



# COMSOL Multiphysics®

*Release Notes*



# COMSOL Multiphysics Release Notes

© 1998–2013 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](http://www.comsol.com/sla)) and may be used or copied only under the terms of the license agreement.

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 COMSOLAB 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/tn](http://www.comsol.com/tn).

Version:

May 2013

COMSOL 4.3b

## Contact Information

Visit the Contact Us page at [www.comsol.com/contact](http://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](http://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](http://www.comsol.com/support/case).

Other useful links include:

- Support Center: [www.comsol.com/support](http://www.comsol.com/support)
- Download COMSOL: [www.comsol.com/support/download](http://www.comsol.com/support/download)
- Product Updates: [www.comsol.com/support/updates](http://www.comsol.com/support/updates)
- COMSOL Community: [www.comsol.com/community](http://www.comsol.com/community)
- Events: [www.comsol.com/events](http://www.comsol.com/events)
- COMSOL Video Center: [www.comsol.com/video](http://www.comsol.com/video)
- Support Knowledge Base: [www.comsol.com/support/knowledgebase](http://www.comsol.com/support/knowledgebase)

Part No. CM010001

# Release Notes

COMSOL<sup>®</sup> version 4.3b contains 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 4.3b*

---

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

- The Multibody Dynamics Module extends the Structural Mechanics Module with functionality for multibody dynamics modeling an assembly of rigid and flexible bodies. See [Multibody Dynamics Module](#) for more information.
- The Semiconductor Module, a new module for modeling semiconductor devices. See [Semiconductor Module](#) for more information.
- The Electrochemistry Module provides tools for modeling of electrochemical cells. See [Electrochemistry Module](#) for more information.
- The Wave Optics Module is designed for modeling of electromagnetic wave propagation in linear and nonlinear optical media. See [Wave Optics Module](#) for more information.
- The Molecular Flow Module is a new module for modeling of low-pressure rarefied gas flows in vacuum systems. See [Molecular Flow Module](#) for more information.

## *New Functionality in Version 4.3b*

---

### **DOCUMENTATION**

- The *COMSOL Multiphysics Reference Manual* replaces the earlier *COMSOL Multiphysics User's Guide* and *COMSOL Multiphysics Reference Guide* and provide a single comprehensive resource for documentation of the functionality in COMSOL Multiphysics
- The *COMSOL Installation Guide* replaces the earlier *COMSOL Installation and Operations Guide* and *COMSOL Quick Installation Guide* with a focus on information about how to install the COMSOL software on the supported platforms. For information about the COMSOL commands and running COMSOL in general, see the *COMSOL Multiphysics Reference Manual*.

### **GENERAL COMSOL DESKTOP FUNCTIONALITY**

- File locking: Only one instance of COMSOL Multiphysics Model File (MPH-file) can be open at the same time. If the file is locked, you can open it in a read-only mode. When an MPH-file is locked, COMSOL creates a separate lock file with the

same filename as the MPH-file plus the extension `.lock`, stored in the same directory as the locked MPH-file.

- Curvilinear coordinate systems for curve geometric shapes make it easy to define anisotropic material properties in bent geometric shapes. This functionality is available as a Curvilinear Coordinates user interface with four different methods:
  - Diffusion method
  - Elasticity method
  - Flow method
  - User-defined coordinates
- In the Model Library view, you can use identifiers for nodes in a model to help searching for models that make use of specific functionality. To limit the search to tag names and identifiers, use the prefix `@`, for example, `@genext` or `@ehs`. To search for a specific physics user interface, use the scoping syntax `@physics:<identifier>`. For example, type `@physics:jh` to find all models that use the Joule Heating physics.
- New units:
  - For force, *kilopond*: `kp`, `kpF`
  - For pressure, *inches water*: `inAq`, `inH2O`
  - For dynamic viscosity, *poise*: `P` (also `cP` for centipoise, for example)
- 

## GEOMETRY AND MESH

### *New Geometry Tools and Functionality*

- Boolean operations (union, difference, intersection, and general set operations) are now available in the geometry sequence and can be used to operate on directly on the geometry objects.
- New *Partition* operation in the geometry sequence, where you can partition a geometry using other geometry objects (tool objects) or, in 3D, a work plane.
- To create a 2D cross section from a plane intersection, a *Cross Section* node is available in work planes and in 2D and 2D axisymmetric geometries.
- When drawing in a work plane, the intersections of all 3D objects with the work plane can now be visualized as blue curves (in addition to the projection of 3D edges onto the work plane or 3D edges that lie in the work plane).

- The Finalize operation now reports—in the Messages window—statistics for the geometry such as the number of domains, boundaries, edges, and vertices in the finalized geometry. You can use this information to detect errors in the geometry—for example, an unwanted thin domain that has been created due to a mismatch between geometry objects.
- In 1D and 2D axisymmetric geometries, any geometry in the  $r < 0$  half-plane is removed automatically.
- Collapse Faces and Merge Edges are new virtual operations that can be used to simplify the geometry for analysis.
- It is possible to use named selections in other operations in the geometry sequence.
- If your license included the CAD Import Module, you can export the geometry to an ACIS file (as .sat or .sab files).

#### *New Meshing Tools and Functionality*

- Much improved mesh sweep operation for creating swept meshes using automatic source and destination detection.
- Export of STL meshes from isosurfaces and other triangle meshes of 3D plots
- For mesh import, a *Cylinder* partitioning node has been added to complement the Ball and Box options.
- Support for importing 2D Nastran meshes.
- New virtual geometry operations: *Collapse Faces* and *Merge Edges*: Using the Collapse Faces operation or the analogous Merge Edges operation, you can remove small or narrow faces from a 3D geometry that cannot be, or that are not suitable for being, incorporated in a composite face.
- Mesh import without introducing any additional boundary partitioning: A new option, *Minimal*, is available in the *Boundary Partitioning* list in the Import node's settings. If you choose this option and import a mesh from a file, the import operation does not introduce any additional boundary partitioning than what is specified in the file and what is required by COMSOL.
- *Continue meshing* is now also used for the Swept, Boundary Layer, and Mapped meshers. Continue meshing means that the mesher does not stop with an error if it encounters a problem for a geometric entity. Instead, it continues meshing the remaining entities and returns the resulting mesh together with error and warning nodes containing information on the encountered problems.

## PHYSICS USER INTERFACES

- *Thin Impermeable Barrier on Interior Boundaries*: This boundary condition is available as a “slit condition” on interior boundaries for the mass transport user interfaces. It applies no flux on both sides of an interior boundary and can be added to solid interior walls.
- The *Crosswind diffusion* formulation has been updated for all interfaces supporting Crosswind diffusion. Models solved with the new formulation can give different results compared to models solved in version previous to version 4.3b. Any model created in previous versions will retain the old crosswind formulation if the crosswind diffusion is active. The new formulation becomes available if the physics is replaced.
- The computation of accurate boundary fluxes has been improved. There is now an option to apply a smoothing to the boundary fluxes.

## STUDIES AND SOLVERS

- New *iterative SCGS* (symmetrically coupled Gauss-Seidel) *solver*, which is useful as an efficient preconditioner for solving the Navier-Stokes equations and similar fluid-flow problems using linear elements.
- New *auxiliary space Maxwell solver* (AMS), which uses the auxiliary space Maxwell solver preconditioner from the Lawrence Livermore National Laboratory linear solver/preconditioner library *hypr*. This solver provides improved performance for magnetic field simulations.
- New *domain decomposition solver* using a Schwarz method. Domain decomposition divides the modeling domain into subdomains where each PDE in the subdomains is easier to solve. The total solution is then obtained by iterating between the computed solutions for each domain using as boundary conditions the currently known solutions from the other domains. The domain decomposition solver is efficient for distributed memory systems (cluster computing) and as a more memory-efficient alternative to a direct solver for large problems.
- Improved stop condition:
  - In the *Stop Condition* node you can now add multiple stop expressions, and you can choose between a logical expression (where the solver stop if the expression is true) or an expression where the solver stop if it becomes negative (as in earlier versions of COMSOL Multiphysics).
  - There is also an *Implicit event* table (with the time-dependent solver) where you can specify at which events the solver should stop. All implicit events defined in

the model automatically appear in the table. The stepping stops when any event marked as active is triggered.

- Under *Output at Stop*, Use the *Add solution* option to make the solver additionally store the corresponding solutions before and after the stop condition was fulfilled.
- For sensitivity analysis, you can add a *Sensitivity* study step directly from a main Study node's context menu (by first selecting the *Advanced Study Options* on the Model Builder's Show menu). In the settings for the Sensitivity study node you can specify the gradient method (forward or adjoint) and also add the objective functions and control variables. The *Sensitivity Analysis of a Communication Mast Detail* model now uses the Sensitivity study node for the sensitivity analysis.
- The solution memory for the solvers now appears in the *Messages* window.
- In the tree view under *Physics and Variables Selection* in the study step node's Settings window, there is now a *Go to Source* button that you can click to move to the corresponding node in the model tree. The geometry in the *Graphics* window also displays the selection for the nodes that you click in the study step node's physics tree.
- An option to *Keeps warnings in stored log* has been added to the *Log* section in the settings window for solver nodes such as *Store Solution* and *State Space*. Select this check box if you want to store the warnings from these nodes in the model file.
- You can now specify the path for saving solutions at intermediate parameter values to file during parametric sweeps.

## RESULTS

- Color expressions are available for the following 1D plot types: Line Graph, Point Graph, and Global.
- TIFF and GIF added as image export formats: TIFF, GIF
- It is now possible to control the quality of exported JPEG images.
- Height attributes can now be added 2D contour plots.
- Reversed arc length can be the  $x$ -axis parameter in line graphs.
- Function data sets can now point to a specific function or to all functions in a list.
- The sector number is available as a variable in sector data sets.
- Element refinement settings are now available in the *Advanced* section in the maximum and minimum values nodes under *Derived Values* and *Data Sets*. The



Maximum and minimum probes now propagate the refinement setting to the corresponding data sets.

- Matrix Histogram is a new 2D plot type for visualizing fatigue results. Requires a license for the Fatigue Module.

### *Backward Compatibility vs. Version 4.3a*

---

#### **TERMINATION CRITERION FOR STATIONARY SOLVERS**

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

#### **BACKWARD EULER INITIALIZATION TIME STEP**

A new setting in the *Advanced* section of the settings window for the *Time-Dependent Solver*, the *Fraction of initial step for Backward Euler* field, provides the possibility to enter 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, which differs from earlier versions, which used a value of 1. When opening models created in version 4.3a or earlier versions, the value for this fraction is set to 1 to maintain compatibility with those earlier versions.

### *Backward Compatibility vs. Version 4.3*

---

#### **NEW TERMINOLOGY FOR CONSTRAINT TYPES**

The following constraint types in version 4.3 have new names in version 4.3b:

- *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 constraints formulation 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 previously used Lagrange multiplier `un_1m` has been removed.

Models saved in version 4.3 can be opened and postprocessing 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, generate a new automatic solver sequence. Models saved in versions previous to version 4.3 must either be re-solved in version 4.3b to be possible to postprocess, or opened and re-saved in version 4.3 before opened in version 4.3b.

#### **AUTOMATICALLY CREATED IDENTITY/CONTACT PAIRS**

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

#### *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.

### *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 4.2a. 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 4.2a interface, the *Linear Displacement* and *Similarity Transform* boundary conditions are not yet available as preset conditions. Those boundary conditions are planned for version 4.3.

In version 4.2a, you can create the corresponding conditions by manually entering variables.

#### **BACKWARD COMPATIBILITY FOR PRE-3.5A MODELS**

COMSOL 4.2a 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 35.a Command” for Windows, “COMSOL Convert pre 35a Command” for Linux, or the section “COMSOL 35.a Convert pre35a Command” for the Mac in the *COMSOL Multiphysics Reference Manual* for more information.

# AC/DC Module

*New Functionality in Version 4.3b*

---

## **MAGNETIC FIELD FORMULATION**

The *Magnetic Field Formulation* user interface is a new physics user interface based on a different formulation of Maxwell's equations and solves for the magnetic field. This formulation has applications in materials with nonlinear resistivity such as superconductors. In addition, a new *E-J characteristic* material property group is available.

## **RLC COIL GROUP**

The RLC Coil Group is a coil group that includes, in the current balance, the in-plane current flow driven by the potential difference between turns (for example, capacitive coupling). The coil group uses an automatic analysis of the geometry. This is a new feature in the Magnetic and Electric Fields user interface.

## **FLOATING POTENTIAL GROUP**

The Floating potential group check box is a new setting for the *Floating Potential* node, which considers each group of connected boundaries as a separate group with its own potential. The floating potential group simplifies set-up of models with many floating electrodes.

## **BOUNDARY COILS AND COIL GROUP FUNCTIONALITY**

- *Boundary coils* are thin cross-section coils that can now be modeled as boundaries. Both single-turn and multi-turn coils are available in 2D and 3D. The 2D single-turn coil includes a lateral skin effect.
- Coil Group is now a setting in the Single-Turn Coil node and no longer a separate node. The Coil Group setting is also available for multi-turn coils and the new boundary coils. A new Domain Group subnode makes it possible to group domains in a single turn (parallel and series connections of different domains).

## **ELECTRICAL CONTACT**

*Electrical contact* computes the electric current between two surfaces in contact. The conductance depends on the surface properties and the contact pressure (it represents the fact that when the pressure increases, the contact is better). This contribution can be evaluated following a predefined correlation and can be coupled with structural

mechanics contact. The Electrical Contact node is available on pair boundaries and on interior boundaries.

### **GAUGE FIXING AT TOP LEVEL OF THE PHYSICS USER INTERFACE**

The Gauge Fixing feature can now be added at the top level of the Magnetic Fields, Magnetic and Electric Fields, and Rotating Machinery, Magnetic user interfaces to provide faster set-up and collect all settings in a single node. This also makes it easier to globally enable and disable gauge fixing.

### *New and Updated Models in Version 4.3b*

---

- *Electrodynamic Bearing*: This model illustrates the working principle of a passive electrodynamic bearing. An electrically conducting rotor rotating in a magnetic field produced by a permanent magnets induces eddy currents on the conducting rotor.
- *Homopolar Generator*: A homopolar generator is composed of an electrically conductive rotating disc placed in a uniform magnetic field that is perpendicular to the plane of rotation. This example models the flow of current through the copper conductor and the rotating disc.
- *Static Field Modeling of a Halbach Rotor*: This model presents the static-field modeling of an outward-flux-focusing magnetic rotor using permanent magnets, a magnetic rotor also known as a Halbach rotor.
- *Axial Magnetic Bearing Using Permanent Magnets*: This model illustrates how to calculate design parameters like magnetic forces and stiffness for an axial permanent magnet bearing.
- *Superconducting Wire*: This model has been recreated using the new Magnetic Field Formulation physics user interface.

### *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.

There is no backward compatibility for models created using the 2D formulations based on the magnetic field  $H$  as dependent variable in version 3.5a. Such models will not include any physics interface settings when opened in version 4.3b. Geometrical

data and definitions (constants and expressions) will be imported and the physics can manually be added using the Magnetic Field Formulation user 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 4.3b*

---

- The *Pipe Acoustics, Frequency Domain* user interface (requires the Pipe Flow Module) supports frequency-domain and eigenfrequency studies and is designed for modeling propagation of sound waves in flexible pipe systems. The equations are implemented in 1D along edges and solve for the cross-section averaged acoustic pressure and acoustic particle velocity. The equations are formulated in a general way to include the possibility of a stationary background flow. This physics user interface is available in 3D on edges and points and in 2D on boundaries and points, and it includes a series of engineering relations for the end impedance at the pipe terminations.
- The material input for the *Thermoacoustics* user interface has been updated to include options for entering the compressibility and the coefficient of thermal expansion. This makes it possible to model the propagation of compressible waves (sound waves) in any fluid with user-defined inputs and nonstandard constitutive relations. Several options exist for entering these coefficients.
- Thermal expansion in the Piezoelectric Devices user interfaces.
- All existing point sources have been collected as one *monopole source* with a type selection (Flow, Intensity, Power, and User defined) including an additional input field for the phase of the source.
- The total pressure is now continuous when the background pressure field is used only on certain domains. That is, it need not be used on all domains. It is also possible to ensure that the wave number  $k$  for the background pressure field is constant across different material domains.
- A Symmetry boundary conditions is now available in the Pressure Acoustics user interface (Frequency Domain, Transient, and Boundary Mode).

## *New Models in Version 4.3b*

---

- *Vibrating Particle in Water: Correct Thermoacoustic Material Parameters*: A tutorial model that simulates a small vibrating hemispherical particle in water. The vibrations induce acoustic waves in the fluid. The model shows how to set up the



material parameters in a thermoacoustic model to get the correct acoustic (compressible) waves in the fluid.

- *Organ Pipe Design*: Tutorial model that uses the Pipe Acoustics, Frequency Domain user interface to model an organ pipe. The half wave resonance frequency is studied with respect to dependency on the pipe radius and the pipe wall thickness. The acoustic properties of the system can easily be studied with respect to other parameters like the pipe cross section shape, the pipe material, or even a steady flow of air in the pipe.

### *Backward Compatibility vs. 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 vs. Version 4.3*

---

- The symbols for the acoustics monopole ( $Q_m$ ; was  $Q$ ) and dipole ( $q_d$ ; was  $q$ ) 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.

### *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.

### *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 and Improved Functionality in Version 4.3b*

---

## **NEW PHYSICS USER INTERFACES AND FEATURES**

- The Electroanalysis user interface has the equations, boundary conditions, and rate expression terms for modeling mass transport of diluted species in electrolytes using the diffusion-convection equation, solving for electroactive species concentrations. This physics user interface is applicable for electrolyte solutions containing a large quantity of inert “supporting electrolyte.” Ohmic loss is assumed to be negligible. The physics user interface includes tailor-made functionality for modeling cyclic voltammetry problems.
- Additional Intercalating Material feature in the Lithium Ion Battery and Battery with Binary Electrolyte user interfaces.
- Switching between Primary and Secondary Current Distributions in the Secondary Current Distribution user interface can now be achieved by changing the Current Distribution Type property in the Secondary Current Distribution user interface. Both the Primary Current Distribution and the Secondary Current Distribution entries in the Model Wizard now both make use of the Secondary Current Distribution user interface but with different default values for the Current Distribution Type property. COMSOL API code for user with Java created before version 4.3b using the Primary Current Distribution (pic) user interface may experience backward compatibility issues due to different behavior of the Porous Electrode feature and different default boundary features.
- Limiting Current Density feature in the Electrode Kinetics section of the Porous Electrode Reaction and the Electrode Reaction nodes.
- Edge Electrode feature in the Secondary Current Distribution user interface. Use the Edge Electrode with 3D models to model electric current conduction in the tangential direction on an edge. The feature is suitable for geometries such as long pipes and thin wires where the electric potential variation within the electrode in the normal direction to the electrode surface is negligible. This assumption allows for the thin electrode domain to be replaced by a lumped one-dimensional partial differential equation formulation on the edge. In this way it is possible to reduce the problem size and avoid potential problems with mesh anisotropy in the thin layer can be avoided. Edge Electrode Reactions can be added to the Edge Electrode to model electrode reactions.

## GENERAL IMPROVEMENTS AND CORRECTIONS

- Improved Physics Induced Mesh settings for Porous Electrodes in all physics user interfaces. Improved Automatic Mesh Settings in 1D for the Battery user interfaces on boundaries between electrodes and electrolytes.
- The Electrolyte-Electrode Boundary Interface now supports Harmonic Perturbation (in the Current Distribution and Battery user interfaces).
- Slip Tangential Velocity Condition in the Electrode-Electrolyte Interface Coupling boundary feature (in the Free and Porous Media Flow and Reacting Flow user interfaces).
- The Continuity pair feature in the Tertiary Current Density, Nernst-Planck user interface has been updated.
- Intercalated concentrations are now solved for in Frequency Domain, Linearized study steps (in the Lithium-Ion and Battery with Binary Electrolyte user interfaces).

## NEW AND UPDATED MODEL

- Vanadium Redox Flow Battery model
- Fuel Cell with Serpentine Flow Field model
- Lithium-Ion Battery model with Multiple Intercalating Electrode Materials
- Single Particle model of a Lithium-Ion Battery

## *Backward Compatibility vs. 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  $\varepsilon_p$  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 formulation 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 following the Nernst-Einstein relation. The default setting when making new models has been changed to use the Nernst-Einstein relation. Model Java-files generated prior to 4.3b using Migration will have to be modified manually to account for these changes.

# LiveLink Products for CAD

## *New Functionality in LiveLink™ for SpaceClaim® in Version 4.3b*

---

Geometry dimensions created by the user in SpaceClaim are automatically transferred to COMSOL during synchronization. This means that they appear in the table in the settings window of the LiveLink feature node, such that the user does not need to manually enter parameters to control in the SpaceClaim geometry. Corresponding global parameters are also generated automatically. Thus it is now much faster to set up a geometric parameter sweep or optimization when working with LiveLink™ for SpaceClaim®.

## *New Functionality in LiveLink™ for Inventor® in Version 4.3b*

---

The One Window interface for Inventor provides an embedded COMSOL modeling environment in the user interface of the Inventor CAD program. Users of LiveLink™ for Inventor® can now set up and compute models and analyze results inside the Inventor user interface.

## *New Functionality in LiveLink™ for SolidWorks® in Version 4.3b*

---

Synchronizing the geometry between SolidWorks and COMSOL now also encompasses the synchronization of material selections. Selections that contain synchronized geometry objects (bodies) are created in COMSOL based on the material definitions from the CAD design. The selections are named after the material name in SolidWorks. Use these selections as input for geometry features requiring object selections, or for any model definitions, physics or material settings requiring domain selections. The LiveLink node contains a table with a list of the synchronized selections.

## *New Functionality in LiveLink™ for Solid Edge® in Version 4.3b*

---

To allow the user to select which parameters should be synchronized between Solid Edge and COMSOL a dialog box has been implemented in Solid Edge. Selected parameters are automatically transferred to COMSOL during synchronization. This means that they appear in the table in the settings window of the LiveLink feature node, such that the user does not need to manually enter parameters to control in the Solid Edge geometry. Corresponding global parameters are also generated

automatically. Thus it is now much faster to set up a geometric parameter sweep or optimization when working with LiveLink™ *for* Solid Edge®.

# CFD Module

*New Functionality in Version 4.3b*

---

## **SST TURBULENCE MODEL**

The *SST turbulence model* (shear stress transport model) is a two-equation, low-Reynolds number (that is, it does not use wall functions) RANS model. It is formally a mix between the  $k-\omega$  model and the  $k-\varepsilon$  model, but the  $k-\varepsilon$  part has been reformulated so that the model solves for turbulent kinetic energy,  $k$ , and specific dissipation rate,  $\omega$ . The underlying idea is that  $k-\omega$  is better than  $k-\varepsilon$  close to walls, while  $k-\varepsilon$  behaves better in the free stream. The model achieves this swapping between the models by introducing a blending function which has the effect that the model is equivalent to  $k-\omega$  close to walls and equivalent to  $k-\varepsilon$  far from walls. The SST model has mainly been used to simulate exterior flows, such as wings and wing-like geometries, and turbines, but it is also a common tool in engineering and has been reported successful in many other situations.

## **FROZEN ROTOR STUDY TYPE**

The *Frozen Rotor* is a new study type for Rotating Machinery, Fluid Flow physics. The functionality is equivalent to solving for the stationary Navier-Stokes where centrifugal and Coriolis forces have been added to the rotating domains. The solution is pointwise continuous over boundaries between rotating and nonrotating domains. The term “frozen” refers to the fact that the rotor will be frozen in position compared to surrounding nonrotating parts. If the nonrotating part of the geometry is rotationally invariant, or the model only contains a rotating domain, then frozen rotor gives the exact steady-state solution. Otherwise, the frozen rotor gives an approximate solution that depends on the exact position of the rotor. Examples where the true solution can be obtained are many mixers without baffles and systems that rotates, for example lab-on-a-chip. Examples where the frozen rotor solution can be strongly dependent on the rotor position are turbines and mixers with baffles.

## **SWIRL FLOW IN MIXTURE MODEL AND BUBBLY FLOW**

You can now activate the out-of-plane velocity component in the 2D axisymmetric versions of the Mixture Model and Bubbly Flow user interfaces to include swirl flow.



### **SCREEN ON INTERIOR BOUNDARIES**

The Screen node is available for modeling wire-gauzes, grilles, or perforated plates as thin permeable barriers in the Single-Phase Flow, Non-Isothermal Flow, and Conjugate Heat Transfer user interfaces. Correlations are included for resistance and refraction coefficients.

### **ACCURATE BOUNDARY FLUXES FOR MASS TRANSPORT**

Support for computing and postprocessing accurate boundary fluxes has been added to the following physics user interfaces:

- Transport of Diluted Species
- Transport of Concentrated Species
- Reacting Flow
- Rotating Machinery, Reacting Flow
- Reacting Flow in Porous Media, Diluted Species
- Reacting Flow in Porous Media, Concentrated Species

Accurate boundary fluxes are solved for when you have selected the **Compute boundary fluxes** check box (in the **Discretization** section and enabled by default). The boundary flux solved for corresponds to all contributions to the flux term.

### **STEFAN VELOCITY FOR WALLS IN REACTING FLOW**

Stefan velocities are automatically computed and prescribed on walls. The Stefan velocity is defined when applying Mass Fraction and Flux features.

### **NEW SOLVER DEFAULTS FOR FLOW PROBLEMS**

- The new SCGS smoother, which provides performance improvements, is the default option for fluid-flow models using P1+P1 (linear) elements.
- The absolute tolerance for time-dependent problems has been decreased to  $5 \cdot 10^{-4}$  (from  $10^{-3}$ ) to reduce the numerical dissipation.
- For pseudo time stepping, the initial CFL numbers have been lowered (from 5 to 3 for turbulent flow and from 10 to 5 for laminar flow). The new initial CFL numbers mean that the number of nonlinear iterations typically increases somewhat, but the increased robustness should more than compensate for this.

### *New Models in Version 4.3b*

---

- *Flow Around an Inclined NACA 0012 Airfoil*: A new benchmark model of a NACA 0012 wing profile showing the use of the SST turbulence model.
- *Turbulent Mixing*: This model shows how the mixing capability of a stirred reactor can be visualized. Modeling the turbulent flow using the  $k$ - $\varepsilon$  turbulence model, a traces species is added when the flow field has developed. This model also demonstrates the use of the new Frozen Rotor study step and the Thin Impermeable Barrier condition.
- *Flow Through a Uniform Inclined Screen*: A new benchmark model using the Screen feature.

### *Backward Compatibility vs. 4.3a*

---

#### **MIXTURE MODEL EQUATION FORMULATION**

The Mixture Model equation formulation has been revised to increase the accuracy and to facilitate solution of a number of problem setups. As a consequence of the revision, old models can return slightly different results compared to previous versions. As a part of the revision, a penalty diffusion on the dispersed phase has been deactivated because it—after the other revisions—turned out to make the equation system too nonlinear and thereby hard to converge. However, you can still activate the penalty diffusion in the Advanced section of the Mixture Model settings.

### *Backward Compatibility vs. 4.3*

---

#### **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 Java-files 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 4.3b. 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 4.3b.

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  $-Q_{br} \cdot \mathbf{u} / \varepsilon_p^2$  in the right hand side previously lacked the factor  $1/\varepsilon_p^2$ , where  $\varepsilon_p$  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 constraints formulation 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, $k$ - $\varepsilon$ and Turbulent Flow, $k$ - $\omega$*

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

#### *High Mach Number Flow*

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

These boundary condition 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.

Models saved in version 4.3 can be opened and postprocessing 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, generate a new automatic solver sequence. Models saved in versions previous to version 4.3 must either be re-solved in version 4.3b to be possible to postprocess, or opened and re-saved in version 4.3 before opened in version 4.3b.

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 4.3b compared to version 4.3 and, the result can differ slightly between the versions.

#### *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 user 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

#### **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 4.3b*

---

## **ACCURATE BOUNDARY FLUXES FOR MASS TRANSPORT**

Support for computing and postprocessing accurate boundary fluxes has been added to the following physics user interfaces:

- Transport of Diluted Species
- Transport of Concentrated Species
- Nernst-Planck Equations
- Reacting Flow
- Reacting Flow in Porous Media, Diluted Species
- Reacting Flow in Porous Media, Concentrated Species

Accurate boundary fluxes are solved for when you have selected the **Compute boundary fluxes** check box (in the **Discretization** section and enabled by default). The boundary flux solved for corresponds to all contributions to the flux term.

*Backward Compatibility vs. 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  $\varepsilon_p$  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 formulation 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, and Nernst-Planck Equations) anisotropic mobility is now supported, and the mobility can be set to following the Nernst-Einstein relation. The default setting when making new models has been changed to use the Nernst-Einstein relation. Model Java-files generated prior to 4.3b using Migration will have to be modified manually to account for these changes.

# Corrosion Module

*New and Improved Functionality in Version 4.3b*

---

## **ELECTROANALYSIS USER INTERFACE**

The Electroanalysis user interface has the equations, boundary conditions, and rate expression terms for modeling mass transport of diluted species in electrolytes using the diffusion-convection equation, solving for electroactive species concentrations. This physics user interface is applicable for electrolyte solutions containing a large quantity of inert “supporting electrolyte.” Ohmic loss is assumed to be negligible. The physics user interface includes tailor-made functionality for modeling cyclic voltammetry problems.

## **OTHER NEW FUNCTIONALITY**

- Switching between Primary and Secondary Current Distributions in the Secondary Current Distribution user interface can now be achieved by changing the Current Distribution Type property. Both the Primary Current Distribution and the Secondary Current Distribution entries in the Model Wizard now both make use of the Secondary Current Distribution user interface, but with different default values for the Current Distribution Type property. API code for use with Java created before version 4.3b using the Primary Current Distribution (piec) user interface may experience backward compatibility issues due to different behavior of the Porous Electrode feature, and different default boundary features.
- Solving for the corroding concentration can now be disabled in the Corroding Electrode Surface and in the External Corroding Electrode nodes.
- A new Limiting Current Density feature in the Electrode Kinetics section of the Porous Electrode Reaction and Electrode Reaction nodes.
- A new Infinite Electrolyte boundary condition in the Secondary Current Distribution and the Corrosion, Secondary user interfaces.
- Edge Electrode feature in the Secondary Current Distribution user interface. Use the Edge Electrode with 3D models to model electric current conduction in the tangential direction on an edge. The feature is suitable for geometries such as long pipes and thin wires where the electric potential variation within the electrode in the normal direction to the electrode surface is negligible. This assumption allows for the thin electrode domain to be replaced by a lumped one-dimensional partial differential equation formulation on the edge. In this way it is possible to reduce the

problem size and avoid potential problems with mesh anisotropy in the thin layer can be avoided. Edge Electrode Reactions can be added to the Edge Electrode to model electrode reactions.

#### **IMPROVEMENTS**

- Improved Physics Induced Mesh settings for Porous Electrodes in all user interfaces.
- The Electrolyte-Electrode Boundary Interface feature now supports Harmonic Perturbation (in the Current Distribution user interfaces).
- A Slip Tangential Velocity Condition in the Electrode-Electrolyte Interface Coupling boundary feature (in the Free and Porous Media Flow user interface).
- Improved default solver settings for Automatic Remeshing (in the Corrosion user interfaces).

#### *New and Updated Models*

---

- Isolator Thickness Effect: this model demonstrates the effect of Al isolator thickness on galvanic corrosion using a parametric study.
- Corrosion Protection of a Ship Hull: this example models a secondary current distribution around a ship hull using an impressed current cathodic protection (ICCP) system for protection of the shaft and propeller.
- Localized Corrosion: This example models galvanic corrosion between two different phases in a magnesium alloy for a representative cross-sectional microstructure configuration.

#### *Backward Compatibility vs. 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  $\epsilon_p$  is the porosity.

#### **WEAK CONSTRAINTS UPDATE FOR FLUID FLOW INTERFACES**

The weak constraint formulation 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, Corrosion, and Tertiary Nernst-Planck) anisotropic mobility is now supported, and the mobility can be set to following the Nernst-Einstein relation. The default setting when making new models has been changed to use the Nernst-Einstein relation. Model Java-files generated prior to 4.3b using Migration will have to be modified manually to account for these changes.

# Electrochemistry Module

The Electrochemistry Module extends the COMSOL Multiphysics environment with customized physics user interfaces for modeling of electrochemical cells, with tools for building detailed models of the configuration of electrodes and electrolyte. The physics user interfaces include chemical species transport, charge balances, heat transfer, and fluid flow. You can use this module to model electrochemical and electroanalytical procedures in electrochemical cells at different operating conditions for different electrode configurations, structures and dimensions; and different electrode and electrolyte materials and chemistry.

# Electrodeposition Module

*New and Improved User Interfaces and Features in 4.3b*

---

## **ELECTROANALYSIS USER INTERFACE**

The Electroanalysis user interface has the equations, boundary conditions, and rate expression terms for modeling mass transport of diluted species in electrolytes using the diffusion-convection equation, solving for electroactive species concentrations. This physics user interface is applicable for electrolyte solutions containing a large quantity of inert “supporting electrolyte.” Ohmic loss is assumed to be negligible. The physics user interface includes tailor-made functionality for modeling cyclic voltammetry problems.

## **OTHER NEW FUNCTIONALITY**

- Switching between Primary and Secondary Current Distributions in the Secondary Current Distribution user interface can now be achieved by changing the Current Distribution Type property. Both the Primary Current Distribution and the Secondary Current Distribution entries in the Model Wizard now both make use of the Secondary Current Distribution user interface, but with different default values for the Current Distribution Type property. API code for use with Java created before version 4.3b using the Primary Current Distribution (piec) user interface may experience backward compatibility issues due to different behavior of the Porous Electrode feature, and different default boundary features.
- Solving for the deposited concentration can now be disabled in the Depositing Electrode Surface and in the External Depositing Electrode nodes.
- A new Limiting Current Density feature in the Electrode Kinetics section of the Porous Electrode Reaction and Electrode Reaction nodes.
- Edge Electrode feature in the Secondary Current Distribution user interface. Use the Edge Electrode with 3D models to model electric current conduction in the tangential direction on an edge. The feature is suitable for geometries such as long pipes and thin wires where the electric potential variation within the electrode in the normal direction to the electrode surface is negligible. This assumption allows for the thin electrode domain to be replaced by a lumped one-dimensional partial differential equation formulation on the edge. In this way it is possible to reduce the problem size and avoid potential problems with mesh anisotropy in the thin layer

can be avoided. Edge Electrode Reactions can be added to the Edge Electrode to model electrode reactions.

#### **IMPROVEMENTS**

- Improved Physics Induced Mesh settings for Porous Electrodes in all user interfaces.
- The Electrolyte-Electrode Boundary Interface feature now supports Harmonic Perturbation (in the Current Distribution user interfaces).
- A Slip Tangential Velocity Condition in the Electrode-Electrolyte Interface Coupling boundary feature (in the Free and Porous Media Flow user interface).
- Improved default solver settings for Automatic Remeshing (in the Electrodeposition user interfaces).

#### *New Model*

---

- Rotating Cylinder Hull Cell: This axisymmetric example models and compares primary, secondary, and tertiary current distributions in a rotating cylinder Hull cell.

#### *Backward Compatibility vs. 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  $\varepsilon_p$  is the porosity.

#### **WEAK CONSTRAINTS UPDATE FOR FLUID FLOW INTERFACES**

The weak constraint formulation 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, Electrodeposition, and Tertiary Nernst-Planck) anisotropic mobility is now supported, and the mobility can be set to following the

Nernst-Einstein relation. The default setting when making new models has been changed to use the Nernst-Einstein relation. Model Java-files generated prior to 4.3b using Migration will have to be modified manually to account for these changes.



# Fatigue Module

## *New Functionality in Version 4.3b*

---

- Rainflow analysis with the Palmgren-Miner rule. With the Cumulative Damage feature you can evaluate fatigue caused by a variable or random load. The rainflow algorithm reduces the variable stress history into a discrete stress distribution. The fatigue limitation is defined with a S-N curve that incorporates the R-value dependence. The damage is calculated using the Palmgren-Miner linear damage rule. For large load histories significant computation time gains can be made through use of the option generalized loads. This technique utilizes the principle of superposition to calculate the influence of the load cycle on the structure.
- Fatigue load cycle simulation together with the Shell, Plate, and Multibody Dynamics user interfaces.
- Matrix Histogram plots, a new 2D plot type for visualizing fatigue results.

## *New Models in Version 4.3b*

---

- *Frame with Cutout*: A thin-walled frame member with a central cutout is subjected to a random load scenario. Although the stresses are expected to be far below the yield level of the material, a concern arises whether or not the component will fail due to fatigue. This example demonstrates an approach to damage quantification of a long load history. The Rainflow counting algorithm is used to define the load scenario, and the Palmgren-Miner linear damage model quantifies the damage.
- *Cycle Counting Benchmark*: A benchmark model of the rainflow counting algorithm compares results between ASTM and the COMSOL Fatigue Module using a flat tensile test specimen. An extension is made for the cumulative damage calculation following the Palmgren-Miner model and results are compared with analytical expressions.
- *Rim Fatigue*: A high-cycle fatigue (HCF) analysis of a spoke in a wheel rim. As the wheel rotates, the stress histories in the structure will become non-proportional. The risk of fatigue is evaluated using the Findley criterion.

# Heat Transfer Module

*New and Improved Functionality in Version 4.3b*

---

## **MULTIPLE SPECTRAL BANDS AND SURFACE-TO-SURFACE RADIATION ENHANCEMENTS**

Enhancements are available for surface-to-surface radiation:

- The surface temperature used by surface-to-surface radiation features now account for the sides of a slit when they are active on the same boundary.
- Surface-to-surface radiation supports non unity transparent media refractive index.

In addition surface-to-surface radiation supports multiple spectral bands:

- it provides a predefined interface for solar and ambient spectral bands.
- it supports up to five wavelength intervals.
- the surface properties (emissivity) can be defined for each spectral band.
- it provides predefined fractional emissive power for blackbody and greybody surfaces or user-defined emissive power.

## **THERMAL CONTACT**

The thermal contact feature calculates the thermal conductance used to define thermal contact. The conductance  $h$  consists in 3 contributions:

$$h = h_c + h_g + h_r$$

- $h_c$  = Thermal Constriction Conductance: represents the actual contact term which depends on the surface properties and the contact pressure.
- $h_g$  = Gap Conductance: conductance through the fluid between surfaces. It is negligible when the contact pressure is significant but can play a role just before a good contact (high pressure) is established.
- $h_r$  = Radiative Conductance: surface-to-surface radiation contribution.

Each contribution can be evaluated following a predefined correlation and can be coupled with structural mechanics contact.

Friction heat source is supported with predefined heat partition coefficient definition.

Thermal contact can be used on pairs and on internal boundaries.

## HEAT TRANSFER IN POROUS MEDIA

Domain feature to model heat transfer in porous media. This feature replaces the previous **Heat Transfer in Fluids** and **Porous Matrix** domain features.

## HEAT TRANSFER WITH PHASE CHANGE

This feature implements the apparent heat capacity method for modeling phase change. It provides automatically equivalent thermodynamic properties and account for phase change latent heat.

The Phase Change and Continuous Casting models now use this feature.

## ACCURATE FLUXES

New postprocessing variables are available to postprocess conductive heat flux without the need to estimate the temperature gradient explicitly. This option, activated by default, requires solving for additional internal degrees of freedom. It provides high accuracy for conductive heat flux evaluation on exterior and interior boundaries.

## HEAT TRANSFER IN THIN SHELLS ENHANCEMENTS

Support for total power and total source have been added **Heat Transfer in Thin Shell** features.

**Heat Transfer in Thin Shell** also supports frame conversions, like domain heat transfer user interfaces.

## MOIST AIR FUNCTIONS

Three functions have been added to convert relative humidity into vapor concentration or moisture content and to get saturation pressure.

- Relative humidity to vapor concentration:  $fc(RH,T,pA)$
- Relative humidity to moisture content:  $fxvap.(RH,T,pA)$
- Saturation pressure function:  $psat(T)$

## NEW VARIABLES FOR THERMAL DIFFUSIVITY

Predefined postprocessing variables are available for the mean thermal diffusivity,  $\alpha = k_{\text{mean}}/(\rho C_p)$  and for the thermal diffusivity tensor,  $\alpha = \mathbf{k}/(\rho C_p)$ .

## **THERMAL RESISTANCE INPUT IN THIN THERMALLY RESISTIVE LAYER FEATURE**

An alternative method is available to define thin thermally resistive layers properties. Instead of specifying the layer properties as thickness and thermal conductivity, it is possible to define its thermal resistance (SI unit:  $\text{m}^2\text{K}/\text{W}$ ).

## **STRUCTURED POSTPROCESSING MENUS**

In the Heat transfer interfaces, plot menu paths have been added to classify variables.

### *New Models in Version 4.3b*

---

## **CONDENSATION DETECTION IN AN ELECTRONIC DEVICE**

This example simulates the thermodynamical evolution of moist air in an electronic box with the aim of detecting whether condensation occurs when the external environment properties change.

The model imports measured data for the air temperature, pressure, and water vapor concentration.

For the first part of the simulation, the water vapor concentration is considered to be homogeneous inside the box and equal to the external concentration. In the second part, transport and diffusion of the water vapor are taken into account.

## **FREEZE DRYING**

This example models the process of ice sublimation in a vial under vacuum-chamber conditions, a test case for many freeze-drying setups. The model uses the **Deformed Geometry** interface to compute the coupled heat and mass balances to handle the advancing vapor-ice interface.

## **TIN MELTING FRONT**

This example demonstrates how to model phase transition by a moving boundary interface according to the Stefan problem.

A square cavity containing both solid and liquid tin is submitted to a temperature difference between left and right boundaries. Fluid and solid parts are solved in separate domains sharing a moving melting front. The position of this boundary through time is calculated according to the Stefan energy balance condition.

In the melt, motion generated by natural convection is expected due to the temperature gradient. This motion, in turn, influences the front displacement.

### **SHELL AND PIPE HEAT EXCHANGER**

Shell and tube heat exchangers are commonly used in oil refineries and other large chemical processes as well as in many other applications.

Two separated fluids with different temperatures flow through the heat exchanger, one flows through the tubes (tubeside) and one through the shell around the tubes (shellside). There are several design parameters and operating conditions influencing the optimal performance of a shell and tube heat exchanger.

### **ELECTRONIC ENCLOSURE COOLING**

This study simulates the thermal behavior of a computer power supply unit (PSU). Most of such electronic enclosures include cooling devices to avoid electronic components to be damaged by excessively high temperatures. In this model, an extracting fan and a perforated grille cause an air flow in the enclosure to cool internal heating.

### **CROSS-FLOW HEAT EXCHANGER**

This model solves the fluid flow and heat transfer in a micro heat exchanger made of stainless steel. These types of heat exchangers are found in lab-on-chip devices in biotechnology and microreactors, for example, for micro fuel cells. The model takes heat transferred through both convection and conduction into account.

### **PLATE HEAT EXCHANGER**

One of the most common devices in unit operations is the heat exchanger, which can both cool and heat fluids and solid materials. Cooling is a needed process for several reasons: materials are sensitive to high temperatures; operating conditions might not be optimal for high temperatures; or it might just be for practical or environmental reasons that you want to keep a low temperature. Heat exchangers are most often used in processes to remove heat when convective cooling from fans and fins is not adequate. Heating is also used in many processes to keep a fluid at a given elevated temperature.

### **THERMAL CONTACT RESISTANCE BETWEEN AN ELECTRONIC PACKAGE AND A HEAT SINK**

This example reproduces parts of the study of a publication on the thermal contact resistance at the interface between a heat sink and an electronic package. Eight cooling fins equip the cylindrical heat sink and contact is made at the radial boundaries of the package. The efficiency of the device depends on the cooling of the fins and the heat transfer from the package to the heat sink. This model focuses on the heat transfer

through the contact interface where four parameters influence the joint conductance: contact pressure, microhardness of the softer material, surface roughness, and surface roughness slope.

### **CONTACT SWITCH**

This model illustrates how to implement a multiphysics contact. It models the thermal and electrical behavior of two contacting parts of a switch. The electrical current and the heat cross from one part to the other only at the contact surface. The contact switch device has a cylindrical body and plate hook shapes at the contact area. There, the thermal and electrical apparent resistances are coupled to the mechanical contact pressure at the interface, which is solved by the model.

### *Backward Compatibility vs. 4.3a*

---

### **HEAT TRANSFER IN POROUS MEDIA**

Old models using the Porous feature will be supported in COMSOL Multiphysics interface. It is however recommended to replace this feature by **Heat Transfer in Porous Media** (see [Obsolete features](#)). 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.

4.3 and 4.3a models containing surface to surface models will support the old syntax in multiphysics interfaces. It is, however, recommended to update the models to use the new syntax. Model created with a version before 4.2a needs to be opened in version 4.3 or 4.3a and resaved before opening in 4.3b.

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

### **RADIATION IN PARTICIPATING MEDIA**

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

4.3 and 4.3a models containing radiation in participating media models will support the old syntax in multiphysics interfaces. It is, however, recommended to update the models to use the new syntax. Model created with a version before 4.2a needs to be opened in version 4.3 or 4.3a and resaved before opening in 4.3b. In addition, for all versions, it may be necessary to regenerate solver default to get 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 quadrature 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 vs. 4.3*

---

#### **THE DEFAULT MODEL LIST HAS BEEN REMOVED**

Heat transfer user interfaces no longer have a default model list. This list was used to change the default feature of models. A similar result can be obtained by adding the corresponding domain feature immediately below the default feature and by setting the domain selection to *All domains*.

Any user Model Java-files that modify the default model value will 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 to account for the contributions from linear source terms from *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 of the models when these contributions are large. This change can modify the convergence behavior of existing models.

#### **FRAME SELECTION IN DISCRETIZATION SECTION**

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

#### **UPDATE OF FEATURES VARIABLES NAMES**

In order to avoid name conflicts between features variable names some of them have been renamed.

This change concerns contributive features. Since they are contributing, it is possible to have to similar features active on the same boundary. In order to make it possible to distinguish the variables from each feature, the feature scope has been introduced in

the variable name. When the feature tag was appended at the end of the variable name, the variable name uses now a prefix. These changes affect following features 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 `ht.bhs1.surf` (assuming that `ht` is the physics interface tag).

Another example, in the *Heat Flux* feature, the variable that was previously named `ht.q0_hf1` is now `ht.hf1.q0`.

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.

#### **NEW DEFAULT FLUID FEATURES AND OPAQUE SUBFEATURE**

In relation to the improved default settings, all Model MPH-files will be automatically converted to be identical to the 4.3 version.

Due to these new default features, user Model Java-files can be simplified. In addition, in the user Model Java-files that adds the *Fluid* feature or the *Opaque* subfeature with the default tag, a manual update is required to avoid a duplicate tags conflict.

#### **WEAK CONSTRAINTS UPDATE FOR FLUID FLOW USER INTERFACES**

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

- Laminar Flow
- Turbulent Flow,  $k$ - $\epsilon$



- Turbulent Flow, low-Re  $k$ - $\epsilon$
- 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 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 4.3b compared to version 4.3 and, the result can differ slightly between the versions.

#### *Backward Compatibility vs. 4.2a*

---

#### **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 Conjugate Heat Transfer interface.

#### *Backward Compatibility vs. 3.5a*

---

#### **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 4.3b. 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.

## **SURFACE-TO-SURFACE AND OPAQUE FEATURE**

The opaque subfeature is expected to set by default under **Heat Transfer in solids**. However, when 3.5a models are loaded, the opaque subfeature is added under all the domain features. This can lead to wrong radiation direction definition.

### *Obsolete features*

---

Following feature are still supported when models created with older version are loaded in COMSOL Multiphysics. However they will be removed in future version. Hence they should be replaced 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**.

# MEMS Module

## *New Functionality in Version 4.3b*

---

- Thermal expansion in the Piezoelectric Devices and Electromechanics user interfaces.
- The Floating potential group check box is a new setting for the *Floating Potential* node in the Electric Currents and Electrostatics user interfaces, which considers each group of connected boundaries as a separate group with its own potential. The floating potential group simplifies set-up of models with many floating electrodes.

## *New Model in Version 4.3b*

---

- *Thin-Film BAW Composite Resonator*: Bulk acoustic wave (BAW) resonators are useful components for many radio-frequency applications, where they can operate as narrow band filters. This example shows how to perform eigenfrequency and frequency-response analyses of a composite thin-film BAW resonator.

## *Backward Compatibility vs. Version 3.5a*

---

### **FILM DAMPING**

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

# Microfluidics Module

## *New Model in Version 4.3b*

---

*Tesla Microvalve Optimization:* This model performs a topological optimization for a Tesla microvalve. A Tesla microvalve inhibits backward flow using friction forces rather than moving parts. The design can be optimized by distributing a specific amount of material within the modeling domain. The goal is to maximize the ratio of the pressure drop for the forward and backward flow across the device.

## *Backward Compatibility vs. 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.

## *Backward Compatibility vs. 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  $\varepsilon_p$  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 following the Nernst-Einstein relation. The default setting when making new models has been changed to use the Nernst-Einstein relation. Model Java-files generated prior to 4.3b using Migration will have to be modified manually to account for these changes.

# Molecular Flow Module

The Molecular Flow Module is a collection of tailored physics user interfaces for the simulation of kinetic gas flows. The module includes two physics user interfaces: the Transitional Flow and Molecular Flow. The Transitional Flow user interface uses a discrete velocity/lattice Boltzmann approach to solve low velocity gas flows in the transitional flow regime. The Molecular Flow user interface uses the angular coefficient method to solve molecular flows. The Model Library and supporting documentation explain how to use the interfaces to model a range of problems encountered by vacuum practitioners.

Gas rarefaction is of critical importance for both high velocity gas flows around aircraft traveling at high altitudes and for low velocity flows at low gas pressures, such as those encountered in vacuum systems. Historically most academic research has been focused on aerospace applications, and consequently modeling software appropriate for such flows is readily available. Unfortunately the techniques usually employed to model the high velocity flows around aircraft are inappropriate for the modeling of low velocity flows, such as those typically encountered in vacuum systems because the particle-based methods employed produce an unacceptable level of statistical noise at low velocities. The Molecular Flow Module is designed to offer previously unavailable simulation capabilities for the accurate modeling of low pressure, low velocity gas flows. The module facilitates accurate simulations of molecular flows in complex geometries.

# Multibody Dynamics Module

The Multibody Dynamics Module, which requires the Structural Mechanics Module, is designed for performing static and dynamic analyses for a system of flexible and rigid components connected through a set of joints with certain degrees of freedom. The components can then undergo complex combinations translational and rotational motions with respect to one another. Some major application areas include automotive applications, aerospace engineering, biomechanics, biomedical instruments, robotics, and general dynamic simulations.

The joints that connect the different structural parts can have many different properties, including elastic connections, viscous damping, stop conditions, and lock conditions.

# Nonlinear Structural Materials Module

## *New Functionality in Version 4.3b*

---

- Three new hyperelastic material models:
  - Gent
  - Gao
  - Storakers

## *New Model in Version 4.3b*

---

*Arterial Wall Mechanics:* This example shows how to implement an anisotropic hyperelastic material for modeling collagenous soft tissue in arterial walls. Anisotropic hyperelasticity is based on the Holzapfel-Gasser-Ogden model. The new Curvilinear Coordinates user interface is used to visualize the fiber directions.

# Optimization Module

## *New Functionality in Version 4.3b*

---

- An *Optimization* study node that you can add directly from the Study node now includes all optimization methods: SNOPT and Levenberg-Marquardt, in addition to the gradient-free optimization methods Nelder-Mead, Monte Carlo, and Coordinate search.
- For the gradient-free optimization methods, you can choose a method for multiple objectives: sum of objective (the default), minimum of objectives, or maximum of objectives.

## *Updated Models in Version 4.3b*

---

The following models now use the Optimization study node to set up the optimization:

- Tuning Fork Optimization
- Curve Fit Mooney-Rivlin
- Horn Shape Optimization (also requires the Acoustics Module)
- Optimal Cooling (also requires the Chemical Reaction Engineering Module)
- Pipeline Insulation (also requires the Pipe Flow Module)

## *Backward Compatibility vs. 4.3a*

---

The *Optimization* check box in the Stationary and Time Dependent study steps have been removed. Instead use the Optimization study step for including optimization in a model. The Optimization check box does appear in models created using earlier versions of COMSOL where you have selected that check box



# Particle Tracing Module

*New Functionality in Version 4.3b*

---

## **MONTE CARLO ELASTIC COLLISIONS**

A *Monte Carlo elastic collision* option is now available in the Elastic Collision Force node in the Charged Particle Tracing user interface. The particle velocity changes by a discrete amount if a collision occurs with a suitably high probability. The collision probability is determined by the collision cross section and background number density. The Monte Carlo elastic collision option is useful for modeling ion motion where the background pressure is above vacuum, such as ion funnels and ion mobility spectrometers.

## **VELOCITY REINITIALIZATION**

The velocity of a particle can be changed arbitrarily if a certain expression is fulfilled, which can be used for general-purpose Monte Carlo simulations. This feature is available in all particle tracing user interfaces.

## **CHANGING AUXILIARY VARIABLES**

It is now possible to change the value of an auxiliary dependent variable when it crosses or touches a boundary using a Wall node. The value can be a function of any combinations of particle variables and variables defined in the physical geometry. An application for this functionality is to use it to count the number of times a particle strikes the wall.

## **PERFORMANCE IMPROVEMENTS**

Models including wall bounces will solve up to 10 times faster than in earlier versions.

*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

## *New Functionality in Version 4.3b*

---

- The *Pipe Acoustics, Frequency Domain* user interface (requires the Acoustics Module) supports frequency-domain and eigenfrequency studies and is designed for modeling propagation of sound waves in flexible pipe systems. The equations are implemented in 1D along edges and solve for the cross-section averaged acoustic pressure and acoustic particle velocity. The equations are formulated in a general way to include the possibility of a stationary background flow. This physics user interface is available in 3D on edges and points and in 2D on boundaries and points, and it includes a series of engineering relations for the end impedance at the pipe terminations.
- Homogenized multiphase models added to the Pipe Flow interface: With a Newtonian fluid model, you can choose a gas-liquid model with a friction factor multiplier or an effective Reynolds number. This gas-liquid model solves for one fluid's velocity/pressure field and adjusts either the friction factor or Reynolds number.

## *New Model in Version 4.3b*

---

*Organ Pipe Design*: Tutorial model that uses the Pipe Acoustics, Frequency Domain user interface to model an organ pipe. The half wave resonance frequency is studied with respect to dependency on the pipe radius and the pipe wall thickness. The acoustic properties of the system can easily be studied with respect to other parameters like the pipe cross section shape, the pipe material, or even a steady flow of air in the pipe.

# Plasma Module

*New Functionality in Version 4.3b*

---

## **COLLISIONLESS HEATING**

A collisionless heating feature has been added based on the Hagelaar model. The feature adds an additional heat source for the electrons to mimic the effect of collisionless heating. Collisionless heating is only available for 2D axially symmetric models, and it introduces an extra dependent variable for the out-of-plane electron drift velocity.

# RF Module

## *New Functionality in Version 4.3b*

---

- A *passive lumped port* is now available without generating unnecessary S-parameters. The port can be of a uniform, coaxial, or user-defined type.
- Analytical circular port on 3D models providing predefined modes as rectangular waveguide ports do. These ports are available for TE and TM modes in frequency-domain studies.
- Ports for periodic structures in 3D. The port computes all possible two-dimensional diffraction orders from a reciprocal lattice structure. Using an available Reference Point subnode it is possible to characterize the reciprocal lattice. The periodic port supports both elevation and azimuth angle configuration for wave incidence.

## *New Model in Version 4.3b*

---

*Optimizing a Bow-Tie Antenna:* A bow-tie antenna patterned on a dielectric substrate is optimized by changing the length of the arms and the flare angle to reduce the magnitude of the reflection coefficient.

## *Backward Compatibility vs. 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

## *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 4.3b 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.

# Semiconductor Module

The new Semiconductor Module is a collection of physics user interfaces and predefined models that you can use for modeling of semiconductor devices based on charge-carrier transport methods.

It includes a dedicated interface to model semiconductor devices with the drift diffusion equations, enhanced capabilities for modeling electrostatics, and an electrical circuits interface (with a SPICE import facility) that can be coupled to the device level models.

The Semiconductor Module Model Library contains a suite of models that demonstrate how to use the interface for modeling a range of practical devices. These models provide step-by-step instructions demonstrating how to use the interfaces to model practical devices. The module is well documented and includes a detailed outline of the underlying theory on which it is based. Furthermore the software is uniquely transparent because the model equations are always visible, and you have complete freedom in the definition of phenomena that are not predefined.

# Structural Mechanics Module

## *New Functionality in Version 4.3b*

---

- New *Bolt Pre-Tension* functionality for modeling pre-stressed bolts.
- Thermal expansion is now available in the Piezoelectric Devices user interface.
- The center of rotation for a rigid connector can be defined as the centroid of a set of geometrical entities, which does not need to be part of the selection for the Rigid Connector itself. This functionality is available only in the Solid Mechanics user interface and not in the multiphysics interfaces.
- The settings for contact analysis have been improved:
  - The setting of the penalty function is more user friendly using preset penalty factor control by default. The Friction subnode includes the same options for the penalty factor but the default is to inherit the settings from its parent Contact node.
  - The solver might spend a lot of time iterating on a solution from which it is impossible get back to the true solution path. You can now create a Boolean expression that will force the solver to immediately stop the iterations and jump back in time or parameter value.
  - The number of segregated iteration before an attempt is rejected has been reduced from 40 to 15. A well-behaved problems should seldom need more than 10 iterations, so now a cutback will come sooner in a non-converging situation.
- A *Beam Cross Section* user interface, where you can draw the cross section in 2D, and the software then computes the beam data, including moments of inertia, torsional rigidity, section modulus, and warping section modulus. It is also possible to perform a detailed evaluation of the stresses on a beam cross section given the section forces. When coupled with beam elements in 3D this provides a high-fidelity method for stress evaluation in critical cross-sections of the beam.

## *New and Updated Models in Version 4.3b*

---

- A new tutorial model, *Stress Dependent Elasticity*, shows how to implement an explicit dependence of the stress in your material description.

- A new verification model, *Elliptic Membrane*, gives a detailed convergence study for solid elements in elasticity.
- The *Tube Connection* model has been significantly reworked to display the use of the new Bolt Pre-Tension feature.

### *Backward Compatibility vs. 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** with the contact normal penalty factor set to the expression used in the model.

### *Backward Compatibility vs. Version 3.5a*

---

#### **DISPLACEMENTS CONTROL THE SPATIAL FRAME**

In the Solid Mechanics physics, COMSOL now always assumes that the displacements control the spatial frame. See Support Knowledge Base article 1156.



# Subsurface Flow Module

*New Functionality in Version 4.3b*

---

## **HEAT TRANSFER IN POROUS MEDIA**

Domain feature to model heat transfer in porous media. This feature replaces the previous **Heat Transfer in Fluids** and **Porous Matrix** domain features.

## **HEAT TRANSFER WITH PHASE CHANGE**

This feature implements the apparent heat capacity method for modeling phase change. It provides automatically equivalent thermodynamic properties and account for phase change latent heat.

The Phase Change model now uses this feature.

*Backward Compatibility vs. 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  $\varepsilon_p$  is the porosity.

## **WEAK CONSTRAINTS UPDATE FOR FLUID FLOW INTERFACES**

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

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

# Wave Optics Module

The Wave Optics Module solves problems in the field of electromagnetic waves at optical frequencies (corresponding to wavelengths in the nanometer to micrometer range). The module is useful for simulations and design of optical applications in virtually all areas where you find electromagnetic waves, such as:

- Optical fibers
- Photonic waveguides
- Photonic crystals
- Nonlinear optics
- Laser resonator design
- Active devices in photonics

The physics user interfaces cover the following types of electromagnetics field simulations and handle time-harmonic, time-dependent, and eigenfrequency/eigenmode problems:

- In-plane, axisymmetric, and full 3D electromagnetic wave propagation
- Full vector mode analysis in 2D and 3D

Material properties include inhomogeneous and fully anisotropic materials, media with gains or losses, and complex-valued material properties. In addition to the standard postprocessing functionality, the Wave Optics Module supports direct computation of S-parameters and far-field patterns. You can add ports with a wave excitation with specified power level and mode type, and add PMLs (perfectly matched layers) to simulate electromagnetic waves that propagate into an unbounded domain. For time-harmonic simulations, you can use the scattered wave or the total wave.

# LiveLink for MATLAB®

## *New Functionality in COMSOL 4.3b*

---

The following new function is available:

- Support do solution information: Using the new function `mphsolution` you can retrieve information about the solution object for both inner and outer parameters in parametric sweeps.
- Evaluate expressions on particle trajectories: The new function `mphparticle` makes it possible to evaluate, along particle trajectories, information such as particle position, particle velocity, or any postprocessing expression. The `mphparticle` function supports only datasets created with the Particle Tracing Module.
- Table data access: Use the new function `mphtable` to extract data from the columns and headers of a table contained in the model.

## *Known Issues*

---

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

## *Backward Compatibility vs. Version 3.5a*

---

### **MATLAB SUPPORT WITH COMSOL MULTIPHYSICS VERSION 3.5A**

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

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 4.3b.

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 4.3b with MATLAB.

If you have any problems with this conversion process, please contact COMSOL's technical support team at [support@comsol.com](mailto: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 4.3b. This functionality will be available in a later release.

#### **COMSOL API**

See the section “The COMSOL API for use with Java<sup>®</sup>” on page 70 regarding changes in the COMSOL API.

# LiveLink for Excel®

## *New Functionality in Version 4.3b*

---

The LiveLink™ for Excel® includes the following new functionality:

- Support for multiple models from Excel with one COMSOL model per workbook (.xlsx file).
- Support for Excel 2013 (only the desktop version is supported. This makes this LiveLink compatible with Excel 2007, 2010, and 2013 on Windows).
- Material data available in Excel sheets can be imported into a material library in COMSOL.
- File locking is supported when opening a COMSOL model from the COMSOL ribbon tab in Excel.
- Excel files can be used in interpolation functions.
- Excel files can be exported from the Table window.
- An option is now available to separate values and units during import and export of parameters to and from Excel.
- Easier handling of Excel import and export in tables in the COMSOL Desktop settings windows.

# The COMSOL API for use with Java®

## *COMSOL 4.3b API Changes*

---

### **PHYSICS USER 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 compared to models solved in version previous to version 4.3b. Java files can be modified to retain the 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 get 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` is deprecated and replaced by `order`.

## *COMSOL 4.3a API Changes*

---

### **GENERAL API CHANGES**

- Class files compiled with COMSOL 4.3 or lower version need to be recompiled if they contain code that change the model of selections.
- 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_1m`. now create a vectorial Lagrange multiplier field with default name `model.u_1m`. The default component names in 3D are `model.u_1m`, `model.v_1m` and `model.w_1m` respectively. Java files must be updated accordingly.

- Weak constraints for the Interior Wall feature are no longer available. Any references to the features weak constraint parameter (`weakConstraints`) or its Lagrange multipliers must be removed.
- The **Automatic** setting for the **Pseudo time stepping** property now sets the variable `<phtag>.locCFL` equal to the built-in variable `CFLCMP` which in turn triggers a PID regulator via the automatic solver suggestions. Here, `<phtag>` 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("CFLNumbExpr", 1, "Manual");
```

 before calling a solver.

### COMSOL 4.3 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.
- From version 4.3 the methods
 

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

 are depreciated 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 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 `ei1`.
- 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 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 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.