COMSOL Multiphysics®

Release Notes

COMSOL Multiphysics Release Notes

Version: October 2012 COMSOL 4.3a

Contact Information

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

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

Other useful links include:

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

1

Release Notes

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

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

COMSOL Multiphysics

New Products in Version 4.3a

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

- Fatigue Module extends the Structural Mechanics Module with functionality for fatigue analysis. See Fatigue Module for more information.
- LiveLink™ *for* Excel® gives you access to COMSOL model definitions and results data from an Excel spreadsheet. See LiveLink for Excel for more information.
- LiveLink™ *for* Solid Edge® provides a bidirectional interface to the Solid Edge CAD software. See CAD Import Module and LiveLink Products for CAD for general information about the LiveLink products for interfacing CAD software.
- ECAD Import Module for importing ECAD geometries into COMSOL. See ECAD Import Module for more information.

New Functionality in Version 4.3a

GENERAL COMSOL DESKTOP FUNCTIONALITY

- A *Cylinder* selection feature makes it possible to create a cylinder for selecting objects in a geometry. This feature is similar to the Box and Ball selection features and can simplify selection of geometric entities in suitable geometries.
- You can select boundaries (faces) and edges in the geometry by continuous tangent in the *Explicit*, *Ball*, *Box*, and *Cylinder* selection features. That makes it possible to select, by just selecting a single face, for example, an entire sheet that consists of several faces (boundaries) that are connected so that their tangents are continuous (within an angular tolerance that you can specify).
- You can specify the *browser* to use for viewing COMSOL documentation in the Preferences dialog box. By default, COMSOL uses the system default browser, but you can also select to use a custom browser.
- The predefined physical constants in COMSOL now use the 2010 CODATA recommended values, which in some cases differ slightly from the previous values (from 2006). The *COMSOL Multiphysics User's Guide* contains a list of all supported physical constants and their values, which now are the 2010 CODATA recommended values.

- The global variable `numberofdofs` reports the total number of degrees of freedom (DOFs) in the model, including any internal DOFs in the solver.

- It is now possible to plot the *inverse function* in addition to plotting the function itself for interpolation functions that you define.

- A warning is now issued if you use a reserved name for a built-in function (such as `abs` or `cos`) as the name for a user-defined function. The name then turns orange, and a tooltip indicates that the chosen name is a reserved function name. Such warnings also appear for user-defined parameters and variables that use names that are reserved names for built-in variables such as `i`, `j`, and `pi`.

- New operators:

  - `integrate`: Evaluates an integral of a general expression with respect to an integration variable over a real interval specified by lower and upper limits.

  - `circumcenter`: Evaluates an expression in the circumcenter of the mesh element to which the point belongs.

  - `centroid`: For simplex meshes, this operator evaluates an expression in the centroid of the mesh element to which the point belongs.

  - `env`: Evaluates an expression at the coordinates of a particle in a domain. Requires the Particle Tracing Module.

  - `bndenv`: Evaluates an expression at the coordinates of a particle at a boundary. Requires the Particle Tracing Module.

GEOMETRY AND MESH

New Geometry Tools and Functionality

- Global x-, y-, and z-axis as predefined options for axis direction in 3D geometry features such as 3D geometry primitives (Block, for example) and applicable geometry operations (Rotate, for example) and selection types (Cylinder, for example) are now available from the *Axis type* lists in addition to general Cartesian and spherical axes.

- The Interpolation Curve feature can now read curve coordinates from file on the sectionwise data format in addition to the spreadsheet data format. You can also specify the curves as vectors of x, y, and z coordinates.

- The behavior when moving geometry objects directly in the Graphics window has been improved so that a move updates the position of the original geometry object instead of creating an explicit Move feature.

New Meshing Tools and Functionality

- Ball and Box wireframe visualization in mesh import sequences for partitioning of imported meshes by splitting geometric entities by creating at least one new geometric entity for the elements enclosed in the specified ball or box.

- Logical Expression feature for mesh import so that the imported mesh can be partitioned into separate domains, for example, by creating at least one new geometric entity for the mesh elements that fulfill the specified logical expression.

- Automatic refinement of the starting mesh at sharp corners.

- Support for boundary layers on isolated boundaries.

PHYSICS INTERFACES

- Improved constraint settings for Dirichlet-type boundary conditions. If you activate the Advanced Physics Options, boundary conditions such as Dirichlet Boundary Condition or a Temperature condition includes a Constraint Settings section with new options. From the *Apply reaction terms on* list:

  - Select *All physics (symmetric)* to apply reaction terms bidirectionally on *u* as well as on any other dependent variables appearing in *r*. This corresponds to the bidirectional (symmetric) constraint in version 4.3.

  - Select *Current physics (internally symmetric)* to apply reaction terms on all components of *u*. This is a new constraint type.

  - Select *Individual dependent variables* to apply reaction terms from each active component in the Dirichlet condition only on the corresponding component of *u*. This is the default setting and corresponds to the unidirectional constraint in version 4.3.

- New Wave Form PDE interfaces, mathematical interfaces that are based on the discontinuous Galerkin method for use with the time-explicit solver.

- Predefined flux variables for accurate evaluation of boundary fluxes are now available for the Coefficient Form PDE and General Form PDE user interfaces, where it is enabled by default. The flux variables are also available in the structural mechanics user interfaces but there you need to activate them first. For example, `dflux.u` is the boundary flux in the down direction for a dependent variable *u* in a Coefficient Form PDE. To create variables for boundary fluxes, select the *Compute boundary fluxes* check box in the physics interfaces *Discretization* section.

STUDIES AND SOLVERS

- *Get Initial Value* is available as an option on the Study level and for each study step node. Using this option makes COMSOL evaluate the solution and variables using

the initial values, which makes it possible to plot and evaluate the solution and any solution-dependent variables using the initial values as the solution.

- Compute commands are now available for individual study steps (without opening the solver sequence): When applicable, each study step includes options to *Compute to Selected*, *Compute Selected Step*, and *Compute from Selected* for easier testing and debugging of large models with several study steps.

- Support for running COMSOL on the Amazon Elastic Compute Cloud™ (Amazon EC2™). This is made possible by the addition of new remote access tools for optimized data transfer, which you can run from the COMSOL Desktop or in batch mode. See the separate *Running COMSOL on the Amazon Cloud* document for details.

- Improvements to solution selection in the Study and Dependent Variables nodes:
  - The possibility to select a solution number manually. This input can be used for parametric sweeping with time-dependent and eigenvalue and parametric solutions as input.
  - The options *First* and *Last* are available as options for the solution numbers to select.
  - Specification of *Initial values of variables solved for* is now available also on the study step level and not only in the solver sequence.

- Export of system matrices (stiffness matrices, mass matrices, and so on) from the Modal Solver, which can now provide mode-reduced system matrices, and from the Assemble feature. Under Derived Values in the Results branch, you can add a System Matrix node where you specify which of the computed system matrices to output and if you want to output them using a sparse of full format.

- Input of system matrices generated outside of COMSOL using an Input Matrix subnode to a solver node, where you can specify which system matrices and vectors that should use external data in Java. The saved Model Java-file contains code for inputting the selected matrices and vectors.

- Pseudo time stepping is now available for the stationary solver when using a fully coupled or segregated solver. Pseudo time stepping is used in transport problem to stabilize the convergence toward steady state. Here an adaptive feedback regulator controls a CFL (Courant–Friedrichs–Lewy) number which is then used for pseudo time stepping. In the CFD Module, solver defaults control the pseudo time stepping for stationary fluid-flow models.

- Improved parallel performance for shared-memory/multicore computers and for distributed computing:
  - Multicore computing: Greatly improved handling of constraint boundary conditions such as fixed temperature, electric potential, and displacement speeds up computations for most physics. Performance increase is thanks to new constraint elimination algorithms
  - Distributed computing: Solvers have been optimized by the introduction of a very efficient sparse matrix reordering algorithm for direct solvers, and communication for matrix-vector data has been optimized.
- Domain point probes and boundary point probes can now follow a certain frame type (spatial, material, geometry, or mesh) in a model with multiple frames.

RESULTS

- Transparent background for image snapshots using PNG format, making it easier to integrate COMSOL images into documents and combining them with other graphics.
- The Join data set now supports the *norm of difference* as a method for joining two solutions, so that you can evaluate and plot that quantity when comparing solutions.
- Improved placing and display of labels on logarithmic x- and y-axes.
- More flexible *legend positioning*: Middle left and Middle right have been added as additional legend positions.
- Options to invert the phase (change the sign) when rotating and reflecting are now available as options in Sector 2D and Sector 3D data sets for more flexibility when rotating and reflecting results data.
- In the Filter subnode for the Particle Trajectories plot (for the Particle Tracing Module), it is now possible to specify a logical expression for the particle to include.
- Improved settings for animation, with better support for animating parametric solutions and added possibility to specify the number of frames in the generated movie.

Backward Compatibility vs. Version 4.3

NEW TERMINOLOGY FOR CONSTRAINT TYPES

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

- *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 condition 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_lm has been removed.

Models saved in version 4.3 can be opened and postprocessing will include un_lm until the model is re-solved. In some cases, occurrences of un_lm in the solver sequence must be replaced manually. This is the case if un_lm 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 4.3a to be possible to postprocess, or opened and re-saved in version 4.3 before opened in version 4.3a.

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 Convertpre35a Command" for Windows, the section "COMSOL Convertpre35a Command" for Linux, or the section "COMSOL Convertpre35a Command" for the Mac in the *COMSOL Installation and Operations Guide* for more information.

AC/DC Module

New Functionality in Version 4.3a

- The Electrostatics physics now supports the Frequency Domain study.

Backward Compatibility vs. Version 3.5a

CHANGE IN DEPENDENT VARIABLES

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

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

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

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

- New fluid model, *Boundary layer approximation*, for pressure acoustics that you can use for modeling the thermal and viscous losses at boundaries of a duct as a bulk loss (an equivalent fluid). You can choose between *wide duct* and *narrow duct* approximations

- Enter viscous characteristic length directly in Biot equivalent fluid models: It is now possible to enter the viscous characteristic length L_v or the viscous characteristic length parameter s.

- Domain heat sources are now available in the physics interfaces for thermoacoustics. Heat sources in the domain are useful when modeling optoacoustics and acoustic heat exchangers. In connection with this addition of domain heat sources, the symbols for acoustic monopole dipole volume sources in pressure acoustics have changed (see below).

- Editing of advanced pressure acoustics equation settings when the equation form is set to *Frequency domain* in the Pressure Acoustics node's settings window. You can then edit the expression for the scaling and the order of the non-reflecting boundary condition approximation, which is necessary for a correct set up of the modal solver.

New and Updated Models

- A model (bk_4134_microphone) of the Brüel & Kjær 4134 condenser microphone, comparing the modeled sensitivity level to measurements performed on an actual microphone, which are in good agreement. The membrane deformation, pressure, velocity, and electric field are also determined.

- A model (probe_tube_microphone) of a probe tube attached to a microphone. This model requires the Acoustics Module and the Pipe Flow Module.

- An example (nonlinear_acoustics_westervelt_1d) demonstrating how to model transient nonlinear propagation of finite-amplitude acoustic waves in fluids, solving a 1D Westervelt equation.

- A model (piezoacoustic_spherical) of a spherical piezoacoustic transducer. The device is poled along the radial direction of the sphere, requiring the definition of a new local system of coordinates.

- A model (acoustics_particulate_filter) demonstrating acoustics of a particulate-filter-like system. This updated version computes the acoustic transmission loss through a particulate-filter-like system using the Poroelastic Waves user interface.

Backward Compatibility vs. Version 4.3

- The symbols for the acoustics monopole (Q_m; was Q) and dipole ($\mathbf{q}_d$; was $\mathbf{q}$) volume sources in pressure acoustics have changed.

- The default value for the Typical wave speed property in the Acoustic-Piezoelectric Interaction, Frequency Domain interface has changed to 343 m/s.

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.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.

Backward Compatibility vs. Version 4.2

- The Pressure Acoustics and Transient Pressure Acoustics interfaces in version 4.2 are now called Pressure Acoustics, Frequency Domain and Pressure Acoustics, Transient, respectively. The names for other acoustics interfaces have changed in a similar way.

- When creating an orthotropic or anisotropic linear elastic material in the Elastic Waves or Poroelastic waves interfaces, the default is now to use Voigt notation. This will cause a model created using an old Java script to fail. Adjust the Java file so that it explicitly selects Standard notation instead of Voigt notation when using orthotropic or anisotropic materials.

Batteries & Fuel Cells Module

New Functionality in Version 4.3a

NEW PHYSICS FEATURES

- Film Resistance is now available in the Electrolyte-Electrode Boundary Interface and Electrolyte-Electrode Domain Interface features in all Electrochemistry physics (except Primary Current Distribution and Electrode, Shell).

- New Separator domain node in Tertiary Current Distribution, Nernst-Planck physics.

- New `liion.cs_average/batbe.cs_average` variable introduced, defined by the porous electrode node in the Lithium-Ion Battery and Battery with Binary Electrolyte Interfaces. The variable represents the average solid particle concentration in the electrode particles.

GENERAL IMPROVEMENTS AND CORRECTIONS

- The governing equations for the Porous Electrode feature in Tertiary Current Distribution, Nernst-Planck have been corrected. The time derivative is now multiplied by the electrolyte volume fraction. Also the reaction sources of the Reaction feature are multiplied by the electrolyte volume fraction (that is, implying that the sources correspond to reactions taking place in the fluid phase only). Models created prior to 4.3a with nonunity electrolyte volume fractions will render different results compared to 4.3.

- In the Lithium Ion Battery and the Battery with Binary Electrolyte physics interfaces the transport number variable (`tplus`) is now correctly assigned to the value in the **Materials** node, if **From material** has been chosen for this parameter. The Batteries And Fuel Cells material library value for `tplus` for the three lithium-ion electrolytes has been changed to 0.363. A space-dependent `tplus` is now supported.

UPDATED MODEL

- SEI layer resistance increase has been added to the "Capacity Fade of a Li-ion Battery" (capacity_fade) model.

BRINKMAN EQUATIONS AND FREE AND POROUS MEDIA FLOW

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

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 4.3a using Migration will have to be modified manually to account for these changes.

CAD Import Module and LiveLink Products for CAD

New LiveLink for Solid Edge

The new LiveLink™ *for* Solid Edge® provides a bidirectional interface for transferring CAD data to a from the Solid Edge CAD software.

New Functionality in the CAD Import Module

- During import of external CAD data you can select the *Remove redundant edges and vertice*s check box to remove such edges and vertices when importing the geometry. By default this option is not selected, so COMSOL imports the geometry keeping all edges and vertices.
- The file-based CAD import (in the CAD Import Module and all MCAD LiveLink products) now supports the following updated file formats:
    - Catia V5 R22
    - Parasolid V25

New Functionality in LiveLink for AutoCAD®

- Added support for AutoCAD 2013.

New Functionality in LiveLink for Inventor®

- Associativity is now maintained without writing information to the CAD file.

New Functionality in LiveLink for Pro/ENGINEER®

- Parameters are transferred together with their units to COMSOL.
- User-defined parameters are now correctly transferred to COMSOL.

New Functionality in LiveLink for Creo Parametric

- Support for Creo Parametric 2.0.

- Parameters are transferred together with their units to COMSOL.
- User-defined parameters are now correctly transferred to COMSOL.

CFD Module

NEW STUDY TYPES FOR ONE-WAY FSI

For improved performance for certain fluid-structure interaction (FSI) models, two new study types simplify setting up and solving one-way coupled FSI models:

- Stationary,. One-Way Coupled
- Time Dependent,. One-Way Coupled

Both study types solve for the fluid flow first, the for the solid mechanics. Other physics can be included in either or both of these study steps.

REACTING FLOW USER INTERFACE

The new Reacting Flow user interface is a multiphysics interface for mass transport in fluids. The Reacting Flow user interface combines the physics from Single Phase Flow and Transport of Concentrated Species. Key features:

- A new reaction feature.
- Pseudo time stepping for both the species and the momentum equations.
- Turbulence modeling by RANS:
  - k-ε
  - k-ω
  - low-Reynolds number k-ε
- Includes two algebraic models for the turbulent Schmidt number:
  - Kays-Crawford
  - High-Schmidt number model.
- Turbulent reaction modeling using Eddy Dissipation Concept (EDC).
- Wall functions for turbulent mass transport.

SPALART-ALLMARAS TURBULENCE MODEL WITH ROTATION CORRECTION

The Spalart-Allmaras model has been supplemented with a "rotation correction." The correction is passive in thin shear layers while it gives superior predictions in for example core regions. The rotation correction can effect the solutions of old models.

IMPROVED MESHING DEFAULTS

The meshing defaults have been improved to include corner refinement, trimming of mesh elements at sharp corners (instead of splitting), and generally improved settings for robust and accurate meshing.

IMPROVED PSEUDO TIME-STEPPING

The pseudo time-stepping algorithm, used per default for turbulence models and for Non-Isothermal Flow and Conjugate Heat Transfer models, has been improved. The new implementation uses a PID regulator, which compared to the 4.3 implementation makes the convergence more robust without using additional iterations. The additional robustness reduces the need for manual tuning of the solvers and makes it easier to solve stationary problems, especially for turbulent flow.

As a result of this update, some old models may experience different convergence behavior compared to version 4.3.

FLEXIBLE DEFAULT SOLVER SETTINGS

The default solver settings for all fluid-flow user interfaces are now based on the number of mesh elements and use a direct solver for small models (up to 100, 000 elements in 3D or 300,000 elements in2D). For larger models, additional multigrid levels are added for an iterative solver. The first additional level is added at 600,000 elements in 3D, and the solver uses a maximum of four levels, including the finest mesh.

THIN-FILM FLOW INTERFACES

A major upgrade has been performed on the Thin-Film Flow interfaces. New functionality has been added and errors in the weak form implementation, present in the previous versions, have been corrected. Correspondingly results may differ in comparison to previous versions. It is recommended that models from version 3.5a and earlier are rebuilt in the current version. The Thin-Film Flow user interfaces now have easier to understand names and new options for nonidentical slip coefficients at the wall and the base.

PHASE FIELD INTERFACE: TRANSIENT WITH INITIALIZATION STUDY

The 3D Phase Field user interface now includes a preset Transient with Initialization study in addition to the Time Dependent study.

New Models in Version 4.3a

- A new benchmark model, blasius_boundary_layer, models a Blasius boundary layer for a 2D laminar flow. The model investigates the effect of mesh refinement as well as changes in the basis function order, and the results are compared with a theoretical similarity solution.

- A new benchmark model for the Thin-Film Flow interfaces has been added (slider_bearing_1d). This model computes the flow and forces in a 1D slider bearing and compares the results with exact analytic solutions.

- A second new benchmark model for the Thin-Film Flow interfaces has been added (step_bearing_1d). This model computes the flow and forces in a 1D step bearing and compares the results with exact analytic solutions.

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 4.3a. 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 4.3a.

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 ε_p is the porosity.

REACTING FLOW INTERFACE NAME CHANGE

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

WEAK CONSTRAINTS UPDATE FOR FLUID FLOW INTERFACES

The weak 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-ε and Turbulent Flow, k-ω
- Symmetry
- The Wall functions boundary condition in the Wall feature
- The Moving wall (wall functions) boundary condition in the Wall feature
- The Slip boundary condition in the Wall feature

Turbulent Flow, low-Re k-ε and Turbulent Flow, Spalart-Allmaras
- Symmetry
- The Slip option in the Wall feature

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

Bubbly Flow
- Symmetry

- The Slip boundary condition for the liquid phase in the Wall feature
- 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-ε and turbulent flow k-ω only)

- The Moving wall (wall functions) boundary condition in the Wall feature (turbulent flow k-ε and turbulent flow k-ω 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-ε only)

- The Moving wall (wall functions) boundary condition in the Wall feature (turbulent flow k-ε 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_lm has been removed.

Models saved in version 4.3 can be opened and postprocessing will include un_lm until the model is re-solved. In some cases, occurrences of un_lm in the solver sequence must be replaced manually. This is the case if un_lm 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 4.3a to be possible to postprocess, or opened and re-saved in version 4.3 before opened in version 4.3a.

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 4.3a 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 4.3a

- A new Reacting Flow user interface: a multiphysics interface for mass transport in fluids. The Reacting Flow user interface combines the physics from Single Phase Flow and Transport of Concentrated Species and includes a unique reaction feature and pseudo time stepping for both the species and the momentum equations.

- Many input fields in the Reaction and Species nodes in the Reaction Engineering interface have been equipped with an Automatic/User Defined option. The User Defined option for these fields is in addition equipped with an *Reset to Default* button in order to generate the automatic expression, which can then be manually edited.

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 ε_p is the porosity.

REACTING FLOW INTERFACE NAME CHANGE

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

WEAK CONSTRAINTS UPDATE FOR FLUID FLOW INTERFACES

The weak constraint 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.

M O B I L I T Y

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 4.3a using Migration will have to be modified manually to account for these changes.

Corrosion Module

NEW PHYSICS FEATURE

- Film Resistance is now available in the Electrolyte-Electrode Boundary Interface and Electrolyte-Electrode Domain Interface features in all Electrochemistry physics (except Primary Current Distribution and Electrode, Shell).

GENERAL IMPROVEMENTS AND CORRECTIONS

- The governing equations for the Porous Electrode feature in Tertiary Current Distribution, Nernst-Planck have been corrected. The time derivative is now multiplied by the electrolyte volume fraction. Also the reaction sources of the Reaction feature are multiplied by the electrolyte volume fraction (that is, implying that the sources correspond to reactions taking place in the fluid phase only). Models created prior to 4.3a with nonunity electrolyte volume fractions will render different results compared to 4.3.

- In the Lithium Ion Battery and the Battery with Binary Electrolyte physics interfaces the transport number variable (`tplus`) is now correctly assigned to the value in the *Materials* node, if *From material* has been chosen for this parameter. The Batteries And Fuel Cells material library value for `tplus` for the three lithium-ion electrolytes has been changed to 0.363. A space-dependent `tplus` is now supported.

New Models

- New models of crevice corrosion: a tutorial model of crevice corrosion with electrode deformation (crevice_corrosion_with_deformation) and a verification model of crevice corrosion of iron in an acetic acid/sodium acetate solution (crevice_corrosion_fe).
- A new tutorial model of corrosion protection of an oil platform using sacrificial anodes (oil_platform).
- A new 1D verification model of a diffuse double layer (diffuse_double_layer).
- A new example model of a galvanized nail (galvanized_nail).

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 ε_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 4.3a using Migration will have to be modified manually to account for these changes.

ECAD Import Module

Overview

The new ECAD Import Module product extends the functionality of COMSOL Multiphysics by enabling modeling using imported ECAD design files.

The import capabilities cover the *GDS II*, *NETEX-G*, and *ODB++* (.xml) file formats and include the creation of 3D geometry during the import operation.

Compared to earlier ECAD import capability in the AC/DC Module, RF Module, and MEMS Module, the ECAD Import Module includes the following new functionality:

• Save and Load buttons have been added below the Layers table. The Save button saves that table to a text file. The Load button updates the user-editable fields in the table with information from the text file. Information in the loaded file that is not applicable is ignored.

• For GDS/NETEX-G files, when *Create selections* is enabled in the Selections of Resulting Entities section, in addition to the object/layer selections that were created in 4.3, selections corresponding to the cells in the imported file are also created. These selections are created for domains and faces.

• For GDS/NETEX-G files, *Net to import* is now a combo box (list) where you can select among the valid values for the given file.

Electrodeposition Module

New and Improved Functionality in Version 4.3a

- New Film Resistance in the Electrolyte-Electrode Boundary Interface and Electrolyte-Electrode Domain Interface features in all Electrochemistry physics (except Primary Current Distribution and Electrode, Shell)

- The Electrodeposition, Secondary and Electrodeposition, Tertiary Nernst-Planck physics interfaces now implement deformation of the geometry frame instead of the mesh frame. The name of the Electrodeposition, Moving Mesh branch in the Model Wizard has been changed accordingly to Electrodeposition, Deformed Geometry.

New Model

- A new model, Fountain Flow Effects on a Rotating Wire, explores the convective flow effects in a cell with a rotating wafer. This model requires the CFD Module.

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 ε_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 4.3a using Migration will have to be modified manually to account for these changes.

Fatigue Module

Overview

The Fatigue Module, new in version 4.3a, is intended for fatigue analysis of structures. The term fatigue is used to describe the phenomenon where a component fails after repeated loadings and unloadings, even though the magnitude of each individual load is smaller than the ultimate stress of the material. The Fatigue Module is an add-on module to the Structural Mechanics Module.

Using the Fatigue interface, you can compute the risk of fatigue cracks occurring in a structure where the stress and strain state during a single load cycle has been computed. One of the following physics interfaces—Solid Mechanics, Thermal Stress, Joule Heating and Thermal Expansion, or Piezoelectric Devices—is used along with the Fatigue interface to accomplish this analysis.

Two basic types of evaluation can be performed:

• Stress-based analysis for high-cycle fatigue (HCF). The result is a usage factor, which tells you how close to the fatigue limit the load cycle is.

• Strain-based analysis for low-cycle fatigue (LCF). The result is a lifetime prediction in terms of the number of cycles to fatigue.

When using the Fatigue Module in combination with the Structural Mechanics Module for fatigue analysis, the procedure begins with a stress analysis, followed by an evaluation against fatigue criteria as a separate results and analysis step. All load cycles are assumed to be equal and it is possible to handle non-proportional loading. Non-proportional loading occurs when the directions of the principal stresses and strains are not constant during the load cycle.

Geomechanics Module

- For soil plasticity, a new option to use dilatation angle in the plastic potential is available for the Drucker-Prager and Mohr-Coulomb plasticity models. Using this option, the dilatation angle replaces the angle of internal friction when defining the plastic potential.

Heat Transfer Module

New Functionality in Version 4.3a

MOVING FRAME SUPPORT

A new option, activated by default, is available when moving mesh is detected.

When it is active, the heat transfer features automatically account for deformation effects on heat transfer properties. In particular the effects for volume changes on the density are considered. Rotation effects on thermal conductivity of an anisotropic material and, more generally, deformation effects on arbitrary thermal conductivity are also covered.

IMPROVED PSEUDO TIME STEPPING

The pseudo time-stepping algorithm, used per default for turbulence models and for Non-Isothermal Flow and Conjugate Heat Transfer models, has been improved. The new implementation uses a PID regulator, which compared to the 4.3 implementation makes the convergence more robust without using additional iterations. The additional robustness reduces the need for manual tuning of the solvers.

As a result of this update, some old models may experience different convergence behavior compared to version 4.3.

SUPPORT FOR LOAD CASES

Loads and constraints can act conditionally in models using load groups and constraint groups, which you can include in load cases using various combinations of the defined groups.

Use load groups for heat sources and heat fluxes. The heat sources and heat fluxes can be scaled in the load cases using a scalar weight.

Use constraint groups for temperature conditions.

Load groups and constraint groups are available in the Heat Transfer in Solids, Heat Transfer in Fluids, Heat Transfer in Porous Media, Bioheat Transfer, Heat Transfer in Thin Shells, Conjugate Heat Transfer, Non-isothermal Flow, and Rarefied Flow physics.

ACCURATE ENTHALPY CALCULATION

A new integration operator is used to compute the enthalpy. This operator is faster and more accurate compared to the previous implementation. Accuracy gains are observed when the heat capacity is temperature dependent or when the density is pressure dependent.

The following variables benefit from this improvement:

- Internal energy
- Enthalpy
- Sensible enthalpy
- Convective heat flux
- Total energy flux
- Total heat flux

MOIST AIR

The Heat Transfer in Fluids feature now contains a new fluid type, *Moist air*. This fluid type provides thermodynamics properties of unsaturated humid air. It also provides dedicated postprocessing variables. In particular it is possible to verify if the saturation level has been reached during the simulation. This is useful to avoid water formation to prevent corrosion for example.

This new fluid type replaces the previously existing humid air material from the *Liquids and Gases* material library.

IMPROVED DEFAULT SETTINGS

The *Conjugate Heat Transfer* physics now contains a *Heat Transfer in Solid* and a *Fluid* feature. By default *Heat Transfer in Solid* is active in all domains and a *Fluid* has an editable empty selection.

In models containing surface to surface radiation, solid domains are considered as opaque: an *Opaque* subfeature (with a editable selection set to *All domains*) is now added by default under *Heat Transfer in Solid* and *Biological Tissue* features. Fluid domains are transparent by default.

The total heat flux is now defined as the default expression for vector plots.

Model inputs support is now available in all heat transfer features. It has been added to the *Thin Thermally Resistive Layer* and *Pair Thin Thermally Resistive Layer* features.

New Models in Version 4.3a

- Heating Circuit: this model includes DC-induced Joule heating, heat transfer, and structural mechanics analysis of the thin resistive layer covered on a solid glass plate.

- Buoyancy Flow in Water: this tutorial provides tools and explanations for successful modeling of natural convection flows.

- Composite Thermal Barrier: this model demonstrates the use of thin thermally resistive layer boundary condition to model efficiently very thin resistive structure.

- Phase change: this example demonstrates how to model a phase change and predict the impact of the latent heat on energy transport.

- Vacuum flask: this model treats the natural convection cooling using two approaches. The first approach is to use heat transfer coefficients to describe the thermal dissipation. The second approach is to model the convective flow of air outside the flask.

Backward Compatibility vs. 4.3

THE DEFAULT MODEL LIST HAS BEEN REMOVED

Heat transfer interfaces no longer have a default model list. This list box 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, see Moving Frame support for details.

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 *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 theses 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 INTERFACES

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

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

See the CFD Module Release Notes for details.

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

REVISION 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 4.3a 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.

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

MEMS Module

New Functionality in Version 4.3a

- A new Thermoelasticity user interface implements the equations of linear thermoelasticity in an easy-to-use interface.

- A major upgrade has been performed on the Thin-Film Flow interfaces. New functionality has been added and errors in the weak form implementation, present in the previous versions, have been corrected. Correspondingly results may differ in comparison to previous versions. It is recommended that models from version 3.5a and earlier are rebuilt in the current version. The Thin-Film Flow user interfaces now have easier to understand names and new options for nonidentical slip coefficients at the wall and the base.

- Modal Reduced Order Model study step added for obtaining equivalent circuit parameters from eigenvalue studies.

New Models in Version 4.3a

- A new model (thermoelastic_damping_3d) demonstrates how to use the Thermoelasticity user interface for modeling thermoelastic damping in a MEMS resonator using fully coupled thermal and structural equations.

Backward Compatibility vs. Version 3.5a

FILM DAMPING

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

Microfluidics Module

New Functionality in Version 4.3a

NEW MODELS
- A new model (split _recombine_mixer) computes the flow in a 3 dimensional mixer in the absence of physical diffusion. The results are compared with a benchmark paper from the literature.
- A new model (rotating_channel) computes fluid convection in a channel embedded within a spinning disk. The results are compared with a benchmark paper from the literature.

CHANGES TO THE FREE MOLECULAR FLOW AND TRANSITIONAL FLOW INTERFACES
Updates to the Free Molecular Flow and Transitional flow have produced minor changes in functionality that are not currently documented. The following changes have been made. In the Free Molecular Flow interface, integration groups are no longer supported. The dependent variables for Transitional flow are no longer displayed in the discretization section and can no longer be renamed.

PHASE FIELD INTERFACE: TRANSIENT WITH INITIALIZATION STUDY
The 3D Phase Field user interface now includes a preset Transient with Initialization study in addition to the Time Dependent study.

WEAK CONSTRAINTS UPDATE FOR FLUID FLOW INTERFACES
The weak constraint formulation for some boundary conditions in the following interfaces have been updated:

- Laminar Flow
- Creeping Flow
- Two-Phase Flow, Level Set
- Two-Phase Flow, Phase Field
- Two-Phase Flow Moving Mesh
  - Symmetry
  - The Slip option in the Wall feature
- Brinkman Equations

- Free and Porous Media Flow
- Slip Flow
  - Symmetry
  - The Slip option in the Wall feature

See CFD Module for further details.

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 ε_p is the porosity.

MOBILITY

In physics user interfaces modeling migration (Transport of Diluted Species) anisotropic mobility is now supported, and the mobility can be set to 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 4.3a using Migration will have to be modified manually to account for these changes.

Nonlinear Structural Materials Module

New Functionality in Version 4.3a

- Three new hyperelastic material models:
  - Yeoh
  - Varga
  - Blatz-Ko (for compressible materials like "rubber foam")

New Models in Version 4.3a

- Temperature-Dependent Plasticity in Pressure Vessel, which models a pressure vessel subjected to a rapid temperature change.
- Plastic Deformation in a Biomedical Stent, a model of the deformation of a stent subjected to a radial outward pressure using an elasto-plastic material model with linear hardening.

Optimization Module

New Functionality in Version 4.3a

An Optimization study for gradient-free (derivative-free) optimization can now be added directly from the Study node. The objective function and constraints are added in the study node. The Optimization solver collects the settings necessary for solving optimization problems based on model parameters. This solver allows parameters that control the geometry or mesh sequence, compared to the more limited impact of global constraint variables in the Optimization user interface. For gradient-free optimization, three new optimization methods are available:

- Nelder-Mead (the default)
- Monte Carlo
- Coordinate search

You can specify the objective and the control parameters directly in the Optimization study node's settings window, and it is also possible to select if the optimization minimizes or maximizes the values of the objective.

New Model in Version 4.3a

- An extension of the Tuning Fork model, the Shape Optimization of a Tuning Fork model (tuning_fork_optimization) shows how to set up an Optimization study to determine the prong length at which the fork vibrates at the standard concert pitch, 440 Hz.

Particle Tracing Module

New Functionality in Version 4.3a

- New boundary conditions are available for the following types of particle reflections:

  - Diffuse reflection.

  - General reflection, where you can supply your own expressions for the particle velocity after collision with a wall.

  - Pass through, which can be used on interior boundaries in conjunction with a sticking probability or expression.

- New variables have been added for the particle release time, particle stop time, and particle status. This improves the handling of the particle tracing when remeshing occurs. To activate these variables, set the *Store particle status data* option to *On*.

- In the Filter subnode for the Particle Trajectories plot, it is now possible to specify a logical expression for the particle to include.

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

- SCCM (standard cubic centimeter per minute) option for inflow makes it possible to specify a normal-volume based flow.

- It is now possible to apply internal temperature and pressure constraints, which is needed for circulating flow and other configurations.

- Y-junctions are now supported in addition to T-junctions. Y-junctions are useful for modeling nonperpendicular junctions.

Plasma Module

New Functionality in Version 4.3a

PLASMA MODELING

• An updated Electric Potential feature is available in the DC Discharge and Capacitively Coupled Plasma user interfaces. It contain quick choices for the following common external circuits:

 - None (the default)

 - Series RC circuit

 - Ballast resistor

 - Blocking capacitor

• A finite volume discretization is available for the DC Discharge and Capacitively Coupled Plasma user interfaces if you activate the Advanced Physics Options. Note that the following boundary conditions are not compatible with the finite volume discretization:

 - Terminal

 - Distributed Capacitance

 - Thin Low-Permittivity Gap

 - Floating Potential

UPDATED MODEL

The positive_column_2D model has been updated to use the updated Electric Potential feature.

RF Module

New Functionality in Version 4.3a

- A new general 2.5D formulation for waves where the out-of-plane wave number (azimuthal mode number) is greater than 0. This is useful for disk antenna modeling, accurate scattering modeling, laser (Gaussian) beam models, and cavity model analysis for accelerators, for example.

- The new Electomagnetic Waves, Time Explicit physics interface solves a system of two first-order partial differential equations (Faraday's law and Maxwell-Ampère's law) for the electric and magnetic fields using the *time explicit discontinuous Galerkin* method. The sources can be in the form of volumetric electric or magnetic currents or electric surface currents or fields on boundaries. It is used primarily to model electromagnetic wave propagation in linear media. Typical applications involve the transient propagation of electromagnetic pulses.

- Port for periodic structures in 2D models. This port feature makes it easier to model excitation of Floquet periodic structures and includes automatic setup of diffraction orders.

- Support for volumetric currents, making it possible to set a volumetric external current density in domains.

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 4.3a you need to click OK to close the error message, then clear the existing solutions, and finally re-solve the model.

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

Structural Mechanics Module

New Functionality in Version 4.3a

- In all structural mechanics user interfaces, it is now possible to give a harmonic perturbation to initial stresses and strains.
- Load and constraint groups, for load case modeling, for example, are now supported in the Membrane user interface.

New Models in Version 4.3a

- Contacting Rings, a benchmark model involving stick-slip friction of a ring rolling inside another ring.
- Oscillating FSI, a transient fluid-structure interaction simulation of a beam vibrating in a flow.
- Blasting Rock, a model with a transient analysis of the wave propagation in rock mass caused by a short duration load on the surface.

Backward Compatibility vs. Version 3.5a

PLANE STRESS

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

Subsurface Flow Module

Note: The Subsurface Flow Module replaced the Earth Science Module in version 4.2.

New Functionality in Version 4.3a

- A full set of edge conditions (in 3D) or point conditions (in 2D) for fracture flow are now available as subnodes for the Fracture Flow node used as a boundary condition in the Darcy's Law user interface.

- The Creeping Flow user interface is now included in the Subsurface Flow Module.

Updated Model

- The Pore Scale model has been improved. The second part of the model now calculates porosity and permeability from an imported image.

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 ε_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.

Material Library

- This version adds a number of material properties to the following materials (where the added properties appear within parentheses):

  - Tecamax (TC)
  - Manganin alloy 130 (dL/L, mte, density)
  - Silafont 36 (dL/L, mte, density, TD)
  - Si (emiss)
  - Ti (rho-liq)
  - Mo (SR)
  - Mo La2O3 ODS (SR)
  - Y2O3 (TC)
  - Y2O3-9La2O3 (dl/L, mte, density, E, SH)
  - Kevlar 49 (E, UTS)
  - 304L, 310 (e-N)
  - Ti-6Al-4V, Ti-4Al-2.5Sn (e-N)
  - Maraging 300 (e-N)
  - Ni (e-N)
  - Inconel 718 (e-N)
  - C10100 (OFHC) (e-N)
  - 2219, 2014 (e-N)
  - G-10CR and G-11CR (E, v, UTS)
  - Pyrolytic graphite (TC, rho, dL/L, mte, den, Cp, SH)
  - Several glasses (dL/L, mte, TC)
  - Ti-4Al-1V-3Mo (dL/L, mte, density)
  - 201, 201L, 321 (HE)
  - TZM (SN)
  - V2C, Mo2C, Ta2C, W2C (dL/L, mte)
  - 4340, 4130 (UTS, YS)

- AM50 (YS)
- Inconel MA 754 (CSS)
- 9Cr-1Mo, 9Cr-2Mo, 9Cr-3Mo (UTS, YS, % elong)
- 255 (E)
- 309, 304, 304L, DP 780 (SSC)
- CsCl (TC)
- Ti (SSC)

LiveLink for MATLAB

New Functionality in COMSOL 4.3a

The following new function is available:

- `mphinputmatrix`: Adds a linear matrix system to a 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 4.3a.

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 4.3a.

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 `flsave`, 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 4.3a with MATLAB.

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

GEOMETRY

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

COMSOL API

See the section "The COMSOL Java API" on page 55 regarding changes in the COMSOL API.

LiveLink for Excel

Overview

The new LiveLink™ *for* Excel®, which extends your COMSOL modeling environment with an interface between COMSOL Multiphysics and Microsoft Excel 2010. With LiveLink for Excel, you can use a spreadsheet to access model data in cell, such as definitions or results. You can edit the data and update the model to compute the solution. You can also access the Excel data directly from the COMSOL Desktop: import or export the model definition from, respectively to, an Excel spreadsheet.

A COMSOL 4.3a toolbar becomes available in the Excel menu, and on that toolbar you find utilities to operate to a COMSOL model.

From the Excel interface you can open a COMSOL model (saved in the MPH-format) in a COMSOL server. The first time you open a file, Excel starts a COMSOL server where the model is loaded. The data transfer between Excel and the COMSOL server is performed using a TCP/IP communication protocol.

Once the model is loaded in the COMSOL server you can import into Excel the model definitions such as model parameters, variables, and functions. The data name, expression, units, and description are listed in the Excel in separate cells. These cells can be edited as in any Excel spreadsheet. Data transfer from Excel to the COMSOL model is then done using a manual update operation.

The COMSOL Java API

COMSOL 4.3a Java 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

The fluid flow features listed in CFD Module and in Microfluidics Module previously created a scalar Lagrange multiplier field with default name `model.un_lm`. They now instead create a vectorial Lagrange multiplier field with default name `model.u_lm`. The default component names in 3D are `model.u_lm`, `model.v_lm` and `model.w_lm` respectively. Java files must be updated accordingly.

Weak constraints for the Interior Wall feature are no longer available. Any references to 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 Java API Changes

- In the Acoustics Module the far-field variables `pfar` and `Lp_far` get new names with full scope. They are now referred to as `acpr.ffc1.pfar` and `acpr.ffc1.Lp_far`, respectively. Any user Model Java-files that use the old variable

names in expressions (such as expressions to plot or evaluate expressions including these variables) will require a manual update.

- 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 Java 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 (`el`, `eel`, `Sl`, `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, `elX2` is replaced by `el22` and `elx2x3` is replaced by `el23`. The tensors si and ei have changed names to `Sil` and `eil`.

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

```
model.physics("dl").feature("smm1").set(chif_mat,userdef);
```

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

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

COMSOL 4.1 Java API Changes

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

- In version 4.0a the property `nonlin` in the stationary solver could have the values `auto`, `on`, `off`, and `linearized`. The default was in most cases `auto`, but in some cases, specifically in a frequency-domain study, the default was `linearized`. In version 4.1 the possible values are `auto`, `on`, `off`, and `linper`, with `auto` as default, except in the *Frequency Domain, Linearized* study step where the default is `linper`. The value `linearized` can still be set, but this is treated as `auto`. If the problem is linear and the linearization point is zero, `auto` should give the same solution as `linearized`. For nonlinear problems where the linearization point is zero, the solution given by `auto` might be slightly different, and probably more accurate, while changing to `off` should give exactly the same solution as `linearized`. When there is a nonzero linearization point it is relevant to use option `linper`. This option works like `linearized` except that source terms which are to contribute in the linearized problem must be enclosed by the `linper` operator.

- In this CFD Module, Subsurface Flow mode, in the Darcy's Law interface, Brinkman interface, Richards' Equation interface, and the Poroelasticity interface, the permeability and porosity are now material parameters and they do no longer have a default value. If the default value was used in a Java-file you now have to set the value. The following example sets the permeability to the old default value:

```
model.physics("dl").feature("dlm1").set(kappa_mat,userdef);
model.physics("dl").feature("dlm1").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`.