INTERFACES

New multiphysics couplings and predefined multiphysics interfaces replace and upgrade the existing multiphysics capabilities of the Acoustics Module. For example, you can now couple an acoustic fluid domain to a solid mechanics domain by adding an acoustics physics interface and a Solid Mechanics interface separately and then coupling them at the boundary using the new multiphysics coupling. With this approach you get more flexibility by increasing the complexity of a model sequentially. This new formulation also makes it possible to decouple the two contributing physics and gives access to all functionality of the acoustics interface and the functionality included in the solid interface (depending on the available licenses).

The new multiphysics couplings available under the *Multiphysics* node with the Acoustics Module include:

- *Acoustic-Structure Boundary* for coupling the Pressure Acoustics interfaces to solid structures including the Solid Mechanics, Shell (interior and exterior), Membrane, and Multibody Dynamics interfaces.
- *Thermoacoustic-Structure Boundary* for coupling Thermoacoustics, Frequency Domain to solid structures including Solid Mechanics, Shell (interior and exterior), Membrane, and Multibody Dynamics.
- *Aeroacoustic-Structure Boundary* for coupling Linearized Navier-Stokes, Frequency Domain to solid structures including Solid Mechanics, Elastic Waves, Shell (interior and exterior), Membrane, and Multibody Dynamics.
- *Acoustic-Porous Boundary* for coupling the Pressure Acoustics interfaces to a Porous Material domain from the Poroelastic Waves (or Elastic Waves) interface.
- *Porous-Structure Boundary* for coupling Solid Mechanics to Porous Material domains from the Poroelastic Waves (or Elastic Waves) interface.
- *Acoustic-Thermoacoustic Boundary* for coupling the Pressure Acoustics interfaces to Thermoacoustics, Frequency Domain.

- *Background Potential Flow Coupling* for the one-way coupling of Compressible Potential Flow to the Linearized Potential Flow interfaces.
- *Piezoelectric Effect* connects a Solid Mechanics interface and an Electrostatics interface for modeling piezoelectric materials.



The Shell (interior and exterior) and Membrane interfaces require the Structural Mechanics Module. The Multibody Dynamics requires the Multibody Dynamics Module.

Several predefined multiphysics interfaces have been added, which automatically add the contributing physics interfaces and the multiphysics couplings. These include:

- Acoustic-Solid Interaction, Frequency Domain and Transient
- Acoustic-Shell Interaction, Frequency Domain and Transient (requires the Structural Mechanics Module)
- Acoustic-Piezoelectric Interaction, Frequency Domain and Transient
- Acoustic-Elastic Waves Interaction
- Acoustic-Poroelastic Waves Interaction
- Acoustic-Thermoacoustic Interaction, Frequency Domain
- Thermoacoustic-Solid Interaction, Frequency Domain
- Thermoacoustic-Shell Interaction, Frequency Domain (requires the Structural Mechanics Module)

Existing models with the old multiphysics interfaces (`acs1`, `astd`, `acsh`, `acshtd`, `acpz`, `acpztd`, `ta`, `tas`, `tash`, and `e1w`) still use the old interfaces for backward compatibility purposes.

NEW ACOUSTICS PHYSICS INTERFACES

This version includes four new physics interfaces: *Linearized Navier-Stokes, Frequency Domain*; *Linearized Navier-Stokes, Transient*; *Ray Acoustics*; and *Acoustic Diffusion Equation*.

A new *Geometrical Acoustics* branch has been added to the Acoustics Module with the Acoustic Diffusion Equation and the Ray Acoustics interfaces. These are interfaces for modeling acoustics in the high frequency limit where the wavelength is much smaller than the characteristic geometric features. This is at frequencies above the Schroeder frequency for rooms. Both physics interfaces are suited for modeling acoustics in rooms and concert halls. The Acoustic Diffusion Equation interface is

restricted to indoor applications whereas the Ray Acoustics interface can be used, for example, in ocean acoustics and atmosphere acoustics.

The *Acoustic Diffusion Equation* (*ade*) interface includes the following features:

- Define rooms and their acoustic properties
- Absorption properties of walls
- Coupling of rooms with transmission loss
- Domain and wall sources
- Enter material properties and sources in octave and 1/3 octave bands

The *Ray Acoustics* (*rac*) interface includes the following features:

- Detailed wall treatment for specular, diffuse, and mixed reflections
- Intensity calculations
- Frequency and ray direction dependent wall conditions
- Release features for the acoustic rays including options for the frequency distributions, intensity, and wave front of the rays
- Material discontinuity conditions
- Determine impulse response and plot with new 1D Ray Plots

Under the *Aeroacoustics* branch you find the new Linearized Navier-Stokes, Frequency Domain (*lnsf*) and Linearized Navier-Stokes, Transient (*lnst*) interfaces. These physics interfaces are used to compute the acoustic variations in pressure, velocity, and temperature in the presence of any stationary isothermal or non-isothermal background mean flow. You can use these physics interfaces for aeroacoustic simulations that can be described by the linearized Navier-Stokes equations. Coupling the frequency domain interface to structures using the new Aeroacoustic-Structure Boundary multiphysics coupling makes it possible to perform detailed vibration analyses of structures in the presence of flow, such as FSI in the frequency domain. Boundary conditions include:

- Wall
- Interior Wall and Interior Impedance
- Mechanical conditions such as no slip, slip, stress, and impedance
- Thermal conditions (isothermal, adiabatic, heat flux)
- Domain sources

UPDATED AND IMPROVED PHYSICS INTERFACES

Several physics interfaces are updated and improved: Linearized Euler, Frequency Domain; Linearized Euler, Transient; Thermoacoustics, Frequency Domain; Poroelastic Waves; and Elastic Waves:

- Under the *Aeroacoustics* branch the Linearized Euler interfaces are updated with new boundary conditions:
 - Interior Wall
 - Interior Normal Impedance
 - Asymptotic Far-Field Radiation
 - Pressure (Isentropic)
- The Poroelastic Waves (*pe1w*) and Elastic Waves (*e1w*) interfaces are updated. The predefined multiphysics interfaces coupling to pressure acoustics replace the old interfaces.
- Thermoacoustics, Frequency Domain (*ta*) is updated with new Wall, Interior Wall, and Interior Impedance conditions. The Acoustic-Thermoacoustic Interaction predefined multiphysics interface replaces the old Thermoacoustics interface.

NEW FLUID MODELS

The *Poroacoustics* and *Narrow Region Acoustics* domain features in the Pressure Acoustics, Frequency Domain interface are updated with several new fluid models.

In Poroacoustics for modeling porous materials in a homogenized manner:

- Zwikker-Kosten
- Attenborough
- Wilson
- Johnson-Champoux-Allard-Lafarge
- Johnson-Champoux-Allard-Pride-Lafarge

In Narrow Region Acoustics for including thermal and viscous losses of the acoustic boundary layer:

- Slit
- Circular duct
- Rectangular duct
- Equilateral duct
- User defined, for entering complex wave number and specific impedance

PIEZOELECTRIC DEVICES

The Piezoelectric Devices interface is replaced by a multiphysics coupling called *Piezoelectric Effect*, which connects a Solid Mechanics interface and an Electrostatics interface.

- The new formulation makes it possible to decouple the structural and electrical effects and gives access to all functionality in the Solid Mechanics and Electrostatics interfaces.
- In the Solid Mechanics interface, there is a new node *Piezoelectric Material*, with optional subnodes *Thermal Expansion* (requires the Structural Mechanics Module or the MEMS Module), *Initial Stress and Strain*, *Mechanical Damping*, *Coupling Loss*, *Dielectric Loss*, and *Conduction Loss (Time Harmonic)*.
- The *Dielectric Loss* node has an option to use the dielectric dispersion, which you can use for modeling electrical losses in both the time domain and the frequency domain.
- In the Electrostatics interface, there is a new *Charge Conservation, Piezoelectric* node.

Existing models that include a Piezoelectric Devices interface will still use the old interface.

New Models in Version 5.0

- Piezoelectric Tonpitz Transducer
- Acoustic Reflections off a Water-Sediment Interface
- Lumped Receiver Connected to Test Setup with an 0.4-cc Coupler
- Test Bench Car Interior

Backward Compatibility With Version 4.3a and Older

- The old style perfectly matched layer (PML) that is located under the physics node will be discontinued in the next version of COMSOL. From the next version on only the PMLs defined under the Definitions node are supported.
- In order for the old style perfectly matched layer (PML) to be backward compatible, the PML feature must be placed after any domain Monopole source, domain Dipole source, or Background Pressure Field features.

Backward Compatibility With Version 4.3

- The symbols for the acoustics monopole (Q_m ; was Q) and dipole (\mathbf{q}_d ; was \mathbf{q}) volume sources in pressure acoustics have changed.
- The default value for the Typical wave speed property in the Acoustic-Piezoelectric Interaction, Frequency Domain interface has changed to 343 m/s.

Batteries & Fuel Cells Module

New Functionality in Version 5.0

NEW PHYSICS INTERFACE FOR MASS TRANSPORT AND REACTIONS

This version includes a major overhaul of the mass transport and reactions functionality. The old *Transport of Diluted Species* for free flow and *Species Transport in Porous Media* for porous media are now unified in one physics interface called *Transport of Diluted Species in Porous Media*. This physics interface has many new strengths in addition to better user-friendliness: In addition to support for *convection* and *migration in electric fields* it now also includes adsorption, dispersion, and volatilization in saturated and partially saturated porous media:

- New entry point in the Model Wizard: *Transport of Diluted Species in Porous Media*. This is a Transport of Diluted Species interface with a default Porous Media Transport Properties domain feature activated.
- Replaces and unifies the old Solute Transport and Species Transport in Porous media interfaces.
- Introduces unified nomenclature for all mass transport.
- Stabilization introduced for porous media mass transport.
- The new *Dispersion* section in the Settings window makes it possible to model dispersion in isotropic and transverse anisotropic porous media.
- The *Diffusion* section now supports modeling of diffusion in isotropic and anisotropic porous media.
- The dependent variables for concentration are defined in the unit mol/m³ in all mass transport interfaces.
- A new *Mass-Based Concentrations* feature provides results output in mass-based quantities such as mass fractions and mass concentrations.
- Improved nomenclature for porous media mass transfer.
- Existing Model MPH-files created with the now retired Species Transport in Porous Media and Solute Transport open in those physics interfaces, but they are not available in the Model Wizard for new models.

The names (default prefix) for the mass transport interfaces are now more intuitive:

- Transport of Diluted Species: **tds** (previously: **chds**)

- Transport of Diluted Species in Porous Media (tds)
- Transport of Concentrated Species: tcs (previously: chcs)
- Surface Reactions: sr (previously: chsr)

IMPROVED BATTERY INTERFACES

The following improvements are available in the Lithium-Ion Battery and Battery with Binary Electrolyte interfaces:

- Updated Porous Electrode and Additional Intercalating Material features:
 - These domain nodes now use the extra dimensions machinery to implement the porous electrode. This allows improved postprocessing capabilities and space dimension dependency in the particle dimension.
 - The intercalating concentration discretization order default has changed from 2 to 1. Model MPH-files from version 4.4 and earlier are migrated to 5.0 to use linear elements for the extra dimension discretization, with the number of extra dimensions discretization elements multiplied by the original element order. In this way the number of degrees of freedom is preserved; however, small differences in `cs_surface`, `cs_center`, and `cs_average` may arise due to different meshes used when changing the element order. Model Java API code is not migrated; that is, the number of degrees of freedom may change.
 - Updated solver defaults. The scale for the intercalation concentration is now set manually to 10000. Model MPH-files from earlier versions using different manual scales are set to the new default.
 - Changed initial value for electrolyte concentration dependent variable to 1000.
- Electrolyte and Separator: Model inputs for concentration not matched by default, and a list for material selection is added to be consistent to how the Porous Electrode node works.
- Electrolyte, Separator, and Porous Electrode: Concentration dependence of the activity dependence (`dlnfd1nc`) can now come from materials. The default value of `dlnfd1nc` has changed to 1. This may affect Java API backwards compatibility.

ADDITIONAL NEW FUNCTIONALITY

- A new *Current Distribution on Edges*, BEM interface for modeling of currents on edges using the boundary element method.
- New *Danckwerts* inflow boundary condition in the Transport of Diluted Species interface's Inflow node.

- New *Equilibrium Reaction* and *Surface Equilibrium Reaction* nodes in the Transport of Diluted Species, Electroanalysis, and Tertiary Current Distribution, Nernst-Planck interfaces.
- New *Counter electrode* boundary condition in the Electrode Surface feature in the Electroanalysis interface.
- New *Periodic Condition* boundary node in the Primary Current Distribution and Secondary Current Distribution interfaces.
- Updated Equilibrium Potential section in Electrode Reaction and Porous Electrode Reaction nodes' Settings windows. The equilibrium potential parameter is now split from the dE_{eq}/dT parameter. Both parameters can be taken from the material.
- Improved solver suggestions and changed naming of the global potential degrees of freedom created by the total or average current boundary conditions in Electrolyte Current, Electrode Current, Electrode Power, Electrode-Electrolyte Boundary Interface, Electrolyte-Electrode Boundary Interface and Electrode Surface nodes. The changes may affect API backward compatibility.
- Improved automatic meshing: shared points (2D and edges (3D) between Electrode Reaction boundaries are now set to an *extremely fine* mesh size by default (in all electrochemistry interfaces that include the Electrode Reaction boundary node).
- Updated Batteries and Fuel Cells material library. New lithium ion battery electrode and electrolyte materials added. Added and updated parameter values in existing materials. Lithium-ion battery models in the model library have been updated to use the new materials

New Models in Version 5.0

- Lithium-Ion Battery Impedance Model tutorial model with parameter estimation using features from the Optimization Module.
- Lithium Rate Capability, Internal Resistance, and Drive Cycling tutorial models.

Backward Compatibility With Version 4.3

BRINKMAN EQUATIONS AND FREE AND POROUS MEDIA FLOW

The momentum equations in the Brinkman Equations interface and the Free and Porous Media Flow interface have been corrected. The term $-Q_{br} \cdot \mathbf{u} / \epsilon_p^2$ in the right-hand side previously lacked the factor $1/\epsilon_p^2$, where ϵ_p is the porosity.

REACTING FLOW INTERFACE NAME CHANGE

The Reacting Flow interfaces have been renamed *Reacting Flow in Porous Media*. Opening a model using either the Reacting Flow, Diluted Species or the Reacting Flow, Concentrated Species interface, the interface is renamed Reacting Flow in Porous Media.

WEAK CONSTRAINTS UPDATE FOR FLUID FLOW INTERFACES

The weak constraint formulations for some boundary conditions in the following interfaces have been updated:

- Laminar Flow
- Brinkman Equations
- Free and Porous Media Flow

See [CFD Module](#) for details.

MOBILITY

In physics user interfaces modeling migration (Transport of Diluted Species; Transport of Concentrated Species; Reacting Flow in Porous Media, Diluted Species; Reacting Flow in Porous Media, Concentrated Species; Tertiary Current Distribution; and Nernst-Planck) anisotropic mobility is now supported, and the mobility can be set to follow the Nernst-Einstein relation. The default setting when making new models is changed to use the Nernst-Einstein relation. Model files for Java generated prior to 4.3b using Migration will have to be modified manually to account for these changes.

CAD Import Module and LiveLink Products for CAD

New Product in Version 5.0

The LiveLink™ for Revit® is new in version 5.0, and provides the possibility to link COMSOL models with geometries created using Autodesk's Revit® Architecture building design software for building information modeling (BIM).

New Functionality in the CAD Import Module and LiveLink Products for CAD in Version 5.0

EXTENDED CAD FILE IMPORT

The CAD Import Module and the LiveLink™ products for CAD can now import AutoCAD® (.dwg and 3D .dxf files), and NX® (-prt) files.

These products also support new versions for many of the existing file formats (see <http://www.comsol.com/products/specifications/cad/>).

CAD IMPORT MODULE GEOMETRY KERNEL UPGRADE

The CAD Import Module and the LiveLink™ products for CAD utilizes 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 5.0 includes an upgraded version of the Parasolid kernel and as a result a number of stability issues have been fixed, which makes import of CAD models and solid operations more robust.

New Functionality in LiveLink™ for AutoCAD®

- Parameters from the AutoCAD® model can now be selected to be transferred to the COMSOL software during synchronization. In the COMSOL model, global parameters are created and linked automatically to the AutoCAD parameters. This provides a more effective set-up of models with parametric sweeps or optimization. Units for parameters are now also supported.
- The LiveLink™ for AutoCAD® interface now supports AutoCAD 2015.

New Functionality in LiveLink™ for Inventor®

- Expanding on the functionality that synchronizes selections based on material assignments to the CAD design in Inventor®, the LiveLink™ interface now adds support for user-defined selections. In the added COMSOL Selections interface in Inventor, you can define selections that are synchronized to the COMSOL model. You can choose to synchronize selections for assembly components, features, bodies, faces, edges, or points, which become selections in the model when the design is synchronized with the COMSOL Desktop. Setting up a model becomes more efficient as you can also create selections from features of the Model Builder or from components of an assembly.
- To the extent possible, the LiveLink interface now synchronizes only changed geometric objects. This results in improved synchronization times for larger geometries.
- Synchronization of solid and surface bodies and assembly components can now be controlled by their visibility setting in the Inventor design. Only bodies and components with their visibility turned on are synchronized.
- The LiveLink™ interface now supports Inventor 2015.

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

The LiveLink™ interface now supports PTC® Creo® Parametric™ 3.0.

New Functionality in LiveLink™ for Solid Edge®

The LiveLink™ interface now supports Solid Edge® ST7.

CFD Module

New Functionality in Version 5.0

ALGEBRAIC TURBULENCE MODELS

The two new turbulence models, the *Algebraic yPlus* model and the *L-VEL* model, are so-called enhanced viscosity models. The models are based on the momentum balance in the wall layer, and are suitable for interior flows such as in electronic cooling applications. The algebraic turbulence models are computationally less expensive and more robust but, in general, less accurate than transport equation models like the *k-ε* model or the Spalart-Allmaras model. Since they are algebraic, no additional boundary or initial conditions are needed. The no slip condition is applicable on walls. The new turbulence models are available in the following physics interfaces:

- Single-Phase Flow
- Non-Isothermal Flow
- Conjugate Heat Transfer

ABSOLUTE PRESSURE ANNOUNCEMENT

From now, the Fluid Flow interfaces announce the absolute pressure. Hence, each interface using a relative pressure internally has a user input to define the reference pressure level. From that the physics interface announces the absolute pressure as the sum of the relative pressure and the reference pressure level.

CAVITATION FOR THIN-FILM FLOW

The ability to perform simple modeling of cavitation effects in bearings has been added to the Thin Film Flow interfaces. Gaseous cavitation can be treated using Elrond's algorithm.

FAN, INTERIOR FAN, AND GRILLE FEATURES FOR TURBULENT FLOW

The Fan, Interior Fan, and Grille features have been updated to take turbulence into account. The supported turbulence models are: Algebraic yPlus, L-VEL, *k-ε*, *k-ω*, low-Reynolds *k-ε*, and SST.

FAN AND GRILLE FEATURES UPDATED

For the Inlet flow direction, it is now possible to specify if the flow is normal or to set a particular direction for the inlet velocity.

NEW INLET FEATURE FOR FLUID FLOW

The new Inlet feature only has one pressure option compared to two options in the old feature. The Settings window includes an option to suppress backflow analogous to the option available on outlets. The new inlet pressure condition requires a flow direction, which can be either normal flow (the default), or a user-defined direction.

The pressure point constraint used by the mass flow option is replaced by a normal stress condition in order to improve mass conservation. It now also supports weak constraints.

The velocity and laminar inflow options remain the same as before.

IMPROVEMENTS TO TURBULENCE MODELS

- The formulations of all turbulence models have been reviewed and revised for improved stability and nonlinear convergence. The main changes include regularizations and tuning of stabilization constants for both turbulence equations and the Navier-Stokes equations.
- The SST turbulence model is available in the Reacting Flow interface. SST is a low-Reynolds number model, which for reacting flow means that it supports the high Schmidt number model for turbulent mixing.

IMPROVED DEFAULT VALUES FOR PHASE VISCOSITIES

The default values for the continuous phase viscosity and the dispersed phase viscosity (in cases for which the dispersed phase is set to bubbles/liquid droplets) in the Euler-Euler model now use Krieger-type expressions for better robustness.

IMPROVED WALL DISTANCE FUNCTIONALITY

- The Continuity feature in the Wall Distance interface is updated with a discontinuous Galerkin formulation, which is more robust for iterative solvers.
- There is a new Periodic Condition feature in the Wall Distance interface. It also uses a discontinuous Galerkin formulation, which means that it works well with iterative solvers and for non-matching meshes.
- The technology used in these new features in the Wall Distance standalone interface has also been implemented for all turbulence models using a wall distance equation.

PIPE FLOW IMPROVEMENTS

- The 1D Pipe Flow interface can now be coupled to the 3D Laminar Flow interface through the Pipe Connection feature.
- In harmony with the Non-Isothermal Flow interface, the Non-Isothermal Pipe Flow interface has an optional pressure work term that can be activated. The term can be important to activate if the pressure drop is expected to be considerable and the fluid is compressible. The pressure-volume work is then added to the energy balance.

PSEUDO TIME STEPPING FOR MIXTURE MODEL AND BUBBLY FLOW

The Bubbly Flow and Mixture Model interfaces now support pseudo time stepping. This makes it much easier to solve stationary models using these interfaces, especially if the flow is turbulent. Pseudo time stepping is active per default for both laminar and turbulent flow.

MORE INTUITIVE NAMES OF MASS TRANSPORT INTERFACES

The names (default prefix) for the mass transport interfaces are now more intuitive:

- Transport of Diluted Species: `tds` (previously: `chds`)
- Transport of Concentrated Species: `tcs` (previously: `chcs`)

New Model in Version 5.0

A Journal Bearing Cavitation model demonstrates the new functionality for modeling of cavitation effects.

Backward Compatibility With Version 4.4

INLET FEATURE

The old inlet features still exist, but have been excluded from the physics context menu. Old models retain old Inlet features, but adding a new Inlet feature gives the new functionality. Old Java files create Inlet features of the old types. The Normal stress condition can still be prescribed on inlets using an Open Boundary or a Boundary Stress feature.

NEW NAMES FOR MASS TRANSPORT INTERFACES

Running Model Java-files from previous versions may fail due to the new default Name in the Transport of Diluted Species and Transport of Concentrated Species interfaces.

This can be avoided by adding a command that specifies the identifier in accordance with the Name of the interface. For example, when creating a Transport of Concentrated Species interface using the old Name (chcs):

```
model.physics().create("chcs", "ConcentratedSpecies", "geom1",
    new String[][]{{"wO2", "wH2O", "wN2"}});
```

Add the following line to specify the identifier accordingly:

```
model.physics("chcs").identifier("chcs");
```

WALL DISTANCE EQUATION

The old Continuity node is still included in the interface but is excluded from the feature list. Hence, old models retain their old Continuity node when opened in version 5.0 and old Model Java-files are fully backward compatible.

Backward Compatibility With Version 4.3b

BUBBLE-INDUCED TURBULENCE IN BUBBLY FLOW

Equation terms accounting for bubble-induced turbulence in the Turbulent Bubbly Flow interface are now added correctly. As a result, models with Turbulent Bubbly Flow now show higher levels of turbulence.

Backward Compatibility With Version 4.3a

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 than they did in the previous versions. As a part of the revision, a penalty diffusion on the dispersed phase has been deactivated because after the other revisions, it made the equation system too nonlinear and therefore more difficult to make it converge. However, you can still activate the penalty diffusion in the Advanced section of the Mixture Model settings.

FLUID-STRUCTURE INTERACTION

The Fluid-Structure Interaction (FSI) multiphysics interface has been updated. The separate `vWall` field is no longer required and has been removed. FSI models from 4.3 and earlier versions that include Time Dependent study steps will be affected in the following way:

- Model files for Java will fail. Any reference to the `vWall` field must be removed.
- Models older than 4.3 (that is, 4.2a and older) must either clear their time-dependent solutions when opened in 5.0. An alternative procedure is to:
 - Open the model in version 4.3.
 - Right-click any study with a time-dependent study step and select *Update Solution*.
 - Save the model.
 - Open the model in version 5.0.

FSI models with only stationary study steps will not be affected. Note that `vWall` will still be available as a variable. Hence, references to `fsi.vWall` made in, for example, another physics still work.

BRINKMAN EQUATIONS AND FREE AND POROUS MEDIA FLOW

The momentum equations in the Brinkman Equations interface and the Free and Porous Media Flow interface have been corrected. The term $-\mathbf{Q}_{br} \cdot \mathbf{u} / \epsilon_p^2$ in the right hand side previously lacked the factor $1/\epsilon_p^2$, where ϵ_p is the porosity.

REACTING FLOW INTERFACE NAME CHANGE

The Reacting Flow interfaces have been renamed Reacting Flow in Porous Media. If you open a model using either the Reacting Flow, Diluted Species or the Reacting Flow, Concentrated Species interface, the interface is renamed Reacting Flow in Porous Media.

WEAK CONSTRAINTS UPDATE FOR FLUID FLOW INTERFACES

The weak constraint formulations for the following boundary conditions in the following interfaces has been updated:

Laminar Flow and Creeping Flow

- Symmetry
- The Slip boundary condition in the Wall feature

Turbulent Flow, k - ε and Turbulent Flow, k - ω

- Symmetry
- The Wall functions boundary condition in the Wall feature
- The Moving wall (wall functions) boundary condition in the Wall feature
- The Slip boundary condition in the Wall feature

Turbulent Flow, low-Re k - ε and Turbulent Flow, Spalart-Allmaras

- Symmetry
- The Slip option in the Wall feature

Rotating Machinery

- Symmetry
- The Slip boundary condition in the Wall feature
- The Wall functions boundary condition in the Wall feature (turbulent flow only)
- The Moving wall (wall functions) boundary condition in the Wall feature (turbulent flow only)
- Rotating Wall (turbulent flow only)

Bubbly Flow

- Symmetry
- The Slip boundary condition for the liquid phase in the Wall feature
- The Wall functions boundary condition for the liquid phase in the Wall feature (turbulent flow only)

Mixture Model

- Symmetry
- The Slip boundary condition for the mixture in the Wall feature
- Wall functions boundary condition for the mixture in the Wall feature (turbulent flow only)

Euler-Euler Model, Laminar Flow

- Symmetry
- The Slip boundary condition for either phase in the Wall feature
- The Slip boundary condition for either phase in the Inlet feature

Brinkman Equations and Free and Porous Media Flow

- Symmetry
- The Slip boundary condition in the Wall feature

Two-Phase Flow, Level Set

- Symmetry
- The Slip boundary condition in the Wall feature
- The Wetted wall boundary condition in the Wall feature
- The Moving wetted wall boundary condition in the Wall feature
- The Wall function boundary condition in the Wall feature (turbulent flow only)
- The Moving wall (wall functions) boundary condition in the Wall feature (turbulent flow only)

Two-Phase Flow, Phase Field

- Symmetry
- The Slip boundary condition in the Wall feature
- The Wall function boundary condition in the Wall feature (turbulent flow only)
- The Moving wall (wall functions) boundary condition in the Wall feature (turbulent flow only)

Non-Isothermal Flow and Conjugate Heat Transfer

- Symmetry
- The Slip boundary condition in the Wall feature
- The Wall functions boundary condition in the Wall feature (turbulent flow $k-\epsilon$ and turbulent flow $k-\omega$ only)
- The Moving wall (wall functions) boundary condition in the Wall feature (turbulent flow $k-\epsilon$ and turbulent flow $k-\omega$ only)

High Mach Number Flow

- Symmetry
- The Slip boundary condition in the Wall feature
- The Wall functions boundary condition in the Wall feature (turbulent flow $k-\epsilon$ only)
- The Moving wall (wall functions) boundary condition in the Wall feature (turbulent flow $k-\epsilon$ only)

These boundary conditions are now formulated using the same set of Lagrange multipliers as all the other boundary conditions for the dependent velocity variables. The previously used Lagrange multiplier `un_1m` has been removed.

When you open models saved in version 4.3, they include `un_1m` until the model is re-solved. In some cases, occurrences of `un_1m` in the solver sequence must be replaced manually. This is the case if `un_1m` was the only Lagrange multiplier component in a segregated group or the only Lagrange multiplier component of a Vanka smoother. Alternatively, you can generate a new automatic solver sequence. Models saved in versions previous to version 4.3 must either be re-solved in version 5.0 for postprocessing, or opened and re-saved in version 4.3 before being opened in version 5.0.

Weak constraints for the Interior Wall feature are no longer available.

REVISION OF THE TURBULENCE MODELS

The formulations of some variables in the turbulence models have been revised in order to improve accuracy. Models using a turbulence model can display a different convergence behavior in version 5.0 than in version 4.3 and the result can differ slightly between the versions.

Chemical Reaction Engineering Module

New Functionality in Version 5.0

GENERAL NEW FUNCTIONALITY

- A new feature, *Reactive Pellet Bed*, is introduced to model packed-bed reactors. For an example, see the *Multiscale 3D Packed Bed Reactor* model.
- A new Danckwerts Inflow condition in the Inflow feature of the Transport of Diluted Species interface makes reacting flow problems faster and more stable. For an example see the *Dialysis* model.
- New Equilibrium Reactions and Surface Equilibrium Reactions in the Transport of Diluted Species and Nernst-Planck interfaces. This feature makes is easier and more stable to model equilibrium reactions. For an example, see the *Water Purification by Silver Complexation* model.

UPDATED AND IMPROVED REACTION ENGINEERING INTERFACE

The *Reaction Engineering* interface is updated with the following improvements:

- Faster, more workable user interface.
- The concentration (mol/m^3) is replaced by activity in the law of mass action for the determination of the reaction rate. The default activity coefficient is unity, which means ideality.
- More physical property calculations and export options.
- Stronger support for large amounts of species and reactions.
- The CSTR reactors now support multiple inlets and outlets as well as filling and draining. See the *Ideal Tank Flow System* example model.
- Reversible Reaction Group, Equilibrium Reaction Group, Species Group: These group features have been introduced to easier handle large amounts of reactions and species, which are now presented in tables. CHEMKIN imports are done automatically in groups. For an example, see the *Compression Ignition of Methane* model.
- Better species thermodynamics and transport property export to 1D, 2D, and 3D models.

- The Reaction feature can be both volumetric and surface based.
- New improved surface reactions solved for: If a species is set to surface species, the rate expression of reactions is used instead of a bulk mass balance, and the species production exports correctly to influx conditions in space-dependent models.
- Heat flux is included in the energy balance when there is a surface reaction.
- The space-dependent model created by the Generate Space-Dependent Model feature is not linked back to the Reaction Engineering interface anymore once the generation is done. Instead, at the creation of a space-dependent model a Chemistry node is created, which is similar to a Material node in COMSOL Multiphysics. The Chemistry node contains the reaction properties provided by the Reaction Engineering interface.
- You can also add the new Chemistry interface directly to a space-dependent model without doing any analyses in a Reaction Engineering interface. For an example, see the updated *Thermal Decomposition* model.
- The Reaction feature in Transport of Diluted Species can now pick up kinetic expressions automatically from the Chemistry interface. For an example, see the model *Fine Chemical Production in a Plate Reactor* model.

UPDATED AND IMPROVED MASS TRANSPORT INTERFACES

This version includes a major overhaul of the mass transport and reactions functionality. The old *Transport of Diluted Species* for free flow and *Species Transport in Porous Media* for porous media are now unified in one physics interface called *Transport of Diluted Species in Porous Media*. This physics interface has many new strengths in addition to better user-friendliness: In addition to support for *convection* and *migration in electric fields* it now also includes adsorption, dispersion, and volatilization in saturated and partially saturated porous media:

- New entry point in the Model Wizard: *Transport of Diluted Species in Porous Media*. This is a Transport of Diluted Species interface with a default Porous Media Transport Properties domain feature activated.
- Replaces and unifies the old Solute Transport and Species Transport in Porous media interfaces.
- Introduces unified nomenclature for all mass transport.
- Stabilization introduced for porous media mass transport.
- The new Dispersion section in the Settings window makes it possible to model dispersion in isotropic and transverse anisotropic porous media.

- The Diffusion section now supports modeling of diffusion in isotropic and anisotropic porous media.
- The dependent variables for concentration are defined in the unit mol/m³ in all mass transport interfaces.
- A new Mass-Based Concentrations feature provides results output in mass-based quantities such as mass fractions and mass concentrations.
- Improved nomenclature for porous media mass transfer.
- Existing Model MPH-files created with the now retired Species Transport in Porous Media and Solute Transport open in those physics interfaces, but they are not available in the Model Wizard for new models.

The names (default prefix) for the mass transport interfaces are now more intuitive:

- Transport of Diluted Species: **tds** (previously: **chds**)
- Transport of Diluted Species in Porous Media: **tds**
- Transport of Concentrated Species: **tcs** (previously: **chcs**)
- Nernst-Planck Equations: **npe** (previously: **chnp**)
- Surface Reactions: **sr** (previously: **chsr**)

New and Updated Models in Version 5.0

- A Multiscale 3D Packed Bed Reactor.
- Water Purification by Silver Complexation
- Ideal Tank Flow System
- Level Control of a Tank Using Events
- Chlorine Scrubber
- Updated Dialysis model with a new more stable technique for handling membrane solubility.
- Most reacting flow models have been updated with the more stable Danckwerts inflow condition.

Backward Compatibility With Version 4.4

MASS TRANSPORT INTERFACES

Running Model Java-files from previous versions may fail due to the new default Name in the mass transport interfaces. This can be avoided by adding a command that

specifies the identifier in accordance with the Name of the interface. For example, when creating an interface using the old Name (chcs):

```
model.physics().create("chcs", "ConcentratedSpecies", "geom1",
    new String[][]{{"wO2", "wH2O", "wN2"}});
```

Add the following line to specify the identifier accordingly:

```
model.physics("chds").identifier("chcs");
```

The Java API syntax for creating and accessing vectors and tensors has changed as well as the syntax for settings physics properties. This does not affect Model MPH-files. See the following examples:

Specifying the Diffusion Tensor

Old syntax:

```
model.physics("chnp").feature("cdm1").set("D_0",
    new String[]{"D1", "0", "0", "0", "D1", "0", "0", "0", "D1"});

model.physics("chnp").feature("cdm1").set("D_1",
    new String[]{"D2", "0", "0", "0", "D2", "0", "0", "0", "D2"});

model.physics("chnp").feature("cdm1").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("npe").feature("cdm1").set("D_c1",
    new String[]{"D1", "0", "0", "0", "D1", "0", "0", "0", "D1"});

model.physics("npe").feature("cdm1").set("D_c2",
    new String[]{"D2", "0", "0", "0", "D2", "0", "0", "0", "D2"});

model.physics("npe").feature("cdm1").set("D_c3",
    new String[]{"D3", "0", "0", "0", "D3", "0", "0", "0", "D3"});
```

Accessing Tensor Components

Accessing tensor components in the definitions of other variables, for example.

Old syntax: Dxx_c1, Dxy_c1, Dxz_c1, and so on.

New syntax (component index after the species name): D_c1xx, D_c1xy, D_c1xz, and so on.

Specifying the Reaction Rate Vector

Old syntax:

```
model.physics("chnp").feature("react1").set("R_0",
    new String[]{"R1"});
```

```

model.physics("chnp").feature("react1").set("R_1",
    new String[]{"R2"});
model.physics("chnp").feature("react1").set("R_2",
    new String[]{"R3"});

```

New syntax (includes the species name in the first argument of the set method):

```

model.physics("npe").feature("react1").set("R_c1",
    new String[]{"R1"});
model.physics("npe").feature("react1").set("R_c2",
    new String[]{"R2"});
model.physics("npe").feature("react1").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("chnp").feature("init1").setIndex("c2", "c20", 0);
model.physics("chnp").feature("init1").setIndex("c3", "c30", 0);
model.physics("chnp").feature("init1").setIndex("V", "V0", 0);

```

New syntax:

```

model.physics("npe").feature("init1").setIndex("initc", "c20", 1);
model.physics("npe").feature("init1").setIndex("initc", "c30", 2);
model.physics("npe").feature("init1").setIndex("initV", "V0");

```

Selecting Properties

Old syntax:

```

model.physics("chnp").prop("Convection").
    set("Convection", 1, "0");
model.physics("chnp").prop("Migration").set("Migration", 1, "0");
model.physics("chnp").prop("ConvectiveTerm").
    setIndex("ConvectiveTerm", "noncons", 0);

```

New syntax:

```

model.physics("npe").prop("TransportMechanism").
    set("Convection", 1, "0");
model.physics("npe").prop("TransportMechanism").
    set("Migration", 1, "0");
model.physics("npe").prop("AdvancedSettings").
    setIndex("ConvectiveTerm", "noncons", 0);

```

Corrosion Module

New Functionality in Version 5.0

NEW PHYSICS INTERFACE FOR MASS TRANSPORT AND REACTIONS

This version includes a major overhaul of the mass transport and reactions functionality. The old *Transport of Diluted Species* for free flow and *Species Transport in Porous Media* for porous media are now unified in one physics interface called *Transport of Diluted Species in Porous Media*. This physics interface has many new strengths in addition to better user-friendliness: In addition to support for *convection* and *migration in electric fields* it now also includes adsorption, dispersion, and volatilization in saturated and partially saturated porous media:

- New entry point in the Model Wizard: *Transport of Diluted Species in Porous Media*. This is a Transport of Diluted Species interface with a default Porous Media Transport Properties domain feature activated.
- Replaces and unifies the old Solute Transport and Species Transport in Porous media interfaces.
- Introduces unified nomenclature for all mass transport.
- Stabilization introduced for porous media mass transport.
- The new Dispersion section in the Settings window makes it possible to model dispersion in isotropic and transverse anisotropic porous media.
- The Diffusion section now supports modeling of diffusion in isotropic and anisotropic porous media.
- The dependent variables for concentration are defined in the unit mol/m^3 in all mass transport interfaces.
- A new Mass-Based Concentrations feature provides results output in mass-based quantities such as mass fractions and mass concentrations.
- Improved nomenclature for porous media mass transfer.
- Existing Model MPH-files created with the now retired Species Transport in Porous Media and Solute Transport open in those physics interfaces, but they are not available in the Model Wizard for new models.

The names (default prefix) for the mass transport interfaces are now more intuitive:

- Transport of Diluted Species: **tds** (previously: **chds**)

- Transport of Diluted Species in Porous Media: `tds`
- Transport of Concentrated Species: `tcs` (previously: `chcs`)
- Surface Reactions: `sr` (previously: `chsr`)

ADDITIONAL NEW FUNCTIONALITY

- A new *Current Distribution on Edges, BEM* interface for modeling of currents on edges using the boundary element method.
- New *Danckwerts* inflow boundary condition in the Transport of Diluted Species interface's Inflow node.
- New *Equilibrium Reaction* and *Surface Equilibrium Reaction* nodes in the Transport of Diluted Species, Electroanalysis, and Tertiary Current Distribution, Nernst-Planck interfaces.
- New *Counter electrode* boundary condition in the Electrode Surface feature in the Electroanalysis interface.
- New *Periodic Condition* boundary node in the Primary Current Distribution and Secondary Current Distribution interfaces.
- Updated Equilibrium Potential section in Electrode Reaction and Porous Electrode Reaction nodes' Settings windows. The equilibrium potential parameter is now split from the dE_{eq}/dT parameter. Both parameters can be taken from the material.
- Improved solver suggestions and changed naming of the global potential degrees of freedom created by the total or average current boundary conditions in Electrolyte Current, Electrode Current, Electrode Power, Electrode-Electrolyte Boundary Interface, Electrolyte-Electrode Boundary Interface and Electrode Surface nodes. The changes may affect API backward compatibility.
- Improved automatic meshing: shared points (2D) and edges (3D) between Electrode Reaction boundaries are now set to an *extremely fine* mesh size by default (in all electrochemistry interfaces that include the Electrode Reaction boundary node).
- New Noncorroding Boundary default feature in the Corrosion, Deformed Geometry interfaces.
- Improved solver default settings in the Corrosion, Deformed Geometry interfaces. The scale of the deforming coordinate variables are based on the smallest initial element mesh size.

New Models in Version 5.0

The following new models are included in the Corrosion Module Model Library:

- A CO₂ Corrosion model
- A Multiple Oil Platforms model

Backward Compatibility With Version 4.3

BRINKMAN EQUATIONS AND FREE AND POROUS MEDIA FLOW

The momentum equations in the Brinkman Equations interface and the Free and Porous Media Flow interface have been corrected. The term $-Q_{br} \cdot \mathbf{u} / \varepsilon_p^2$ in the right hand side previously lacked the factor $1/\varepsilon_p^2$, where ε_p is the porosity.

WEAK CONSTRAINTS UPDATE FOR FLUID FLOW INTERFACES

The weak constraint formulations for some boundary conditions in the following interfaces have been updated:

- Laminar Flow
- Brinkman Equations
- Free and Porous Media Flow

See [CFD Module](#) for details.

MOBILITY

In physics user interfaces modeling migration (Transport of Diluted Species; Tertiary Current Distribution, Nernst-Planck; and Corrosion, Tertiary Nernst-Planck) anisotropic mobility is now supported, and the mobility can be set to follow the Nernst-Einstein relation. The default setting when making new models has been changed to use the Nernst-Einstein relation. Model files for Java generated prior to 4.3b using Migration will have to be modified manually to account for these changes.

ECAD Import Module

New Functionality in Version 5.0

- The Import settings window for GDS files is updated to use terminology for GDS import. Settings and terminology that are not applicable to GDS import have been removed or updated. The GDS import settings is now separated from the settings window for NETEX-G file.
- For GDS import arc recognition is improved, and straight lines are recognized even with the remove interior boundaries option.
- For ODB++ import the handling of certain symbols and mixed units is fixed, and construction of geometry objects is improved.

Electrochemistry Module

New Functionality in Version 5.0

NEW PHYSICS INTERFACE FOR MASS TRANSPORT AND REACTIONS

This version includes a major overhaul of the mass transport and reactions functionality. The old *Transport of Diluted Species* for free flow and *Species Transport in Porous Media* for porous media are now unified in one physics interface called *Transport of Diluted Species in Porous Media*. This physics interface has many new strengths in addition to better user-friendliness: In addition to support for *convection* and *migration in electric fields* it now also includes adsorption, dispersion, and volatilization in saturated and partially saturated porous media:

- New entry point in the Model Wizard: *Transport of Diluted Species in Porous Media*. This is a Transport of Diluted Species interface with a default Porous Media Transport Properties domain feature activated.
- Replaces and unifies the old Solute Transport and Species Transport in Porous media interfaces.
- Introduces unified nomenclature for all mass transport.
- Stabilization introduced for porous media mass transport.
- The new Dispersion section in the Settings window makes it possible to model dispersion in isotropic and transverse anisotropic porous media.
- The Diffusion section now supports modeling of diffusion in isotropic and anisotropic porous media.
- The dependent variables for concentration are defined in the unit mol/m^3 in all mass transport interfaces.
- A new Mass-Based Concentrations feature provides results output in mass-based quantities such as mass fractions and mass concentrations.
- Improved nomenclature for porous media mass transfer.
- Existing Model MPH-files created with the now retired Species Transport in Porous Media and Solute Transport open in those physics interfaces, but they are not available in the Model Wizard for new models.

The names (default prefix) for the mass transport interfaces are now more intuitive:

- Transport of Diluted Species: `tds` (previously: `chds`)

- Transport of Diluted Species in Porous Media: **tds**
- Transport of Concentrated Species: **tcs** (previously: **chcs**)
- Surface Reactions: **sr** (previously: **chsr**)

ADDITIONAL NEW FUNCTIONALITY

- A new *Current Distribution on Edges, BEM* interface for modeling of currents on edges using the boundary element method.
- New *Danckwerts* inflow boundary condition in the Transport of Diluted Species interface's Inflow node.
- New *Equilibrium Reaction* and *Surface Equilibrium Reaction* nodes in the Transport of Diluted Species, Electroanalysis, and Tertiary Current Distribution, Nernst-Planck interfaces.
- New *Counter electrode* boundary condition in the Electrode Surface feature in the Electroanalysis interface.
- New *Periodic Condition* boundary node in the Primary Current Distribution and Secondary Current Distribution interfaces.
- Updated Equilibrium Potential section in Electrode Reaction and Porous Electrode Reaction nodes' Settings windows. The equilibrium potential parameter is now split from the dE_{eq}/dT parameter. Both parameters can be taken from the material.
- Improved solver suggestions and changed naming of the global potential degrees of freedom created by the total or average current boundary conditions in Electrolyte Current, Electrode Current, Electrode Power, Electrode-Electrolyte Boundary Interface, Electrolyte-Electrode Boundary Interface and Electrode Surface nodes. The changes may affect API backward compatibility.
- Improved automatic meshing: shared points (2D) and edges (3D) between Electrode Reaction boundaries are now set to an *extremely fine* mesh size by default (in all electrochemistry interfaces that include the Electrode Reaction boundary node).

Electrodeposition Module

New Functionality in Version 5.0

NEW PHYSICS INTERFACE FOR MASS TRANSPORT AND REACTIONS

This version includes a major overhaul of the mass transport and reactions functionality. The old *Transport of Diluted Species* for free flow and *Species Transport in Porous Media* for porous media are now unified in one physics interface called *Transport of Diluted Species in Porous Media*. This physics interface has many new strengths in addition to better user-friendliness: In addition to support for *convection* and *migration in electric fields* it now also includes adsorption, dispersion, and volatilization in saturated and partially saturated porous media:

- New entry point in the Model Wizard: *Transport of Diluted Species in Porous Media*. This is a Transport of Diluted Species interface with a default Porous Media Transport Properties domain feature activated.
- Replaces and unifies the old Solute Transport and Species Transport in Porous media interfaces.
- Introduces unified nomenclature for all mass transport.
- Stabilization introduced for porous media mass transport.
- The new Dispersion section in the Settings window makes it possible to model dispersion in isotropic and transverse anisotropic porous media.
- The Diffusion section now supports modeling of diffusion in isotropic and anisotropic porous media.
- The dependent variables for concentration are defined in the unit mol/m^3 in all mass transport interfaces.
- A new Mass-Based Concentrations feature provides results output in mass-based quantities such as mass fractions and mass concentrations.
- Improved nomenclature for porous media mass transfer.
- Existing Model MPH-files created with the now retired Species Transport in Porous Media and Solute Transport open in those physics interfaces, but they are not available in the Model Wizard for new models.

The names (default prefix) for the mass transport interfaces are now more intuitive:

- Transport of Diluted Species: `tds` (previously: `chds`)

- Transport of Diluted Species in Porous Media: `tds`
- Transport of Concentrated Species: `tcs` (previously: `chcs`)
- Surface Reactions: `sr` (previously: `chsr`)

ADDITIONAL NEW FUNCTIONALITY

- A new *Current Distribution on Edges, BEM* interface for modeling of currents on edges using the boundary element method.
- New *Danckwerts* inflow boundary condition in the Transport of Diluted Species interface's Inflow node.
- New *Equilibrium Reaction* and *Surface Equilibrium Reaction* nodes in the Transport of Diluted Species, Electroanalysis, and Tertiary Current Distribution, Nernst-Planck interfaces.
- New *Counter electrode* boundary condition in the Electrode Surface feature in the Electroanalysis interface.
- New *Periodic Condition* boundary node in the Primary Current Distribution and Secondary Current Distribution interfaces.
- Updated Equilibrium Potential section in Electrode Reaction and Porous Electrode Reaction nodes' Settings windows. The equilibrium potential parameter is now split from the dE_{eq}/dT parameter. Both parameters can be taken from the material.
- Improved solver suggestions and changed naming of the global potential degrees of freedom created by the total or average current boundary conditions in Electrolyte Current, Electrode Current, Electrode Power, Electrode-Electrolyte Boundary Interface, Electrolyte-Electrode Boundary Interface and Electrode Surface nodes. The changes may affect API backward compatibility.
- Improved automatic meshing: shared points (2D) and edges (3D) between Electrode Reaction boundaries are now set to an *extremely fine* mesh size by default (in all electrochemistry interfaces that include the Electrode Reaction boundary node).
- New Nondeposition Boundary feature in the Electrodeposition, Deformed Geometry interfaces
- Improved solver default settings in the Electrodeposition, Deformed Geometry interfaces. The scale of the deforming coordinate variables are based on the smallest initial element mesh size.

New Model in Version 5.0

A new Microconnector Bump in 3D model is included in the Electrodeposition Module Model Library.

Backward Compatibility With Version 4.3

BRINKMAN EQUATIONS AND FREE AND POROUS MEDIA FLOW

The momentum equations in the Brinkman Equations interface and the Free and Porous Media Flow interface have been corrected. The term $-Q_{br} \cdot \mathbf{u} / \epsilon_p^2$ in the right hand side previously lacked the factor $1/\epsilon_p^2$, where ϵ_p is the porosity.

WEAK CONSTRAINTS UPDATE FOR FLUID FLOW INTERFACES

The weak constraint formulations for some boundary conditions in the following interfaces have been updated:

- Laminar Flow
- Brinkman Equations
- Free and Porous Media Flow

See [CFD Module](#) for details.

MOBILITY

In physics user interfaces modeling migration (Transport of Diluted Species; Tertiary Current Distribution, Nernst-Planck; and Electrodeposition, Tertiary Nernst-Planck) anisotropic mobility is now supported, and the mobility can be set to follow the Nernst-Einstein relation. The default setting when making new models has been changed to use the Nernst-Einstein relation. Model files for Java generated prior to 4.3b using Migration will have to be modified manually to account for these changes.

Fatigue Module

New Functionality in Version 5.0

FATIGUE STUDY TYPE

A new Fatigue study type is added. It has the following advantages:

- A load cycle can be selected from the underlying parametric or time dependent analysis. There is no longer a need to run a separate extra study for a single cycle.
- No other physics interface are solved in a Fatigue study.

STRESS-LIFE AND STRAIN-LIFE FATIGUE MODELS

A new family of fatigue models called *Stress-Life* has been added. It is intended for computing the number of cycles to fatigue, based on the stress range. There are three models:

- S-N curve
- Basquin
- Approximate S-N curve

Another new family of fatigue models called *Strain-Life* has been added. It is intended for computing the number of cycles to fatigue, based on the strain range. There are three models:

- E-N curve
- Coffin-Manson (available already in version 4.4, but moved to this family of models)
- Combined Basquin and Coffin-Manson

Also, the Matrix Histogram plot in 3D, used for presenting load and damage distributions, is improved.

New Models in Version 5.0

- Fatigue Failure of an Eyeglass Frame
- High-cycle Fatigue of a Reciprocation Piston Engine
- Bracket—Fatigue Evaluation

Geomechanics Module

New Functionality in Version 5.0

SMALL STRAIN NONLINEAR ELASTICITY MODELS

A new class of material models intended for small strain nonlinear elasticity have been added. There are a number of different material models available under Nonlinear Elastic Material:

- Ramberg-Osgood
- Hyperbolic Law
- Hardin-Drnevich
- Duncan-Chang
- Duncan-Selig
- User defined

Heat Transfer Module

New and Improved Functionality in Version 5.0

HEAT TRANSFER IN THIN LAYERS

The new *Thin Layer* feature, available on exterior and interior boundaries, is a combination of the obsolete Highly Conductive Layer and Thin Thermally Resistive Layer features.

It offers options for cost effective, lumped, models to represent thin layers when they are either acting as a thermal resistance or as the highly conductive layer. A third option is available for the general case where no particular assumption is made so the full heat transfer equations are solved on the layer. In this case a dedicated mesh is automatically generated for the layer without adding any complexity to the geometry meshing process.

A similar technology is available in the Heat Transfer in Thin Shells interface with the *Thin Layered Shell* feature.

THIN LAYERED SHELL

The *Thin Layered Shell* feature is available in the Heat Transfer in Thin Shells interface. This boundary feature overrides *Thin Conductive Shell* to enable modeling of shells formed of multiple layers without making any assumption on the heat transfer properties. In particular a mesh is automatically generated to discretize the shell thickness.

HEAT TRANSFER IN THIN FILMS

The new *Thin Film* feature behaves like the obsolete Highly Conductive Layer feature, and models heat transfer in thin fluid regions. Using this feature prevents representing the fluid film explicitly in the geometry. It provides a lumped model for heat transfer to achieve cost effective modeling, and dedicated settings to define the thermal properties of the fluid as well as the flow properties.

HEAT TRANSFER IN FRACTURES

The new *Fracture* feature is only active when the Heat transfer in porous media check box is selected in the Physical Model section. It models heat transfer in fractures, which correspond to thin porous media. Using this feature prevents representing the fracture thickness in the geometry. It provides a lumped model for cost effective modeling, and

dedicated settings to define the fluid and the solid parts as well as the flow properties. The flow can be defined by a flow interface, in particular by the Fracture Flow interface.

HEAT TRANSFER IN HIGHLY CONDUCTIVE ROD

The new *Thin Rod* feature provides a lumped heat transfer model to model highly thermally conductive rods as edges. It is similar to the obsolete feature Highly Conductive Layer but applies on edges.

When a solid with low thermal conductivity contains rods made of a highly thermally conductive material, the heat transfer contribution of the rod cannot be neglected. Using this feature prevents representing the rod thickness in the geometry for cost effective modeling.

CRYOGENIC DAMAGE ANALYSIS IN BIOLOGICAL TISSUE

The *Temperature threshold* form for damage integral analysis in the Biological Tissue feature now provides options to perform cryogenic analysis. Two sets of *Damaged temperature*, *Damaged time*, and *Temperature of necrosis* parameters can be entered simultaneously for the modeling of combined hyperthermia and cryogenic processes.

FAN, INTERIOR FAN AND GRILLE FEATURES AVAILABLE FOR TURBULENT FLOW

Fan, *Interior Fan*, and *Grille* features are updated so that turbulence models are supported. These features provide lumped models that replace an explicit device description by a boundary condition.

Moreover, when the *Non-Isothermal Flow* coupling feature is used, the mixing effect of the temperature field at the fan outlet is included.

ISOTHERMAL DOMAINS

The new *Isothermal Domain* feature can be used to model regions where the temperature is assumed to be homogeneous. Instead of solving a partial differential equation over the domain, a global heat equation is computed to determine the domain temperature.

It is only active when the Isothermal Domain check box is selected in the Physical Model section, and it is compatible with most of the classical domain, boundary, edge, and point features.

Nevertheless a new dedicated boundary condition is introduced to set global conditions on isothermal domains: the *Isothermal Domain Interface* feature provides

options to define thermal insulation, convective heat flux, thermal contact, or ventilation between two isothermal domains.

SURFACE-TO-SURFACE RADIATION ENHANCEMENT

Several improvements are added to the surface-to-surface capabilities. These cover the computational performances, the modeling capabilities, and postprocessing:

- Parallelization of radiation code: the code used to compute the view factor for surface-to-surface radiation is improved to support shared memory parallelization. Assembly time for surface-to-surface radiation models is reduced when COMSOL Multiphysics has access to multiple cores.
- Radiation support for shell and solid interfaces coupling: it is now possible to couple a domain interface (for example, Heat Transfer in Solids) and a shell interface (Heat Transfer in Thin Shells) that share the same temperature and radiosity fields. This makes it possible to build surface-to-surface models containing shells and solids that are separated by a region (usually air or vacuum) that is represented in the geometry (not meshed).
- View factor postprocessing: The heat transfer interfaces where surface-to-surface radiation is enabled have a set of operators that are evaluated like the irradiation variables in surface-to-surface radiation. These operators make it possible to retrieve the irradiation variables values but also to compute the geometrical view factor in a given geometry.
- Accurate postprocessing of radiative quantities: new postprocessing variables enable to plot the irradiation variables at Gauss points where they are defined. Doing so avoids unnecessary smoothing induced by Lagrange point based plots and facilitates the analysis of such plots.

SURFACE-TO-SURFACE RADIATION—PREDEFINED CITY LIST FOR SOLAR POSITION

A new option is available to automatically define the location parameters (latitude, longitude, and time zone) from a city name in models using surface-to-surface radiation with the sun as an external radiation source. This option provides a list of many large cities to choose from.

Since the time zone is not entered manually, the *Include daylight saving time* check box is available under the city list to solve a model during summer time. In this case, the time zone is updated.

OPTION TO SPECIFY DIMENSION OF POINT/LINE HEAT SOURCE

For *Line Heat Source* and *Point Heat Source* features, a new option allows the specification of the radius of the source. Doing so avoids having non-physical, infinity-concentrated heat sources that lead to mesh dependent solutions. It does not require that the mesh element size matches the heat source radius: the heat source is handled even with coarse meshes. Furthermore, refining the mesh close to the source location leads to a stable solution matching the solution of the corresponding geometry where the source is represented by a domain instead of a point.

A new *Heat Source Radius* section is available under the Line Heat Source section. When the *Specify heat source radius* check box is selected, the value of the variable `radius` can be set.

The tutorial model *Heat Conduction with a Localized Heat Source on a Disk* has guidelines about how to use this option depending on the ratio between the source and surrounding geometry typical sizes.

NEW DISCRETIZATION ORDER FOR THE DISCRETE ORDINATES METHOD

Constant discretization order for the Discrete Ordinate Method is available in Heat Transfer in Participating Media and Radiation in Participating Media interfaces. This low order discretization enables you to speedup computation since it reduces the number of degrees of freedom.

THERMOELASTIC DAMPING AND PRESSURE WORK

In Heat Transfer in Solids the new subfeature *Thermoelastic Damping* replaces Pressure Work. It allows the modeling of solids heating due to the compression generated by vibrations. In particular this feature accurately handles materials with a nonconstant thermal expansion coefficient. When the *Thermal Expansion* multiphysics node is used, the thermoelastic damping contribution is automatically taken from it.

In Heat Transfer in Fluids the subfeature Pressure Work does not support the low Mach number formulation option anymore.

The *Non-Isothermal Flow* multiphysics feature now overrides both Pressure Work (under Heat Transfer in Fluids) and Viscous Dissipation since it contains those heat source contributions.

VISCOUS DISSIPATION IN FLUID FLOW

The *Viscous Dissipation* subfeature replaces the obsolete Viscous Heating feature. It is available in Heat Transfer in Fluids and Heat Transfer in Porous Media interfaces.

The viscous dissipation contribution can be taken straight from the fluid flow interfaces or set manually.

THERMAL MATERIAL PROPERTIES AVAILABLE FOR ALL MATERIALS IN THE BUILT-IN MATERIAL LIBRARY

The built-in material library is updated so that all materials available in it contain all properties needed for thermal analysis.

OUT-OF-PLANE HEAT TRANSFER UPDATE

The **Out-of-Plane Heat Transfer** property, which was available in 2D and 1D geometries, is obsolete in COMSOL Multiphysics 5.0. Now all geometries are considered as 3D geometries, and thickness, cross-sectional perimeter or cross-sectional area are always available through the variables d , P_c , and A_c .

DIFFUSE SURFACE

The *Diffuse Surface* feature combines both Surface-to-Surface Radiation and Surface-to-Ambient Radiation features. It is used to model the interaction with the ambient medium unless *Surface-to-surface radiation* is enabled in the Physical Model section, in which case the default options include radiation between surfaces. To only model interaction with the ambient medium, clear the Include surface-to-surface radiation check box. This leads to fast computations but ignores any surface-to-surface radiation effects.

ADDITION OF RECIPROCAL AVERAGE MODEL FOR EFFECTIVE THERMAL CONDUCTIVITY IN POROUS MEDIA

A average model is added to the *Volume average* option for the computation of the effective thermal conductivity in porous media. The *Reciprocal average* option evaluates the effective thermal conductivity as the weighted harmonic mean of fluid and porous matrix conductivities, and is suitable when the thermal flux is perpendicular to the matrix structure.

In addition, the *Power law* option, previously only available in the Subsurface Flow Module, is now added in Heat Transfer in Porous Media.

NUSSELT-BASED CORRELATIONS FOR EFFECTIVE THERMAL CONDUCTIVITY

In order to reduce the computational cost of conjugate heat transfer models, the natural convection mixing effect in cavities can be modeled from a thermal point of view as an immobile fluid with a larger thermal conductivity. With this approximation

it is not necessary to compute the flow in the cavities. The Nusselt correlations are provided for two configurations: parallelepiped cavity with horizontal and vertical temperature gradient. You can also use your own correlation to handle a different configuration.

MISCELLANEOUS

- The default Arrow Surface plot is removed from Isothermal Contours in all heat transfer interfaces. Only the Contour plot is available.
- It is now possible to use customized coordinate names from the component instead of the default x, y, and z when the Radiation in Participating Media feature is used.
- In surface-to-surface radiation features, the issue obtained when setting T_{amb} as T where T is a model dependent variable is fixed. This fix also applies to the new *Diffuse Surface* feature.
- In the *Radiation in Participating Media* feature, the name of the refractive index variable $ht.nr$ is changed to $ht.nr_{rad}$ to remove conflicts in 2D axisymmetric models where $ht.nr$ is the radial normal component.
- In the now obsolete Highly Conductive Layer feature, the accurate flux computation was fixed for the particular geometrical configurations leading to inaccurate heat fluxes. This fix applies to the new *Thin Film* feature.
- The definition of the weighted average temperature variable T_{ave} is corrected for 2D axisymmetric models.

New Models in Version 5.0

HEAT CONDUCTION WITH A LOCALIZED HEAT SOURCE ON A DISK

This conductive heat transfer example shows how to define a heat source localized on a small domain using the Heat Transfer interface. The accuracy and the numerical cost of the different features to model the heat source are compared. The results bring guidelines to select the suitable option depending on the ratio between the source and surrounding geometry typical sizes.

PARAMETERIZED DOUBLE-PIPE HEAT EXCHANGER GEOMETRY

This is a template MPH-file containing the physics interfaces and the parameterized geometry for the model Double-Pipe Heat Exchanger.

DOUBLE-PIPE HEAT EXCHANGER

Double-pipe heat exchangers, with their typical U-turn shape, are one of the simplest and cheapest type of heat exchangers used in the chemical process industry. This model studies the cooling of engine oil entering at 130°C by a cold transformer oil entering at 60°C in counter-current. The meshing part presented here handles possible difficulties due to high aspect ratio of the device. In postprocessing, various quantities describe the performance and efficiency of this double-pipe heat exchanger.

MICROCHANNEL HEAT SINK

This example studies the air flow and its cooling effect in a microchannel heat sink with perpendicular manifolds. This particular configuration, acting as a flow divider before air enters the heat sink, improves efficiency in cooling an electronic component.

ANISOTROPIC HEAT TRANSFER THROUGH WOVEN CARBON FIBERS

Carbon-fiber-reinforced polymers contain woven carbon fibers that have a thermal conductivity along the fiber axis is much higher than perpendicular to it. This tutorial model shows how to use the curvilinear coordinates interface to compute the local fiber orientation and to use it to define anisotropic thermal conductivity of fibers. Because the carbon-fiber-reinforced polymer sample dimensions are rather small, infinite elements are used to avoid setting boundary conditions too close to the heat source.

RADIATIVE COOLING OF A GLASS PLATE

A glass melt is cooled down by radiation to form the final shape. The simulation of radiation is performed using the three radiation in participating media models provided in COMSOL Multiphysics to compare the accuracies and computational costs.

MIXED DIFFUSE-SPECULAR RADIATION BENCHMARK

This model shows how to use the Mathematical Particle Tracing interface to simulate mixed diffuse-specular reflection between surfaces in an enclosure. This model is separated into two parts. The first part compares the heat fluxes computed by the Mathematical Particle Tracing interface with the exact analytical solution for two identical infinitely long parallel gray plates under mixed diffuse-specular reflection at constant temperature. The second part couples the Mathematical Particle Tracing interface with the Heat Transfer in Solids interface for the parallel plate geometry but with different characteristics and spatially varying temperatures.

This model requires the Particle Tracing Module.

GROUND HEAT RECOVERY FOR FLOOR HEATING

This model shows how to set-up a heat transfer analysis for different patterns of heat collectors. It couples heat transfer in subsurfaces with heat transfer inside pipes. For different configurations, geometry subsequences are used and run by a parametric study. The results for the operating temperatures are compared.

EVAPORATIVE COOLING

This tutorial shows how to couple three physics interfaces to model evaporative cooling. The effects that need to be taken into account are heat transfer, transport of water vapor and fluid flow. User-defined expressions are used to implement the source term for the water vapor and evaporative heat source, as well as the moist air feature to accurately describe the material properties.

Backward Compatibility With Version 4.4

OBSOLETE FEATURES

A number of features are now obsolete because they have been merged in an existing feature or replaced by a more general one. However, when models from COMSOL Multiphysics 4.4 or before contain these features, they'll be visible in version 5.0 with a warning sign notifying that the feature is now obsolete. It is recommended to update the models. Use the newly available feature and remove the obsolete one to warranty that the model will remain compatible with the future versions of COMSOL Multiphysics.

The table below summarize the changes:

TABLE 1-1: OBSOLETE AND NEW FEATURES

OBSOLETE FEATURE	REPLACE IN 5.0 BY
Convective Heat Flux	Heat Flux with 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

Note that it is not possible to add one of the obsolete feature from the COMSOL Multiphysics 5.0 GUI, only the new features are available.

OUT-OF-PLANE HEAT TRANSFER UPDATE

The Out-of-plane heat transfer property, which was available in 2D and 1D geometries is removed in COMSOL Multiphysics 5.0. Now all geometries are considered as 3D geometries and thickness or area parameters are always available.

API Users

For the following features:

- **Heat Source**
- **Heat Flux**
- **Boundary Heat Source**
- **Layer Heat Source**
- **Inflow Heat Flux**

With the **Overall heat transfer rate** option (previously named **Total power** or **Total heat flux**), the thickness parameters (`dz ENTR` and `Ac INPUT`) have been removed. Compatibility of previous code is maintained for most of the cases. However, three configurations require manual update of the code:

- If you specified 2 times the thickness parameter
- If you specified first the thickness and in a second command the total power value
- If you specified first the thickness and then activate the out of plane heat transfer option.

For these 3 cases you need to rewrite this part of the java code by doing the following, for heat source example (2D and 1D axi symmetry geometries):

```
Ptot_50 = Ptot_44_value*ht.d/dz ENTR_44_value
```

where `Ptot_44_value` is the total power value used in previous versions, `Ptot_50` is the total power value to use in COMSOL Multiphysics 5.0 and `dz ENTR_44_value` is the value that were used in previous versions. In 1D geometries, the following update should be done :

```
Ptot_50 = Ptot_44_value*ht.d/Ac INPUT_44_value
```

where `Ac INPUT_44_value` is the value that were used in previous versions.

Flux variables

The flux variables are always given in W/m^2 (also for 2D or 1D geometries). To obtain the corresponding flux variables in W/m on a boundary in 2D for example, the flux variables have to be multiplied by the thickness `ht.d`.

The following variables have been updated following this convention:

- Conductive heat flux, `ht.dflux`

- Convective Heat flux , `ht.cf Flux`
- Translational heat flux, `ht.tr1flux`
- Total energy flux, `ht.teflux`
- Total heat flux, `ht.tflux`
- Boundary convective heat flux, `ht.chflux`
- Radiative heat flux, `ht.rflux`

All flux variables have now the following sign convention : positive heat flux corresponds to heating, whereas negative heat flux to cooling. `chflux` and `rflux` may have opposite sign compared to previous versions depending on the context were they are defined.

INFLOW HEAT FLUX IS ONLY APPLICABLE ON NON-SOLID DOMAINS

The **Inflow Heat Flux** feature is no longer applicable on boundaries adjacent to the following domain features: **Heat Transfer in Solids**, **Biological Tissue** and **Isothermal Domain**. This restriction avoids ill-posed configurations due to the null velocity field in these features. This may change a model results if this boundary condition was previously applied to boundary adjacent to a non-fluid domain.

HEAT TRANSFER WITH PHASE CHANGE NOW USES A VOLUMETRIC FORMULATION

The variables `ht.thetai` now represent volume fractions (they were mass fractions before). This implies changes in the definition of density and heat capacity at constant pressure close to the phase interface. Far from the phase change interface the density and heat capacity are unchanged.

Backward Compatibility With Version 4.3b

INTERIOR FLUXES UPDATED

Because the interior flux variable definitions have been revised, models that use interior fluxes in definition, physics interfaces, or postprocessing show different results in version 5.0 than in earlier versions.

IMPROVED ENERGY BALANCE

Changes introduced in the equation formulation to enable better energy balance may slightly modify the results.

Among the changes, the reference enthalpy, H_{Ref} , the value of which is defined up to a constant, is now set to 0 J/kg at the reference temperature and pressure. This changes the absolute values of the enthalpy and several variables based on it. In particular, the value of the convective heat flux is changed. Even the sign may be changed in some cases. However, the quantity of interest, the net convective heat flux, is identical because the shift induced by the change of H_{Ref} applies for the enthalpy definition at the inlet as well as at the outlet.

Backward Compatibility With Version 4.3a

HEAT TRANSFER IN POROUS MEDIA

Old models using the Porous feature will supported in COMSOL Multiphysics interface. It is however recommended to replace this feature by Heat Transfer in Porous Media (see [Obsolete Features and Variables](#)). For a model using the API, it will be needed to update the model to use Heat Transfer in Porous Media.

SURFACE TO SURFACE RADIATION

The surface radiosity, previously named J , is now named $ht.J$ by default (assuming that ht is the physics interface tag).

Models created using versions 4.3 and 4.3a that contain surface-to-surface radiation support the old syntax in multiphysics interfaces. It is, however, recommended that you update the models to use the new syntax. Models created in a version before 4.2a need to be opened in version 4.3 or 4.3a and resaved before being opened in 5.0.

For a model using the API, you must update the model to use the new syntax.

RADIATION IN PARTICIPATING MEDIA

The radiative intensity variables names, previously named $I1$ and so on, became $ht.I1$ (when there are less than 10 discrete directions) or $ht.I01$ (when there are 10 or more discrete directions) by default.

Models created in versions 4.3 and 4.3a that include radiation in participating media support the old syntax in multiphysics interfaces. It is, however, recommended that you update the models to use the new syntax. Models created in a version before 4.2a need to be opened in version 4.3 or 4.3a and resaved before being reopened in 5.0. In addition, for all versions, it may be necessary to regenerate the default solver to get the correct solver settings.

The S2 quadrature in 2D has been replaced by the LSE symmetric quadrature. It has been updated so that it coincides with the other quadratures. LSE symmetric quadratures were already used for S4, S6, and S8 in 2D and for all 3D quadratures. This quadrature update can result in significant changes in 2D models using S2 quadrature. However, the theoretical error is not different and the 2D and 3D implementations are now consistent.

Backward Compatibility With Version 4.3

THE DEFAULT MODEL LIST HAS BEEN REMOVED

Heat transfer user interfaces no longer have the default model list. This list was used to change the default feature for heat transfer models. You can obtain a similar result by adding the corresponding domain feature immediately below the default feature and setting its domain selection to *All domains*.

Any model files for Java that modify the default model value require a manual update.

IMPROVED STABILIZATION OF HEAT TRANSFER IN SOLIDS

The streamline diffusion stabilization for *Heat Transfer in Solids* and *Biological Tissue* features has been improved. It now accounts for contributions from linear source terms from the *Heat Source*, *Out-of-Plane Convective Cooling*, *Out-of-Plane Radiation*, and *Out-of-Plane Heat Flux* features. This improves the robustness of the convergence when these contributions are large. This change may modify the convergence behavior of existing models.

FRAME SELECTION IN DISCRETIZATION SECTION

The *Frame type* list, which was previously available when moving mesh was detected, has been removed. The frame type is now controlled by the features.

UPDATE OF FEATURE VARIABLE NAMES

In order to avoid name conflicts between feature variable names some of them have been renamed. This change concerns contributive features. Because they are contributing, it is possible to have similar features active on the same boundary. To make it possible to distinguish the variables from each feature, the feature scope has been introduced in the variable name so that variable names now use a prefix. These changes affect the following feature variables:

- Heat flux
- Out-of-plane heat flux

- Convective cooling
- Out of plane convective cooling
- Boundary heat source
- Heat source
- Line heat source
- Point heat source
- Edge/Point heat flux
- Electrochemical reaction heat flux
- Reaction heat flux

For example, in a *Boundary Heat Source* feature, `ht.surf` has been renamed into `ht.bhs1.surf` (assuming that `ht` is the physics interface tag). In the *Heat Flux* feature, the variable that was previously named `ht.q0_hf1` is now `ht.hf1.q0`.

Any model files for Java that use the old variable names in expressions (such as expressions used for plotting or data evaluation that include such old variable names) need to be updated manually.

NEW DEFAULT FLUID FEATURES AND OPAQUE SUBFEATURE

All Model MPH-files created in version 4.3 are converted to appear and behave the same way as in 5.0.

Due to these new default features, model files for Java can be simplified. In addition, model files for Java that add the *Fluid* feature or the *Opaque* subfeature with the default tag require a manual update to avoid duplicate tag conflicts.

WEAK CONSTRAINTS UPDATE FOR FLUID FLOW USER INTERFACES

The weak constraint formulations for some boundary conditions in the following interfaces have been updated:

- Laminar Flow
- Turbulent Flow, k - ϵ
- Turbulent Flow, low-Re k - ϵ
- Non-Isothermal Flow
- Conjugate Heat Transfer

See the CFD Module Release Notes for details.

Weak constraints for the Interior Wall feature are no longer available.

REVISION TO THE TURBULENCE MODELS

The formulations of some variables in the turbulence models have been revised in order to improve accuracy. Simulations using a turbulence model can display a different convergence behavior in version 5.0 compared to version 4.3, and the numerical result can differ slightly between the versions.

Obsolete Features and Variables

The following features are still supported when models created in older versions are loaded in version 5.0. However, they will be removed in future versions. Hence, you should replace them by the new corresponding feature.

- **Porous Matrix** is replaced by **Heat Transfer in Porous Media**.
- **Infinite Element** domain feature should be removed and **Infinite Element Domain** should be declared in the **Definitions** node.
- **Electrochemical Heat Flux** is replaced by **Boundary Heat Source**.
- **Reaction Heat Flux** is replaced by **Boundary Heat Source**.
- **Heat Transfer in Participating Media** is replaced by **Heat Transfer with Radiation in Participating Media**.

The table below contains a list of variables that are available but obsolete. They should each be replaced by the new corresponding expressions because they will be removed in future versions.

TABLE I-2: OBSOLETE VARIABLES AND CORRESPONDING EXPRESSIONS

OBSOLETE VARIABLE	CORRESPONDING EXPRESSION
qin	-qr_in
qout	qr_out
qw	-qr_net
ndflux_acc	ndflux
ndflux_acc_u	ndflux_u
ndflux_acc_d	ndflux_d
ntflux_acc	ntflux
ntflux_acc_u	ntflux_u
ntflux_acc_d	ntflux_d
nteflux_acc	nteflux

TABLE 1-2: OBSOLETE VARIABLES AND CORRESPONDING EXPRESSIONS

OBSOLETE VARIABLE	CORRESPONDING EXPRESSION
nteflux_acc_u	nteflux_u
nteflux_acc_d	nteflux_d

MEMS Module

New Functionality in Version 5.0

PIEZOELECTRIC DEVICES

The Piezoelectric Devices interface is replaced by a multiphysics coupling called *Piezoelectric Effect*, which connects a Solid Mechanics interface and an Electrostatics interface.

- The new formulation makes it possible to decouple the structural and mechanical effects, and gives access to all functionality in the Solid Mechanics interface and an Electrostatics interface.
- In the Solid Mechanics interface, there is a new node *Piezoelectric Material*, with optional subnodes *Thermal Expansion*, *Initial Stress and Strain*, *Mechanical Damping*, *Coupling Loss*, *Dielectric Loss*, and *Conduction Loss (Time Harmonic)*.
- The *Dielectric Loss* node has an option to use the dielectric dispersion, which can be used for modeling electrical losses in both the time domain and the frequency domain.
- In the Electrostatics interface, there is a new *Charge Conservation*, *Piezoelectric* node.
- Existing models that include a Piezoelectric Devices interface will still use the old interface.

MATERIAL PROPERTIES FOR QUARTZ

Material properties have been added for quartz to support both left-handed and right-handed polarized material as well as the two standards commonly used for defining the material properties. The 1949 IRE standard is frequently used in the literature when describing the quartz material properties. The 1978 IEEE standard is used for all other materials in COMSOL (and is typically used in the literature for all materials other than quartz).

FLUID-STRUCTURE INTERACTION, FIXED GEOMETRY

The new multiphysics coupling Fluid-Structure Interaction, Fixed Geometry is a simplified FSI coupling which does not involve a deformed mesh for the fluid. It is

intended for situations where the structural deformations are small enough not to affect the geometry of the fluid domain. The coupling contains two effects:

- The force from the fluid on the solid, that is pressure and viscous force.
- The structural velocity as a boundary condition on the fluid.

New and Updated Models in Version 5.0

- A new Surface Micromachined Accelerometer model has been added, which shows how to use geometry subsequences in a COMSOL model to generate repeated structures (in this case an array of differential capacitive sensors).
- The geometry for the 2D Biased Resonator model sequence is simplified to make these models more accessible as tutorial examples.

Backward Compatibility With Version 4.3b

For models using contact, plot expressions are not updated when you open an old model where the pair names have been edited manually. Also, opening an old model where a pair name has been manually edited might affect the solver configuration. It is recommended that you regenerate a new default solver configuration.

Microfluidics Module

Improved Functionality in Version 5.0

In the Two-Phase Flow, Moving Mesh interface, the Navier-Slip, External Free Surface, Fluid-Fluid Interface, and Wall Contact features are improved for curved boundaries.

Several improvements have been made to the Slip Flow interface:

- **Symmetry, Flow** and **Symmetry, Heat** are merged in a single feature called **Symmetry**. Features are migrated automatically from previous versions.
- **Flow Continuity** and **Heat Continuity** are merged in one feature called **Continuity**. You need to manually update models from previous versions to use the new feature.
- **Periodic Flow Condition** and **Periodic Heat Condition** are merged in one feature called **Periodic Condition**. You need to manually update models from previous versions to use the new feature.
- The Open Boundary feature now includes settings for both heat transfer and fluid flow.
- Shallow channel approximation is not supported by the interface (previously it was unsupported but present in the user interface).
- Models with Stokes flow now use heat crosswind diffusion by default.

Backward Compatibility With Version 4.4

See the improvements to the Slip Flow interface above.

Backward Compatibility With Version 4.3a

The Free Molecular Flow and Transitional Flow user interfaces now form the Molecular Flow Module. A Molecular Flow Module license is required to use these interfaces.

BRINKMAN EQUATIONS AND FREE AND POROUS MEDIA FLOW

The momentum equations in the Brinkman Equations interface and the Free and Porous Media Flow interface have been corrected. The term $-Q_{br} \cdot \mathbf{u} / \epsilon_p^2$ in the right hand side previously lacked the factor $1/\epsilon_p^2$, where ϵ_p is the porosity.

MOBILITY

In physics user interfaces modeling migration (Transport of Diluted Species) anisotropic mobility is now supported, and the mobility can be set to follow the Nernst-Einstein relation. The default setting when making new models has been changed to use the Nernst-Einstein relation. Model files for Java generated prior to 4.3b using Migration will have to be modified manually to account for these changes.

Mixer Module

New Functionality in Version 5.0

MULTIPHASE FLOW CAPABILITY IN THE MIXER MODULE

The new physics branch Rotating Machinery, Multiphase Flow includes two new physics interfaces

- Mixture Model, Laminar Flow
- Mixture Model, Turbulent Flow

Use these physics interfaces to model the flow of a multiphase mixture in equipment containing one or more rotating parts. The mixture consists of a liquid with dispersed particles—bubbles, liquid droplets, or solid particles—which are assumed to travel with their terminal velocity. There are two study types available for this physics interface. For the Time Dependent study type the rotation is achieved by moving mesh functionality, also known as sliding mesh. For the Frozen Rotor study type the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces.

NEW PHYSICS INTERFACES IN ROTATING MACHINERY

The physics branch Rotating Machinery now includes two new physics interfaces:

- Rotating Machinery, Turbulent Flow, SST
- Rotating Machinery, Turbulent Flow, Spalart-Allmaras

The available study types are Transient with Initialization and Frozen Rotor with Initialization. Both turbulence models are low-Reynolds number models, which means that they resolve the flow field all the way down to the wall.

NEW PHYSICS INTERFACES IN ROTATING MACHINERY, NON-ISOTHERMAL FLOW

The physics branch Rotating Machinery, Non-Isothermal Flow now includes two new physics interfaces:

- Turbulent Flow, SST
- Turbulent Flow, Spalart-Allmaras

The available study types are Transient with Initialization and Frozen Rotor with Initialization. Both turbulence models are low-Reynolds number models, which means that they resolve the flow and heat transfer all the way down to the wall.

SST TURBULENCE MODEL IN ROTATING MACHINERY, REACTING FLOW

The physics branch Rotating Machinery, Reacting Flow now contains the Turbulent Flow SST interface. The available study types are Transient with Initialization and Frozen Rotor with Initialization. SST is a low-Reynolds number model, which for reacting flow means that it supports the high Schmidt number model for turbulent mixing.

Molecular Flow Module

New Functionality in Version 5.0

NUMBER DENSITY RECONSTRUCTION

It is now possible to reconstruct the number density on Axial Symmetry boundaries.

New and Updated Models in Version 5.0

NEW MODEL—CHARGE EXCHANGE CELL

Gas cells have several applications in scientific instruments design. A gas cell is used to define a high pressure region within the instruments main vacuum system, for example in this application note we are designing a high pressure region 100 mm long with an operating pressure of 10^{-3} Torr inside the collision cell and a main vacuum system pressure of 10^{-5} Torr. In mass spectrometry typical applications are the removal of mass spectral interferences in Inductively Coupled Plasma Mass Spectrometry (ICPMS) or as a collision cell promoting ion molecule reactions or fragmentation in tandem mass spectrometry (MS-MS).

This model requires the Particle Tracing Module.

MODEL UPDATE—S-BEND BENCHMARK

The *s_bend_benchmark* model is updated to use the new Accumulator feature in the Particle Tracing interfaces. This makes the Monte-Carlo particle tracing approach easier to set up.

Multibody Dynamics Module

New Functionality in Version 5.0

- A new feature *Spring-Damper* has been added. It can connect two points with an elastic spring, viscous damper, or both.
- It is now possible to connect to flexible parts from the Shell and Beam interfaces, in which an Attachment node has been added.
- A Rigid Body DOF Summary table is added to the Multibody Dynamics interface. This makes it easier to check the consistency of complex rigid body assemblies.
- A new section Joint Elasticity is added to the joint settings. Using this option, the constrained degrees of freedom in a joint can now be made elastic. This can serve several purposes:
 - Add a physical stiffness in a joint, like a bushing.
 - Resolve overconstraints in situations where there are closed loops.
 - Create new joint types. By assigning a degree of freedom as elastic, but with no stiffness, it is possible to release a degree of freedom in an existing joint.
- The new Base Motion allows the source side of a joint or Spring-Damper to have a prescribed displacement, velocity, or acceleration.
- The Prescribed Motion in a joint can now be enforced also by weak and unidirectional constraints.
- It is now possible to initialize a transient analysis of a rigid body assembly so that initial conditions are enforced for only a few degrees of freedom, while the rest of the systems adapts to these values. A new Initially Rigid node is available, in which an elastic part can defined to be rigid while setting up the initial values with the new method.
- It is now possible to connect a Multibody Dynamics interface to a Heat Transfer in Solids or Pressure Acoustics interface through couplings under the Multiphysics node.
- The attachment listing mechanism in joints and spring-dampers is improved by grouping similar type of attachments and highlighting the disabled ones.

New Models in Version 5.0

- Mechanics of a Golf Swing

- Biomechanical model of the human body in a sitting posture
- Truck Mounted Crane
- Optimization of a Crane Link Mechanism
- Walking Instability in a Washing Machine
- Vibration in a Washing Machine Assembly

Nonlinear Structural Materials Module

New Functionality in Version 5.0

SMALL STRAIN NONLINEAR ELASTICITY MODELS

A new class of material models intended for small strain nonlinear elasticity have been added. There are a number of different material models available under Nonlinear Elastic Material:

- Ramberg-Osgood
- Power Law
- Uniaxial data
- Bilinear Elastic
- User Defined

HYPERELASTICITY IN MEMBRANES

Hyperelasticity has been added to the Membrane interface.

New Model in Version 5.0

- Inflation of a Spherical Rubber Balloon—Membrane Version

Optimization Module

New Functionality in Version 5.0

OPTIMIZATION PROBLEM FORMULATION

- Multistudy optimization support for derivative-free optimization: using Study Reference features, each objective function contribution and constraint can be evaluated in its own study. This allows mixing studies of different types in the same optimization problem.
- Nested and parameterized optimization: using Study Reference features, it is now possible to include one optimization study as a subproblem in another study and to perform parametric sweeps over optimization problems.
- Simplified parameter estimation: a new Parameter Estimation study step provides an easy-to-use interface for common parameter estimation problems, in particular when measured data is not stored in a .csv file.
- Control variable scaling: a manual scale is now requested for each control variable state and field; the solvers only see a rescaled problem. Given reasonable scales, solvers now treat tolerance settings in a uniform way.

OPTIMIZATION SOLVERS

- COBYLA—Constrained Optimization BY Linear Approximation: a new derivative-free trust-region solver, which uses linear approximation of both objective function and constraints, based on sampling at vertices of a well-formed simplex.
- Augmented Lagrangian method: an augmented Lagrangian wrapper, solving a sequence of modified problems with only simple bounds, provides improved constraint handling for derivative-free solvers, in particular for BOBYQA, which previously did not handle general constraints.
- Improved stop and restart: optimization solvers can now be stopped and restarted in the same way as other solvers.

New Models in Version 5.0

- *Multistudy Optimization of a Bracket*: new model in the Optimization Module Model Library showing weight minimization under constraints on both static stress and fundamental natural frequency.
- *Optimization of a Crane Link Mechanism*: new model in the Multibody Dynamics Module Model Library which among other things illustrates use of the new Augmented Lagrangian constraint handling.

Backward Compatibility With Version 4.3a

The Optimization check box in the Stationary and Time Dependent study steps have been removed. Instead, use the Optimization study step. The Optimization check box does appear in models created in earlier versions of COMSOL where you have selected that check box.

Particle Tracing Module

General New Functionality in Version 5.0

ACCUMULATORS

The Accumulator features are general-purpose tools that define dependent variables, called accumulated variables, on domains or boundaries. The values of these variables are then updated as particles move through the domains or collide with the boundaries.

An Accumulator that is added directly to a particle tracing physics interface can be assigned to a set of domains. The accumulated variables are updated while particles move through these domains. The accumulated variables may be dependent on properties of both the domains and the particles. Built-in settings are available to either compute the accumulated variables based on the current position of each particle, or based on the particle positions at all previous time steps. The Accumulator can be used either to count particles within a certain region, or to compute the density of various particle properties, such as mass density, number density, and charge density.

Alternatively, an Accumulator can be added to a Wall, Outlet, or Axial Symmetry boundary condition. In this case, the accumulated variable is only defined on the boundary, and is only changed when particles interact with boundaries. This provides an efficient means of counting collisions with walls, and can also be used to compute the flux of any particle property at the boundary. For example, an Accumulator can compute the momentum flux at a boundary, which can then be used to compute pressure.

The Accumulators can work with in conjunction with ALE, so the geometry can deform based on the flux of particles onto the surface.

PARTICLE ID PLOTS

The new Particle ID plot type exists which allows for visualization of the individual or collective behavior of a system of particles.

OVERRIDING PARTICLE PROPERTIES

The new Override Properties node can conveniently introduce multiple species of particle into a model, each species having different physical properties. Every feature that releases particles, such as the Release, Inlet, and Secondary Emission features, has

a new setting called Inherit Properties that assigns properties to the particles released by that feature.

SECONDARY EMISSION FEATURE ON BOUNDARIES

Secondary emission of particles is now implemented by adding one or more Secondary Emission nodes to a Wall or Outlet node. The particles released by each Secondary Emission node can be given unique settings for the initial velocity and values of auxiliary dependent variables. This allows multiple distributions of secondary particles to be released during a single particle-wall interaction. By using the new Override Properties feature, it is also possible to release multiple species of secondary particle.

SECONDARY EMISSION ON DOMAINS

The Velocity Reinitialization and Elastic Collision Force features now support secondary emission of particles. Whenever a particle's velocity is reinitialized due to either of these features, the particle can release secondary particles. In addition, the Velocity Reinitialization feature can now make the primary particle stick, freeze, or disappear, instead of reinitializing its velocity.

FALLBACK CONDITIONS ON INLETS

The Inlet node no longer overrides the settings of Wall and Outlet nodes that share the same selection. This allows custom settings to be applied for particles that return to the inlet boundary at a later time. When older models are opened, a Wall node with the Disappear wall condition is automatically added to the Model Builder to ensure backwards compatibility.

CONE AND HEMISPHERICAL PARTICLE RELEASE FROM BOUNDARIES

It is now possible to release particles on Inlets with a hemispherical or conic velocity direction.

UNITS FOR ALL AUXILIARY DEPENDENT VARIABLES

Units can now be specified in the settings window for each Auxiliary Dependent Variable feature. A different unit can be assigned to each auxiliary dependent variable.

ARROW POINT TYPE

In addition to points and comet tails, it is now possible to plot arrows at particle positions in the Particle Trajectories, Particle Tracing, and Particle Tracing with Mass plots.

NEW DRAG LAWS

New options are available when computing drag forces with the Particle Tracing for Fluid Flow interface. In addition to the Stokes, Schiller-Naumann, and Haider-Levenspiel drag laws from COMSOL v4.4, the following new drag laws are available:

- Oseen correction: An alternative to the Stokes drag law at low relative Reynolds numbers.
- Hadamard-Rybczynski: Useful for computing the drag force on extremely pure gas bubbles or fluid droplets.
- Standard drag correlations: A set of piecewise-continuous functions of the Relative Reynolds number, valid over many orders of magnitude.

EROSION FEATURE

An Erosion feature is now available for the Particle Tracing for Fluid Flow interface. The Erosion feature can be added to any Wall or Outlet boundary condition, causing it to compute the eroded particle mass or rate of erosive wear on those boundaries. This feature includes four built-in erosion models:

- Expression
- Finnie
- E/CRC
- Oka
- DNV

DRAG FORCES IN RAREFIED GASES

It is now possible to apply a correction factor to the drag force when the Knudsen number in the surrounding fluid is large. After selecting the Include Rarefaction Effects check box in the physics interface settings, the following correlations for this drag force correction factor become available:

- Basset: Applicable for near-continuum flow.
- Epstein: An asymptotic solution for particles in a free molecular flow.

- Phillips: Shares the same asymptotic behavior as the Basset and Epstein correlations, making it applicable over a wide range of Knudsen numbers.
- Cunningham-Millikan-Davies: Includes three user-defined parameters that can be used to obtain a best-fit to empirical data.

RAREFACTION EFFECTS IN THERMOPHORESIS

Corrections for high Knudsen numbers are now available for the Thermophoretic Force feature. When the Include rarefaction effects check box is selected, the following thermophoretic force models are available:

- Epstein: The low Knudsen-number limit.
- Waldmann: The free-molecular limit.
- Talbot: Shares the same asymptotic behavior as the Waldmann and Epstein models, and includes three tunable user-defined parameters.
- Linearized BGK: Derived from the linearized Bhatnagar-Gross-Krook (BGK) and S model.

IMPROVED DIELECTROPHORETIC FORCE FOR PARTICLES WITH THIN SHELLS

A Shell node can now be added to the Dielectrophoretic Force node. This enables more accurate calculation of the dielectrophoretic force on particles with thin outer layers, such as biological cells.

DEDICATED ACCUMULATOR-BASED FEATURES

The following features are available and use the accumulators to allow easy coupling between the Particle Tracing for Fluid Flow and other physics:

- Erosion: This allows the erosion rate to be computed on selected boundaries (see above).
- Mass Deposition: The allows the total deposited mass to be computed on selected boundaries.
- Boundary Load: This computes the force per unit area or pressure on selected boundaries due to an incident particle flux. The computed value can be used in, for example, a structural mechanics simulation.
- Mass Flux: This computes the mass flux vector or normal mass flux incident on selected boundaries.

New Functionality in Charged Particle Tracing

ETCH FEATURE

An Etch feature is now available for the Charged Particle Tracing interface. The Etch feature can be added to any Wall or Outlet boundary condition, causing it to compute the etch rate on those boundaries. The Etch feature supports both collisional and collisionless plasmas.

INITIAL PHASE SHIFT FOR ELECTRIC AND MAGNETIC FORCES

In the settings for the Electric Force and Magnetic Force nodes, a user-defined phase shift can now be applied to time-harmonic fields. The Initial phase angle setting is available when the Multiply force by phase angle check box is selected.

DEDICATED ACCUMULATOR-BASED FEATURES

The following features are available which use the accumulators to allow easy coupling between the Charged Particle Tracing and other physics:

- Etch: This feature computes the etch rate on selected boundaries (see above).
- Current Density: This feature computes the current density or normal current density on selected boundaries. The computed value can be used in, for example, an electric currents simulation.
- Heat Source: This feature computes the heat source on selected boundaries due to an incident particle flux. The computed value can be used in, for example, a heat transfer simulation.

New Models in Version 5.0

The following new models are available in the Particle Tracing Module's model library:

- Trapped Protons—this model shows the trajectory of nonrelativistic protons within the Earth's magnetic field. The magnetic field of the Earth is computed from an external C function.
- Dielectrophoretic Separation—This model demonstrates the continuous separation of platelets (PLTs) from red blood cells (RBCs) using dielectrophoresis (DEP). This model requires either the CFD Module, Microfluidics Module, or Subsurface Flow Module.

Pipe Flow Module

New Functionality in Version 5.0

- The Pipe Flow interface: Pipe connections for seamless coupling of pipe flow domains to 3D fluid flow domains. In conjunction with this new feature, the Laminar inflow and Laminar outflow boundary condition on the Single Phase Flow interface is now available to licensees of the Pipe Flow Module.
- The Pipe Flow, Non-isothermal Pipe Flow, and Reacting Pipe Flow interfaces: New postprocessing variables for heat transfer coefficients, total friction drag, and more.
- The Pipe Flow, Frequency Domain interface: New postprocessing variables for sound pressure level, intensity, and instantaneous intensity. New property to set the reference pressure for the sound pressure level.
- The Pipe Flow, Transient interface: New postprocessing variable for instantaneous intensity.
- The Non-isothermal Pipe Flow and Reacting Pipe Flow interfaces: Pressure work contribution added for compressible gases expanding over large pressure drops.

New Models and Applications in Version 5.0

- Ground Heat Recovery for Floor Heating: This model compares three different coil patterns embedded in the subsurface with typical thermal properties of an uppermost soil layer in a garden.
- Gas Box: This is a new application available in the Application Libraries. The application models gas flow in a mass flow verifier system.

Plasma Module

New Functionality in Version 5.0

NEW EQUILIBRIUM DISCHARGE INTERFACES

Three new physics interfaces are now available for modeling discharges that are in local thermodynamic equilibrium (LTE):

- Equilibrium DC Discharge—The counterpart of the DC Discharge interface.
- Equilibrium Inductively Coupled Plasma (Requires the AC/DC Module)—The counterpart of the Inductively Coupled Plasma interface.
- Combined Inductive/DC Discharge (Requires the AC/DC Module)—A combination of the two discharges above. This is primarily intended for welding applications.

PHYSICS INTERFACE IMPROVEMENTS AND REORGANIZATION

- The boundary conditions available to drive the plasma have been reorganized and improved.
- Cross Section Import is now a Global level feature instead of being in the Heavy Species Transport context menu.
- Boundary conditions for Electrostatics that can be applied on boundaries adjacent to a Plasma Model domain are now one level higher than the boundary conditions that can only be applied on boundaries adjacent to a Charge Conservation feature. The following boundary condition can now only be applied on boundaries adjacent to a Charge Conservation feature:
- The Terminal feature is improved, and it is now possible to drive the terminal with a fixed current. This makes it possible to compute the DC bias on a capacitively coupled plasma. Simply use the Terminal, specify a fixed current (make sure to activate weak constraints if using the finite element discretization), and once the model has reached a periodic steady-state solution, the period-averaged electric potential on that boundary represents the DC bias (see the model *Plasma Module*>*Capacitively Coupled Plasmas*>*ccp_benchmark*).

- The finite volume discretization, which was previously only available under Advanced Physics settings, is available everywhere and also for the Inductively Coupled Plasma and Microwave Plasma interfaces.
- A thickness setting is now available for 1D and 1D axisymmetric models, which you can use to compute the current flowing through contacts. This is a necessary condition to be able to use a fixed current or fixed charge boundary conditions.

Metal Contact Boundary Condition

The new Metal Contact boundary condition is similar to the Electric Potential boundary condition in version 4.4, but it can only be applied on boundaries adjacent to a Plasma Model feature. The Metal Contact can be driven with a fixed current or an external circuit in addition to an electric potential. Driving certain types of discharges with a fixed current instead of a fixed potential can result in a more stable system and less convergence problems when solving (the reverse can also be true).

Dielectric Contact Boundary Condition

A new Dielectric Contact boundary condition replaces the External Surface Charge Accumulation feature and by default acts in exactly the same way. It is intended for boundaries that represent a dielectric material in contact with the plasma. An additional option has been added that allows for an additional distributed capacitance on the surface. This makes it possible to apply a bias voltage on the back side of a dielectric material without the need to include the dielectric material in the geometry.

Backward Compatibility With Version 4.4

Old models will be updated to use the new boundary conditions, but re-solving a model still gives the same results as before. The following changes to the model are made upon opening:

- If an Electric Potential feature is applied to boundaries adjacent to a Plasma Model feature, it is converted to a Metal Contact feature. The settings from the original model are retained.
- If an Electric Potential feature is applied to boundaries adjacent to a Charge Conservation feature, it remains the same.
- If an Electric Potential feature is applied to a combination of boundaries adjacent to a Plasma Model and Charge Conservation feature, a Metal Contact feature is created for the boundaries adjacent to the Plasma Model feature, and the Electric

Potential feature remains on boundaries adjacent to the Charge Conservation feature.

- Any External Surface Charge Accumulation boundary conditions is replaced by a new Dielectric Contact feature. The settings from the original model are retained. The External Surface Charge Accumulation is now obsolete.

RF Module

New Functionality in Version 5.0

AUTOMATED MESHING

Meshing of models is now optionally automated through the physics-controlled auto mesh suggestion:

- Mesh is scaled by material properties (resolving local wavelength).
- Copy mesh is applied for periodic conditions.
- Swept (3D) or mapped (2D) meshing is applied for perfectly matched layers.

AUTOMATIC SOLVER SUGGESTIONS

Automatic solver suggestion are now available with three options: *Robust*, *Intermediate*, and *Fast*. The Fast option can provide up to 40% faster simulation speed for well-posed models compared to the Robust option.

NUMERIC TEM PORT FOR TRANSMISSION LINES

A Numeric TEM port feature for transmission lines is now available for the following applications:

- Supporting TEM mode between two conductors
- Analyzing S-parameters scaled by input reference impedance

LINEARIZED POLARIZED PLANE WAVE BACKGROUND FIELD

A linearly polarized plane wave background field option has been added for scattering problem. You specify the field by a few parameters rather than by numerical expressions.

OTHER NEW FUNCTIONALITY

- The usability of the periodic port is enhanced. It is no longer necessary to use a reference point to analyze a reciprocal lattice for a periodic cell perfectly aligned to the major axis.
- The Far-field feature now works out-of-the-box without applying specific domain and boundary settings unless the geometry has interior voids.
- Nonlinear eigenvalue problems do not need manual solver configuration to include the linearization point.

New Models in Version 5.0

More than 10 new models in the RF Module Model Library are shipped with version 5.0 or in the first 5.0 model update. The following models are available directly with the 5.0 release:

- MRI Birdcage Coil
- Detecting the Orientation of a Metallic Cylinder Embedded in a Dielectric Shell (example model showing how to use linearly polarized plane wave background field)
- Notch Filter Using a Split Ring Resonator (example model showing how to use numeric TEM ports)
- Car Windshield Antenna Effect on a Cable Harness

Backward Compatibility With Version 4.3a

The following models have been moved from the RF Module Model Library to the Wave Optics Module Model Library:

- Beam Splitter
- Photonic Crystal
- Scattering on Substrate
- Optical Scattering Off of a Gold Nanosphere
- Dielectric Slab Waveguide
- Fabry-Perot Cavity

Semiconductor Module

New Functionality in Version 5.0

DOPING MODELS

The Semiconductor Doping Model feature is replaced by two features: the *Analytic Doping Model* and the *Geometric Doping Model*. The Analytic Doping Model provides similar functionality to the Semiconductor Doping Model available in previous releases, but it is possible to use a rotated coordinate system to specify the region of constant doping (now referred to as the box region). The Geometric Doping Model creates a profile that varies as a function of the distance from a boundary defined in the geometry. For both new features, additional functional forms for the dopant decay are supported.

SEMICONDUCTOR INITIALIZATION STUDY

A new study, *Semiconductor Initialization*, is available, which in two dimensions makes it possible to refine the mesh in regions where the dopant concentration varies rapidly.

NEW SEMICONDUCTOR OPTOELECTRONICS INTERFACES

Two new coupled multiphysics interfaces have been added: *Semiconductor Optoelectronics, Beam Envelopes* and *Semiconductor Optoelectronics, Frequency Domain*. These interfaces enable the modeling of bulk, direct band-gap semiconductors interacting with optical fields.

- The Semiconductor Optoelectronics, Beam Envelopes interface combines the Semiconductor interface with the Electromagnetic Waves, Beam Envelopes interface.
- The Semiconductor Optoelectronics, Frequency Domain interface combines the Semiconductor interface with the Electromagnetic Waves, Frequency Domain interface.
- The coupling occurs through the new *Optical Transitions* node, which works in conjunction with a *Semiconductor-Electromagnetic Waves Coupling* multiphysics node.
- The Optical Transitions node adds a stimulated emission generation term (appropriate for direct band-gap materials) on domains in the Semiconductor

interface. This term is proportional to the optical intensity in the corresponding electromagnetic waves interface.

- Additionally, spontaneous emission (for direct band-gap materials) is accounted for.
- The effect of the light adsorption or emission is accounted for by a corresponding change in the complex permittivity or refractive index in the electromagnetic waves interface.
- In cases where it is not necessary to model the electromagnetic field directly (for example, if the absorption is small, or if you are concerned primarily with emission across a spectrum) it is possible to use the optical transitions feature in an uncoupled Semiconductor interface.
- The new *GaAs PIN Photodiode* model demonstrates the functionality.

TUNNEL CURRENTS

Two new boundary conditions, *Insulator Interface* and *Floating Gate*, have been added to the Semiconductor interface, which make it possible to model tunnel currents through insulating barriers and accumulation of charge on floating gates. The model of an *Electrically Erasable Programmable Read Only Memory (EEPROM)* device demonstrates the use of these boundary conditions.

MODELING OF TRAPS

Detailed modeling of traps on the domain or boundary level is now supported. On the domain level, the Shockley-Read-Hall recombination feature is renamed to *Trap-Assisted Recombination*, and in addition to the Shockley-Read-Hall model, an additional option is available called Explicit trap distribution. Boundary level traps can be added to the Insulator, Thin Insulator Gate, and Insulator Interface features using the applicable check boxes. On both domains and boundaries the feature works in a similar manner:

- Traps can be modeled as discrete energy levels, as a continuum of different energy levels, or as a mixture of the two approaches.
- The *Discrete Energy Level* subfeature is used to specify single discreet levels. Several can be added to the parent feature (either a boundary condition or the Trap-Assisted Recombination feature on the domain).
- The *Continuous Energy Level* feature is available only when Specify continuous or discrete levels is selected in the parent feature. When this is the case one or more Continuous Energy Level features can be added in conjunction with any number of Discrete Energy Level features. The continuous energy levels are represented internally in COMSOL by means of an extra dimension, which makes it possible to

plot quantities such as the trap occupancy as a function of energy at a particular point in space.

- The charge on the traps can be specified by entering a neutral energy or by choosing one of the four trap types with the energy level subfeature.
- The *Analytic Trap Density* and *Geometric Trap Density* features enable the specification of a spatial distribution of trapping impurities in a similar manner to the corresponding features for specifying the doping.

BAND-GAP NARROWING MODELS

The Semiconductor Material model includes two new models for band-gap narrowing:

- The *Slotboom* model, an empirical model frequently used for silicon. Material data is added for silicon.
- The *Jain-Roulston* model, a physics-based model that applies for a range of materials. Material data is available for all materials except GaN.

OTHER IMPROVEMENTS

- The direction of current flow in the Metal Contact feature is changed to be consistent with other COMSOL products. Now a positive terminal current corresponds to conventional current flow into the simulation domain. Note that this change causes a reversal of the sign of the terminal current variable in voltage-biased contacts.
- Several stability improvements have been made in the electrostatics features that apply in Charge Conservation domains. Additionally, the Surface Charge and Space Charge features can now be added to both semiconducting and insulating domains.
- A new *User-Defined Generation* feature is available for more convenient specification of user-defined generation terms (previously, a negative user-defined recombination rate had to be specified).

Structural Mechanics Module

New Functionality in Version 5.0

HYGROSCOPIC SWELLING

Hygroscopic Swelling has been added to all structural mechanic interfaces.

Hygroscopic Swelling is a subnode to Linear Elastic Material. Together with the Nonlinear Structural Materials Module, Hygroscopic Swelling is also available with Hyperelastic Material and Nonlinear Elastic Material.

PIEZOELECTRIC DEVICES

The Piezoelectric Devices interface is replaced by multiphysics coupling called *Piezoelectric Effect*, which connects a Solid Mechanics interface and an Electrostatics interface.

- The new formulation makes it possible to decouple the structural and mechanical effects, and gives access to all functionality in the Solid Mechanics interface and an Electrostatics interface.
- In the Solid Mechanics interface, there is a new node *Piezoelectric Material*, with optional subnodes *Thermal Expansion*, *Initial Stress and Strain*, *Mechanical Damping*, *Coupling Loss*, *Dielectric Loss*, and *Conduction Loss (Time Harmonic)*.
- The Dielectric Loss node has an option to use the dielectric dispersion, which you can use to model electrical losses in both the time domain and the frequency domain.
- In the Electrostatics interface, there is a new *Charge Conservation*, *Piezoelectric* node.
- Existing models that include a Piezoelectric Devices interface will still use the old interface.

FLUID-STRUCTURE INTERACTION, FIXED GEOMETRY

The new multiphysics coupling Fluid-Structure Interaction, Fixed Geometry is a simplified FSI coupling which does not involve a deformed mesh for the fluid. It is

intended for situations where the structural deformations are small enough not to affect the geometry of the fluid domain. The coupling contains two effects:

- The force from the fluid on the solid, that is pressure and viscous force
- The structural velocity as a boundary condition on the fluid

GENERAL IMPROVEMENTS

- The menu structure is improved for all structural mechanics interfaces. All interfaces now share the same grouping of similar menu entries.
- *Spring Foundation* and *Thin Elastic Layer* now support input of full spring and damping matrices, which makes it possible to model coupling between force in one direction and displacement in another direction.
- In axisymmetric models, it is now possible to add a point load on the axis of rotation in the Solid Mechanics interface. This load is called *Point Load (on Axis)*. The old Point Load (which could only be applied to off-axis points) is renamed *Ring Load*.
- For all loads in the structural mechanics interfaces, it is now possible to enter a *total force*.

SHELL, PLATE, AND BEAM IMPROVEMENTS

- Rotational springs have been added to Spring Foundation for the Beam interface.
- The Beam interface supports geometric nonlinearity.
- There are built-in features for coupling beams and shells. In the Shell interface there is a *Beam Connection* node, and in the Beam interface there is a *Shell Connection* node. The coupling appears in several variants:
 - Coupling along a common edge, which is used in both the Shell interface and the Beam interface,
 - Coupling along two parallel edges,
 - Coupling a point on a beam to a section of a shell edge,
 - Coupling a point on a beam to a section of a shell boundary.
- A *Point Mass* node is added in the Shell and Plate interfaces. In this node, it is also possible to enter a mass moment of inertia tensor.
- The *Point Mass* in the Beam interface is extended to allow that the moment of inertia tensor can be entered as a full tensor, and not only along principal axes.
- The definition of offset in the Shell interface has been changed. Previously, it was assumed that the degrees of freedom were located on the midsurface of the shell. Instead, the degrees of freedom are located on the surface described by the mesh.

This makes the offset property more useful for modeling shells on solid geometries without generating the midsurfaces.

- A new boundary condition, *Simply Supported*, has been added to the Shell and Plate interfaces.
- An *Attachment* condition has been added to the Shell and Beam interfaces. This makes it possible to use shells and beams as flexible parts from the Multibody Dynamics interface. This requires the Multibody Dynamics Module.
- The Beam interface has a new default plot where the geometry is shown with line thickness scaled by beam stiffness. The local beam axes are also shown.
- The default plots for the Shell interface have been changed:
 - A new default plot where the physical thickness (including offset) is shown has been added. On the top surface the local shell system directions are also shown.
 - The two plots of von Mises stress on top and bottom surface are replaced by a single plot with the maximum von Mises stress from both surfaces.
- Many new postprocessing variables have been added to the Shell and Plate interfaces.

IMPROVEMENTS TO THE MEMBRANE INTERFACE

The Membrane interface is significantly updated. The main changes are:

- New features: *Prescribed Velocity*, *Prescribed Acceleration*, *Symmetry*, *Antisymmetry*, and *Hygroscopic Swelling*.
- The previous version of the interface always assumed geometric nonlinearity. The new version listens to the *Include geometric nonlinearity* setting in the study step settings in the same way as other structural mechanics interfaces.
- The Linear Elastic material can now also be Orthotropic or Anisotropic. This also affects the settings of the Damping subnode, where non-isotropic loss factors are now allowed.
- When used together with the Nonlinear Structural Materials Module, a number of hyperelastic material models are available.
- *Prescribed Displacement* now supports General Notation, similar to Solid Mechanics.
- For 2D axisymmetry, Edge Load is renamed to *Ring Load*.
- For 2D axisymmetry there is a new load type called *Point Load (on Axis)*.
- The new Membrane interface is not backward compatible with the old one. When opening an old model, the old interface is used.

CONTACT AND FRICTION IMPROVEMENTS

Significant changes have been made to the *Contact* and *Friction* nodes. The changes, however, mostly appear in the *Equation View*. Several variables have a new scope. As an example, a contact pressure is now named `solid.cnt1.Tn` instead of `solid.Tn_p1`. This means that there is now one contact pressure for each Contact node, rather than one for each contact pair. In most cases, these changes do not affect you as a user. In some special situations where you explicitly have referenced contact variables, the automatic conversion of an old model into a new one could fail. You will then have to replace some expressions manually. In the Heat Transfer in Solids interface, *Thermal Contact* and *Pair Thermal Contact* nodes that use the contact pressure from Solid Mechanics may also need to be updated.

New Models in Version 5.0

- Mixed-Mode Debonding of a Laminated Composite
- Connecting Shells and Solids
- Connecting Shells and Beams

Backward Compatibility With Version 4.3b

- If an old model using the Beam interface is opened, the Euler-Bernoulli formulation is used. The Beam Formulation selection is still shown but cannot be changed from Euler-Bernoulli. If Advanced Physics Options is enabled, then the new section **Backward Compatibility** is shown. If you clear the **Use pre 4.4 formulation** check box, then the new formulation is used. Doing so enables the use of Timoshenko beams, but you must manually handle solver settings like segregation and scaling. This legacy option does not support beams mixed with solids or shells in cases where the same names for degrees of freedom were used in both physics interfaces.
- For models using contact, plot expressions are not updated when you open an old model where the pair names have been edited manually. Also, opening an old model where a pair name has been manually edited might affect the solver configuration. It is recommended that you regenerate a new default solver configuration.

Backward Compatibility With Version 4.3a

- In the Solid Mechanics physics (SolidMechanics), the property `ControlsSpatialFrame` no longer exists, and COMSOL now always assumes that

the displacements control the spatial frame. Remove all lines setting the value of `ControlsSpatialFrame`. See Support Knowledge Base article 1156.

- When opening old models that include contact, the penalty factor control is set to **User defined** and the contact normal penalty factor to the expression used in the model.

Subsurface Flow Module

New Functionality in Version 5.0

UNIFIED MASS TRANSPORT INTERFACES

Using the Transport of Diluted Species interface in free flow, you can now add saturated and partially saturated porous domains. The old Solute Transport interface is replaced by a new interface, called *Transport of Diluted Species in Porous Media*. This interface has many strengths added to model convection, adsorption, dispersion, and volatilization in saturated and partially saturated porous media:

- New entry point in the Model Wizard: *Transport of Diluted Species in Porous Media*. This is a Transport of Diluted Species interface with a default *Porous Media Transport Properties* domain feature.
- Replaces and unifies the old Solute Transport and Species Transport in Porous Media interfaces.
- Unified nomenclature for all mass transport interfaces.
- Consistent and inconsistent stabilization methods are introduced for transport of solutes in porous media.
- Using the new *Dispersion* section, you can model dispersion in isotropic and transverse isotropic porous media.
- The *Diffusion* section makes it possible to model diffusion in isotropic and anisotropic porous media.
- The dependent variables for concentration are defined in mol/m^3 , and with a new feature, *Mass Based Concentrations*, you can specify the molar mass per species.
- Improved nomenclature for porous media mass transfer.

HEAT TRANSFER IN FRACTURES

The new *Fracture* feature is active when the Heat Transfer in Porous Media check box is selected in the Physical Model section. The Fracture feature models heat transfer in fractures and thin porous media. By using this feature, you avoid meshing the fracture thickness. It provides a lumped model for cost-effective modeling and dedicated settings to define the fluid and the solid parts as well as the flow properties. The flow can be defined by a fluid-flow interface, in particular by the Fracture Flow interface. Consistent and inconsistent stabilization methods are also available.

Backward Compatibility With Version 4.4

SOLUTE TRANSPORT

The Solute Transport interface has been replaced by the new interface Transport of Diluted Species in Porous Media.

Wave Optics Module

New Functionality in Version 5.0

AUTOMATED MESH SUGGESTIONS

Meshing of models is now optionally automated through the physics-controlled auto mesh suggestion:

- Mesh is scaled by material properties (resolving local wavelength).
- Copy mesh is applied for periodic conditions.
- Swept (3D) or mapped (2D) meshing is applied for perfectly matched layers.

LINEARIZED POLARIZED PLANE WAVE BACKGROUND FIELD

A linearly polarized plane wave background field option has been added for scattering problem. You specify the field by a few parameters rather than by numerical expressions.

OTHER IMPROVEMENTS

- The usability of the periodic port is enhanced. It is no longer necessary to use a reference point to analyze a reciprocal lattice for a periodic cell perfectly aligned to the major axis.
- The Far-Field feature now works out-of-the-box without applying specific domain and boundary settings unless the geometry has interior voids.
- Nonlinear eigenvalue problems do not need manual solver configuration to include the linearization point.

New Models in Version 5.0

The following new models are included in the Wave Optics Module's Model Library:

- Bandgap Analysis of a Photonic Crystal
- Gaussian Beam Incident at the Brewster Angle
- Transverse Modes for a Symmetric Laser Cavity

Material Library

New Materials and Material Data in Version 5.0

- Added data properties and materials for the following: CMSX-8, CMSX-7, CMSX-486, Waspaloy, Haynes 282, Haynes 263, R-41, C52100, C51000, C71500, C63800, C15000, C11000, C10100, C10200, SiC, Zr, Pb(Mg_{0.33}Nb_{0.66})O₃, Kovar, Re, Ru, Co, Ni, Dy, Er, TiC, water, air, He, Ne, CO, F, Ar, Methane, IN-100, Udimet 500, Udimet 700, GX12CrMoVNbN9-1 (GP91), P91, P92, Ta-10W, Ti-13V-11Cr-3Al, molybdenum, Rene 95, Waspaloy and Astroloy.
- Thermal conductivity has been added for the following materials: Ge (liquid), n-heptane (C₇H₁₆), cyclohexane, ethylbenzene, o-xylene, m-xylene, p-xylene, isopropylbenzene (cumene C₉H₁₂), chlorobenzene (C₆H₅Cl), MoSi₂, Ti C-110M, Udimet 710, hexane, octane, Vaseline (white petroleum jelly), and the following copper alloys: copper alloys: C28000, C31400, C43300, C46400, C37300, C52400, C36000, C54400, C16210, C90500, and C51100.
- Heat capacity has been added for the following materials: GaSb, AlSb, Mg₂Pb, CdS, Fe₃S, HgS, CdSe, HgSe, ZnTe, HgTe, SnS, SnSe, SnTe
- Fatigue data has been added for the following materials: 1018, C10200, Ti, 347, Ni TD, MAR-M200; 301, 304, 316NG, and CF-8M stainless steels; 63SN-Pb solder.
- Expansion data has been added or improved for the following materials: Pb-5Sn, Pb-10Sn, Pb-60Sn, Pb-63Sn, solid Hg, Sb, Be, Bi, Cr, Au, Fe, Pb, Ni, Nb, Pd, Pt, Na, W, Cd, and low carbon steels
- Stress rupture has been added for the following materials: 301, 17-4 PH, M252, Inconel 700, Inconel 700C, Astroloy, Udimet D-979, Nimonic 81, Nimonic 115, Nimonic 942, Pyromet 860, Rene 95, Nickel TD, Udimet 400, Udimet 700, Udimet 710, Unitemp AF2-1 DA, L-605, Nimonic 901, and Incoloy 807.

LiveLink for MATLAB

New Functionality in Version 5.0

The following sections list new functionality in LiveLink™ for MATLAB®.

GENERAL UTILITIES

- `mphevaluate`: Evaluate parameters and perform unit conversion on the command line.
- `mphopen`: Enhanced way of opening model files. Support for searching the most recent files.
- `mphxmeshinfo`: Support for internal degrees of freedom.
- `mphmean`, `mphmax`, `mphmin`, and `mphint2` now support the `dataseries` option.

GEOMETRY AND FILE HANDLING

- `mphwritestl`, `mphreadstl`: write and read STL files.
- `mphsurf`: Create surfaces, which can be saved as STL files and later imported into COMSOL.
- `mphinterpolationfile`: Saves MATLAB data as text files that can later be imported into COMSOL. Supported formats are `grid`, `sectionwise`, and `spreadsheet`.

ADDITIONAL NEW FUNCTIONALITY

- A number of wrapper functions have improved error handling that result in better-looking error messages, which should be easier to understand.
- When saving and loading models a message appears in the COMSOL Multiphysics Server window that includes the filename. This makes it easier to follow what models are loaded in the server.

LiveLink for Excel

New Functionality in Version 5.0

The following major new functionality is now available in the LiveLink™ for Excel®:

- Localization is added so the COMSOL ribbon in Excel supports all the languages that COMSOL Multiphysics supports.
- Support for VBA (Visual Basic for Applications) is added. This means that you can use VBA to control all aspects of COMSOL modeling. Up to 99% of the COMSOL API is supported. This functionality makes it possible to preprocess data in Excel, submitting the data and parameters to the COMSOL model, and retrieve results from COMSOL, which can be further processed in VBA or inserted into Excel work books.
- Sweeps have been improved. Now parametric sweeps as well as material and function sweeps are supported.

The COMSOL API for Use with Java®

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 are deprecated in COMSOL 5.0:

TABLE I-3: DEPRECATED METHODS AND THEIR REPLACEMENTS

DEPRECATED METHOD	NEW METHOD
<code>com.comsol.model.OptFeature.field()</code>	<code>OptFeature.comp()</code>
<code>com.comsol.model.OptFeature.field(String)</code>	<code>OptFeature.comp(String)</code>
<code>com.comsol.model.Group.identifier()</code>	<code>Group.paramName()</code>
<code>com.comsol.model.Material.identifier()</code>	<code>ModelEntity.tag()</code>
<code>com.comsol.model.MaterialModel.identifier()</code>	<code>ModelEntity.tag()</code>
<code>com.comsol.model.ModelNode.identifier()</code>	<code>ModelEntity.tag()</code>
<code>com.comsol.model.physics.Physics.identifier()</code>	Use <code>tag()</code> instead.
<code>com.comsol.model.physics.MultiphysicsCoupling.identifier()</code>	<code>ModelEntity.tag()</code>
<code>com.comsol.model.Group.identifier(String)</code>	<code>Group.paramName(String)</code>
<code>com.comsol.model.Material.identifier(String)</code>	<code>ModelEntity.tag(String)</code>
<code>com.comsol.model.MaterialModel.identifier(String)</code>	<code>ModelEntity.tag(String)</code>
<code>com.comsol.model.ModelNode.identifier(String)</code>	<code>ModelEntity.tag(String)</code>
<code>com.comsol.model.physics.Physics.identifier(String)</code>	Use <code>tag(string)</code> instead.
<code>com.comsol.model.physics.MultiphysicsCoupling.identifier(String)</code>	<code>ModelEntity.tag(String)</code>
<code>com.comsol.model.ModelEntity.name()</code>	<code>ModelEntity.label()</code>
<code>com.comsol.model.ModelEntity.name(String)</code>	<code>ModelEntity.label(String)</code>

TABLE I-3: DEPRECATED METHODS AND THEIR REPLACEMENTS

DEPRECATED METHOD	NEW METHOD
<code>com.comsol.model.ParameterEntity.set(String, int, double)</code>	Use the 0-based version (<code>setIndex</code>) instead.
<code>com.comsol.model.ParameterEntity.set(String, int, double)</code>	Use the 0-based version (<code>setIndex</code>) instead.
<code>com.comsol.model.ParameterEntity.set(String, int, double[])</code>	Use the 0-based version (<code>setIndex</code>) instead.
<code>com.comsol.model.ParameterEntity.set(String, int, int)</code>	Use the 0-based version (<code>setIndex</code>) instead.
<code>com.comsol.model.ParameterEntity.set(String, int, int[])</code>	Use the 0-based version (<code>setIndex</code>) instead.
<code>com.comsol.model.ParameterEntity.set(String, int, int, double)</code>	Use the 0-based version (<code>setIndex</code>) instead.
<code>com.comsol.model.ParameterEntity.set(String, int, int, int)</code>	Use the 0-based version (<code>setIndex</code>) instead.
<code>com.comsol.model.ParameterEntity.set(String, int, int, String)</code>	Use the 0-based version (<code>setIndex</code>) instead.
<code>com.comsol.model.ParameterEntity.set(String, int, String)</code>	Use the 0-based version (<code>setIndex</code>) instead.
<code>com.comsol.model.ParameterEntity.set(String, int, String[])</code>	Use the 0-based version (<code>setIndex</code>) instead.

COMSOL 4.4 API Changes

PHYSICS INTERFACES

- The default Initial Value feature in the following interfaces have been extended
 - Transport of Concentrated Species
 - Reacting Flow in Porous Media (rfcs)
 - Reacting Flow
 - Rotating Machinery, Reacting Flow

Previously, only the initial mass fractions could be specified. Now the initial values can be specified in terms of mass fractions, mole fractions, molar concentrations, number densities, or densities. Opening an old Model MPH-file, *Mixture specification* is set to *Mass fractions*, and the mass fractions specified are entered in the respective text fields. In old model files for Java, commands specifying the initial value of a mass fraction return an error in version 5.0. To specify the initial mass

fraction of a species `w2`, which is the second in the list of species (seen in the Dependent Variables section when clicking the interface), change the COMSOL Java command

```
model.physics("chcs").feature("init1").set("w2", 1, "0.1");
```

into

```
model.physics("chcs").feature("init1").set("w0", 2, "0.1");
```

- The Pressure Acoustics, Frequency Domain interface and the Boundary Mode Acoustics interface have a new default feature. If the default feature has been edited in a Java file, the following line should be added to the Java file to obtain the old behavior:

```
model.physics("acpr").feature().create("pam1",  
"PressureAcousticsModel").selection().all();
```

for Pressure Acoustics, Frequency Domain, or

```
model.physics("acbm").feature().create("pam1",  
"PressureAcousticsModel").selection().all();
```

for Boundary Mode Acoustics.

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

- For models using contact, the variable field name in the solver settings for the old contact variables is different than in earlier versions. For example:
 - In version 5.0: `mod1_solid_contact_old_p1`, `mod1_solid_cm_old_p1`.
 - In earlier versions: `mod1_solid_contact_p1_old`, `mod1_solid_cm_p1_old`.
- The parameter form in the property `EquationForm` can no longer be set to `ModeAnalysis` in 3D and 2D axisymmetric calculations for Pressure Acoustics, Frequency Domain, and Pressure Acoustics, Transient. In the same cases, the parameter `modeFreq` has been removed.

COMSOL 4.3b API Changes

PHYSICS INTERFACES

- The *Crosswind diffusion* formulation has been updated for all physics user interfaces supporting Crosswind diffusion. Models solved with the new formulation can give different results than models solved in version earlier than 4.3b. Java files can be modified to retain old crosswind formulations. Please contact support for details.

- The default settings have changed for the Background Pressure Field in the Pressure Acoustics interfaces. Add the following line to obtain the old behavior of this feature:

```
model.physics("acpr").feature("bpf1").set("c", 1, "acpr.c_c");
```
- In the Acoustics Module, the Far-Field Calculation feature is no longer available in 1D and 1D axisymmetry.
- The shape function property `border` has been deprecated and replaced by `order`.

COMSOL 4.3a API Changes

GENERAL API CHANGES

- Class files compiled with COMSOL 4.3 or earlier need to be recompiled if they contain code that changes the contents of `model.selection()`.
- The `solutionintitle` property governs whether to show solution information in plot titles. But since solution information has never been shown for 1D plots, regardless of the content of the property, the property has been removed from 1D plot groups.

FLUID FLOW INTERFACES API

- Fluid flow features in the CFD Module and Microfluidics Module that previously created a scalar Lagrange multiplier field with default name `model.un_lm`, now create a vectorial Lagrange multiplier field with default name `model.u_lm`. The default component names in 3D are `model.u_lm`, `model.v_lm` and `model.w_lm` respectively. Java files must be updated accordingly.
- Weak constraints for the Interior Wall feature are no longer available. Any references to its weak constraint parameter (`weakConstraints`) or Lagrange multipliers must be removed.
- The **Automatic** setting for the **Pseudo time stepping** property now sets the variable `<ptag>.locCFL` to the built-in variable `CFLCMP`, which in turn triggers a PID regulator via the automatic solver suggestions. Here, `<ptag>` is the physics interface tag. Java files where pseudo time stepping is active and have local CFL number set to **Automatic** must be modified by adding the command

```
model.physics(<tag>).prop("PseudoTimeProperty").set("CFLNumExpr", 1, "Manual");
```

before calling a solver.

COMSOL 4.3 API Changes

- In the Acoustics Module, the far-field variables `pfar` and `Lp_far` have new names with full scope. They are now referred to as `acpr.ffc1.pfar` and `acpr.ffc1.Lp_far`, respectively. Any model files for Java that use the old variable names in expressions (such as expressions used for plotting or evaluation that include such old variable names) require a manual update.

- The following methods

```
model.physics(<tag>).feature(<ftag>).params();  
model.physics(<tag>).prop(propname).params();
```

are deprecated and replaced by the methods

```
model.physics(<tag>).feature(<ftag>).param();  
model.physics(<tag>).prop(propname).param();
```

COMSOL 4.2a API Changes

- Class files compiled with COMSOL 4.1 or lower version need to be recompiled.
- The far-field feature in the Electromagnetic Waves interface has changed from being a boundary feature with a boundary selection only, to a domain feature with a domain selection. It also has a subfeature—a boundary selection.
- The dependent variable associated with gauge fixing in the Magnetic Fields, Magnetic and Electric Fields, Electromagnetic Waves, and Transient Electromagnetic Waves interfaces is now unique to each interface. It is no longer available in the model scope, for example, `mod1.psi`. Instead the gauge fixing field is only available in the interface scope, for example, as `mod1.mf.psi`.
- In the scattered field formulation in the Electromagnetic Waves interface, the scattered field is no longer available in the model scope (for example, `mod1.relEx`). Instead the scattered field is only available in the interface scope as, for example, `mod1.emw.relEx`.
- In the Solid Mechanics interfaces (`SolidMechanics`, `PiezoelectricDevices`, `AcousticStructure`, `Poroelasticity`, `ThermallyInducedStresses`, `JouleHeatingThermalExpansion`, `TransientAcousticStructure`, `FluidStructureInteraction`), tensors in local coordinate systems (`e1`, `ee1`, `s1`, `si`, and `ei`) have new names. The coordinates of the local system (for example `x1`, `x2`, `x3`) are no longer used. Instead 1, 2, and 3 are used together with double indexing for all components. As an example, `e1x2` is replaced by `e122` and `e1x2x3` is replaced by `e123`. The tensors `si` and `ei` are now called `si1` and `ei1`.

- In the Darcy's Law interface and the Richards' Equation interface in the Subsurface Flow Module, the compressibility of fluid is now a material parameter and no longer has a default value. If the default value was used, you now have to set the value. The following example sets the permeability to the old default value:

```
model.physics("d1").feature("smm1").set(chif_mat,userdef);
model.physics("d1").feature("smm1").set(kappa,4e-10);
```

- In the Poroelasticity interface in the Subsurface Flow Module, the compressibility of fluid and the Biot-Willis coefficient are now material parameters and no longer have a default value. If the default value was used you now have to set the value. The following example sets the permeability to the old default value:

```
model.physics("d1").feature("smm1").set(chif_mat,userdef);
model.physics("d1").feature("smm1").set(kappa,4e-10);
```

- The Level Set and Phase Field interfaces now include the Initial Interface feature by default. If you have a model that was created in an earlier version of COMSOL, it will fail to create a feature with the same tag name.

COMSOL 4.1 API Changes

The following changes were made to the COMSOL API between version 4.0a and 4.1:

- In version 4.0a the property `nonlin` in the stationary solver could have the values `auto`, `on`, `off`, and `linearized`. The default in most cases was `auto`, but in some cases, specifically in a frequency-domain study, the default was `linearized`. In version 4.1 the possible values are `auto`, `on`, `off`, and `linper`, with `auto` as default, except in the *Frequency Domain, Linearized* study step where the default is `linper`. The value `linearized` can still be set, but this is treated as `auto`. If the problem is linear and the linearization point is zero, `auto` should give the same solution as `linearized`. For nonlinear problems where the linearization point is zero, the solution given by `auto` might be slightly different, and probably more accurate, while changing to `off` should give exactly the same solution as `linearized`. When there is a nonzero linearization point, it is relevant to use the `linper` option. This option works like `linearized` except that source terms that contribute to the linearized problem must be enclosed by the `linper` operator.
- In the Darcy's Law interface, Brinkman interface, Richards' Equation interface, and Poroelasticity interface (in the CFD Module and Subsurface Flow Module), the permeability and porosity are now material parameters and no longer have a default value. If the default values were used in a model file for Java, you now have to set the value. The following example sets the permeability to the old default value:

```
model.physics("d1").feature("d1m1").set(kappa_mat,userdef);
model.physics("d1").feature("d1m1").set(kappa,3e-11);
```

- The pressure variable in solid mechanics, typically `solid.pw`, now only gets allocated degrees of freedom for incompressible materials. If you have referred to it, for example, during segregation in the solver, the code will have to be changed to remove the field.
- The solution to eigenvalue and eigenfrequency problems may now appear in a different order.
- The Batteries & Fuel Cells Module's boundary feature, `BoundaryReactionCoefficient`, is tagged by `rc` instead of `brc` by default. This means that files that explicitly refer to the old default tag name must be modified to refer to the new tag name.

COMSOL 4.0a Java API Changes

The following changes were made to the COMSOL API between version 4.0 and 4.0a:

- The load face parameter (`Fph`) of the *Phase* feature in `AcousticStructure`, `TransientAcousticStructure`, `ThermallyInducedStresses`, `SolidMechanics`, `JouleHeatingThermalExpansion`, `Poroelasticity`, and `FluidStructureInteraction` has changed interpretation from degrees to radians.
- The `physics.field()` of `Beam` and `Truss` are not backward compatible with 4.0. Scalar fields are now vector fields.
- The variables for strain and stress in the `Shell` interface have been renamed. Variable names that began with a lowercase `s` now begin with an uppercase `S`. For variables names that ended in lowercase `l` (`L`), the letter was moved to the third position. For example, `emXXl` is now `emLXX`.
- Force loads on solids and trusses have been moved from the material to the spatial frame. This implies that the index notation of some variables has changed from uppercase `XYZ` to lowercase `xyz`.
- The acoustics interfaces (`PressureAcoustics`, `TransientPressureAcoustics`, `AcousticStructure`, and `TransientAcousticStructure`) have been moved from the material to the spatial frame. This implies that index notation of vector variables has changed from uppercase `XYZ` to lowercase `xyz`.

This concludes the release notes for COMSOL Multiphysics version 5.0.

