

# COMSOL Multiphysics®

*Release Notes*



VERSION 4.1



**Benelux**  
COMSOL BV  
Röntgenlaan 19  
2719 DX Zoetermeer  
The Netherlands  
Phone: +31 (0) 79 363 4230  
Fax: +31 (0) 79 361 4212  
info@comsol.nl  
www.comsol.nl

**Denmark**  
COMSOL A/S  
Diplomvej 381  
2800 Kgs. Lyngby  
Phone: +45 88 70 82 00  
Fax: +45 88 70 80 90  
info@comsol.dk  
www.comsol.dk

**Finland**  
COMSOL OY  
Arabiankatu 12  
FIN-00560 Helsinki  
Phone: +358 9 2510 400  
Fax: +358 9 2510 4010  
info@comsol.fi  
www.comsol.fi

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

**Germany**  
COMSOL Multiphysics GmbH  
Berliner Str. 4  
D-37073 Göttingen  
Phone: +49-551-99721-0  
Fax: +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  
Phone: +91-80-4092-3859  
Fax: +91-80-4092-3856  
info@comsol.co.in

**Italy**  
COMSOL S.r.l.  
Via Vittorio Emanuele II, 22  
25122 Brescia  
Phone: +39-030-3793800  
Fax: +39-030-3793899  
info.it@comsol.com  
www.it.comsol.com

**Norway**  
COMSOL AS  
Postboks 5673 Sluppen  
Søndre gate 7  
NO-7485 Trondheim  
Phone: +47 73 84 24 00  
Fax: +47 73 84 24 01  
info@comsol.no  
www.comsol.no

**Sweden**  
COMSOL AB  
Tegnérgatan 23  
SE-111 40 Stockholm  
Phone: +46 8 412 95 00  
Fax: +46 8 412 95 10  
info@comsol.se  
www.comsol.se

**Switzerland**  
COMSOL Multiphysics GmbH  
Technoparkstrasse 1  
CH-8005 Zürich  
Phone: +41 (0)44 445 2140  
Fax: +41 (0)44 445 2141  
info@ch.comsol.com  
www.ch.comsol.com

**United Kingdom**  
COMSOL Ltd.  
UH Innovation Centre  
College Lane  
Hatfield  
Hertfordshire AL10 9AB  
Phone: +44-(0)-1707 636020  
Fax: +44-(0)-1707 284746  
info.uk@comsol.com  
www.uk.comsol.com

**United States**  
COMSOL, Inc.  
1 New England Executive Park  
Suite 350  
Burlington, MA 01803  
Phone: +1-781-273-3322  
Fax: +1-781-273-6603

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

COMSOL, Inc.  
744 Cowper Street  
Palo Alto, CA 94301  
Phone: +1-650-324-9935  
Fax: +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)

**Company home page**  
[www.comsol.com](http://www.comsol.com)

**COMSOL user forums**  
[www.comsol.com/support/forums](http://www.comsol.com/support/forums)

## *COMSOL Release Notes*

© COPYRIGHT 1998–2010 COMSOL AB.

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

The software described in this document is furnished under a license agreement. The software may be used or copied only under the terms of the license agreement. No part of this manual may be photocopied or reproduced in any form without prior written consent from COMSOL AB.

COMSOL and COMSOL Multiphysics are registered trademarks of COMSOL AB. LiveLink and COMSOL Desktop are trademarks of COMSOL AB..

Other product or brand names are trademarks or registered trademarks of their respective holders.

Version:                      October 2010                      COMSOL 4.1

Part number: CM010001

# Release Notes

COMSOL Multiphysics version 4 is in many aspects an entirely new product. This document outlines the differences between version 3.5a and version 4.1. It also outlines additions that we intend to make in COMSOL Multiphysics version 4.2.

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

# COMSOL Multiphysics

Version 4 introduces many new ideas and concepts to make it easier for you to create models. The following are the most important updates to the graphical user interface:

**New layout:** In version 4 it is easier to organize and design models with the intuitive structure of the *COMSOL Desktop*. The *Model Builder* displays all the features of your model in one place.

**Personalized desktop:** Control how you organize the *COMSOL Desktop* layout—your preferences are saved for the next time you open COMSOL Multiphysics.

**Sequences of operations:** You can now build custom model sequences to generate your geometry, physics, mesh, studies, and results. The sequences can be edited and changes are automatically updated across the model. Operations in the sequences can be modified by the parametric solver.

**Geometric parameter sweeps:** With geometry sequences COMSOL can perform geometric parameter sweeps with full associativity from the user interface.

**Material settings:** Materials are now administrated in one node in the Model Builder. You can select a material and its properties for each domain and for all physics in that domain.

**Multiple solutions and meshes:** With the new layout, you can save and view multiple solutions and meshes, and compare and contrast the results in the *Results* branch.

**Probes:** Use probes to visualize scalar quantities during computations. The quantities can be defined as integrals, max/min, the average of a field quantity, or the value at a point. This now works for time stepping and parameter sweeps.

**Dynamic help:** The new context-dependent help enables easy browsing with extended search functionality. The *Help Desk* provides access to the complete documentation set.

**Improved graphics:** Faster, better looking graphics.

**Model settings:** Access to the model settings is easier and more intuitive. When you select a node in the Model Builder, a docked window containing associated settings displays at the same time.

**Predefined selections:** Define selections of domains, boundaries, edges, and points. These predefined selections are available in the **Settings** windows for the physics interfaces, meshing, and when studying the results.

**LiveLink™ family of products for integrating with CAD and MATLAB®** The new LiveLink™ for SolidWorks®, LiveLink™ for Inventor®, and LiveLink™ for Pro/ENGINEER® products connect COMSOL Multiphysics directly with these CAD programs for interactively linking parameters specified in a CAD system with simulation geometry. In addition, LiveLink for MATLAB is available for incorporating COMSOL Multiphysics models in the MATLAB technical computing and programming environment.

**Cluster support** A floating network license for COMSOL Multiphysics can be extended at no additional cost to computational nodes for clusters on the Windows Cluster Server 2003, Windows HPC Server 2008, Windows HPC Server 2008 R2, and Linux platforms.

Version 4 also introduces:

- New and enhanced physics interfaces and predefined multiphysics interfaces
- New parallel solvers and higher solver performance:
  - A modal solver for frequency response and time domain (for structural and acoustics simulations, for example)
  - The MUMPS and SPOOLES direct solvers for parallel and cluster computing
  - A time discrete solver
  - A fast-frequency sweep solver using AWE (asymptotic waveform evaluation) for electromagnetic waves simulations, for example

### *New Functionality in Version 4.1*

---

#### **GENERAL COMSOL DESKTOP FUNCTIONALITY**

- Undo/redo of many operations in the **Model Builder** and the **Settings** window.



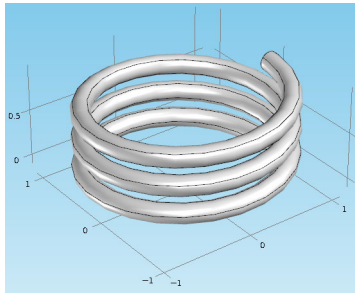
Undo and Redo in the Main Toolbar

- Copy-paste and duplication of selected nodes in the Model Tree, such as functions, selections, study steps, plot groups and plots, and images and data for export.
- Automatic save and recovery of models during solver operations.

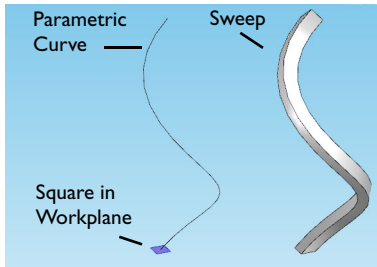
- The `dtang` operator is a complement to the `d` and `pd` differentiation operators. `dtang(f,x)` computes the derivative of `f` with respect to `x`, in the tangential direction along a boundary, when `f` is defined only on the boundary and therefore cannot be differentiated using the `d` operator.
- License borrowing is now available directly in the user interface. This functionality allows floating network license users to borrow a license and run COMSOL, for example, on a laptop or a home computer that is not connected to a network.

## GEOMETRY

- 3D Helix predefined geometric primitive.



- Parametric curves in 2D and 3D. Curves can now be described as functions of a parameter  $s$ .
- Geometry sweep, in 3D, based on parametric curves.



- Polygon nodes for easily creating polygons from coordinate data.
- Rendering of work planes in 3D.
- COMSOL geometries can now be converted to the Parasolid format in the CAD Import Module. This allows for the parameterization of geometries that combine

imported parts from CAD files and parts created using COMSOL geometry operations.

- Export to the Parasolid file format using the CAD Import Module is now supported for most COMSOL geometries, including objects created by extrude and revolve. You can then access geometries created in COMSOL in your CAD software.

## **MESH**

- Physics-controlled meshing, which automatically creates meshes that are adapted to the physics in the model. This is, for example, utilized by the Fluid Flow interfaces that create a somewhat finer mesh than the default and in addition create boundary layer meshes on no-slip boundaries.

For more information on how to use physics-controlled meshing for fluid flows, see “Physics-Controlled Meshing” in the *COMSOL Multiphysics User’s Guide*.

- Improved and more robust boundary layer meshing. The boundary-layer mesher can now handle cases where interior boundaries intercept the boundary layer.
- A deformed mesh can now be used as a geometry. Using the Moving Mesh interface, you are able to manually remesh in order to keep a high quality mesh as the geometry deforms.

## **PHYSICS INTERFACES**

- Display of equations in physics interfaces, which gives a compact and precise description of your physics settings.
- Support for additional element types in the PDE interfaces: Argyris, bubble, curl, divergence, discontinuous, Hermite, and Gauss point data elements.
- The PDE interfaces now support setting the field name and dependent variables independently. You can create a vector field, under one name, that encompasses several dependent variables.
- Additional functionality in the physics interfaces for each module, see the module sections in these release notes.

## STUDIES AND SOLVERS

- Improved usability in the **Study** branch:
  - Settings in the study steps and the corresponding settings in the solvers are synchronized by default.
  - The relative tolerance for the time-dependent solver can now be set in the study step.
  - The **Time discrete solver** is now enabled by adding a **Time Discrete** study step. This solver gives you full control of the time discretization and can be used, for example, in the projection method for fluid flow. To use this method for fluid flow, you also have to select the memory efficient form option in a **Fluid Flow** interface.
  - A computation is easily resumed by right-clicking on the study or solver nodes and selecting **Continue**.
  - Computations can also be started from any solver step by selecting **Compute From Selected** in the solver sequence.
  - Time-dependent simulations can be segregated for one-way coupled multiphysics models. The solver picks the value of the variable not solved at every time step during the solution of a second variable. This is available by selecting the **All** option for the variables not solved for in the **Dependent Variables** settings.
  - **Show default solver** creates a new active solver if the current solver has been edited. The old solver is kept in the tree but it is no longer active. If the study step or the physics have been changed, but the current solver has not been edited, the current solver is refreshed with the settings from the study step and physics. If a solver setting has been changed manually, an edit overlay is displayed on the corresponding solver.
- Scaling of eigenvectors can be done using mass scaling and maximum values, in addition to the root mean square (RMS). The mass scaling gives eigenvectors that can exchange with other software in a standard fashion. The scaling with maximum values visualize eigenmodes using the same range (max = 1).
- Mass participation factors are computed for eigenvalue/eigenfrequency studies. A mass participation factor for an eigenmode gives an estimate of how much vibrational energy is dissipated through an eigenmode in each direction.
- Linearized frequency-domain problems can be solved about a general stationary bias solution. This bias solution can be the solution to a nonlinear problem. A perturbation operator allows you to specify the load that should be applied in the frequency domain. In **Results**, you are able to visualize the load and the response to



the load in the frequency domain as well as the bias solution and combinations of the bias solution and the frequency response.

- The AMG preconditioner now support complex-valued problems. This allows for the use of memory efficient iterative solvers for problems solved in the frequency domain.
- The choice of damping factor in the automatic damped Newton method is improved. This yields a more robust solution process for nonlinear problems.
- Automatic and manual scaling can now be mixed for more robust solution of multiphysics problems.
- Explicit time stepping solver using the Runge-Kutta method is now available. This solution method may give a higher performance for problems that require very small time steps.

## RESULTS

- Labels have been added to contour plots.
- You can now display several plot windows that visualize a plot group. (The **Plot In** option in 4.0a only allowed several plot windows with snapshots of the graphics window.)
- Polar plots that displays graphs in polar coordinates specified by radius and angle.
- Frequency-domain transformation using FFT for computing frequency spectra from time-dependent simulations.
- Data sets for visualization of results on surfaces and edges.
- Data series operations for computing the maximum, minimum, average, or integral of, for example, the values of a quantity in a point over time for a time-dependent model.
- Export of table data to file.
- Syntax for special characters (Greek letters, mathematical symbols, and others) are now enabled in plots. Unicode characters can also be used and will then be printed with the current font. Special characters always use our symbol font.
- Transparency has been improved to correctly show several overlapping faces. This works for OpenGL rendering and software rendering.

#### **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 4.1. 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), which is new functionality in version 4.1.

In the version 4.1 interface, the *Linear Displacement* and *Similarity Transform* boundary conditions are not yet available as preset conditions. Those boundary conditions are planned for version 4.2.

In version 4.1, you can create the corresponding conditions by manually entering variables.

#### **PAIRS BOUNDARY CONDITIONS**

*Pairs* are used to connect boundaries between domains that are separated by an *assembly boundary* (boundary between different parts in an assembly). To create an assembly, you have to either import it from a CAD package or actively form an assembly as a final step in the COMSOL geometry sequence.

*Pair boundary conditions* are available as general conditions for all application modes in versions 3.5a. Most pair boundary conditions that were available in COMSOL 3.5a can be found in the physics interfaces in version 4.1. Exceptions from this are listed under backward compatibility for each product.

#### **PERIODIC PAIR BOUNDARY CONDITIONS**

*Periodic boundary conditions* are used to model repetitive structures, where one boundary in a domain is identical to another boundary in the same domain.

Periodic boundary conditions are available as general conditions for all application modes in version 3.5a. In version 4.1, this functionality is a tailored condition for each physics interface.

However, some physics interfaces may lack periodic boundary conditions in version 4.1; see the *Backwards Compatibility* section for the modules below.

Note that all physics interfaces will include periodic boundary conditions in the future.

### **AXISYMMETRIC MODELS**

In version 3.5a equations in axisymmetric application modes use the *independent variable* for the radius,  $r$ , to account for axisymmetry. In version 4.1, the equations are compensated by using the factor  $2\pi r$ .

If you have manually multiplied expressions by  $2\pi$  for a version 3.5a model, please note that these may be incorrect when the model is opened in version 4.1.

### **REPORT GENERATOR**

The *report generator* is not yet implemented in version 4.1. A report generator is planned for 4.2.

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

COMSOL 4.1 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 4.1*

---

- The Electric Currents physics interface contains the possibility to model electric currents in saturated porous media for fluids and solids with different conductivities and permittivities.
- Archie's law for porous media
- The formulation of the boundary conditions in the Magnetic and Electric fields interface has been updated to make them more user-friendly.

The following addition to the new functionality in COMSOL Multiphysics is of special interest for users of the AC/DC Module:

- Frequency-domain transformation using FFT for computing frequency spectra from time-dependent simulations.

## *New Functionality in Version 4*

---

This release of the AC/DC Module includes a number of new capabilities. Some of the most important ones are the following:

- *Electric and magnetic point dipoles* allow for easy and computationally efficient modeling of electromagnetic sources. Many coils and electrostatic applications can be approximated by one or more point dipoles.
- The *Transition* boundary condition for metallic layers of arbitrary thickness provides accurate modeling of geometrically thin metal sheets without having to mesh the thickness. This substantially helps to reduce the number of elements and the problem size. The layer can be of arbitrary electrical thickness (measured relative to the skin depth) and have a more or less pronounced discontinuity in the tangential electric field.
- The *Contact Resistance* boundary condition for efficient modeling of thin resistive sheets provides accurate modeling of geometrically thin highly resistive sheets without having to mesh the thickness. This substantially helps to reduce the number of elements and the problem size. The type of layer modeled using the contact resistance condition typically has a discontinuity in the normal electric field.
- Discontinuous interior boundary conditions like *electric insulation* for electric currents and *perfect magnetic conductor* for magnetic fields are now available.

- *Single-turn coil domain* and *multi-turn coil domain* modeling in 2D allow for easy modeling of coil domains in 2D and also support connection to electrical circuits (integrated with the new Electrical Circuit interface, described below). One or more geometrical domains can be assigned as voltage or current-driven coil. More than one geometrical domain can be treated as belonging to the same coil. This functionality replaces the use of manual model couplings in previous versions.
- The new *Electrical Circuit* physics interface with predefined passive and active components is integrated with all the other interfaces in the AC/DC Module via the terminal boundary condition and the single-turn and multi-turn coil domain functionality. It is now possible to build electrical circuits directly in the Model Builder. SPICE netlists are imported and translated into native and editable COMSOL circuit elements in the Model Builder.
- A new moving mesh interface for *rotating machinery* is now implemented in version 4.0 where the Moving Mesh (ALE) physics interface is much easier to use compared to previous versions. You can use this interface to implement new rotating machinery models. For backward compatibility for old rotating machinery models, see below.
- *Port/Terminal sweep* for frequency-dependent lumped parameter calculations in the graphical user interface makes it easy to compute full lumped parameter matrices. These can be exported in the text-based Touchstone file format.

### *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 4.1*

---

- The Acoustic-Structure Interaction interface now supports eigenfrequency studies.

## *New Functionality in Version 4*

---

- The definition of material properties has been reworked and extended, both for frequency and time domains.
- *Thin interior walls*—*hard*, with acceleration, impedance or perforations—can now be defined directly as boundary conditions. No modification to the geometry is necessary.
- The *Scattered field formulation* no longer makes any assumptions about the incident field. This makes it easier to model complicated situations with both fluid-fluid and fluid-solid interfaces.
- A new multiphysics interface for acoustic-structure interaction has been added. The coupling between fluids and solids is fully automated.

## *Backward Compatibility*

---

### **ULTRAWEAK FORMULATION**

The *ultraweak formulation* is not implemented in version 4.1 and is not planned for re-implementation in its previous form. The reason is that this formulation is very difficult to use without basis element order adaptivity (p-adaptivity).

# Batteries and Fuel Cells Module

## *New Functionality in 4.1*

---

- A new *Tertiary Current Distribution, Nernst-Planck* interface is introduced to model electrochemical reactions and ion transport in an electrolyte using the Nernst-Planck equations with an electroneutrality condition. This allows for the solution of charge balances in dilute electrolytes of arbitrary composition in cells with porous and non-porous electrodes.
- A new *Lead-Acid Battery* interface is introduced. The electric potential in the electrolyte and electrodes, the acid concentration, and the porosity in the electrodes, separators, and reservoirs in lead-acid batteries are modeled. This physics interface provides a state-of-the-art formulation for the modeling of lead-acid batteries.
- In the *Lithium-ion battery* and *Battery with Binary Electrolyte* interfaces *flakes* and *cylinders* shapes for the electrode insertion particles are introduced.
- The Electrochemistry interfaces support the *Frequency Domain, Linearized* study step, for running AC-impedance analysis.



# CAD Import Module and LiveLink Products for CAD

## *New Functionality in Version 4.1*

---

- COMSOL geometries can now be converted to the Parasolid format. This allows for the parameterization of geometries that combine imported parts from CAD files and parts created using COMSOL geometry operations.
- Export to the Parasolid file format is now supported for most COMSOL geometries, including objects created by extrude and revolve. You can then access geometries created in COMSOL in your CAD software.

## *New Functionality in Version 4*

---

- Support for reading SolidWorks, Pro/ENGINEER, and Inventor files now included.

# CFD Module

## *New Functionality in Version 4.1*

---

- A new *Spalart-Allmaras Turbulence Model* interface is available, which is suited for modeling wings and airfoils.
- Permeability and porosity can be obtained from the Material Library in the *Darcy's Law* and *Brinkman's Equation* interfaces.

## *New Functionality in Version 4*

---

This section describes new functionality and backward compatibility with respect to the implementations in the Chemical Engineering Module in COMSOL 3.5a.

- A new *Low-Reynolds Number k-ε Turbulence Model* physics interface is introduced, which yields high accuracy in the description of the flow close to walls. Mass and heat transport are also accurately described by this formulation.
- The improved *k-ε Turbulence Model* physics interface implementation gives greater robustness when the value of the turbulence intensity is small. The wall functions are also more accurate and require less input compared to previous versions.
- The new turbulence model formulations allow for *time-dependent simulations of turbulent flow*. You can run time-dependent simulations without having to start with a steady flow solution as initial condition.
- The option to use *pseudo time stepping* when computing stationary solutions has been added to the Single-Phase Flow physics interfaces (including Brinkman Equations and Free and Porous Media Flow). Pseudo-time stepping increases the robustness when solving highly nonlinear problems and reduces the need for solving in several steps.
- The *Rotating Machinery* multiphysics interface has been substantially improved. Using the *Rotating Domain* feature one or several rotating domains can be included in a model. The axis of rotation can easily be altered as well as the rotation direction and frequency. The *Rotating Wall* feature automatically adds the proper velocity constraints on boundaries of the Rotating Domain feature which do not correspond to the interface between the rotating and stationary domain.
- The possibility to include a *Forchheimer Drag* extension to the fluid flow resistance in porous media flow has been added.

- It is now possible to include a convective term in the Brinkman Equations. This means that porous media flow including higher fluid velocity can be studied. The inclusion of the term is controlled using the *Neglect inertial term (Stokes-Brinkman)* switch available in the Brinkman Equations and the Free and Porous Media physics interfaces.
- Stabilization has been included for the Brinkman equations. The stabilization provides increased robustness and less computational cost for a given accuracy compared to previous versions. It is also imperative when solving problems including the convective term.
- You can now add and remove species in the *Chemical Species Transport* physics interfaces. This means that you do not have to start with all species in your transport model. Instead you can add species one by one, thus reducing the risks for introducing errors.
- Two new diffusion models, a mixture-averaged diffusion model and a model based on Fick's law, are introduced in the *Transport of Concentrated Species* interface. These diffusion models are less computationally demanding than the Maxwell-Stefan diffusion model, and the latter model require less input data for the interaction between species in the mixture. The new models can be used in convectively dominated problems, where high accuracy in the diffusion interaction is not required, or when interaction data is not available.
- The improved stabilization for *Chemical Species Transport* in version 4.0 yields higher accuracy with a relatively coarse mesh compared to version 3.5a. This also results in increased robustness and less computational cost for a given accuracy compared to previous versions.
- A new *Open Boundary* condition is introduced in the physics interfaces for turbulent Single Phase Flow, Heat Transfer and Chemical Species Transport. The condition is designed to be used on model boundaries including both in and outflow sections. On sections with inflow the solution is not affected, while at occurrences of inflow a user defined exterior condition is enforced.
- A new physics interface for *Species Transport in Porous Media* accounts for the effect of the tortuous path in porous media. This path results in the additional dispersion perpendicular to the main flow of a transported species caused by the convective flux. The dispersion of species in porous media is thus more accurately described compared to previous implementations.
- A new physics interface for *Heat Transfer in Porous Media* can be used to accurately study heat transfer in porous catalysts, filters, and other unit operations involving porous media.

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

The new wall functions have the potential to deliver higher accuracy than the formulation used in 3.5a. They might, however, require finer wall resolution. Hence, a 3.5a turbulence model can often benefit from an additional boundary layer mesh or refined boundary layer mesh when imported into 4.1.

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

The *k- $\omega$  turbulence model* physics interface is not yet implemented in version 4.1. It is planned for the CFD Module in version 4.2.

Version 4.1 includes automatic translation of models built with the previous *k- $\omega$*  turbulence model. When opened, the full model, including initial values and boundary conditions, is converted to the *k- $\epsilon$*  turbulence model. Once opened, the model can also be also be changed to the *Low-Reynolds k- $\epsilon$  Turbulence Model* interface. The latter physics interface present an excellent alternative for higher accuracy in models including confined flows.

#### **PSEUDO APPLICATION MODES**

The *Pseudo* application modes for species transport in version 3.5a allow for the use of the dependent variable for time as a space coordinate in the direction of the flow.

The corresponding physics interfaces are not yet implemented in version 4.1. They are planned for a later version

Meanwhile, you can either create this alternative description manually, by relating time to a position along the length of the reactor using the axial velocity, or you can use a full 2D or 3D model.

#### **SPECIAL BASIS FUNCTIONS OR ELEMENTS**

None of the *special basis functions* or *elements* for the finite element formulation of flow problems included in version 3.5a are available in version 4.0. However, bubble elements and streamline diffusion are equivalent. Hence, old model that utilized bubble elements can be solved using streamline diffusion instead.

Other special elements that were available in 3.5a will not be re-implemented in version 4. The reason for this is that the stabilized formulation in version 4.1 gives high accuracy to a relatively small computational cost compared to the special elements.

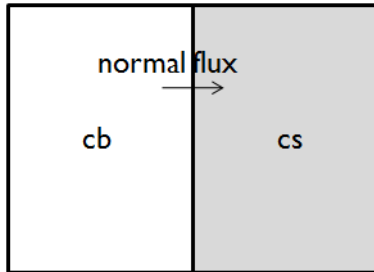
### THIN BOUNDARY LAYER PAIR BOUNDARY CONDITIONS

The thin boundary layer approximation approximates the mass flux perpendicular to an interface according to:

$$\mathbf{n}_i \cdot \mathbf{n} = -k(c_b - c_s)$$

where  $\mathbf{n}_i$  denotes the flux of species  $i$ ,  $\mathbf{n}$  the normal vector,  $c_s$  the surface concentration, and  $c_b$  the bulk concentration of species  $i$ .

In the case where  $c_s$  actually is a concentration in a separate domain, so that the interface between two domains requires a discontinuous concentration but a continuous flux, this condition could be defined in 3.5a using *pair boundary conditions*.



*Figure 1-1: Example of two domains with two separate dependent variables for chemical concentration.*

Version 3.5a models using this functionality are not automatically converted to version 4.1. However, you can convert these models manually in version 4.1 by using separate fields for the surface and bulk concentrations.

# Chemical Reaction Engineering Module

## *New Functionality in Version 4.1*

---

- Permeability and porosity can be obtained from the Material Library in the *Darcy's Law* and *Brinkman's Equation* interfaces.
- The variables defined by the *Reaction Engineering* interface do no longer require scopes. This makes it easier for the user to interpret and edit automatically generated expressions, for instance, describing reaction kinetics.

## *New Functionality in Version 4*

---

- You can now add and remove species in the *Chemical Species Transport* physics interfaces. This means that you do not have to start with all species in your transport model. Instead you can add species one by one, thus reducing the risks for introducing errors.
- Two new diffusion models, a mixture-averaged diffusion model and a model based on Fick's law, are introduced in the *Transport of Concentrated Species* interface. These diffusion models are less computationally demanding than the Maxwell-Stefan diffusion model, and the latter model require less input data for the interaction between species in the mixture. The new models can be used in convection dominated problems, where high accuracy in the diffusion interaction is not required, or when interaction data is not available.
- The improved stabilization for *Chemical Species Transport* in version 4 yields higher accuracy with a relatively coarse mesh compared to version 3.5a. This also results in increased robustness and less computational cost for a given accuracy compared to previous versions.
- The possibility to include a *Forchheimer Drag* extension to the fluid flow resistance in porous media flow has been added.
- It is now possible to include a convective term in the Brinkman Equations. This means that porous media flow including higher fluid velocity can be studied. The inclusion of the term is controlled using the *Neglect inertial term (Stokes-Brinkman)* switch available in the Brinkman Equations and the Free and Porous Media physics interfaces.

- Stabilization has been included for the Brinkman equations. The stabilization provides increased robustness and less computational cost for a given accuracy compared to previous versions. It is also imperative when solving problems including the convective term.
- A new *Open Boundary* condition is introduced in the physics interfaces for chemical species transport. The condition is designed to be used on model boundaries including both convective in and outflow sections. On sections with inflow the species concentration is not affected, while on sections with inflow a user defined exterior concentration is enforced.
- A new physics interface for *Species Transport in Porous Media* accounts for the effect of the tortuous path in porous media. This path results in the additional dispersion perpendicular to the main flow of a transported species caused by the convective flux. The dispersion of species in porous media is thus more accurately described compared to previous implementations.
- A new physics interface for *Heat Transfer in Porous Media* can be used to accurately study heat transfer in porous catalysts, filters, and other unit operations involving porous media.

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

---

#### **PSEUDO APPLICATION MODES**

The *Pseudo* application modes for species transport in version 3.5a allow for the use of the dependent variable for time as a space coordinate in the direction of the flow.

The corresponding physics interfaces are not yet implemented in version 4.1. They are planned for a later version

Meanwhile, you can either create this alternative description manually, by relating time to a position along the length of the reactor using the axial velocity, or you can use a full 2D or 3D model.

#### **SPECIAL BASIS FUNCTIONS OR ELEMENTS**

None of the *special basis functions* or *elements* for the finite element formulation of flow problems included in version 3.5a are available in version 4.0. However, bubble elements and streamline diffusion are equivalent. Hence, old model that utilized bubble elements can be solved using streamline diffusion instead.

Other special elements that were available in 3.5a will not be re-implemented in version 4. The reason for this is that the stabilized formulation in version 4.1 gives high accuracy to a relatively small computational cost compared to the special elements.

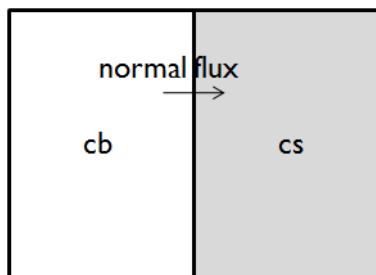
#### THIN BOUNDARY LAYER PAIR BOUNDARY CONDITIONS

The thin boundary layer approximation approximates the mass flux perpendicular to an interface according to:

$$\mathbf{n}_i \cdot \mathbf{n} = -k(c_b - c_s)$$

where  $\mathbf{n}_i$  denotes the flux of species  $i$ ,  $\mathbf{n}$  the normal vector,  $c_s$  the surface concentration, and  $c_b$  the bulk concentration of species  $i$ .

In the case where  $c_s$  actually is a concentration in a separate domain, so that the interface between two domains requires a discontinuous concentration but a continuous flux, this condition could be defined in 3.5a using *pair boundary conditions*.



*Figure 1-2: Example of two domains with two separate dependent variables for chemical concentration.*

Version 3.5a models using this functionality are not automatically converted to version 4.1. However, you can convert these models manually in version 4.1 by using separate fields for the surface and bulk concentrations.

#### EQUILIBRIUM REACTIONS AND MASS TRANSPORT

The equilibrium reaction assumptions defined by entering in the Reaction Engineering interface does not automatically generate space-dependent formulations. This functionality will be implemented in a later version.



# Earth Science Module

## *New Functionality in Version 4.1*

---

- Permeability and porosity can be obtained from the Material Library in all interfaces involving porous media flow: *Darcy's Law*, *Richards Equation*, *Brinkman Equations*, *Fracture Flow*, *Free and Porous Media Flow*, and *Poroelasticity* interfaces.

## *New Functionality in Version 4*

---

- The physics interface for *Heat Transfer in Porous Media* now accounts for multiple immobile solids and fluids. There are two choices for averaging the thermal conductivity: volume average and a power law formulation.
- The physics interface for *Heat Transfer in Porous Media* also incorporates options for describing geothermal heating and thermal dispersion.
- The previous three separate formulations of *Darcy's law* (pressure, pressure head, and hydraulic head) are available in one framework in version 4.1. This makes it possible to mix boundary and initial conditions using any of these formulations. The *Porous Media and Subsurface Flow* physics interfaces are also able to specify isotropic, orthotropic or anisotropic permeabilities or hydraulic conductivities.
- The physics interface for *Poroelasticity* includes a poroelastic material model to account for isotropic poroelastic materials. The material properties can be written in terms of bulk and shear modulus, Young's modulus and Poisson ratio, Lamé constants, or pressure and shear velocities. Orthotropic and fully anisotropic materials can be modeled if used together to the SME module.
- The improved stabilization for *Chemical Species Transport* in version 4 yields higher accuracy with a relatively coarse mesh compared to version 3.5a. This also results in increased robustness and less computational cost for a given accuracy compared to previous versions. You can also add and remove species freely in the *Chemical Species Transport* physics interfaces.
- A new physics interface for *Solute Transport* incorporates the old application modes for solute transport in saturated porous media and variable saturated porous media in one physics interface.
- A new physics interface for *Fracture Flow* enables the solution of Darcy's Law in thin shells and fractures, without having to mesh along the thickness of a thin layer.

This substantially improves performance for a given accuracy compared to previous versions.

- You can specify the *unit of time* in days and years in the time-dependent solver. This improves the usability for geotechnical processes compared to the use of seconds.
- *Darcy's Law* can now handle variable densities.
- The possibility to include a *Forchheimer Drag* extension to the fluid flow resistance in porous media flow has been added.
- It is now possible to include a convective term in the Brinkman Equations. This means that porous media flow including higher fluid velocity can be studied. The inclusion of the term is controlled using the *Neglect inertial term (Stokes-Brinkman)* switch available in the Brinkman Equations and the Free and Porous Media physics interfaces.
- Stabilization has been included for the Brinkman equations. The stabilization provides increased robustness and less computational cost for a given accuracy compared to previous versions. It is also imperative when solving problems including the convective term.

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

---

#### **SPECIAL BASIS FUNCTIONS OR ELEMENTS**

None of the *special basis functions* or *elements* for the finite element formulation of flow problems included in version 3.5a are available in version 4.0. However, bubble elements and streamline diffusion are equivalent. Hence, old model that utilized bubble elements can be solved using streamline diffusion instead.

Other special elements that were available in 3.5a will not be re-implemented in version 4. The reason for this is that the stabilized formulation in version 4.1 gives high accuracy to a relatively small computational cost compared to the special elements.

# Heat Transfer Module

## *New Functionality in Version 4.1*

---

- Surface-to-surface radiation and radiation in participating media are integrated in the main *Heat Transfer* interface for easier coupling to other heat transfer mechanisms.
- New *Fan Boundary Condition* for using fan curves at inlets in fluid-flow models.

## *New Functionality in Version 4*

---

- A new *Conjugate Heat Transfer* multiphysics interface (fluid flow, fluid heat transfer, and solid heat transfer) for modeling electronic cooling and heat sinks, for example.
- A new physics interface for *Heat Transfer in Porous Media* accounts for the effect of a tortuous path, which results in the additional convective thermal dispersion perpendicular to the main flow. The temperature field is thus more accurately described compared to previous implementations.
- A new physics interface for *Radiation in Participating Media* introduces the effect of absorption of energy in a media subjected to radiation. The radiation pattern from one surface to another then depends on the media present between the radiating or reflecting surfaces.
- The new physics interface for *Low-Reynolds Number  $k$ - $\epsilon$  Turbulence Model* yields a higher accuracy in the description of the flow and heat transfer close to walls.
- The new *Open Boundary* condition is useful for modeling an open boundary where heat can flow out from the domain or into the domain with a specified exterior temperature.
- Improved stabilization yields increased robustness and higher accuracy for a given computational cost compared to previous versions.

## *Backward Compatibility vs. 3.5a*

---

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

The new wall functions have the potential to deliver higher accuracy than the formulation used in 3.5a. They might, however, require finer wall resolution. Hence,

a 3.5a turbulence model can often benefit from an additional boundary layer mesh or refined boundary layer mesh when imported into 4.1.

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

The *k- $\omega$  turbulence model* physics interface is not yet implemented in version 4.1. It is planned for the CFD Module in version 4.2.

Version 4.1 includes automatic translation of models built with the previous *k- $\omega$*  turbulence model. When opened, the full model, including initial values and boundary conditions, is converted to the *k- $\epsilon$*  turbulence model. Once opened, the model can also be changed to the *Low-Reynolds k- $\epsilon$  Turbulence Model* interface. The latter physics interface presents an excellent alternative for higher accuracy in models including confined flows.

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

Highly conductive layers on the interface between a solid and a fluid are not yet fully implemented and give incorrect results when used in combination with wall functions in turbulent flow. This is caused by the incorrect definition of the wall temperature as the average of the solid and fluid temperature in these physics interfaces.

In some cases, you can correct this in equation view. Please contact technical support if you run into problems with such a model.

Momentum transport Wall Functions are translated into the revised Wall Functions in 4.1. The Heat equation Wall Functions are also translated, but the translated formulation uses the default turbulence modeling constants, regardless of the values that were defined in **Physics>Scalar Variables** in 3.5a.

#### **SURFACE-TO-SURFACE RADIATION IN COMBINATION WITH TURBULENT FLOW AND HEAT TRANSFER**

It is currently not possible to include surface-to-surface radiation in models using the *k- $\epsilon$  Turbulence Model* and heat transfer interfaces. The reason for this is that the wall temperature is incorrectly defined as the average of the fluid temperature and the solid temperature in version 4.1. However, it is possible to use the *Low-Reynolds k- $\epsilon$  Turbulence Model* and the *Spalart-Allmaras turbulence model* with surface-to-surface radiation.

# MEMS Module

## *New Functionality in Version 4.1*

---

- Automatic generation of coordinate systems for 2D and 2D-axisymmetric *Piezoelectric Devices interface*. This simplifies the orientation of anisotropic piezoelectric material properties.
- It is now possible to specify the remanent electric displacement in the *Piezoelectric Devices interface*.

## *New Functionality in Version 4*

---

- New physics interfaces are added for *Thin-Film Flow* and *Thin-Film Gas Flow* (using a conventional and modified Reynolds equation, respectively). These allow you to account for the influence of thin squeezed films that also include a substantial effect from a flow that runs in parallel to the squeezing surfaces in 2D models.
- A new *Lubrication Shell* interface for modeling lubrication by thin fluid films in narrow geometries that can be represented as shells. This gives high accuracy at a much smaller computational cost compared to full fluid-structure interaction in 3D with moving boundaries.
- Version 4 also includes an improved physics interface for *Piezoelectric Devices*. This new interface gives full access to the functionality of the structural mechanics and electrostatics physics interfaces in addition to the piezoelectric material model.

## *Backward Compatibility*

---

### **PIEZOELECTRIC DEVICES INTERFACE**

The following 3.5a functionality is not available in version 4.1:

- Electrical formulations: Version 4.1 does not support the electrical formulation property from 3.5a. The current Piezoelectric Devices and Electrostatics interfaces have the same electrical formulation, which corresponds to the 3.5a Electrostatic Symmetric formulation. When you open an old model, it will be converted to the Electrostatic Symmetric formulation.

#### **FILM DAMPING**

- The solid predefined coupling is missing but can be performed manually
- *Perforated plates* are not yet available. These have to be modeled in version 4.1 using the full geometric description or using equation view.

# Plasma Module

## *New Functionality in Version 4.1*

---

- *Surface Chemistry* is now available including surface species and bulk species that only exist on boundaries.
- The *Coil Group* feature makes it possible to group domains together to form a connected coil. The coil can be driven via either a total current, a total voltage drop, a fixed power, or an external circuit. This is available in the 2D and 2D axisymmetric Inductively Coupled Plasma interface (requires the AC/DC Module).
- The *tensor form* of the ion transport properties are now automatically computed when a static magnetic field is present.
- Plasma domains are now automatically meshed using *boundary-layer meshing*.
- The mean electron energy can be fixed allowing for calculation of approximate solutions very rapidly.

# RF Module

## *New Functionality in Version 4.1*

---

The following additions to the new functionality in COMSOL Multiphysics are of special interest for users of the RF Module:

- *Polar plots* that displays graphs in polar coordinates specified by radius and angle.
- Frequency-domain transformation using *FFT* for computing frequency spectra from time-dependent simulations.

## *New Functionality in Version 4*

---

- *Electric and magnetic point dipoles* allow for easy and computationally efficient incorporation of electromagnetic sources in electromagnetic wave propagation models. Many antennas can be approximated by one or more point dipoles.
- A *Transition boundary condition* for metallic layers of arbitrary thickness provides accurate modeling of geometrically thin metal sheets without having to mesh the thickness. This substantially helps to reduce the number of elements and the problem size. The layer can be of arbitrary electrical thickness (measured relative to the skin depth) and have a more or less pronounced discontinuity in the tangential electric field.
- The discontinuous interior boundary condition *perfect magnetic conductor* is now available.
- A new variational *S-parameter formulation* supports multimode ports and numerical hybrid-mode ports. This allows for accurate modeling of dielectrically loaded waveguides and optical waveguides.
- *Port sweep* for frequency-dependent S-parameter calculations in the graphical user interface makes it easy to compute full S-parameter matrices. These can be exported in the Touchstone file format.
- The new *Electrical Circuit* physics interface with predefined passive and active components is integrated with other physics interfaces in the RF Module via the port and lumped port boundary conditions. It is now possible to build electrical circuits directly in the Model Builder. SPICE netlists are imported and translated into native and editable COMSOL circuit elements in the Model Builder.



## *Backward Compatibility vs. Version 4.0a*

---

### **OVERRIDDEN PORT VARIABLES REMOVED FROM SOLUTION COMPONENTS**

Overridden port features no longer generate solution components in 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. In order to successfully complete the import of such a model into version 4.1 you need to click OK to close the error message, then clear the existing solutions, and finally re-solve the model.

Java 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.

## *Backward Compatibility vs. Version 3.5a*

---

### **CHANGE IN DEPENDENT VARIABLES**

Version 4.1 has a comprehensive set of electromagnetic wave propagation formulations based on the electric field that covers all modeling situations.

However, it is no longer possible to solve for the *magnetic field*  $H$  as a dependent variable.

Models saved in version 3.5a that are based on an  $H$  formulation are translated to the new  $E$  formulation but need to be solved again to update the solution in version 4.1.

### **FAR-FIELD POSTPROCESSING**

*Far fields* are not automatically read in when importing a 3.5a model into version 4.1. The workaround is to manually add a Far Field Calculation node.

### **PAIR BOUNDARY CONDITIONS**

*Pair boundary conditions* are not yet implemented, except for some basic features. This means that the assembly port boundary condition is not supported in version 4.1.

# Structural Mechanics Module

## *New Functionality in Version 4.1*

---

- New *Tresca stress yield function* for elastoplastic materials.
- *Rigid connector* boundary condition in 2D.
- Accuracy of shell elements has been improved.
- Boundary conditions for *prescribed velocity and acceleration* in the Shell, Plate, Beam, and Truss interfaces.
- Improved support for *cross-section orientation* in beams. It is now possible to use different orientations for the same cross section. You can also specify the orientation as a direction vector in addition to the previous method with a control point.

## *New Functionality in Version 4*

---

- A flexible definition of *elastic material data* enables the solid mechanics interfaces to support bulk modulus and shear modulus, Lamé constants, and pressure-wave and shear-wave speed in addition to Young's modulus and Poisson's ratio.
- Custom *hyperelastic material models* can be created by specifying a strain energy function in terms of deformation tensor components and invariants.
- A flexible *2D solid mechanics* interface allows you to switch between plane stress and plane strain approximations at any point during modeling. For example, it is now easy to compare the results with plane strain and plane stress assumptions.
- *Prescribed velocity and acceleration* boundary conditions are now available also in transient analysis. Previously, these conditions were available only in the frequency domain.
- A new *rigid connector* boundary condition (only in 3D in version 4.0) makes it possible to apply kinematic constraints, forces, and moments on boundaries assumed to displace as a rigid body.
- A new *3D shell and 2D plate element formulations* based on first-order or second-order Lagrange shape functions on both triangles and quadrilaterals gives improved accuracy. Curved surfaces and mixed solid-shell models, in particular, are improved. In addition, the new shell elements are fully multiphysics-enabled and support large deformations and anisotropic materials.

- Frequency-domain and time-domain *modal solvers* let you run fast frequency sweeps and transient response analyses on reduced models.
- The *Elastoplastic Material Model* now allows the use of a non associated flow rule.

### *Backward Compatibility*

---

#### **CROSS-SECTION LIBRARY**

The *cross-section library* for beams is not available in version 4.1. This is planned for version 4.2.

#### **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.

#### **SYMBOLS**

*Symbols* in the graphics window representing constraints and loads are not yet available in version 4.1. Symbols will be available in version 4.2.

#### **DIRECT SOLVER USED IN 3D BY DEFAULT**

Version 4.1 reintroduces the direct solver MUMPS as default linear solver in 3D.

# LiveLink for MATLAB

## *New Functionality in COMSOL 4.1*

---

The functionality that has been added to COMSOL Multiphysics with add-on products are also by default available in the LiveLink for MATLAB.

- Helix geometry operation.
- Sweep geometry operation.
- Utility functions for selections: `mphselectcoords` and `mphselectbox` with related geometry utilities `mphgetcoords` and `mphgetadj`.

## *New Functionality in Version 4*

---

The model object that contains the true representation of the model is available directly from the MATLAB® prompt. This means that all functionality available from the COMSOL Desktop becomes automatically available at the MATLAB prompt, and that the model behaves exactly the same from the MATLAB prompt as it does in the COMSOL Desktop. In the future this will also mean that you will be able to see changes to the model object that have been made in MATLAB, and vice versa.

The interface to MATLAB uses an entirely new syntax for the model. In order to help you convert models to the new syntax, two conversion tools are available:

- The function `mphv4`
- The ability to load 3.5a models in the V4 GUI and then to save them as M-files

These conversion tools are limited to converting the `fem` structure to the new syntax; the rest of the M-files that you have written for 3.5a need to be manually converted.

The ability to start COMSOL with MATLAB in a 3.5a compatibility mode is intended to help you in the process of working with both the old and new syntax.

## *Parameterized Geometries*

---

Version 4 includes built-in support for parameterized geometries. There should be no need to change boundary condition during parameterization; thus the `geomanlyze` function is no longer necessary. Parameterization can be performed both with the built-in parametric solver and by wrapping a for-loop around the solver call.

## *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*

---

### **GEOMETRY**

The functionality corresponding to the following geometry functions from version 3.5a is not available in version 4.1. Most of this functionality will be available in version 4.2.

- `flim2curve`, `flcontour2mesh`, and `flmesh2spline`
- `geomsurf`
- `loft` (partially available as `Sweep`)
- `arc`
- `geom0`, `geom1`, `geom2`, and `geom3`
- `geomedit`
- `geomfile`
- `gencyl`
- The `paircand` property of `geomgroup`

### **MESH**

The functionality corresponding to the following mesh functions from version 3.5a is not available in version 4.1. Most of this functionality will be available in version 4.2.

- `meshembed`
- `meshextrude` and `meshrevolve` (essentially replaced by generalized sweep functionality)
- The `hmesh` and `mesh` properties of `meshinit`

### **COMSOL API**

See the section “The COMSOL API” on page 38 regarding changes in the COMSOL API.

# The COMSOL API

## *COMSOL 4.1 API Changes*

---

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

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

```
model.physics("d1").feature("d1m1").set(kappa_mat,userdef);
model.physics("d1").feature("d1m1").set(kappa,3e-11);
```
- The pressure variable in solid mechanics, typically `solid.pw`, now only gets allocated degrees of freedom for incompressible materials. If you have referred to it, for example, during segregation in the solver, the code will have to be changed to remove the field.
- The solution to eigenvalue and eigenfrequency problems may now appear in a different order.

## COMSOL 4.0a 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, e.g., `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 4.0	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 4.0.
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, and reports such as 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 4.0	NAME IN 3.5A	COMMENTS
Bidirectional, symmetric constraint	Ideal constraint	In 4.0 available as a weak constraint and as a strong (pointwise) constraint.
Unidirectional constraint	Non-ideal constraint	In 4.0 available as a weak constraint and as a strong (pointwise) constraint.

## PHYSICS INTERFACES WITH NEW NAMES

NAME IN 4.0	NAMES IN 3.5A	COMMENTS
Transport of Diluted Species	Convection and Diffusion; Diffusion; Electrokinetic Flow; and 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, Earth Science Module, Heat Transfer Module, and MEMS 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 4.0	NAMES IN 3.5A	COMMENTS
Heat Transfer in Fluids	Convection and Conduction; 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, Stress-Strain; Axial Symmetry Stress-Strain; Plane Stress; and 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 4.0 it is also possible to study fully parameterized geometries using parametric sweeps.

#### MATERIAL LIBRARIES

NAME IN 4.0	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). Included with the Heat Transfer Module.
Liquids and Gases	Liquids and Gases	Included with the Chemical Reaction Engineering Module, Acoustics Module, Earth Science Module, Heat Transfer Module, and MEMS Module.
Material Library	Material Library	The COMSOL Material Library add-on is unchanged.