



# COMSOL Multiphysics®

*Release Notes*



VERSION 4.2a



## Benelux

COMSOL BV  
Röntgenlaan 37  
2719 DX Zoetermeer  
The Netherlands  
☎ +31 (0) 79 363 4230  
☎ +31 (0) 79 361 4212  
✉ info@comsol.nl  
🌐 www.comsol.nl

## Denmark

COMSOL A/S  
Diplomvej 381  
2800 Kgs. Lyngby  
☎ +45 88 70 82 00  
☎ +45 88 70 82 88  
✉ info@comsol.dk  
🌐 www.comsol.dk

## Finland

COMSOL OY  
Arabiankatu 12  
FIN-00560 Helsinki  
☎ +358 9 2510 400  
☎ +358 9 2510 4010  
✉ info@comsol.fi  
🌐 www.comsol.fi

## France

COMSOL France  
WTC, 5 pl. Robert Schuman  
F-38000 Grenoble  
☎ +33 (0)4 76 46 49 01  
☎ +33 (0)4 76 46 07 42  
✉ info@comsol.fr  
🌐 www.comsol.fr

## Germany

COMSOL Multiphysics GmbH  
Berliner Str. 4  
D-37073 Göttingen  
☎ +49-551-99721-0  
☎ +49-551-99721-29  
✉ info@comsol.de  
🌐 www.comsol.de

## India

COMSOL Multiphysics Pvt. Ltd.  
Esquire Centre,  
C-Block, 3rd Floor  
No. 9, M. G. Road  
Bangalore 560001  
Karnataka  
☎ +91-80-4092-3859  
☎ +91-80-4092-3856  
✉ info@comsol.co.in  
🌐 www.comsol.co.in

## Italy

COMSOL S.r.l.  
Via Vittorio Emanuele II, 22  
25122 Brescia  
☎ +39-030-3793800  
☎ +39-030-3793899  
✉ info.it@comsol.com  
🌐 www.comsol.it

## Norway

COMSOL AS  
Postboks 5673 Sluppen  
Søndre gate 7  
NO-7485 Trondheim  
☎ +47 73 84 24 00  
☎ +47 73 84 24 01  
✉ info@comsol.no  
🌐 www.comsol.no

## Sweden

COMSOL AB  
Tegnérsgatan 23  
SE-111 40 Stockholm  
☎ +46 8 412 95 00  
☎ +46 8 412 95 10  
✉ info@comsol.se  
🌐 www.comsol.se

## Switzerland

COMSOL Multiphysics GmbH  
Technoparkstrasse 1  
CH-8005 Zürich  
☎ +41 (0)44 445 2140  
☎ +41 (0)44 445 2141  
✉ info@ch.comsol.com  
🌐 www.ch.comsol.com

## United Kingdom

COMSOL Ltd.  
Broers Building  
21 JJ Thomson Avenue  
Cambridge CB3 0FA  
☎ +44-(0)-1223 451580  
☎ +44-(0)-1223 367361  
✉ info.uk@comsol.com  
🌐 www.uk.comsol.com

## United States

COMSOL, Inc.  
1 New England Executive Park  
Suite 350  
Burlington, MA 01803  
☎ +1-781-273-3322  
☎ +1-781-273-6603

COMSOL, Inc.  
10850 Wilshire Boulevard  
Suite 800  
Los Angeles, CA 90024  
☎ +1-310-441-4800  
☎ +1-310-441-0868

COMSOL, Inc.  
744 Cowper Street  
Palo Alto, CA 94301  
☎ +1-650-324-9935  
☎ +1-650-324-9936  
✉ info@comsol.com  
🌐 www.comsol.com

For a complete list of  
international representatives,  
visit  
🌐 [www.comsol.com/contact](http://www.comsol.com/contact)

## Home Page

🌐 [www.comsol.com](http://www.comsol.com)

## COMSOL User Forums

🌐 [www.comsol.com/  
community/forums](http://www.comsol.com/community/forums)

## COMSOL Multiphysics Release Notes

© 1998–2011 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 Desktop, COMSOL Multiphysics, and LiveLink are registered trademarks or trademarks of COMSOL AB. Other product or brand names are trademarks or registered trademarks of their respective holders.

Version:

October 2011

COMSOL 4.2a

Part No. CM010001



# Release Notes

COMSOL Multiphysics version 4.2a contains many new functions and additions to the COMSOL product suite. These *Release Notes* provide information regarding new functionality in existing products. It also outlines some additions that we intend to make in COMSOL Multiphysics version 4.3.

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

# COMSOL Multiphysics

## *New Products in Version 4.2a*

---

The following new products are introduced with COMSOL Multiphysics version 4.2:

- Particle Tracing Module: A new module for simulation of particle trajectories with specialized interfaces and plot types for particle tracing of charged particles in electric fields, mass particles in fluid flow, optical rays, and more. See [Particle Tracing Module](#) for more information.
- LiveLink for Creo™ Parametric: A new LiveLink for connecting COMSOL Multiphysics to the 3D CAD software Creo Parametric from PTC.

## *New Functionality in Version 4.2a*

---

### **GENERAL COMSOL DESKTOP FUNCTIONALITY**

- The Previous Node and Next Node buttons provide quick navigation between modeling steps in the Model Builder.
- You can now select multiple nodes in the model tree to, for example, quickly delete entire chunks of model definitions.
- A new Image function makes it possible to import images on PNG, BMP, GIF, and JPEG formats. You can use the image data for material distribution or to identify regions with different materials by their color or grayscale.
- A new Elevation (DEM) function for importing elevation data in the DEM format, which was created by the US Geological Survey. A DEM file contains elevation data for a portion of the Earth's surface. This can be used to define a function  $z = f(x,y)$ . The resulting function behaves essentially like a grid-based interpolation function. You can then combine these DEM surfaces with solids to form a volumetric representation of geometry and mesh.
- Improved Model Library search function: The search now uses all words in the Search field, but they do not need to appear together in a phrase. To search for an exact wording or phrase, enclose it in quotes.
- New material container variable `root.material` simplifies access to material data. For example, `root.material.rho` is the density  $\rho$  as defined by the materials in each domain in the geometry. For plotting, you can type the expression `material.rho` to create a plot that shows the density of all materials.

- New additional time-dependent solutions operators that you can use for results evaluation, for example:
  - `timeint()` for time integration
  - `timeavg()` for time averages
- New additional prestressed and small-signal analysis operators:
  - `lntotalavg()` evaluates the average of an expression over all phases for a linearized solution
  - `lntotalrms()` evaluates the RMS (root mean square) of an expression over all phases for a linearized solution
  - `lntotalpeak()` evaluates the maximum of an expression over all phases for a linearized solution

## GEOMETRY AND MESH

- It is now possible to interactively sketch 2D primitives on work planes directly in the 3D geometry. Activate by selecting the **Draw on work plane in 3D** check box. This setting is also available as a preference entry. Two new toolbar buttons provide Work Plane Clipping and Align with Work Plane functionality for simplifying geometry creation using work planes.
- The new Interpolation Curve feature in 2D and 3D creates interpolation curves based on look-up table data (entered as  $x$ ,  $y$ , and  $z$  values in a table or imported from file). The feature constructs a cubic spline that interpolates or approximates the given points.
- You can now include a geometry sequence from another model file. If the geometry sequence contains references to functions or parameters, those functions and parameters are also inserted into the model.
- Cut-and-paste geometry objects and operations in the Model Builder tree.
- Interactively stretch and shape 2D objects in the Graphics window.
- $N$ -to- $M$  mesh sweeping: That is, a swept mesh over a domain with  $N$  faces on the source side and  $M$  faces on the destination side. In general, it is required that the partitioning of the source (into faces) is a refinement of the partitioning on the destination. In particular, it is always true that  $N \geq M$ .
- The new copy mesh (Copy Domain, Copy Face, and Copy Edge in 3D; Copy Domain and Copy Edge in 2D) functionality makes it possible to copy the mesh from a meshed entity to an unmeshed entity. The transformation (for the mesh) must be a rigid body transformation, but a constant isotropic scaling and mirroring is allowed (that is, it is possible to copy from a large circle to a small but not from a

circle to an ellipse). If, for example, a boundary of the destination domain is already meshed, the mesh of the source must match at the corresponding boundary. The feature supports  $N$ -to-1 copy, meaning two adjacent half-spheres will copy to a complete sphere. The destination can be multiple; that is, the destination can be several spheres, each getting identical meshes.  $N$ -to- $N$  copy, meaning an array of cubes can be copied to a similar array of cubes.

### **PHYSICS INTERFACES**

- The PDE interfaces for equation-based models now support units. By declaring quantities for the dependent variables and the source terms (the overall equation), the PDE interfaces defines and displays units for all equation terms and quantities. This makes it possible to mix equation-based modeling with other physics interfaces and at the same time make full use of the unit system in the model (the PDE interface then introduces no unexpected or undefined units). You can switch off the unit handling for working with dimensionless quantities.
- Clear indication of the default nodes for all physics interfaces. In the Model Tree, a *D* in the upper-left corner indicates that the node is a default node.

### **STUDIES AND SOLVERS**

- Improved user interface for parametric sweeps:
  - Accumulated probe tables for numerical results from nested parametric sweeps
  - Response surface plots
  - New interface for “off-line” sweeps
  - Improved multiple-parameter sweeps
- Improved performance for time-dependent problems using new scaling.
- More robust automatic remeshing for time-dependent problems
- Possibility to avoid “complex pollution” (small nonzero imaginary components) by splitting complex variables into real and imaginary parts. You can declare field variables as complex or real when using the splitting of complex variables.
- Improvement to the least-squares solver:
  - Parameters and coordinates can be stored in files.
  - A symbolic gradient is available for stationary problems.

### **RESULTS**

- A new Join data set provides the possibility to do superposition and comparison of solutions from two different data sets. You can form combinations of solutions using

difference, sum, product, quotient, and more general and explicit expressions. For example, using the Join data set you can plot and evaluate the difference between two solutions in a mesh convergence study.

- A new Accumulated Probe Table feature enables a probe to write multiparameter data to tables. For example, the table can include the results from a nested parametric sweep with two independent parameters. From the table you can create a new Table Surface plot for plotting response surfaces, for example, and a new Table Graph for a graph plot of the results versus a parameter, for example.
- Interactive slice and isosurface plots, where you can move the slices and isosurfaces interactively in the Graphics window using a slider, provides easy positioning of the slices and isosurfaces.
- Multislice plots for easy generation of multiple slices in different directions.
- Import of external data to tables. The data can be spreadsheet data or other external data in common text-file formats. Importing external can be useful for analyzing and plotting experimental data, for example.
- For all Max/Min markers you can now select the color of the marker text in the **Coloring and Scale** section, including a custom color. This makes it possible to show a color that is more visible against the background than the default black color.
- Custom plot titles: The Title section for plots now provides a Custom setting for creating a customized plot title. When you select Custom you get a number of options for the typical components of a plot title: the data set (and its phase and solution when applicable) and the type, description, expression, and unit for the plotted quantity. You can also add a user-defined prefix and suffix.

### *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.

#### **NEW GEOMETRY FRAME**

The frame where the geometry keeps its original shape, as defined by the geometry sequence, is now called the *geometry frame*. The Deformed Geometry interface defines the material frame based on the geometry frame, which in turn differs from the mesh frame when using manual or automatic remeshing.

#### *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 4.2a*

---

- Matrix evaluation directly in tables for lumped parameters such as capacitance, inductance, impedance, and admittance.
- Small-signal analysis with automated differential inductance computations (also available for other lumped parameters).

## *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 4.1. A full set (3D, 2D and 2D axisymmetry) 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 4.1. Geometrical data and definitions (constants and expressions) will be imported and the physics can manually be added using the (vector potential based) Magnetic Fields physics interface.

### **PAIR BOUNDARY CONDITIONS**

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

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

# Acoustics Module

## *New Functionality in 4.2a*

---

### **NEW FLUID MODELS FOR PRESSURE ACOUSTICS**

- The Pressure Acoustics interface contains a number of new fluid models:
  - Macroscopic empirical porous models: In addition to the Delany-Bazley model, you can also choose the Miki model or create a user-defined model.
  - Thermally conducting fluid model
  - Thermally conducting and viscous fluid model
  - Biot equivalents using a porous model

### **THERMOACOUSTIC-SOLID INTERACTION INTERFACES**

New multiphysics interfaces for thermoacoustic-solid coupling in the frequency domain for 2D, 2D axisymmetric, and 3D models. The Thermoacoustic-Solid Interaction interfaces combine features from the Thermoacoustics and Solid Mechanics interfaces.

### **VOIGT NOTATION FOR ANISOTROPIC LINEAR ELASTIC MATERIALS**

Voigt notation is available for specification of orthotropic and anisotropic linear elastic materials in the Elastic Waves and Poroelastic waves interfaces, and is now used as the default notation.

## *Backward Compatibility vs. 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 4.2a*

---

- New Power input boundary condition.
- Improved infinite elements for easier modeling of unbounded domains.

# CAD Import Module and LiveLink Products for CAD

## *New Product in Version 4.2a*

---

- LiveLink for Creo™ Parametric. With this new product, COMSOL Multiphysics can be seamlessly integrated with the latest design software from PTC.

## *New Functionality in the CAD Import Module*

---

- The Parasolid® geometry kernel from Siemens PLM is now the default geometry kernel for users of the CAD Import Module and all CAD LiveLink products. The Parasolid kernel enables more advanced geometry operations and makes it possible to create complex shapes within the native COMSOL Multiphysics geometry modeling environment.
- Automatic scaling of geometry objects for handling of CAD models of vastly different length scales ranging from nanodevices to mountains and beyond.

## *New Functionality in LiveLink for SolidWorks®*

---

- Model Library access from the One-Window Interface.
- Animations and images from the One-Window Interface.
- Improved performance for large models.

## *New Functionality in LiveLink for SpaceClaim®*

---

- Improved performance for large models.

# CFD Module

*New Functionality in Version 4.2a*

---

## **NEW PHYSICS INTERFACES**

- The physics interfaces for turbulent flow now includes the  $k$ - $\omega$  turbulence model.
- A new Euler-Euler Model interface for modeling laminar flow of two continuous and fully interpenetrating phases.

## **SINGLE-PHASE FLOW**

- A new boundary condition, Interior Wall, in the Single-Phase Flow physics interfaces. It makes it possible to define a wall condition between two fluid domains. This is especially useful to model thin walls as boundaries. You no longer need to define a solid domain with a wall boundary condition on both sides, which can result in a dense mesh. This boundary condition is also available in the Fluid-Structure Interaction multiphysics interface (requires the Structural Mechanics Module or the MEMS Module).

# Chemical Reaction Engineering Module

## *New Functionality in Version 4.2a*

---

- Improved performance for the CHEMKIN® import.
- Improved parameter estimation using a new global least squares solver.
- New models:
  - Parameter estimation for nonideal reactor models
  - Microchannel H-cell
  - Stefan tube

# Electrodeposition Module

*New Functionality and Model in Version 4.2a*

---

- Improved infinite elements for easier modeling of unbounded domains.
- New model that simulates electrodeposition of a microconnector bump.

# Heat Transfer Module

## *New Functionality in Version 4.2a*

---

- Support for defining external radiation sources when surface-to-surface radiation is used. This option is available in the Heat Transfer physics interface and any physics interface that supports surface-to-surface radiation.
- It is now possible to define radiation on both sides of a boundary when surface-to-surface radiation is used. This option is available in the Heat Transfer physics interface and any physics interface that supports surface-to-surface radiation.
- A new option to constrain temperature on a boundary using a discontinuous Galerkin method. This option is especially useful to prevent oscillations on inlet boundaries where convection is the dominating heat transfer mechanism.
- A new boundary condition, Interior Wall, in the Single-Phase Flow physics interfaces. It makes it possible to define a wall condition between two fluid domains. This is especially useful to model thin walls as boundaries. You no longer need to define a solid domain with a wall boundary condition on both sides, which can result in a dense mesh.
- The convective cooling coefficient for external natural convection has been corrected.

## *Backward Compatibility vs. 3.5a*

---

### **K- $\omega$ TURBULENCE MODEL**

The Turbulent Flow, k- $\omega$  physics interface is part of the CFD Module in version 4.2a.

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

Momentum transport Wall Functions are translated into the revised Wall Functions in 4.2a. 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 4.2a*

---

- New Electromechanics multiphysics interface that combines solid mechanics and electrostatics with a moving mesh to model the deformation of electrostatically actuated structures. Applications include:
  - Biased resonator computations with model and frequency-response analysis
  - Pull-in voltage computations
- Improve and simplified modeling of thin-film damping using the new Thin-Film Damping. You can now add thin-film damping to a boundary directly in the Solid Mechanics interface. In a Fluid-Film Properties subnode you define the fluid properties, gas properties, and rarefaction effects. In a Border subnode you define the border condition: a pressure or a border flow. The Film-Damping Shell interface still exists for coupling film-damping and solid mechanics in the same way as in earlier versions using a multiphysics coupling.
- New electromechanical models: A suite of 2D and 3D models of a biased resonator showing how to model a stationary analysis, the frequency response, the normal modes, the pull-in voltage, and the transient response.

## *Backward Compatibility vs. Version 3.5a*

---

### **FILM DAMPING**

- Perforated plates are not yet available. These have to be modeled in version 4.2a using the full geometric description or using equation view.

# Microfluidics Module

*New Functionality in Version 4.2a*

---

## **MICROFLUIDICS**

- A new Slip Flow interface for modeling thermal and isothermal flows within the slip flow regime. The Slip Flow interface makes it possible to model the flow of the gas, including a thin layer of gas adjacent to the walls (Knudsen layer), where the gas is significantly rarefied. The Slip Flow interface is available in 2D and 3D.
- A new Transitional Flow interface, which you can use to model isothermal flows across the full range of Knudsen numbers from the laminar flow limit to the molecular flow limit. The Transitional Flow interface is available in 2D.

## **NEW MODELS**

- Knudsen's Minium: This model, using the Transitional Flow interface, shows that the flow rate of a rarefied gas between parallel plates exhibits a minimum (Knudsen's minimum) at a Knudsen number of about 1.
- Slip Flow Benchmark: A benchmark model of the flow between to sealed chambers connected by a microchannel with conducting walls. This model uses the Slip Flow interface.

# Particle Tracing Module

This module is new in version 4.2a. The Particle Tracing Module is a general purpose, flexible tool that allows you to trace the trajectories of particles in the presence of an external field.

The Particle Tracing Module includes the following physics interfaces:

- Mathematical Particle Tracing:
  - Specify the equations of motion using massless, Newtonian, Lagrangian, or Hamiltonian formulations.
  - Complete freedom over the equations makes it possible to model ray tracing, for example.
- Charged Particle Tracing
  - Use this interface to model ion and electron trajectories in electric and magnetic fields
  - This interface make it easy to define electric, magnetic, and collisional forces
- Particle Tracing for Fluid Flow
  - Use this interface to model microscopic and macroscopic particles in a fluid
  - Includes drag, gravitational, dielectrophoretic, acoustophoretic, and many other forces.

For visualization of the results two new plot types are available:

- Poincaré Map: You can use a Poincaré Map plot for 2D and 3D models to visualize particle trajectories in a section that is transversal to the particle trajectories.
- Phase Portrait: You can use a Phase Portrait plot for 2D models to visualize large data sets of particle trajectories. A phase portrait is a geometric representation of the particle trajectories in the phase plane.

All of the interfaces provide many different ways of releasing particles, either on the boundary or domain level. Particles can also be released at multiple different times. There are several options available for describing how the particles interact with the wall, including specular reflection. The interfaces can all be used with COMSOL Multiphysics but tracing charged particles benefits from the AC/DC Module because it makes it possible to model complex alternating or direct current systems. Similarly, tracing particles in a fluid system benefits from the CFD Module or the Microfluidics Module due to the number of advanced fluid flow features available in those modules.

# Plasma Module

*New Functionality in Version 4.2a*

---

## **PLASMA MODELING**

- It is now possible to compute the ion energy distribution function and angular distribution function (requires the Particle Tracing Module).
- Plotting of cycle-averaged quantities for capacitively coupled plasmas (CCP).

## **NEW MODEL**

- CCP benchmark model that reproduced benchmark results for a one-dimensional capacitively coupled plasma.

# RF Module

## *New Functionality in Version 4.2a*

---

- Port sweeps with S-parameter matrix display versus frequency in table format provides improved results analysis for models using port sweeps.

## *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 4.2a 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 4.2a*

---

- New functionality in the Shell interface:
  - Using the new offset property it is possible to model shells where the midsurface is offset from the location of the boundary in the COMSOL geometry.
  - Prestressed modal and frequency-response analysis
- Voigt notation for orthotropic and anisotropic linear elastic materials.
- Specification of linear elastic materials using Young's modulus and the shear modulus.
- Principal stress and principal strain quantities are now available as variables that you can use in models.
- The handling of geometric nonlinearities built into the study steps making geometrically nonlinear analyses more efficient and easier to use.
- Better organization of structural mechanics result quantities.

## *New Models in Version 4.2a*

---

- Vibration of an Impeller: A tutorial model that demonstrates the use of dynamic cyclic symmetry with postprocessing on the full geometry.
- Postbuckling Analysis of a Hinged Cylindrical Shell. This model shows how to trace a postbuckling path where neither the load nor the displacement increases monotonously.
- Polynomial Hyperelastic Model. This model shows how to implement a Mooney-Rivlin constitutive material model using a user-defined strain density energy.
- Sheet Metal Forming. This model shows plastic metal forming using a rigid punch, and the results are compared with experimental data.

## *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.

# LiveLink for MATLAB

## *New Functionality in COMSOL 4.2a*

---

The LiveLink for MATLAB® includes the following new functionality and performance improvements:

- New function `mphsolinfo` that returns information about a particular solution. This function supersedes the function `mphgetp`, which may be removed in a future version.
- The function `mphgetu` can now return more than one solution vector.
- Updated plots in `mphplot` and `mphviewselection` to accommodate new plot types and selection types.
- The **MATLAB** node includes additional functionality:
  - The **Minimize calls** option reduces the number of calls from COMSOL to MATLAB.
  - The new **Clear Functions** capabilities makes it easier to update M-files without having to restart MATLAB.
- Numerous optimizations for improved performance and memory handling.
- New model library containing examples that illustrate the LiveLink for MATLAB unique capabilities such as: extracting data at the MATLAB prompt, running models in nested loops, and using previous solution data and external MATLAB function callbacks in the COMSOL Desktop.

## *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*

---

### **GEOMETRY**

The functionality corresponding to the geometry functions `geom2` and `geom3` from version 3.5a is not available in version 4.2a. This functionality will be available in a later release.

**COMSOL API**

See the section “The COMSOL Java API” on page 24 regarding changes in the COMSOL API.

# The COMSOL Java API

## *COMSOL 4.2a Java API Changes*

---

No changes made.

## *COMSOL 4.2 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 (`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 Java API Changes*

---

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

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

```
model.physics("d1").feature("dlm1").set(kappa_mat,userdef);
model.physics("d1").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`.

# New Terminology in Version 4

The following tables include new terminology and names in version 4 and their corresponding equivalents in version 3.5a:

## GENERAL TERMINOLOGY

| NAME IN 4.2A       | NAME IN 3.5A                      | COMMENTS  |
|--------------------|-----------------------------------|---|
| Parameters         | Constants                         | Parameters in 4.0 have a broader scope, including the geometry. Also, predefined <i>physical constants</i> are new in version 4.  |
| Variables          | Expressions, Expression variables | Variables appear in two locations depending on their scope: under Global Definitions with a global scope or under Definitions in each model. In the latter case, scope can be set to the entire model or to selected domains, boundaries, edges, or points.                               |
| Physics interface  | Application mode                  |   |
| Coupling operators | Coupling variables                | You find the coupling operators under Model Couplings in the Definitions branches. A coupling operator, as opposed to a coupling variable, can be reused with different arguments. For example, the same coupling operator can be used to integrate different quantities over a boundary. |
| Hide               | Suppress                          | For hiding part of the geometry in the Graphics window.   |
| Study type         | Analysis type                     | The study type sets up a study step, which controls the generation of a solver sequence for performing a certain type of study.   |
| Results branch     | Postprocessing menu               | The Results branch contains the data sets for evaluation and visualizations of results as well as derived values, plot groups for various plots, tables for numerical results, model reports, and export of animations, data files, and images.   |
| Convert to         | Coerce to                         | For converting geometry objects from one type to another (for example, from a solid to a curve).  |
| Domain             | Subdomain                         | A solid geometry consists of one or several domains.  |

| NAME IN 4.2A                        | NAME IN 3.5A         | COMMENTS  |
|-------------------------------------|----------------------|---|
| Bidirectional, symmetric constraint | Ideal constraint     | In 4.2 available as a weak constraint and as a strong (pointwise) constraint. |
| Unidirectional constraint           | Non-ideal constraint | In 4.2 available as a weak constraint and as a strong (pointwise) constraint. |

#### PHYSICS INTERFACES WITH NEW NAMES

| NAME IN 4.2A                      | NAMES IN 3.5A  | COMMENTS  |
|-----------------------------------|--|---|
| Transport of Diluted Species      | Convection and Diffusion;<br>Diffusion;<br>Electrokinetic Flow; and<br>Nernst-Planck without Electroneutrality | The Transport of Diluted Species interface includes all transport mechanism for mass transport of diluted species. The functionality for migration through electrophoresis and electroosmosis is only available in the Chemical Reaction Engineering Module and MEMS Module.                    |
| Transport of Concentrated Species | Maxwell-Stefan Diffusion and Convection  | The Transport of Concentrated Species interface also includes two other diffusion models for mass transport of concentrated species: the mixture-averaged diffusion model and a diffusion model based on Fick's law.  |
| Laminar Flow                      | Incompressible Navier-Stokes; non-Newtonian Flow; General Laminar Flow; and Stokes Flow                        | The Laminar Flow interface includes all variants of laminar flow. Some of the functionality is only available in the Chemical Reaction Engineering Module, Subsurface Flow Module, Heat Transfer Module, and Microfluidics Module.  |
| Electric Currents                 | Conductive Media DC; Quasi-Statics, Electric   | The Electric Currents interface solves for the electric potential in applications where electric currents flow in a conductive media.   |
| Heat Transfer in Solids           | Conduction; General Heat Transfer  | The Heat Transfer in Solids interface provides default settings for conductive heat transfer in solids. But you can easily switch from heat transfer in solids to heat transfer in fluids as well as add and combine different heat transfer mechanism within the same Heat Transfer interface. |

| NAME IN 4.2A            | NAMES IN 3.5A   | COMMENTS   |
|-------------------------|---|--|
| Heat Transfer in Fluids | Convection and Conduction;<br>General Heat Transfer   | The Heat Transfer in Fluids interface provides default settings for convective and conductive heat transfer in fluids. But you can easily switch from heat transfer in fluids to heat transfer in solids as well as add and combine different heat transfer mechanism within the same Heat Transfer interface. |
| Solid Mechanics         | Solid,<br>Stress-Strain;<br>Axial Symmetry<br>Stress-Strain;<br>Plane Stress; and<br>Plane Strain | In 2D models you can switch between plane stress and plane strain without adding a new physics interface.  |
| Deformed Geometry       | Parameterized Geometry  | The Deformed Geometry interface is available for all geometries, not just 2D. In version 4 it is also possible to study fully parameterized geometries using parametric sweeps.  |

#### MATERIAL LIBRARIES

| NAME IN 4.2A                          | NAMES IN 3.5A                        | COMMENTS   |
|---------------------------------------|--------------------------------------|--|
| Built-in                              | Basic Material Properties            | Included in COMSOL Multiphysics.   |
| MEMS                                  | MEMS Material Properties             | Included with the MEMS Module, Acoustics Module, and Structural Mechanics Module.  |
| Piezoelectric                         | Piezoelectric Material Properties    | Included with the MEMS Module, Acoustics Module, and Structural Mechanics Module.  |
| AC/DC                                 | Electric (AC/DC) Material Properties | Included with the AC/DC Module. The Samarium Cobalt material is no longer included.  |
| Convective Cooling boundary condition | Heat Transfer Coefficients           | In version 4.1 the Convective Cooling boundary condition replaces the library of heat transfer coefficients. The Settings window for the Convective Cooling node contains predefined heat transfer coefficients for natural and forced convection (both internal and external).<br><br>Included with the Heat Transfer Module. |
| Liquids and Gases                     | Liquids and Gases                    | Included with the Chemical Reaction Engineering Module, Acoustics Module, Subsurface Flow Module, Heat Transfer Module, and MEMS Module.   |

| <b>NAME IN 4.2A</b>      | <b>NAMES IN 3.5A</b> | <b>COMMENTS</b>  |
|--------------------------|----------------------|--|
| Material Library         | Material Library     | The COMSOL Material Library add-on is unchanged.                       |
| Batteries and Fuel Cells | N/A                  | New material library available with the Batteries & Fuel Cells Module. |