



COMSOL Multiphysics®

Release Notes



VERSION 4.3



COMSOL Multiphysics Release Notes

© 1998–2012 COMSOL

Protected by U.S. Patents 7,519,518; 7,596,474; and 7,623,991. Patents pending.

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

COMSOL, COMSOL Desktop, COMSOL Multiphysics, and LiveLink are registered trademarks or trademarks of COMSOL AB. Other product or brand names are trademarks or registered trademarks of their respective holders.

Version:

May 2012

COMSOL 4.3

Contact Information

Visit www.comsol.com/contact for a searchable list of all COMSOL offices and local representatives. From this web page, search the contacts and find a local sales representative, go to other COMSOL websites, request information and pricing, submit technical support queries, subscribe to the monthly eNews email newsletter, and much more.

If you need to contact Technical Support, an online request form is located at www.comsol.com/support/contact.

Other useful links include:

- Technical Support www.comsol.com/support
- Software updates: www.comsol.com/support/updates
- Online community: www.comsol.com/community
- Events, conferences, and training: www.comsol.com/events
- Tutorials: www.comsol.com/products/tutorials
- Knowledge Base: www.comsol.com/support/knowledgebase

Release Notes

COMSOL Multiphysics version 4.3 contains many new functions and additions to the COMSOL product suite. These *Release Notes* provide information regarding new functionality in existing products and an overview of new products.

We have strived to achieve backward compatibility with the previous version and to include all functionality that is available there. COMSOL 4.3 is compatible with COMSOL 3.5a, COMSOL 4.0, COMSOL 4.0a, COMSOL 4.1, COMSOL 4.2, and COMSOL 4.2a with a few exceptions. Those exceptions are listed in this document.

COMSOL Multiphysics

New Products in Version 4.3

The following new products are introduced with COMSOL Multiphysics version 4.3:

- Corrosion Module, for modeling of corrosion and corrosion protection. See [Corrosion Module](#) for more information.
- Nonlinear Structural Materials Module, for structural analysis of materials with nonlinear behavior. See [Nonlinear Structural Materials Module](#) for more information.
- Pipe Flow Module, for fluid flow, hydraulic transients, heat transfer, mass transport, and acoustics in pipe and channel systems. See [Pipe Flow Module](#) for more information.

New Functionality in Version 4.3

GENERAL COMSOL DESKTOP FUNCTIONALITY

- The style of the COMSOL Desktop has been updated with a new look-and-feel that includes the names of the settings windows and the look and position of toolbars. If you want to continue using the previous look-and-feel, you can change the **Toolbar style** from **COMSOL** (the default) to **Classic** in the preference settings for **User interface** in the **Preferences** dialog box.
- The models in the model libraries that ship with COMSOL Multiphysics and the add-on modules now include information about the solution times and the computer used for measuring those solution times. You find this information below the model description that the **Model Library** window displays when you select a model.
- The model and documentation directories to synchronize with COMSOL's Model Library Update server are configurable through preference settings or COMSOL command options. In the **Model Library** window, icons indicate if a model is stored in a compact or full format. The compact format means that the models do not include the mesh and the solution data. This makes it possible to include many more models on the product DVD.
- The **Model Wizard** provides more flexibility when adding new physics and new studies: On the **Select Study Type**, you can choose which physics to solve for in the

study from a list under **Selected physics**. You can also choose to deactivate new physics in other existing studies in the model (the default) or to include it in all studies. You specify that behavior using the **Deactivate added physics in other studies** check box.

- To disable and enable nodes in the Model Builder you can now use the shortcut keys F3 and F4, respectively.
- You can now use the keyboard shortcut F7 for **Build Selected** in the geometry and mesh sequences, and for **Compute to Selected** in the solver configurations where F8 is the keyboard shortcut for **Compute All**.
- The nodes under Definitions and in the physics branches can now be sorted: Right-click the **Global Definitions** node or a **Definitions** node in a model and select **Sort by Type** to sort all common definitions nodes under subbranches such as **Variables**, **Functions**, and **Coordinate Systems**. Likewise, right-click a physics interface node and select **Sort by Space Dimension** to sort all physics nodes under **Domains** (for material modes, body loads, space charges, and so on), **Boundaries** (for boundary conditions, face loads, and so on), **Edges**, and **Points** depending on their geometric scope. Right-click and select **No Sorting** to return to an unsorted list.
- You can now copy all text in text fields in the settings windows (for example, feature comments, solver logs, and report documentation) using Ctrl+A (Command-A on Mac).
- The names of the variables for the spatial coordinates can be changed in the Model nodes' settings windows for cases where the default coordinate names (for example, x , y , and z) are not applicable.
- You can reset the preferences settings to the default settings by clicking the **Restore Defaults** button in the **Preferences** dialog box.
- As a convenience, it is now possible to open a model and its associated documentation as a PDF file at the same time by clicking **Open Model and PDF** in the **Model Library** window.

GEOMETRY

- You can now extrude and revolve planar faces directly in the 3D geometry (that is, the faces do not need to be created in a work plane).
- You can now use the **Cone** feature to create cones (including truncated cones or conical frustums) by defining the bottom radius and the top radius in addition to using the bottom radius and the semi-angle.
- The tools for virtual geometry operations are now also available for 2D geometries.

- In the geometry sequences you can now move individual geometry nodes (geometry objects and features) up and down to modify and rearrange the geometry sequence.
- Sweeping can now be done along a sequence of edges.
- For the **Polygon** feature you can use coordinate data from a text file or from a table to specify the polygon vertices (in addition to using vectors of coordinate values).
- You can now turn off the automatic rebuild of the geometry that occurs when, for example, clicking on a material or physics node. This setting, **Automatic rebuild**, is available in the main **Geometry** node and as a preference setting for new geometries in the **Preferences** dialog box. Turning off the automatic rebuild can be useful to avoid rebuilding a complex 3D geometry until the geometry is finalized.
- It is now possible to open Model MPH-files that contain CAD geometries even if the license does not include the CAD Import Module or any of the CAD LiveLink products. Doing so clears all solutions, and any geometry features in the model that require the CAD Import Module kernel (for example, to import a CAD geometry from file) do not work; however, you can still use the model by, for example, replacing the CAD geometry by a COMSOL geometry.
- The selection list in the **Selection List** window is now a tree structure with geometry features as parent nodes and objects as child nodes. Selecting a top-level node in the tree automatically changes the selection to all child nodes. All objects are still selected even if the number of objects later increases.
- The geometry measurements for vertex coordinate values, distances between vertices, and lengths of straight edges are now displayed using 10 digits, and it is possible to select and copy all measured numerical values from the **Measurements** window.
- New functions and operators:
 - `isinf` and `isnan` functions to determine if a value or expression is equal to infinity or $-\infty$, or if it is “not-a-number” (NaN), respectively.
 - The “spatial at” operator `scope.ati(<coordinate expressions>, expr)` evaluates the expression `expr` in the geometry with the given model scope on an i -dimensional entity in the point given by the coordinate expressions. For example, `root.mod1.at1(0,y,dm)` evaluates `dm` in the two-dimensional geometry on an edge in the point $(0, y)$.
 - `linearizedelem`, a geometric variable that returns one inside a linearized mesh element and zero otherwise. Note that the faces of the linearized mesh elements

are not considered to be linearized themselves. You can use this variable to identify mesh elements that has become linearized elements.

MESH

- New tools make it possible to partition an imported mesh into several domains or boundaries, for example, in preparation for modeling a geometry that consists of different parts. You can use a box or ball to define the extent of a partition and also delete and join geometric entities and create vertices for a partitioning of the geometry.
- New tools for mesh control in the geometry sequence. You can add mesh control vertices, edges, faces, or domains to control the local mesh without introducing additional geometric entities in the geometry. This makes it possible to, for example, create a separate domain for a boundary layer mesh or for a structured hexahedral mesh without explicitly dividing a geometry that represents a single material or part.
- The default method for handling sharp corners for boundary layer meshing is splitting, but trimming is now also available as an alternative method.
- You can now export the mesh as an STL binary file or text file.
- Improved element transition for boundary layer meshes and across removed control entities.

PHYSICS

- A new Events interface is available for adding states (discrete states and indicator states) and implicit and explicit events. Implicit events are triggered when a condition for a state is fulfilled (becomes less than zero, for example). Explicit events reinitialize a variable at an explicit time; the explicit events can be periodic. You can use events, for example, to switch or cycle between different operational modes depending on some states in the model.

STUDIES AND SOLVERS

- Support for *load cases*:
 - Loads and constraints can act conditionally in models using load groups and constraint groups, which you can include in load cases using various combinations of the defined groups.
 - Use load groups for loads and other Neumann conditions. The loads can be scaled in the load cases using a scalar weight.
 - Use constraint groups for constraints and other Dirichlet conditions.
 - Available for the Solid Mechanics interface in COMSOL Multiphysics and for structural mechanics in general in the Structural Mechanics Module.
- Improved support for parametric sweeps, including multiparameter sweeps with several layers of parameters, where you can choose to solve for specified combinations of parameter values or for all combinations.
- Support for *analysis cases*—the possibility to modify physics and variables in the study step: In the study step’s settings window, select the **Modify physics tree and variables for study step** check box. This creates a copy of the physics and variables parts of the model tree in the **Model Builder** window, and you can use these nodes to create different analysis cases by disabling and enabling certain physics nodes and variable sets locally for a specific study, in addition to the possibility to enable and disable entire physics interfaces.
- New **Cluster Sweep** and **Batch Sweep** nodes provide tools for starting and stopping cluster jobs and batch jobs for individual parameter combinations, also allowing access to parametric sweep results before completion of full sweep. The cluster sweep starts multiple COMSOL batch processes, one for each parameter set, that solve the current study on a cluster.
- The weighted average (sum of squares) error norm used by the solvers for stationary and time-dependent problems has been modified. The new norm gives equal weight to the error estimate for all fields and states solved for. The old norm gave equal weight to the error estimates for all degrees of freedoms solved for. The difference for many models is small, but for models with a combination of dependent variables with vastly different number of degrees of freedoms the error norm can be rather different. For these models the scaling or the tolerances might need adjustment to produce comparable results.
- A new double dogleg nonlinear solver provides improved convergence for contact modeling in structural mechanics. In the Structural Mechanics Module and the MEMS Module, the double dogleg nonlinear solver is selected by default when a

stationary study is generated and contact features are present in the model. For the majority of contact problems this solver has more stable convergence properties than the Newton solver, which is the general default solver. Using the same settings, the double dogleg solver tends to be somewhat slower than the Newton solver on problems where both solvers converge. It is, however, often possible to use larger load steps when using the double dogleg solver.

- Updated PARDISO direct linear solver, which can improve solver performance for large models and many cores.
- On Linux clusters COMSOL now uses Hydra instead of the MPD. You no longer need to start MPD:s before starting COMSOL.

RESULTS

- The reports can now be saved as .docx files for use with Microsoft® Word, in addition to the HTML format. Saving the reports as .docx files makes it possible to edit and extend the report contents in Microsoft Word and to further save it as a PDF file, for example.
- The result plots contain improved support for plotting data from parametric sweeps. In the plot group nodes' settings window you can now select parameters for the plot in several different ways: all, first, last, from a list of parameter values, or manually (selecting parameter indices). In the graph plots, you can choose the source of the data on the x -axis as the inner solution (typically the time but can also be an expression, frequency spectrum, or phase), outer solutions (typically parameters), or all solutions.
- Sector 2D and Sector 3D data sets provide the possibility to plot and analyze the solution on the full geometry while exploiting sector symmetry when setting up and solving the model. This is useful for reducing solution time and memory requirements for models with a geometry that include sector symmetry. You can then use a Sector 2D or Sector 3D data set to form the solution on the full geometry using rotations and reflections.
- 2D Histogram plot type: You can now create 2D histograms, where the colored surface represents the counts that fall into each bin as a function of two quantities.
- The reports can be extended with text segments that can include formatted text using special character formats for user interface labels, strong (bold), emphasis, code segments, equation variables, constants, subscripts, and superscripts. There is also support for Greek letters and dashes. A new **Equation** node supports creating custom equations by using LaTeX commands or importing an image.

- You can now delete and clear all result tables at once by right-clicking the main **Tables** node and selecting **Delete All** or **Clear All**.
- Under **Export**, **Table** is now available for exporting results data from tables to text files or data files.
- The storage of table data can now be on a file or both in the model and on file in addition to storing it in the model. This can make the model file smaller when evaluating large amount of data in tables. This setting is available under **Storage** in the table nodes' settings windows.
- You can now disable the automatic update of plots when, for example, re-solving the model. This setting is available in the main **Results** node's settings window: Clear the **Automatic update of plots** check boxes to disable updates of the plots. There are also preference settings in the **Preferences** dialog box that control if automatic plot updates should be active for new models and for models loaded from files. Turning off the automatic update of plots can be useful if the plots are complex and require extensive resources for computation and rendering.
- In point graphs and global plots you can now use the x -axis to represent the phase.
- The radius in, for example, the points in a plot of particle trajectories can now be an arbitrary expression so that the point radius represents the magnitude of the chosen expression.
- In plots of particle trajectories, comet tail is a new point style that plots the particles with a "comet tail" to visualize the particle velocities, for example. The tail magnitude and direction can be defined using an arbitrary vector expression; the default vector is the negative particle velocity.
- Individual precision preference settings for graph, 2D axis, 3D grid, and color legend values in the **Graphics** section of the **Preferences** dialog box.
- The row index is now available as the x -axis data for Table Graph plots.
- The Coordinate System Volume (in 3D) and Coordinate System Surface (in 2D) plots can now also be used to plot matrix variables.

ABOUT THE PREDEFINED PHYSICAL CONSTANTS

The predefined physical constants in COMSOL version 4.3 use the values from 2006. The current values available from <http://physics.nist.gov/cuu/Constants/index.html> use a newer revision from 2010 with values recommended for international use by CODATA, which means that there are small differences in the values for some of the physical constants in COMSOL when

comparing with the 2010 revision. For version 4.3a, we plan to update the values for these physical constants to the ones used in the 2010 revision.

Backward Compatibility vs. Version 4.2a

ELCPLGENINT ELEMENT NO LONGER AVAILABLE

The `elcplgenint` element has been removed in version 4.3; use `elcplscalar` instead.

DOMAIN NUMBERING

Domain numbers may change for 3D geometries with a COMSOL representation of the geometry. This may happen if you rebuild the geometry sequence for a model that you created in an earlier COMSOL version. Also, the numbering of all geometric entities may change when importing a mesh.

FACE NORMALS AND EDGE DIRECTIONS

Face normals and edge directions may change for geometries with CAD representation. This may happen if you rebuild the geometry sequence for a model that you created in an earlier COMSOL version. Face normals and edge directions are now consistent between CAD and COMSOL representations.

MESH IMPORT

The numbering of all geometric entities, as well as the face normals and edge directions, may change when importing a mesh.

DELETE ENTITIES

The algorithm for the Delete Entities geometry feature has changed slightly. The main difference is that the old algorithm preserved the object type for solid, surface, and curve objects. Existing Delete Entities features in old models use the old algorithm, while new Delete Entities features use the new algorithm. In the API, you need to set the property `compat` to 4.2a to recover the old behavior.

MATERIAL PROPERTIES

The physical quantity for the bulk viscosity defined under Acoustics, Thermoacoustics Model and Acoustics, Viscous Model has changed to Bulk Viscosity.

TEMPERATURE CONSTRAINTS

The sign of the constraint for temperature has been changed to match the convention used in other physics. This has no effect on the models unless weak constraints are used. In that case, the Lagrange multipliers sign is changed.

Backward Compatibility vs. Version 4.2

NEW SCALING OF ABSOLUTE TOLERANCES

A new technique to automatically compute the scaled absolute tolerance has been added. This mechanism computes the error weights in a new way for the BDF solver (for time-dependent problems) for fields using the automatic scaling method. The new method is enabled by default but you can turn it off in the **Time Stepping** (clear the **Update scaled absolute tolerance** check box). Opening an old model the new mechanism will be turned off automatically. But when regenerating a new solver, it will be enabled. Note that when running a model using the API, the new mechanism will be used. If you want the old behavior you can set the property `ewtrescale` to `off` for the time-dependent solver you are using.

As an effect of this new mechanism the default relative and (global) absolute tolerance for models involving Reaction Engineering Physics interfaces has been changed to $1e-3$ and $1e-4$, respectively. This change will also apply to newly generated solvers, but not when reading and old Model MPH-files. For a model using the API that does not set these tolerances explicitly, the new defaults will apply.

Backward Compatibility vs. Version 3.5a

DEFORMED GEOMETRY INTERFACE

The *Parameterized Geometry* application mode in versions 3.5a, which is limited to 2D, is replaced with the *Deformed Geometry* interface in version 4.3. This interface is available in 2D and 3D. The *Deformed Geometry* interface deforms the mesh using an arbitrary Lagrangian-Eulerian (ALE) method and is not the parameterized geometry using geometric parameter sweeps (see above).

In the version 4.3 interface, the *Linear Displacement* and *Similarity Transform* boundary conditions are not yet available as preset conditions.

In version 4.3, you can create the corresponding conditions by manually entering variables.

BACKWARD COMPATIBILITY FOR PRE-3.5A MODELS

COMSOL 4.3 can load models saved from version 3.5a. For loading models from earlier COMSOL versions than 3.5a you need to load them in COMSOL 3.5a and then save them. For simplifying this task a utility is available where you can convert all files in a directory from versions 3.0–3.5 to version 3.5a. See the section “COMSOL Convertpre35a Command” for Windows, the section “COMSOL Convertpre35a Command” for Linux, or the section “COMSOL Convertpre35a Command” for the Mac in the *COMSOL Installation and Operations Guide* for more information.

AC/DC Module

New Functionality in Version 4.3

ROTATING MACHINERY, MAGNETIC 3D AND MIXED FORMULATION

- The Rotating Machinery, Magnetic physics interface is now applicable to three-dimensional models and can be used to model motors, generators, and other electrical machinery in 3D.
- A mixed potential formulation has been implemented for this physics interface. In addition to the magnetic vector potential formulation, it now supports a scalar potential formulation (similar to the one used in the Magnetic Fields, No Currents interface), applicable in domains without current flow. The two formulations can be used together in the same model and are particularly effective for the modeling of electrical machinery with moving parts. Suitable interface conditions are set up automatically to ensure seamless integration of the two formulations.

COIL FEATURES IN 3D AND AUTOMATIC CURRENT CALCULATION

- The Coil features (Single-Turn Coil Domain, Multi-Turn Coil Domain), previously available only for 2D and 2D axisymmetric models, are now available for 3D models as well.
- The 3D Single-Turn Coil Domain introduces an additional dependent variable to ensure correct conservation in the domain. A Magnetic Fields interface with a 3D Single-Turn Coil Domain feature can effectively replace the Magnetic and Electric Fields interface in many applications.
- The 3D Multi-Turn Coil Domain represents a homogenized model for a coil constituted of a great number of tightly-wound wires. In addition to simple built-in coil shapes, such as Linear or Circular, the new Coil Current Calculation study step can compute automatically the path of the wires for coils with complex geometry.
- The 3D coils are compatible with the new 3D Rotating Machinery, Magnetic interface.

POROUS MEDIA MIXTURE MODEL FOR AC/DC

- The mixture model for Porous Media and Archie's Law, previously available only for the Electric Currents interface, are now available for all the interfaces in the AC/DC Module.

- For Porous Media, a dedicated subnode can be used to compute an averaged value of the electrical conductivity, relative permittivity, and relative permeability of mixtures of up to five materials.
- A dedicated subnode is available to compute the electrical conductivity in saturated nonconductive porous media using Archie's Law.

S-PARAMETERS COMPUTATION FOR TERMINALS AND CONVERSION

- Terminals now support a Terminated (terminating impedance) condition, in addition to the existing Current, Voltage, and Circuit conditions. The value of the impedance can be specified manually for each terminal or can be set globally in the physics interface node.
- When solving a Frequency Domain study with this condition, the Terminals will automatically define an S-parameter matrix variable that can be evaluated using a Global Matrix Evaluation node in postprocessing or exported using the Touchstone Export functionality.
- It is now possible to convert lumped parameter matrices in the Global Matrix Evaluation node (for example, from admittance to impedance). By specifying a characteristic impedance, it is also possible to compute the S-parameter matrix from Z and Y and vice versa.
- When performing a Terminal sweep and a frequency sweep in the same study, the plots of the computed lumped parameters or S-parameters as functions of the frequency are created automatically.

OTHER IMPROVEMENTS AND CHANGES

The following changes apply to all the interfaces in the AC/DC Module, unless explicitly stated:

- All the interfaces now support the new multiphysics Infinite Elements under the Definitions node, introduced in version 4.2a. This improvement simplifies the creation of multiphysics coupling between interfaces in models using infinite elements.
- Features now distinguish between Solid and Non-solid domains. This setting defines their behavior when the domains are subject to, for example, rotations or deformations.
- In 2D axisymmetric models, it is now possible to add specialized boundary and point condition on the symmetry axis (for example, Line Charge on Axis for Electrostatics).

- Improved handling of out-of-plane thickness for 2D models. In addition to the global Out-of-plane Thickness setting in the physics interface node, it is now possible to change the thickness of individual domains or boundaries using specialized features.
- For the Electric Currents, Shell interface, specialized features can be used to locally change the thickness of the shell.

Backward Compatibility vs. Version 4.2a

- Due to a general refactoring and optimization of the entire AC/DC Module, obsolete variables that were hidden but were still available in previous versions have now been removed. Among these are the old coil variables and lumped parameters. In 4.3, all the variables defined by coil features follow the template XCoil_<name>, where X is the variable name (V for voltage, Z for impedance, and so on) and <name> is the value written in the **Coil Name** field. Plots and expressions using the old variables must be updated.
- The AC/DC Module now uses the new-style Infinite Elements, available under the **Definitions** node. 4.2a and earlier models containing the old-style Infinite Elements (in the physics interface) will retain them when opened in version 4.3, but it will not be possible to add more old-style Infinite Elements. It is advised to replace those features with the new-style Infinite Elements whenever possible.
- The Rotating Machinery, Magnetic physics interface is now defined using the Material frame. However, Rotating Machinery, Magnetic interfaces in 4.2a or earlier models, if opened in 4.3, will still be defined using the Spatial frame, to ensure that the results of the model do not change between versions.

Backward Compatibility vs. Version 3.5a

CHANGE IN DEPENDENT VARIABLES

Version 4 has a comprehensive set of inductive formulations based on the magnetic vector potential. These cover all modeling situations except for situations with strongly nonlinear conductivity (superconductors) for which magnetic field H -based formulations are more suitable.

The 2D formulations based on the magnetic field H as dependent variable in version 3.5a are not yet included in version 4.3. A full set (3D, 2D, and 2D axial symmetry) of H -based formulations is planned for a future release.

Models created in version 3.5a that are built using an H -based formulation will not include any physics interface settings when opened in version 4.3. Geometrical data and definitions (constants and expressions) will be imported and the physics can manually be added using the (vector potential-based) Magnetic Fields physics interface.

PAIR BOUNDARY CONDITIONS

The Contact Resistance, Transition, and Perfect Magnetic Conductor boundary conditions are not yet available as assembly *Pair boundary conditions*.

A new set of slit-based, discontinuous boundary conditions, which does not require assemblies is introduced in version 4. In most models, this can replace the assembly pair boundary condition.

Acoustics Module

New Functionality in Version 4.3

- Thermoacoustic-Shell Interaction is a new predefined multiphysics interface in 3D for models that contain thermoacoustic domains and pressure acoustic domains as well as shells. The Thermoacoustic-Shell Interaction interface supports three types of shells: exterior shells, interior shells, and uncoupled shells (shells on boundaries that are not connected to any acoustic domain).
- Boundary mode analysis study for Thermoacoustics, Frequency Domain 2D and 1D axisymmetric models, which you can use to determine propagating and nonpropagating out-of-plane modes.
- New and redesigned periodic boundary conditions for solid, piezo, piezo-acoustic, and thermoacoustics applications, including the following new types of periodic boundary conditions:
 - General Floquet periodicity conditions.
 - Cyclic symmetry condition for sector periodicity and dynamic cyclic symmetry.
 - Also, an option to prescribe the periodic conditions on the entire displacement vector has been added (it is the new default option).
- New far-field plots that are generated independently of the models' mesh elements have been added. These plots make it easier to define far-field plots, and you can also create 3D polar plots directly in the COMSOL Desktop. The Bessel Panel and Cylindrical Subwoofer models have been updated to include new far-field plots.
- Default far-field plots are generated when a far-field domain feature is available in the model.
- New and updated intensity variables for the Pressure Acoustics and Aeroacoustics interfaces:
 - Intensity (RMS)
 - Intensity magnitude (RMS)
 - Instantaneous intensity
 - Instantaneous intensity magnitude
- Low-reflecting boundary condition, mainly intended for letting waves pass out from the model without reflection in time-dependent analysis for acoustics and structural

mechanics. As a default, it takes material data from the domain in an attempt to create a perfect impedance match for both pressure waves and shear waves

New Models

- **Generic 712 Coupler:** The model is based on an actual Brüel and Kjær device and the results are compared to measurements.
- **Condenser Microphone:** Model using the new Membrane interface (requires the Structural Mechanics Module and the AC/DC Module).
- **Porous Absorber:** Model that demonstrates the use of the new Floquet periodic boundary condition.
- **Acoustic Scattering:** Model using the new far-field plots.

Backward Compatibility vs. Version 4.2a

- The far-field variables `pfar` and `Lp_far` get new names with full scope. They are now referred to as `acpr.f fc1.pfar` and `acpr.f fc1.Lp_far`, respectively. Any user Model Java-files that use the old variable names in expressions (such as expressions to plot or evaluate expressions including these variables) will require a manual update.
- The default Coordinate stretching type for a Perfectly Matched Layer has changed to Polynomial.

Backward Compatibility vs. Version 4.2

- The Pressure Acoustics and Transient Pressure Acoustics interfaces in version 4.2 are now called Pressure Acoustics, Frequency Domain and Pressure Acoustics, Transient, respectively. The names for other acoustics interfaces have changed in a similar way.
- When creating an orthotropic or anisotropic linear elastic material in the Elastic Waves or Poroelastic waves interfaces, the default is now to use Voigt notation. This will cause a model created using an old Java script to fail. Adjust the Java file so that it explicitly selects Standard notation instead of Voigt notation when using orthotropic or anisotropic materials.

Batteries & Fuel Cells Module

New Functionality in Version 4.3

NEW INTERFACES AND FEATURES

- New Separator feature in the Lithium-Ion Battery and Battery with Binary Electrolyte interfaces.
- New Electrode, Shell interface for modeling electrodes in shells.
- New Electrode Potential, Reference Electrode, and Electric Reference Potential features for handling potentiostatic control of electrochemical cells in all Electrochemistry interfaces.

IMPROVED FUNCTIONALITY

- To improve the solver performance, the porous electrode solid concentration variable c_s (Battery with Binary Electrolyte and Lithium Ion Battery interfaces) is now treated as a field. Models created prior to 4.3 with solver sequences that have been manually altered to disable solving for the c_s variables will lose that setting.
- Improved stabilization of porous electrode coupling reaction terms in the Transport of Concentrated Species, Transport of Diluted Species, Reacting Flow, and Tertiary Current Distribution, Nernst-Planck interfaces.

New Models

- All-solid-state lithium-ion battery model.
- Capacity fade model, using the new Events interface to simulate battery capacity loss during cycling.

CAD Import Module and LiveLink Products for CAD

New Functionality in the CAD Import Module

- Faster meshing for imported CAD files, particularly when using Virtual Geometry operations.
- The file-based CAD import now supports the following new and updated file formats:
 - Creo Parametric 1.0 (new)
 - Acis R22
 - Catia V5 R21
 - Autodesk Inventor 2012
 - Parasolid 23, 24
 - SolidWorks 2012

New Functionality in LiveLink for SolidWorks®

- Automatically link selected parameters from the CAD file to the COMSOL model for easier set up of parametric sweeps.
- Performance improvements for large geometries by the possibility to turn associativity off for vertices, edges, and faces.
- The One Window for SolidWorks now supports the following functionality:
 - Connection to an external server/cluster
 - Virtual geometry operations
 - Particle tracing
 - Improved selection and selection highlighting
 - Display of progress information in a docked window

- Faster meshing for imported CAD files, particularly when using Virtual Geometry operations.
- The file-based CAD import now supports the following new and updated file formats:
 - Creo Parametric 1.0 (new)
 - Acis R22
 - Catia V5 R21
 - Autodesk Inventor 2012
 - Parasolid 23, 24
 - SolidWorks 2012

New Functionality in LiveLink for AutoCAD®

- Faster meshing for imported CAD files, particularly when using Virtual Geometry operations.
- The file-based CAD import now supports the following new and updated file formats:
 - Creo Parametric 1.0 (new)
 - Acis R22
 - Catia V5 R21
 - Autodesk Inventor 2012
 - Parasolid 23, 24
 - SolidWorks 2012

New Functionality in LiveLink for Pro/ENGINEER®

- Automatically link selected parameters from the CAD file to the COMSOL model for easier set up of parametric sweeps.

- Faster meshing for imported CAD files, particularly when using Virtual Geometry operations.
- The file-based CAD import now supports the following new and updated file formats:
 - Creo Parametric 1.0 (new)
 - Acis R22
 - Catia V5 R21
 - Autodesk Inventor 2012
 - Parasolid 23, 24
 - SolidWorks 2012

New Functionality in LiveLink for Inventor®

- Automatically link selected parameters from the CAD file to the COMSOL model for easier set up of parametric sweeps.
- Faster meshing for imported CAD files, particularly when using Virtual Geometry operations.
- The file-based CAD import now supports the following new and updated file formats:
 - Creo Parametric 1.0 (new)
 - Acis R22
 - Catia V5 R21
 - Autodesk Inventor 2012
 - Parasolid 23, 24
 - SolidWorks 2012

New Functionality in LiveLink for Creo™ Parametric

- Automatically link selected parameters from the CAD file to the COMSOL model for easier set up of parametric sweeps.

- Faster meshing for imported CAD files, particularly when using Virtual Geometry operations.
- The file-based CAD import now supports the following new and updated file formats:
 - Creo Parametric 1.0 (new)
 - Acis R22
 - Catia V5 R21
 - Autodesk Inventor 2012
 - Parasolid 23, 24
 - SolidWorks 2012

New Functionality in LiveLink for SpaceClaim®

- Faster meshing for imported CAD files, particularly when using Virtual Geometry operations.
- The file-based CAD import now supports the following new and updated file formats:
 - Creo Parametric 1.0 (new)
 - Acis R22
 - Catia V5 R21
 - Autodesk Inventor 2012
 - Parasolid 23, 24
 - SolidWorks 2012

CFD Module

New Functionality in Version 4.3

TURBULENT MIXING

The user interfaces for Transport of Diluted Species and Transport of Concentrated Species include a new Turbulent Mixing subnode to the Convection and Diffusion material model. The Turbulent Mixing node models the additional mixing caused by turbulence by adding turbulent diffusivity to the molecular diffusivity.

FLUID FLOW

- Stabilization has been updated to increase accuracy for meshes with high aspect ratio. This update affects the following interfaces:
 - Single Phase Flow
 - High Mach Number Flow
 - Two-Phase Flow
 - Mixture Model
 - Bubbly Flow
 - Euler-Euler Model
- Vacuum Pump is now a separate feature instead of an option in the Outlet feature. The Vacuum Pump feature has enhanced graphical support that shows the intended flow direction during physics set up.
- New scaling has been introduced for time-dependent turbulence problems. The new scaling makes the simulation more robust but still accurate. This makes, for example, the Airlift Loop Reactor model easier to solve.

ROTATING MACHINERY

Automatic support for a time-dependent angular velocity has been added in the Rotating Domain feature. In the new implementation the displacement angle of the rotating domain is defined as an ODE variable and integrated using the rotational frequency input. The implementation supports all types of time-dependent frequencies (analytic, interpolation functions, and so on).

Backward compatibility:

- The new implementation does not solve for the rotating wall velocities. Due to this, along with the addition of the displacement angle in the solution, the number of degrees of freedom (DOFs) in a model created in 4.3 will differ compared to that of a model created in a previous version.
- When you open an model using rotating machinery created in a previous version, the previous implementation will be used, retaining the previous number of degrees of freedom.
- Any attempt to access to the wall velocity variables in a Java script using 4.3 will fail.

NON-ISOTHERMAL FLOW AND HEAT TRANSFER

- A new boundary condition, Interior Wall, is available on interior boundaries in the Non-Isothermal Flow interfaces. It makes it possible to define a wall condition between two fluid domains. This is especially useful to model thin walls as boundaries. You no longer need to define a solid domain with a wall boundary condition on both sides, which can result in a dense mesh. This boundary condition implements wall functions when using the k - ϵ or k - ω turbulence model.
- The inflow heat flux boundary condition has been improved so that heat transport into the domain is controlled by the flow convection. This change prevents unphysical results like computing high temperatures in no-flow parts of inflow heat flux boundaries.
- Enthalpy and internal energy are now calculated using state integrals. This gives increased accuracy for heat and energy balances.

Backward Compatibility vs. 4.2a

DISCONTINUOUS GALERKIN CONDITIONS

The variables used to set up *discontinuous Galerkin conditions* have been revised. Neither of those variables were available as postprocessing variables; they were only visible in the **Equation View** nodes' **Variables** lists. The most notable change is that the variable `K_Stress_Tensor` has changed sign in order to conform with the literature. Please contact support for further information.

This compatibility information affects the following fluid-flow interfaces:

- Single-Phase Flow
- Brinkman Equations

- Free and Porous Media Flow
- Two-Phase Flow, Level-Set
- Two-Phase Flow, Phase Field
- Bubbly Flow
- Mixture Model

FAN, GRILLE, AND VACUUM PUMP FEATURES

The Fan, Grille, and Vacuum Pump features are now separate features instead of options in the Inlet and Outlet features.

Old models with Inlet or Outlet features get the new features instead when opened. The Model Java-file backward compatibility is extensive but not complete. If you experience problems with old Model Java- files, please contact COMSOL's technical support.

This compatibility information affects the following fluid-flow interfaces:

- Creeping Flow
- Laminar Flow
- Fluid-Structure Interaction
- Non-Isothermal Flow
- Slip Flow

HEAT TRANSFER

- Due to the new Inflow Heat Flux boundary condition behavior, the results of models using it are affected. This should lead to better physical results. It is, however, possible to add a compensation flux to the model to retrieve the old behavior. Please contact COMSOL's technical support for this purpose.
- Heat Continuity and Periodicity features from Non-Isothermal flow interface have been updated to support wall function when using a k - ε or k - ω turbulence model.

DEPENDENT VARIABLE CHANGE IN THE ROTATING MACHINERY INTERFACES

When building models in the current release using the Rotating Machinery interfaces, the dependent variable `vwall`, describing the rotating wall velocity, will no longer be solved for. Opening a model built in a previous version, the previous implementation will be retained (including `vwall`). A Java file generated from the previous model may fail to run using the current version. This happens when the no longer present `vwall`

variable is accessed, or if a solver setting for this variable is applied. To avoid this, remove any access to `vwall` in the generated Java file.

Chemical Reaction Engineering Module

New Functionality in Version 4.3

- Improved generation of space-dependent interfaces: Reaction Engineering chemistry can be exported to Free and Porous Media, Heat Transfer in Porous Media, and Diluted Species in Porous Media.
- Reacting systems including equilibrium reactions can now be exported to Transport of Concentrated Species and Nernst-Planck interfaces.
- Improved CHEMKIN import, which is demonstrated in the Compression Ignition of Methane model.
- Uniform scaling of concentration variables has been added to the Reaction Engineering interface and significantly improves the performance of models with many chemical species.

New Models

- Protein adsorption
- Dissociation in a tubular reactor
- Compression ignition chemistry in an engine, with more than 300 reactions

Backward Compatibility vs. 4.2a

Unit support for the characteristic potential length has been corrected. When you open an old model, COMSOL will automatically convert the potential characteristic length to the correct value. However, in previous versions you may have typed 3.3 for the potential characteristic length of Argon (in angstroms); you must now type 3.3[angstrom] in order to obtain identical results.

Corrosion Module

The Corrosion Module, which is new in version 4.3, is intended for the modeling of corrosion and corrosion protection. The descriptions made available by the module are based on current and potential distribution in galvanic cells. Its modeling capabilities covers galvanic corrosion, cathodic protection, anodic protection, and sacrificial anode protection.

The Corrosion Module consists of a number of chemical species transport, fluid flow, heat transfer, electrochemistry, and corrosion interfaces. These physics interfaces describe the potential in the electrolyte and in the corroding or protected metallic structure. The electrode reactions can be described using arbitrary electrode kinetic expressions of the overpotential for the anodic and cathodic reactions. The potential and current distribution can also include the influence of mass transport and heat transfer, which means that the Corrosion Module makes it possible to model composition and temperature in a corrosion cell in detail. The ability to account for mass transport in the electrolyte allows for modeling of corrosion caused by variations in, for example, salt concentration, oxygen concentration, and pH.

The basis of the Corrosion Module is the mass and current balances in the electrolyte and in the corroding or protected metal structure. In the electrolyte, the current balance is defined together with individual species balances for the charged species and the electroneutrality condition. In the metallic structure, the current balance is described using Ohm's law for the current density. The electron transfer reactions, at the interface between the metallic structure and the electrolyte, couple the transport processes in the electrolyte with the current flowing in the metallic structure. The module contains predefined formulations for the abovementioned processes. In addition, the module modifies the Heat Transfer interfaces so that predefined terms for heat sources caused by losses in the electrochemical process are described properly.

Electrodeposition Module

New Functionality in Version 4.3

NEW INTERFACES AND FEATURES

- New Electrode, Shell interface for modeling electrodes in shells.
- New Electrode Potential, Reference Electrode, and Electric Reference Potential features for handling potentiostatic control of electrochemical cells in all Electrochemistry interfaces.

IMPROVED FUNCTIONALITY

- Moving Boundary Smoothing algorithm in the Electrodeposition interfaces.
- The equation formulation for mass conservation of surface concentrations has been changed in order to compensate for boundary stretching.
- The default Zero Normal Mesh Velocity boundary node in the Electrodeposition interfaces is no longer applied to interior boundaries of Free Deformation domains.
- Improved stabilization of porous electrode coupling reaction terms in the Transport of Concentrated Species, Transport of Diluted Species, Reacting Flow, and Electrodeposition, Tertiary interfaces.

New Models

- Superfilling model
- Resistive wafer model
- Inductor coil model

Geomechanics Module

New Functionality in Version 4.3

- Large-strain plasticity.
- Creep modeling support using volumetric, deviatoric, and user-defined creep models.
- Soil Plasticity can now be augmented with an extra yield surface, which limits the maximum principal stress for tension cut-off.

Heat Transfer Module

New Functionality in Version 4.3

- A new boundary condition, Interior Wall, is available on interior boundaries in the Conjugate Heat Transfer and Non-Isothermal Flow interfaces. It makes it possible to define a wall condition between two fluid domains. This is especially useful to model thin walls as boundaries. You no longer need to define a solid domain with a wall boundary condition on both sides, which can result in a dense mesh. This boundary condition implements wall functions when using the k - ϵ or k - ω turbulence model.
- A new parameter, the refractive index, is available in interfaces including radiation in participating media. This parameter makes it possible to define a refractive index different than one like in liquids (for example, water) or solids (for example, glass).
- A new option, solar position, is available to define external radiation source in 3D for all interfaces including surface-to-surface radiation features. This option provides a convenient way to define the direction and intensity of the incident radiation coming from the Sun from the position on the Earth, the date, and the time.
- New choices are available to define the total heat source or the total heat flux in Line Heat Source, Layer Heat Source, and Edge Heat Flux features. These options simplify the definition of models where the total heat source or heat flux is known because you no longer have to convert it into heat flux (or heat source) per surface area or length.
- The surface-to-surface radiation features are now available in the Thin Conductive Shell interface. This is especially useful to model radiative heat transfer between thin structures.
- The previous infinite elements feature has been removed. The Heat Transfer interfaces uses the **Infinite Element Domain** defined in the **Definitions>Domain Properties** section. This simplifies the definition of the infinite domains when a model includes multiple physics.
- The inflow heat flux boundary condition has been improved so that heat transport into the domain is controlled by the flow convection. This change prevents unphysical results like computing high temperatures in no-flow parts of inflow heat flux boundaries.

- Enthalpy and internal energy are now calculated using state integrals. This gives increased accuracy for heat and energy balances.
- A new material, humid air, is available in the Liquids and Gases material library.

Backward Compatibility vs. 4.2a

- The Fan, Grille, and Vacuum Pump features are now separate features instead of options in the Inlet and Outlet features.
Old models with Inlet or Outlet features get the new features instead when opened. The Model Java-file backward compatibility is extensive but not complete. If you experience problems with old Model Java-files, please contact COMSOL's technical support.
This compatibility information affects the Conjugate Heat Transfer interface.
- Due to the new Inflow Heat Flux boundary condition behavior, the results of models using it are affected. This should lead to better physical results. It is, however, possible to add a compensation flux to the model to retrieve the old behavior. Please contact COMSOL's technical support for this purpose.
- The degrees of freedom used for radiative intensities are now internal degrees of freedom of the heat transfer interfaces. This affects all interfaces containing radiation in participating media features except the standalone Radiation in Participating Media interface. The names of the internal degrees of freedom start by the physics interface identifier followed by In, where n is the index of the radiative intensity direction (for example, ht.I1, ht.I2, ...). This change requires updates of the variable names if they have been explicitly used in the model, study, or results nodes.
- In order to avoid conflict in the name of feature internal variables, the feature prefix has been added to the scope of some variables. The list of all feature internal variables and their definition is available in the **Equation View** node of each feature.
- The sign of the constraint for temperature has been changed to match the convention used in other physics. This has no effect on the models unless weak constraints are used. In that case, the Lagrange multipliers sign is changed.
- The default arrow plot in Heat Transfer interfaces now shows the total heat flux instead of the temperature gradient which has opposite sign for purely conductive heat transfer models.
- Heat Continuity and Periodicity features from Non-Isothermal flow interface have been updated to support wall function when using a k - ε or k - ω turbulence model.

K- ω TURBULENCE MODEL

The Turbulent Flow, k- ω physics interface is part of the CFD Module in version 4.3.

TURBULENT FLOW WITH WALL FUNCTIONS AND HEAT TRANSFER IN CONJUGATE HEAT TRANSFER

Momentum transport Wall Functions are translated into the revised Wall Functions in version 4.3. The Heat equation Wall Functions are also translated, but the translated formulation uses the default turbulence modeling constants, regardless of the values that were defined in **Physics>Scalar Variables** in 3.5a.

MEMS Module

New Functionality in Version 4.3

- New functionality for modeling piezoresistivity:
 - Three new Piezoresistivity interfaces for modeling piezoresistivity in shells, in domains, or with boundary currents.
 - Piezoresistive material library for single crystal and polycrystalline p- and n-doped silicon.
- S-parameter calculations: You can now compute S-parameters for piezoelectric, electric, and electromechanical devices.
- Improvements to the Electromechanics physics interface: The Electromechanics interface has been improved and upgraded in a number of ways. The Electromechanical Solid node has been merged with the Linear Elastic Material node to produce a new feature the Linear Elastic Dielectric. Backward compatibility code handles the conversion of models from 4.2a to 4.3 using this feature; however, the electrostrictive material constants will not be transferred because these have been redefined in version 4.3 for improved compatibility with standard reference texts (for example, Stratton, *Electromagnetic Theory*, Adams Press, March 15, 2007; Landau and Lifshitz, *The Electrodynamics of Continuous Media*, Elsevier, 1968). A new Linear Elastic Material node has been added to enable the solution of problems in which solving for the potential within the solid is not required. The Electromechanical Interface node now appears on the top level as a default boundary condition and will automatically obtain the correct selections from both of these domain level features. Finally the Frequency Domain and Eigenfrequency study types are no longer supported. Because the problems solved by the electromechanics interface are inherently nonlinear, it is important to use the linearized approach employed in the Prestressed Analysis, Frequency Domain and the Prestressed Analysis, Eigenfrequency study types. Existing models with these study types will still open and solve as before, but in the case of the Frequency Domain study type, the results are likely to be misleading. As a result of general improvements to the interface, the results obtained from the prestressed analyses may change in version 4.3.
- Coordinate visualizations make it possible to visualize the different material orientations in stacks of piezoelectric materials.

- New periodic boundary conditions: Dynamic cyclic symmetry and Floquet periodicity conditions for solid mechanics and piezoelectric models.

New Models

- Capacitive pressure sensor
- Piezoresistive pressure sensor
- Electrostatically actuated cantilever
- Shear modes in a quartz oscillator

Backward Compatibility vs. 4.2a

- The nonlinear structural mechanics functionality in version 4.2a is now available in the Nonlinear Structural Materials Module, which is an add-on to the Structural Mechanics Module. See [Nonlinear Structural Materials Module](#).

Backward Compatibility vs. Version 3.5a

FILM DAMPING

- Perforated plates are not yet available. These have to be modeled in version 4.3 using the full geometric description or using the equation view.

Microfluidics Module

New Functionality in Version 4.3

MICROFLUIDICS

- Transitional flow is now available in 3D.
- New lattices available for transitional flow.
- The Transitional Flow interface has optimized solver suggestions for controlling trade-off between memory usage and solution time.

New Model

- Controlled diffusion micromixer—simulates an H-shaped arrangement of microchannels designed for controlled diffusive mixing.

Nonlinear Structural Materials Module

The Nonlinear Structural Materials Module, which is new in version 4.3, is an optional package that extends the Structural Mechanics Module to studies that include structural mechanics with nonlinear material behavior. It is designed for researchers, engineers, developers, teachers, and students that want to simulate nonlinear structural materials, including a full range of possible multiphysics couplings.

The module provides an extensive set of nonlinear structural material models, including the following materials:

- Predefined and user-defined hyperelastic materials: neo-Hookean, Mooney-Rivlin, Saint-Venant Kirchhoff, and others.
- Small-strain and large-strain plasticity models using different hardening models.
- Orthotropic plasticity (using Hill's coefficients).
- Predefined and user-defined creep and viscoplastic material models: Norton, Garofalo, Anand, potential, volumetric, deviatoric, user-defined, and others.

The Nonlinear Structural Materials Module includes an extensive model library with verification models and models that show applications for modeling creep, hyperelasticity, plasticity, and viscoplasticity.

Optimization Module

New Functionality in Version 4.3

- The Optimization interface and solver now support time-dependent optimization and sensitivity analysis. To demonstrate this functionality, a new tutorial model, Time-Dependent Optimization, has been added to the Optimization Module Model Library.
- The optimization solvers now allow a choice of forward or adjoint sensitivity in the **Gradient method** list.

New Model

- Time-Dependent Optimization: This tutorial model demonstrates how to compute the periodic steady-state solution of a nonlinear problem using the time-dependent optimization solver.

Particle Tracing Module

New Functionality in Version 4.3

- Particle-particle interactions are now supported (see the Rotating Galaxy model for an example).
- Thermophoretic force, magnetophoretic force, and Brownian force are now available in the Particle Tracing for Fluid Flow interface.
- New release mechanisms in velocity space: spherical and hemispherical.
- Secondary particle emission.
- Sticking expressions and coefficients.
- 2D histogram plots.
- Schiller-Naumann drag force option.
- Distribution functions for auxiliary dependent variables.
- Particles are now released on the deformed mesh for rotating machinery cases.
- Grouping of variables for easier results processing.
- New global variables for the sum of masses and the total energy.
- Filter mechanisms for improved visualization of specific types of particles.
- Comet tail plots.

New Models

- Thermophoresis
- Brownian motion
- Rotating galaxy
- Multipactor

Backward Compatibility vs. 4.2a

- Selections are now respected in the particle tracing interfaces. You may get different results when you re-run models created using version 4.2a.

Pipe Flow Module

The Pipe Flow Module, which is new in version 4.3, is intended for the modeling and simulation of flow of incompressible fluids in pipe and channel systems, as well as compressible hydraulic transients and acoustics waves. Typical simulations yield the velocity, pressure variation, and temperature in systems of pipes and channels.

Hydraulic transients resulting from a valve that is closed rapidly in a pipe network is referred to water hammer, which can be modeled too. The module can be used to design and optimize complex cooling systems in turbines, ventilation systems in buildings, pipe systems in chemical processes, and pipelines in the oil and gas industry.

In common for pipes and channels that can be modeled is that the pipe length is large enough so that the flow inside can be considered fully developed. Piping components such as bends, valves, T-junctions, contractions/expansions, and pumps are also available.

The Pipe Flow Module includes these physics:

- The Pipe Flow physics computes the pressure and velocity field in isothermal pipe systems.
- The Heat Transfer in Pipes physics computes the energy balance in pipe systems but receives the flow field as a value or as a known solved field. Wall heat transfer to the surroundings is included.
- The Transport of Diluted Species in Pipes physics solves a mass balance equation for pipes in order to compute the concentration distribution of a solute in a dilute solution, considering diffusion, dispersion, convection, and chemical reactions.
- The Non-Isothermal Pipe Flow physics is a multiphysics interface that solves the flow, pressure, and temperature simultaneously and fully coupled.
- The Reacting Pipe Flow physics is a multiphysics interface that solves the flow, pressure, temperature, and reacting species transport simultaneously and fully coupled.
- The Water Hammer physics solves rapid hydraulic transients in pipe systems, taking the elastic properties of both the fluid and pipe wall into account.
- The Pipe Acoustics, Transient physics models sound waves in flexible pipe systems.

The physics in the module define the conservation of momentum, energy, and mass of an fluid inside a pipe or channel system.

Plasma Module

New Functionality in Version 4.3

PLASMA MODELING

- New solver for Inductively Coupled Plasmas (ICP), which splits complex degrees of freedom into their real parts. This is much more stable when solving. It is recommended that the coils are driven by a fixed power from now on. To use the new solver on old models you need to regenerate the default solver.
- Mesh control edges allow cathode fall regions to be resolved with unstructured meshes; see the DC Glow Discharge (positive_column_2d) example model.
- It is now possible to compute the ion angular distribution function with the Particle Tracing Module as a 2D histogram.
- Variables have now been grouped for easier results processing.
- There is now a Reduced Electric Fields study type for the Boltzmann Equations, Two-Term Approximation interface.

New Models

New models that use the new solver for Inductively Coupled Plasmas:

- Plasma enhanced chemical vapor deposition
- Electronegative inductively coupled plasma

Backward Compatibility vs. 4.2a

Unit support for the characteristic potential length has been corrected. When you open an old model, COMSOL will automatically convert the potential characteristic length to the correct value. However, in previous versions you may have typed `3.3` for the potential characteristic length of Argon (in angstroms); you must now type `3.3[angstrom]` in order to obtain identical results.

RF Module

New Functionality in Version 4.3

- The new Transmission Line interface solves the time-harmonic transmission line equation for the electric potential. The interface is used when solving for electromagnetic wave propagation along one-dimensional transmission lines and is available in 1D, 2D, and 3D.
- Mixture and Porous media models are available for both the frequency-domain and the transient RF interfaces, when specifying the relative permittivity, the relative permeability, and the conductivity. The conductivity can also be specified using Archie's law.
- New material models for specification of the relative permittivity, including dispersion, have been added to the frequency-domain Electromagnetic Waves interface:
 - The Drude-Lorentz model is suitable for approximating the permittivity of metals.
 - The Debye dispersion model describes the dielectric response of an ensemble of ideal, non-interacting dipoles.
 - Finally, the Sellmeier formula is widely used in optics for specification of the refractive index of optical materials as a function of the wavelength.
- The different models for specifying the relative permittivity, including the new models described above, are now available also when specifying the impedance and the transition boundary condition.
- PMLs are now enabled from the Definitions branch and applies PML scaling to all physics with PML support. Note: Does not apply to the Microwave Heating interface.
- New far-field plots that are generated independently of the models' mesh elements have been added. The directivity parameter is also calculated as part of the new far-field plot generator.
- Default far-field plots are generated when a far-field domain feature is available in the model.
- Conversion between S, Y, and Z parameter output is offered in postprocessing.

Backward Compatibility vs. Version 4.2a

GAUGE FIXING NODE RENAMED

The Gauge Fixing node in version 4.2a has been renamed Divergence Constraint.

Backward Compatibility vs. Version 4.0a

OVERRIDDEN PORT VARIABLES REMOVED FROM SOLUTION COMPONENTS

Overridden port features no longer generate solution components since version 4.1. Models built in version 4.0a and 4.0 with overridden port features will display an error message when opened in version 4.1 or later. In order to successfully complete the import of such a model into version 4.3 you need to click OK to close the error message, then clear the existing solutions, and finally re-solve the model.

Java-files or M-files that create and solve COMSOL models containing overridden port features similarly need to be modified so that the code that generates the solver sequence does not refer to the removed solution components from the overridden ports.

Structural Mechanics Module

New Functionality in Version 4.3

- Support for *load cases*:
 - Loads and constraints can act conditionally in models using load groups and constraint groups, which you can include in load cases using various combinations of the defined groups.
 - Use load groups for loads and other Neumann conditions. The loads can be scaled in the load cases using a scalar weight.
 - Use constraint groups for constraints and other Dirichlet conditions.
 - Available for the Solid Mechanics, Shell, Beam, Truss, and Plate interfaces.
- A new Membrane interface for modeling, for example, prestressed membranes, cladding on solids, and inflatable flexible bags (balloons). The Membrane interface can be seen as a representation of plane stress in a 3D geometry or as a shell without bending stiffness.
- New and redesigned periodic boundary conditions for solid mechanics and piezoelectric applications, including the following new types of periodic boundary conditions:
 - General Floquet periodicity conditions.
 - Cyclic symmetry condition for sector periodicity and dynamic cyclic symmetry.
 - Also, an option to prescribe the periodic conditions on the entire displacement vector has been added (it is the new default option).
- Low-reflecting boundary condition, mainly intended for letting waves pass out from the model without reflection in time-dependent analysis. As a default, it takes material data from the domain in an attempt to create a perfect impedance match for both pressure waves and shear waves.
- Rigid connector improvements:
 - Rigid connectors between shell edges are now available.
 - Rigid connectors from shell and solid interfaces can be attached to each other.
- Linear buckling is now available for the Truss interface.
- The Prestressed Analysis, Frequency Domain study type is now supported also by the Truss interface.

- The *rate of strain* (Ld) and *spin* (Lw) tensors are now available as predefined variables (on the spatial frame) in the solid interfaces.
- Strain energy density is now computed as a result also in the Truss, Beam, Shell, and Plate interfaces.
- Shell curvature data is now available as variables, which you can use to modify material, thickness, or initial stress for deep drawn sheets.
- Gauss point results are now available for almost all stress results in Structural Mechanics, and all Gauss point results have been moved to separate groups in the plot list.
- The use of geometric nonlinearity is now controlled by a setting in the study step also for the Shell and Truss interfaces.

New Models

- Sliding Wedge: This model is a friction benchmark.
- Snap Hook Penalty: Model that shows an alternative simplified contact formulation.
- Truss Tower Buckling: A linear buckling example using the Truss interface.
- Vibrating Membrane: A verification example using the Membrane interface.
- Nonlinear Magnetostriction
- Turbine Stator

Backward Compatibility vs. 4.2a

- The nonlinear structural mechanics functionality in version 4.2a is now available in the Nonlinear Structural Materials Module, which is an add-on to the Structural Mechanics Module. See [Nonlinear Structural Materials Module](#). This applies to hyperelastic and elastoplastic materials, which are now available in the Nonlinear Structural Materials Module.
- There is now a single redesigned Prescribed Rotation node for shells and plates that works with both linear and geometrically nonlinear material.

Backward Compatibility vs. Version 3.5a

PLANE STRESS

Version 4 represents the out-of-plane strain as a separate degree of freedom. *Plane stress* models made with version 3.5a need to be re-solved to correctly display strains and stresses. Displacements are correctly displayed.

Subsurface Flow Module

New Functionality in Version 4.3

- The Transport of Diluted Species interface (chds) now allows for multiple species transport.

New Models

- Pesticide transport
- Sorbing solute

Material Library

New Functionality in Version 4.3

- This version adds a number of material properties to the following materials:
 - Al_2O_3
 - Ti-6Al-4V
 - Ti-46.5Al-2Nb-2Cr
 - Si E
 - Ar solid
 - Kr solid
 - Ti_3SiC_2
 - Si liquid
 - Li_2SO_4
 - AgI
 - Ir
 - Pd
 - Pt
 - V
 - ZrC

LiveLink for MATLAB

New Functionality in COMSOL 4.3

The LiveLink for MATLAB® includes the following new wrapper functions:

- `mphimage2geom`: convert image data to geometries.
- `mphevalpoint`: evaluate expressions at geometric vertices.
- `mphmean`, `mphmin`, `mphmax`: evaluate mean, minimum, and maximum of expressions in a data series, respectively.
- `mphevalglobalmatrix`: evaluate a global matrix, for instance as for an S-parameter matrix.
- `mphsearch`: GUI to search for expression, variables, tags, and so on in a model.

New functionality have also been added to the existing wrapper function, these are listed below:

- More information on more nodes are now available in `mphnavigator`. You can also get solution information such as how the solution object and the solution data sets are connected to each other.
- Selection tags in `mpheval` and `mphint`.
- Extrapolation to evaluate data outside the geometry with `mphinterp`.
- Support for contour labels in 2D and 3D with `mphplot`.

It is now possible to update the MATLAB installation folder directly in the **Preferences** dialog box from the COMSOL Desktop.

A new method to disable automatic variable update is now available. This can help for performance improvements in large model implementations.

Known Issues

The option to plot on a server by starting a COMSOL server with the option `-graphics` is not supported on Mac.

MATLAB SUPPORT WITH COMSOL MULTIPHYSICS VERSION 3.5A

COMSOL makes an effort to be backward compatible: you can load Model MPH-files created in COMSOL Multiphysics 3.5a and later versions in COMSOL Multiphysics 4.3.

When going from version 3.5a to version 4, a major revision was made to the MATLAB interface. This revision was made to reflect changes made to the new user interface and to support parameterized geometry operations. As a result, a new MATLAB interface syntax is used in today's version 4 of COMSOL Multiphysics and its add-on product LiveLink for MATLAB.

In order to assist in the conversion process, a special compatibility mode was created to facilitate the new syntax. This compatibility mode, together with the LiveLink for MATLAB function `mphv4`, is no longer supported in COMSOL Multiphysics 4.3.

If you wish to convert a model defined with an M-file created with version 3.5a to the version 4.3 format, we recommend the following procedure:

- 1** Run the M-file using COMSOL Multiphysics 3.5a and save the model, using `f1save`, as an MPH-file.
- 2** Load the model into COMSOL Multiphysics 4.3 and verify that the model settings have been translated correctly. In addition, verify that the model can be meshed and solved.
- 3** Select **File>Reset History**.
- 4** Select **File>Save as Model M-file**.

The saved M-file can now be tested if you start COMSOL Multiphysics 4.3 with MATLAB.

If you have any problems with this conversion process, please contact COMSOL's technical support team at support@comsol.com or your local COMSOL representative.

GEOMETRY

The functionality corresponding to the geometry functions `geom2` and `geom3` from version 3.5a is not available in version 4.3. This functionality will be available in a later release.

COMSOL API

See the section [The COMSOL Java API](#) regarding changes in the COMSOL API.

The COMSOL Java API

COMSOL 4.3 Java API Changes

- In the Acoustics Module the far-field variables `pfar` and `Lp_far` get new names with full scope. They are now referred to as `acpr.ffc1.pfar` and `acpr.ffc1.Lp_far`, respectively. Any user Model Java-files that use the old variable names in expressions (such as expressions to plot or evaluate expressions including these variables) will require a manual update.
- In the Electrostatics interface, the parameter `DisplacementFieldModel` in the feature `ChargeConservation` has been renamed to `ConstitutiveRelationD` for consistency with the other physics interfaces.
- The newly added Rotating Machinery, Magnetics interfaces in the AC/DC Module will be defined in the Material frame. Vector variables, such as the electric current density `J`, are now defined with components in the Material frame (`JX`, `JY`, and `JZ`). The Electric Field Transformation default feature computes the spatial electric and magnetic fields as postprocessing variables using a Lorentz transformation of the electric field, so the variables `Ex`, `Ey`, `Ez`, `Bx`, `By`, and `Bz` (with components in the Spatial frame) are still defined.
- In the Electric Currents (`ConductiveMedia`) interface, the features `CurrentConservationInMixtures` and `ArchiesLaw` have been replaced by the general mixture model for the AC/DC Module. The same functionality can be obtained by creating a `CurrentConservation` feature, set the required parameter (`epsilon_r_mat`, `mu_r_mat` or `sigma_mat`) to `PorousMedia` (or `ArchiesLaw` for the electrical conductivity) and add a `PorousMedia` (or `ArchiesLaw`) subfeature. When running class files compiled in COMSOL 4.2a or earlier, this conversion is done automatically.
- From version 4.3 the 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();
```

- 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 being a domain feature with a domain selection and, as a sub feature, 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 per interface and is no longer available in the model scope, for example, `mod1.psi`, etc. Instead the gauge fixing field is only available in the interface scope as, for example, `mod1.mf.psi`, etc.
- 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` have changed names to `Si1` and `eil`.
- In the Darcy's Law interface and Richards' Equation interface in the Subsurface Flow Module, the compressibility of fluid is now a material parameter and they do 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);
```
- In the Poroelasticity interface in the Subsurface Flow Module, the compressibility of fluid and the Biot-Willis coefficient are now material parameter and they do 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 as default. If you have a model created in an earlier version of COMSOL, it will fail to create a feature with the same tag name.

COMSOL 4.1 Java API Changes

The following changes were made to the COMSOL Java 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 was in most cases `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 option `linper`. This option works like `linearized` except that source terms which are to contribute in the linearized problem must be enclosed by the `linper` operator.
- In this CFD Module, Subsurface Flow mode, in the Darcy's Law interface, Brinkman interface, Richards' Equation interface, and the Poroelasticity interface, the permeability and porosity are now material parameters and they do no longer have a default value. If the default value was used in a Java-file 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.
- `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 lowercase s now begin with 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 material to spatial frame. This implies that index notation of some variables have changed from uppercase XYZ to lowercase xyz.
- The acoustics interfaces (`PressureAcoustics`, `TransientPressureAcoustics`, `AcousticStructure`, and `TransientAcousticStructure`) have been moved from material to spatial frame. This implies that index notation of vector variables have change from uppercase XYZ to lowercase xyz.

