

COMSOL Multiphysics®

Release Notes



VERSION 4.2



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 80 90
✉ 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.it.comsol.com

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.
UH Innovation Centre
College Lane
Hatfield
Hertfordshire AL10 9AB
☎ +44-(0)-1707 636020
☎ +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
☎ +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

Home Page

🌐 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) and may be used or copied only under the terms of the license agreement.

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

Version:

May 2011

COMSOL 4.2

Part No. CM010001



Release Notes

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

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

COMSOL Multiphysics

New Products in Version 4.2

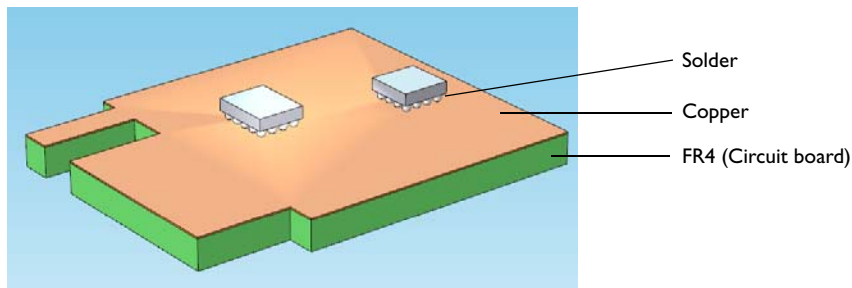
The following new products are introduced with COMSOL Multiphysics version 4.2:

- **Electrodeposition Module:** A new module intended for investigating the influence of different parameters, in an electrodeposition cell, on the thickness and composition of electrodeposited metal layers. See “Electrodeposition Module” on page 21 for more information.
- **Microfluidics Module:** A new module that includes laminar flow capabilities for single-phase and multiphase flow, including Knudsen slip conditions for microchannels. The module also features a tailored molecular flow interface. See “Microfluidics Module” on page 30 for more information.
- **Geomechanics Module:** An add-on to the Structural Mechanics for modeling soil mechanics and other applications in geomechanics. The Geomechanics Module includes a number of well-known material models for soil mechanics and rock mechanics such as the Cam-clay, Drucker-Prager, and Hoek-Brown models. See “Geomechanics Module” on page 23 for more information.
- **LiveLink™ for AutoCAD®**—connects COMSOL Multiphysics with the AutoCAD software.
- **LiveLink™ for SpaceClaim®**—connects COMSOL Multiphysics with the SpaceClaim software.

In addition, the former Earth Science Module is replaced by the Subsurface Flow Module (see “Subsurface Flow Module” on page 37).

GENERAL COMSOL DESKTOP FUNCTIONALITY

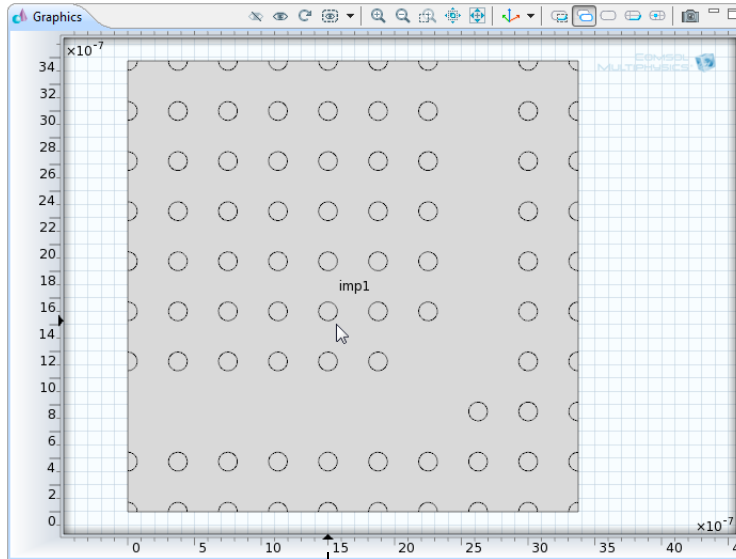
- New selection tools for creating the following selections:
 - Boolean combinations (union, intersection, difference) of previously defined selections
 - geometric entities that are partially or completely enclosed by a box or a sphere
 - complement of other selections
 - geometric entities (boundaries, for example) that are adjacent to some geometric entity (a domain, for example).
 - resulting entities from operations in the geometry sequence
 - duplicates (copies) of existing selections
- “Drag-and-drop” nodes in the Model Tree to copy or duplicate nodes.
- Copy and duplicate nodes in the physics interface branches of the Model Tree.
- Assignment of units for arguments to a function and the function itself, removing the need to “de-dimensionalize” function inputs and treat function outputs as dimensionless.
- Improved separate Log window and Progress window. The Progress window is cleared automatically when the solver or other process that the progress indication concerns has finished. In the Log window it is possible to clear the contents and to stop and resume the scrolling of the log window contents. There are also clear dividers in the log when starting a new model or opening a model to separate the log contents.



- Rendering of materials now include appearance, see the figure above: In 2D that includes color and in 3D color, texture, and reflectance. There are predefined appearance groups for common materials (aluminum, copper, gold, iron, water, air, brick, concrete, plastic, and others) are available, and you can also define custom

appearances. To enable texture rendering, set the **Preferences>Graphics** settings so that **Quality** is selected from the **Optimize for** list.

- New 2D renderer with better axis labels, easier selection of overlapping pair boundaries, and a new drawing table with grid and rulers that show the position of the pointer; see the figure below.

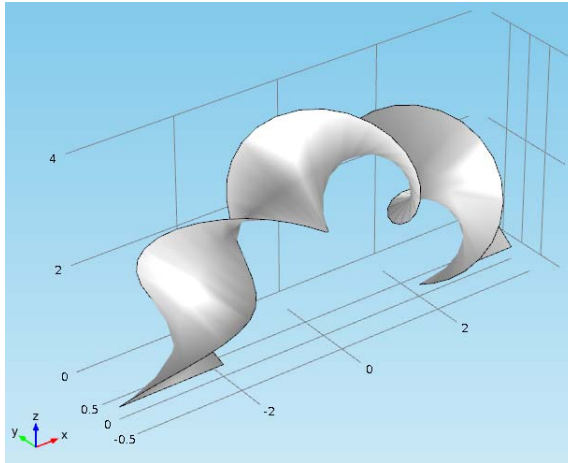


Arrows show the position of the pointer

- Automatic creation of identity and contact pairs in assemblies and easier conversion between pair types.

GEOMETRY AND MESH

- Parametric surfaces make it possible to create 3D surfaces based on coordinate functions using two surface parameters, see the figure below.



- Virtual geometry that can simplify final analyzed 3D geometries for improved meshing and easier physics modeling. You can add additional features to a finalized geometry to:
 - Ignore vertices and edges
 - Form composite edges and faces
 - Collapse edges
 - Merge vertices
- Improved mesh generators that support the virtual geometry operations.
- The import of meshes from NASTRAN files optionally includes materials.

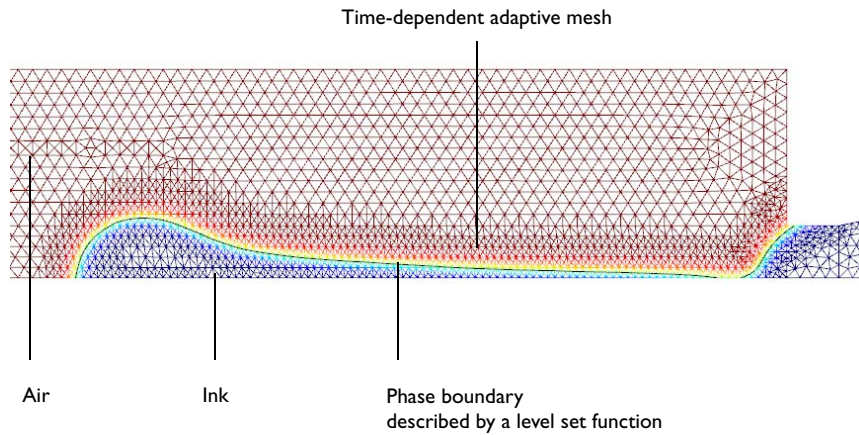
PHYSICS INTERFACES

- New Coefficient Form PDE and General Form PDE interfaces on boundaries, edges (3D), and points (2D and 3D).
- New Domain ODEs and DAEs, Boundary ODEs and DAEs, Edge ODEs and DAEs (3D), and Point ODEs and DAEs (2D and 3D) for solving distributed ODEs and other space-independent equations on domains and boundaries, along edges, or at points in the geometry.

- Heat Equation and Convection-Diffusion Equation are now available in the Classical PDEs section of the Mathematics interfaces.
- New *Thin Diffusion Barrier* boundary condition for interior boundaries in the Transport of Diluted Species interface.

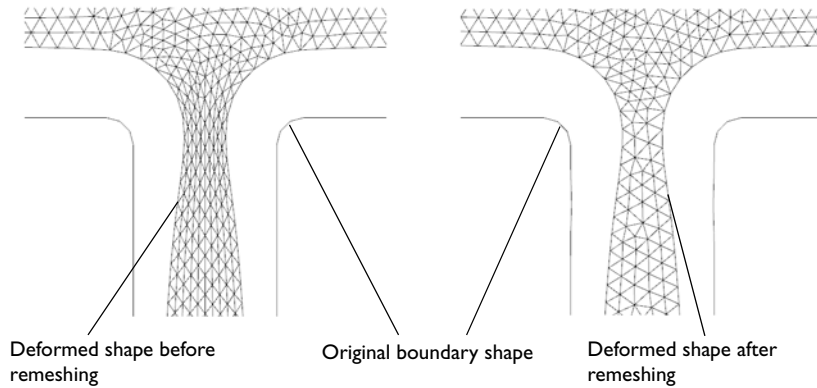
STUDIES AND SOLVERS

- Time-dependent adaptive mesh refinement during solution of time-dependent problems. The example below in the figure below is from the inkjet model in the Microfluidics and CFD Modules Model Libraries. The phase boundary between air and ink is here resolved using time-dependent adaptive meshing.



- Automatic remeshing for models that involve moving meshes. The figure below shows the squeezed element generated by a moving boundary and the resulting

elements after remeshing at the same time step. The model is available in the Electrodeposition Module Model Library.

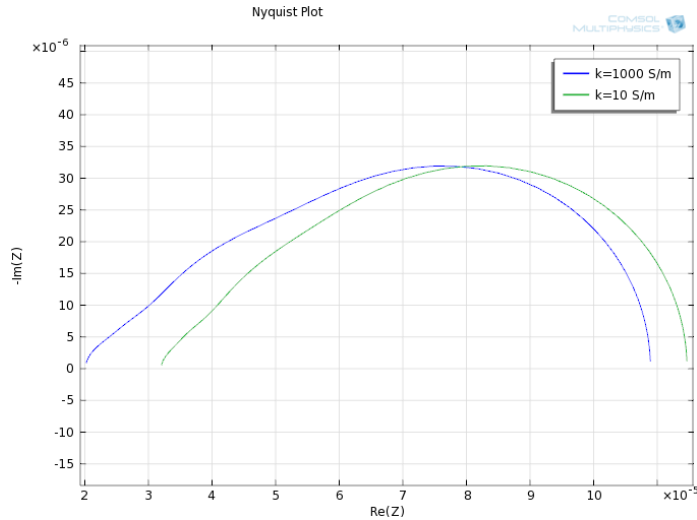


- Parallel assembly and solvers provide much improved scalability of solver performance for cluster computing and distributed-memory parallelism.
- Substantially increased performance for the assembly process.
- Convergence plots for monitoring convergence of nonlinear, iterative, and time-dependent solvers.

RESULTS

- Improved plot defaults that provide plots adapted to the physics in the model with descriptive names of the created plot groups.
- Histogram plots.

- Nyquist plots. The figure below shows the Nyquist plot from an AC-impedance analysis of a fuel cell at two different electrolyte conductivities. The model is available in the Batteries & Fuel Cells Module’s model library.



- New ThermalLight color scale, which is the default color scale for predefined temperature plots.
- RMS (root mean square), square root, and variance of data series.
- Direct results output of location for maximum and minimum.
- Frequency spectra using FFT now include scaling and units.
- Data export using grid format.
- Line markers at data points in graph plots.
- Improved arrow plots: logarithmic arrow scaling, cone arrows in 2D, and optional filtering of data for arrow plots.
- Improved positioning of color legends (color scales): to the right, to the left, above, below, or on alternating sides of the plot.
- Possibility to “symmetrize” the color range in plots (around zero, for example) for improved visualization of wave propagation.
- Direct copying of the contents in the Graphics window to the Windows clipboard for pasting the plot into a word processing document or spreadsheet, for example, using Ctrl+C and Ctrl+V.

Physics Interface Builder (Beta Version)

The Physics Interface Builder is a new tool for building custom applications and physics interfaces directly in the COMSOL Desktop environment. From the **File** menu, choose **New>Physics Interface Builder** to start a new Physics Interface Builder. The Physics Interface Builder window then replaces the Model Builder window. In a similar way that you build models using existing physics interfaces, you can build custom physics interfaces using the building blocks in the Physics Interface Builder and test them interactively. These custom physics interfaces can be fully deployed, including support for localization of the user interface. See the *Physics Interface Builder User's Guide* for more information and examples of how to use the Physics Interface Builder.

Report Generator

Version 4.2 includes a report generator that you can use to generate HTML reports of models. The report generator is an integrated part of the model, and you can add several reports to a model. There are four predefined templates:

- Brief Report: provides an overview of the model
- Intermediate Report: provides the most important information about the model
- Complete Report: provides full information about the model
- Custom Report: an initially empty report that you can fully configure

In all reports you can move, add, disable, and delete report nodes to adjust the report contents.

By default, the reports include a table of contents with clickable links to the contents below. The report generator uses a built-in stylesheet, but you can use a custom stylesheet to change the appearance of the report in the web browser.

For more information, see “Creating Reports” in the *COMSOL Multiphysics User's Guide*.

The reports in version 4.2 do not include the following parts of a model:

- Probes
- Model couplings
- Coordinate systems
- Pairs
- Solver

- Data sets
- Export

The reports will include all applicable parts of the model in version 4.2a.

Note: You find the tools for creating data files, images, and animations under the **Report** branch in version 4.1 under the **Export** branch in version 4.2.

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

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

BACKWARD COMPATIBILITY FOR PRE-3.5A MODELS

COMSOL 4.2 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 4.2

- A *Lumped Port* boundary feature is now available in the Magnetic Fields interface and in the Magnetic and Electric Fields interface. This feature is used for easier excitation of coils and other conducting structures. It also gives access to *S-parameters*.
- *Reduced Potential* formulations of the Magnetic Fields and Magnetic Fields, no currents interfaces as previously available in version 3.5a are now supported.
- *Small signal analysis* is now available. The implementation is different and more general than in version 3.5a; a tutorial example model, *Small Signal Analysis of Inductor*, is provided.
- In the *Electric Currents* interface capacitive terms has been added to the *Distributed Resistance* feature and the name has changed to *Distributed Impedance*. Similarly, the *Contact Resistance* feature has changed and is now called *Contact Impedance*. There is also a *Pair Contact Impedance* available on assembly pairs. All of these features allow for setting dielectric properties for capacitive coupling across the layer in addition to electrical conductivity for resistive coupling. In the stationary study, the dielectric properties are not used.

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

- *Nyquist Plots* are available in 1D Plot Groups.
- *Logarithmic* arrow length scaling is now provided for *Arrow* plots.

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 4.2

THERMOACOUSTICS INTERFACE

A new multiphysics interface for thermoacoustics using frequency-domain analysis.

ACOUSTIC-SHELL INTERACTION INTERFACES

New multiphysics interfaces for acoustic-shell coupling in the frequency domain and the time domain for 3D models. The Acoustic-Shell Interaction interfaces combine features from the Pressure Acoustics and Shell interfaces.

ACOUSTIC-PIEZOELECTRIC INTERACTION INTERFACES

New multiphysics interfaces for acoustic-piezoelectric coupling in the frequency domain and the time domain, which combine features from the Pressure Acoustics, Solid Mechanics, Electrostatics, and Piezoelectric Devices interfaces.

ELASTIC WAVES INTERFACE

New Elastic Waves interface for frequency-domain analysis of elastic waves.

POROELASTIC WAVES INTERFACE

New Poroelastic Waves interface for frequency-domain analysis of poroelastic waves. The Poroelastic Waves interface combines features from the Pressure Acoustics and Elastic Waves interfaces.

Batteries & Fuel Cells Module

New Functionality in 4.2

MATERIAL LIBRARY

- New Batteries and Fuel Cells material library including common battery electrode materials and electrolytes.

ELECTRODE KINETICS

- Concentration-dependent predefined kinetics expressions for porous and planar electrodes.

ELECTROCHEMICAL IMPEDANCE SPECTROSCOPY (EIS)

- AC Impedance Study types simulating EIS using Harmonic Perturbations on Electric Potential and Electrode Current boundary nodes.

NEW PHYSICS INTERFACES AND INTERFACE FUNCTIONALITY

- Surface Reactions interface for modeling of surface reactions on boundaries (surfaces in 3D).
- Reacting Flow, Diluted Species multiphysics interface combining single-phase flow with mass transport of diluted species.
- Reacting Flow, Concentrated Species multiphysics interface combining single-phase flow with mass transport of concentrated species.
- New Thin Diffusion Barrier boundary condition for interior boundaries in the Species Transport in Porous Media interface.

NEW MODELS

- Edge Effects In a Spirally Wound Li-Ion Battery
- Thermal Modeling of a Cylindrical Li-Ion Battery in 2D
- Thermal Modeling of a Cylindrical Li-Ion Battery in 3D (requires the Batteries & Fuel Cells Module and the Heat Transfer Module)
- Electrochemical Impedance Spectroscopy in a Fuel Cell
- Primary Current Distribution in a Lead-Acid Battery Grid Electrode
- Soluble Lead-Acid Redox Flow Battery

CAD Import Module and LiveLink Products for CAD

New Products in Version 4.2

Version 4.2 includes the following new LiveLink products for CAD:

- LiveLink™ for AutoCAD®—connects COMSOL Multiphysics with the AutoCAD software.
- LiveLink™ for SpaceClaim®—connects COMSOL Multiphysics with the SpaceClaim software.

New Functionality in Version 4.2

- The Cap Faces operation feature has been added to help you create fluid or other domains based on solid structures (for example, creating a domain representing the void inside a pipe).
- The LiveLink for SolidWorks interface now embeds the COMSOL modeling environment into the SolidWorks user interface to allow you to create COMSOL models entirely in the SolidWorks design environment.
- Virtual geometry has been introduced in COMSOL Multiphysics to help you mesh imported CAD designs where defeaturing is not possible due to a too complex topology, for example.
- SolidWorks 2011 CAD files are now supported.

CFD Module

New Functionality in Version 4.2

NEW PHYSICS INTERFACES

- High Mach Number Flow interface for laminar and turbulent flow, handling flows with compressibility effects.
- Reacting Flow, Diluted Species multiphysics interface combining single-phase flow with mass transport of diluted species.
- Reacting Flow, Concentrated Species multiphysics interface combining single-phase flow with mass transport of concentrated species.

SINGLE-PHASE FLOW

- Improved stabilization for the single-phase flow interfaces.

MULTIPHASE FLOW

- New Two-Phase Darcy's Law interface for modeling of two-phase flow in porous media.
- New initialization for two-phase flow interfaces using a wall distance function.
- Improved stabilization for the Brinkman Equations and Free and Porous Media Flow interfaces.

NON-ISOTHERMAL FLOW

- Thermal wall functions with turbulence now support the Surface-to-Surface Radiation, Reradiating Surface, and Highly Conductive Layer heat transfer boundary conditions.
- Vacuum pump boundary condition.

POROUS MEDIA FLOW

- New multiphase flow in porous media interface that defines two sets of Darcy equations for the separate phases in porous media.

FLUID-STRUCTURE INTERACTION

- Improved solver settings for the Fluid-Structure Interaction interface.

K- ω TURBULENCE MODEL

The *k- ω turbulence model* physics interface is not yet implemented in version 4.2. It is planned to be reintroduced in a future version of the CFD Module.

Version 4.2 includes automatic translation of models built with the previous *k- ω* turbulence model. When opened, the full model, including initial values and boundary conditions, is converted to the *k- ϵ* turbulence model. Once opened, the model can also be also be changed to the *Low-Reynolds k- ϵ Turbulence Model* interface. The latter physics interface present an excellent alternative for higher accuracy in models including confined flows.

Chemical Reaction Engineering Module

New Functionality in Version 4.2

- Surface Reactions interface for modeling of surface reactions on boundaries (surfaces in 3D).
- Reacting Flow, Diluted Species multiphysics interface combining single-phase flow with mass transport of diluted species.
- Reacting Flow, Concentrated Species multiphysics interface combining single-phase flow with mass transport of concentrated species.
- Parameter estimation for perfectly mixed reactors for perfectly mixed reactors.
- Infinite elements for modeling unbounded domains in the Transport of Diluted Species interface.
- Support for equilibrium reactions in the Transport of Diluted Species interface.
- New *Thin Diffusion Barrier* boundary condition for interior boundaries in the Species Transport in Porous Media interface.
- Improved stabilization for the single-phase flow interfaces.

Electrodeposition Module

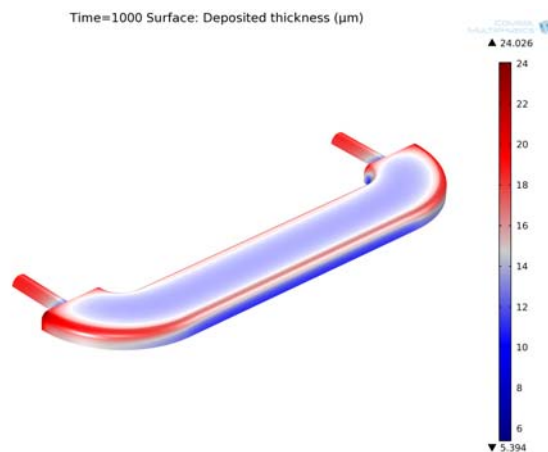
The Electrodeposition Module is a new module intended for investigating the influence of different parameters, in an electrodeposition cell, on the thickness and composition of electrodeposited metal layers.

Important parameters that can be studied with the module are the following:

- Cell geometry
- Electrolyte composition and mixing
- Electrode kinetics
- Operating potential and average current density
- Temperature.

The targeted applications for the Electrodeposition Module are the following:

- Copper deposition for electronics and electrical applications
- Metal deposition for protection of parts, such as corrosion protection and protection against wear
- Decoration of metals and plastic parts
- Electroforming of parts with thin and complex structure
- Metal electrowinning.



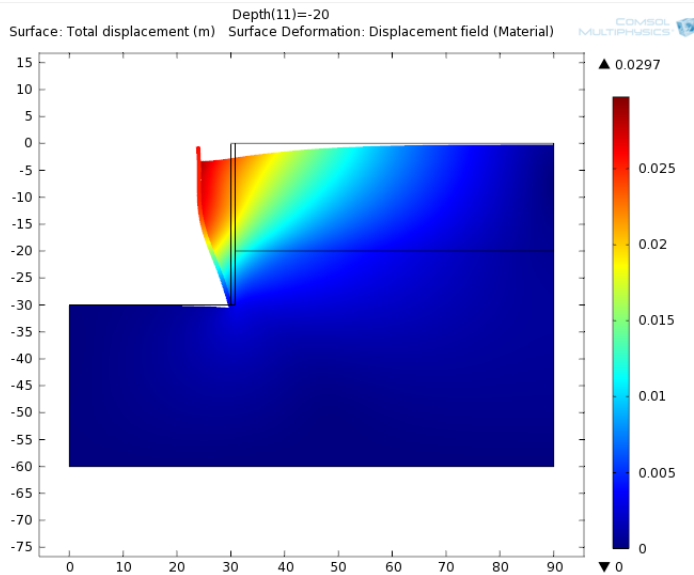
Thickness of the decorative deposited layer in a furniture fitting modeled using the Electrodeposition Module.

Major Functionality

- Electrodeposition, Secondary interface with full electrode kinetics for perfectly mixed electrolyte for fixed and moving boundaries.
- Electrodeposition, Tertiary Nernst-Planck interface for transport of species through diffusion, migration, and convection and with kinetics expressions that account for both activation and concentration overpotential. Also this interface is available for fixed and moving boundaries.
- Simulation of the thickness and composition of the deposited layer at the electrode surfaces for both primary and secondary current distribution in fixed and moving geometries.
- Fluid Flow interfaces that can be combined with the Electrodeposition interfaces to model free and forced convection in the electrolytic cell.
- The Heat Transfer interfaces have ready-made formulations for the contribution of Joule heating, and other electrochemical losses, to the thermal balance in the cell.
- Moving boundaries with and without remeshing.
- Time-dependent studies and capabilities to model AC-impedance for elementary studies of electrodeposition.

Geomechanics Module

The new Geomechanics Module is a specialized add-on to the Structural Mechanics Module aimed at modeling and simulating applications in geomechanics and rock mechanics. The Geomechanics Module includes tailored material models to study plasticity, deformation, and failure of soils and rocks, as well as their interaction with concrete and human-made structures.



Soil model of deep excavation from the Geomechanics Module Model Library.

Major Functionality

The Geomechanics Module includes the following material models:

ELASTOPLASTIC SOIL MODELS

- Mohr-Coulomb
- Capped Mohr-Coulomb
- Drucker-Prager
- Capped Drucker-Prager

- Matsuoka-Nakai
- Lade-Duncan

NON-LINEAR SOIL MODELS

- Cam-clay

CONCRETE MODELS

- Bresler-Pister
- Willam-Warnke
- Ottosen

ROCKS MODELS

- Hoek-Brown
- Generalized Hoek-Brown

DUCTILE MATERIALS AND SATURATED SOILS

- von Mises
- Tresca

Heat Transfer Module

New Functionality in Version 4.2

- Thermal wall functions with turbulence now support the *Surface-to-Surface Radiation* and *Highly Conductive Layer* features and their subfeatures.
- New option to define the total power of a domain *Heat Source* and to define the total *Heat Flux* on a boundary.
- New *Grill Boundary Condition* and *Vacuum Boundary Condition* for setting inlets and outlets properties in fluid-flow models.
- New *Fan Boundary Condition* on interior boundaries.
- Support for tables to define pressure curves in the *Fan Boundary Condition* for using fan curves at inlets in fluid-flow models.
- New option to define structures with multiple layers in the *Thin Thermally Resistive Layer* feature.
- New built-in convective cooling coefficients for external natural convection on inclined walls.
- Update and simplification of the variables names and definitions for heat sources and heat flux. A new section about heat fluxes is available in the documentation.
- *Incident Intensity* boundary condition can be defined using the discrete directions used by the discrete ordinate method.
- *Wall* boundary condition in *Heat Transfer in Participating Media* has been renamed to *Opaque Surface* boundary condition. In version 4.2 it accounts for the thermal contribution of the net radiative heat flux.
- New default plots for Heat Transfer, Heat Transfer in Thin Shells, Surface-to-Surface Radiation, Radiation in Participating Media, Conjugate Heat Transfer, Joule Heating, Inductive Heating, and Microwave Heating interfaces using the new ThermalLight color scale for temperature plots.

Backward Compatibility vs. 4.1

WALL FEATURES IN RADIATION IN PARTICIPATING MEDIA

The *Wall (wall)* feature in the Radiation in Participating Media interface has been renamed *Opaque Surface (os)* in version 4.2. Model Java-files or Model M-files that

create and solve COMSOL models containing Wall features need to be updated to use Opaque Surface instead. When the Opaque Surface is used in the Heat Transfer interface in version 4.2, it adds a new heat flux term that is equal to q_w , which is the net radiative heat flux. If you do not want to keep this term in an imported model from version 4.1, you must add a Heat Flux feature on the same selection as the Opaque Surface, and the parameter q_0 must be set to $-id \cdot q_w$, where id is the physics interface identifier.

Backward Compatibility vs. 3.5a

K- ω TURBULENCE MODEL

The *k- ω turbulence model* physics interface is not yet implemented in version 4.2. It is planned for the CFD Module in version 4.2a.

Version 4.1 includes automatic translation of models built with the previous *k- ω* turbulence model. When opened, the full model, including initial values and boundary conditions, is converted to the *k- ϵ* turbulence model. Once opened, the model can also be changed to the *Low-Reynolds k- ϵ 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

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

Material Library

New Functionality in Version 4.2

- Additional data and properties in the material libraries, such as stress-strain curves for silver, specific heat for various perovskites, and additional data for stainless steel.
- Support for the material appearance that is new in version 4.2.

MEMS Module

Note: Extended microfluidics capabilities are available in the new Microfluidics Module. The MEMS Module now focuses on microelectromechanical applications and does not include any microfluidics capabilities beyond what is included in the main COMSOL Multiphysics software.

New Functionality in Version 4.2

- Perfectly matched layers (PMLs) in the Piezoelectric Devices interface.
- Improved solver defaults in the Fluid-Structure Interaction interface.
- Symbols for boundary conditions and loads in the Graphics window.
- A new study type, *Prestressed analysis, Eigenfrequency*, available for the Solid Mechanics interface.
- A new study type, *Prestressed analysis, Frequency domain*, available for the Solid Mechanics interface.
- Springs and dampers are available for points, edges, boundaries, and domains available for the Solid Mechanics interface.
- Thin elastic layers are available on interior boundaries and between pairs in assemblies. in the Solid Mechanics interface.
- Added mass can be specified for edges, boundaries, and domains for all interfaces available for the Solid Mechanics interface.
- The force on the rigid connector can be given an arbitrary position.
- For the rigid connector you can give mass and inertial properties either directly or through declaring a domain as rigid.
- Improved default solver settings for the Thermal Stress and Joule Heating and Thermal Expansion interfaces.
- Geometric nonlinearity is available also in the Thermal Stress and Joule Heating and Thermal Expansion interfaces.
- Plasticity can now be added to domains selectively without redefining the underlying elastic data.

- Plasticity is now supported for orthotropic and anisotropic materials.
- Damping is available also for hyperelastic and elastoplastic materials.
- Infinite elements for truncating models covering large domains are available.
- The contact pair penalty factors have better defaults that are available for all material models.
- The contact pair can have different offset expressions for the source and destination side boundaries.

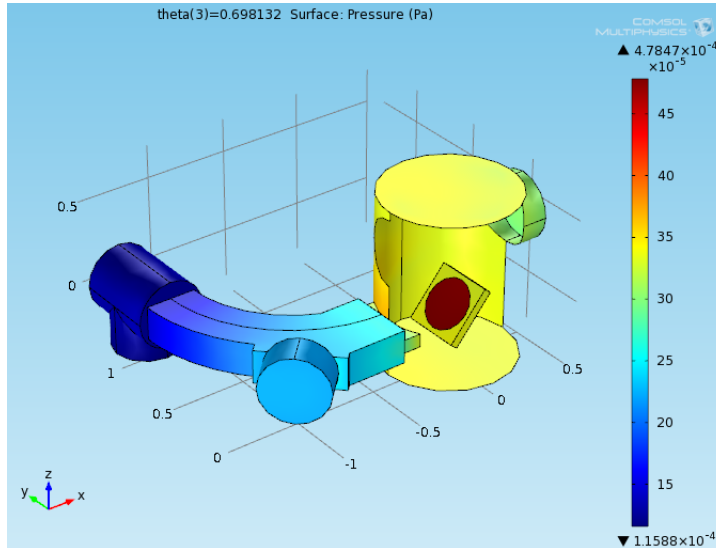
Backward Compatibility vs. Version 3.5a

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 4.2 using the full geometric description or using equation view.

Microfluidics Module

The new Microfluidics Module includes the microfluidics functionality in the version 4.1 of the MEMS Module and new functionality for microfluidics modeling.



Pressure distribution in an ion implanter modeled with the Molecular Flow interface in the Microfluidics Module.

MICROFLUIDICS

- Laminar and creeping flows, compressible ($Ma < 0.3$) and incompressible flows, and non-Newtonian flows supported.
- Multiphase flow with the level set and phase field methods, plus a new multiphase flow interface with the moving mesh method.
- Porous media with Darcy's law and the Brinkman equations, plus a Free and Porous Media Flow interface.
- Enhanced interface for transport of diluted species supporting diffusion, convection, and migration of multiple species.
- Extensive support for electrokinetic effects including electrowetting, electroosmosis, electrophoresis, and dielectrophoresis.

- Support for non-isothermal flows, Joule heating, heat conduction, and heat convection.
- Rarefied flows in the slip-flow and molecular-flow regimes.

NEW MODELS

- Drug Delivery System
- Electrowetting Lens
- Jet Instability (Level Set and Moving Mesh)
- Viscous Catenary
- Ion Implanter
- Vacuum Capillary

Plasma Module

New Functionality in Version 4.2

PLASMA MODELING

- Surface reactions of order greater than 1 are now supported.
- Unit support for reaction and surface reaction rate constants.
- User-defined electron energy distribution functions are now supported.
- Cross section data can be imported directly from the LXCAT database.
- Species transport and thermodynamic data are now included for the most common plasma processing gases.
- Mass flow rate boundary conditions, which allow specification of a flow rate in SCCM units.
- Vacuum pump boundary condition.
- Stabilization of reaction sources has been added for electrons and heavy species.

Note: The stabilization is active by default, so models with very low plasma densities might give slightly different results after re-solving in version 4.2.

NEW MODELS

- Harmonic Content in a Dual Frequency CCP Reactor
- Dipolar Electron Cyclotron Resonance (ECR) Source
- Electrodeless Lamp
- Surface chemistry tutorial model

RF Module

New Functionality in Version 4.2

- The *Far field* feature now supports far field in a medium other than vacuum. It has changed from being a boundary feature to being a domain feature with a (far field medium) domain selection and, as a sub feature, a boundary selection.
- The *Port* feature now supports analytical circular ports in 2D axisymmetry.
- It is now possible to compute S-parameters between ports and lumped ports and to include lumped ports in port sweeps and Touchstone export.

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

- *Nyquist plots* are available in 1D Plot Groups.
- *Logarithmic* arrow length scaling is now provided for arrow plots.

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-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.

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 4.2

- Symbols for boundary conditions and loads in the Graphics window.
- A new study type, *Prestressed analysis, Eigenfrequency*, available for the Solid Mechanics, Shell, and Plate interfaces
- A new study type, *Prestressed analysis, Frequency domain*, available for the Solid Mechanics interface.
- Built-in computation of beam cross-section properties for a number of common cross sections in the Beam interface.
- Springs and dampers are available for points, edges, boundaries, and domains for all interfaces in the Structural Mechanics Module.
- Thin elastic layers are available on interior boundaries and between pairs in assemblies.
- Added mass can be specified for edges, boundaries, and domains for all interfaces in the Structural Mechanics Module.
- The force on the rigid connector can be given an arbitrary position.
- For the rigid connector you can give mass and inertial properties either directly or through declaring a domain as rigid.
- Improved default solver settings for the Thermal Stress and Joule Heating and Thermal Expansion interfaces.
- The Fluid-Structure Interaction interface has been updated and enhanced with improved solver settings.
- Geometric nonlinearity is available also in the Thermal Stress and Joule Heating and Thermal Expansion interfaces.
- Plasticity can now be added to domains selectively without redefining the underlying elastic data.
- Plasticity is now supported for orthotropic and anisotropic materials.
- Damping is available also for hyperelastic and elastoplastic materials.
- Infinite elements for truncating models covering large domains are available.
- Perfectly matched layers (PMLs) in the Piezoelectric Devices interface.

- The contact pair penalty factors have better defaults that are available for all material models.
- The contact pair can have different offset expressions for the source and destination side boundaries.

Backward Compatibility vs. Version 3.5a

PLANE STRESS

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

Subsurface Flow Module

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

New Functionality in Version 4.2

- New option to define structures with multiple layers in the *Thin Thermally Resistive Layer* feature in the Heat Transfer interface.
- New *Thin Diffusion Barrier* boundary condition for interior boundaries in the Solute Transport interface.
- Improved stabilization for the single-phase flow interface.
- Improved stabilization for the Brinkman Equations and Free and Porous Media Flow interfaces.

LiveLink for MATLAB

New Functionality in COMSOL 4.2

GENERAL NEW FUNCTIONALITY IN COMSOL MULTIPHYSICS

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

- New selection methods (ball and box selections, unions, and so on).
- New plot types including ability to view selections in MATLAB figures (`mphviewselection`).
- Parametric surfaces.

IMPROVED OVERVIEW OF THE MODEL OBJECT

- Objects have a more well-defined set of methods because a large number of unnecessary methods have been removed. All methods are now documented in the *COMSOL API Reference Guide* and as Java Doc.
- `mphsave` and `mphcd` functions for easier access to COMSOL model files.
- Functionality and GUI for viewing the Model Object (`mphnavigator`). See Figure 1-1 below.

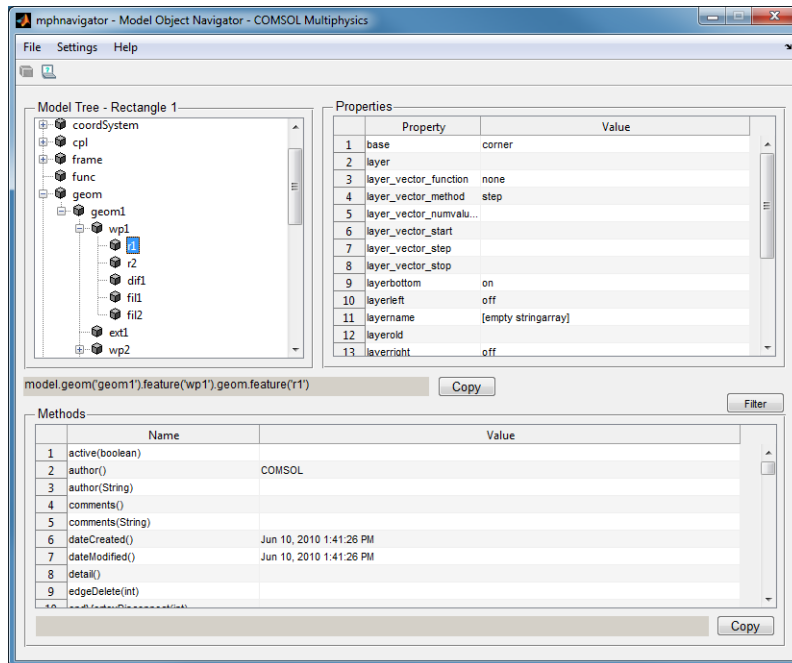


Figure 1-1: The Model Object Navigator (mphnavigator).

IMPROVED ACCESS TO MODEL DATA

- `mphinterp`, `mpheval`, and `mphglobal` are now able to return complex values.
- `mpheval` can return data at Gauss points.
- `mphstate` and `mphmatrix` now support a custom linearization point.
- Mesh and Xmesh (extended mesh) information retrieval.

IMPROVED STARTUP

- Startup message and customizable startup files have been added.
- Improved memory settings for MATLAB through the use of a `java.opts` file.
- When using the MATLAB node inside a COMSOL model: Addition of model path to the MATLAB path, improved error messages, and support for complex-valued data.

Parameterized Geometries

Version 4 includes built-in support for parameterized geometries. There should be no need to change boundary conditions during parameterization unless the geometry is built from scratch using the LiveLink for MATLAB. Parameterization can be performed both with the built-in parametric solver and by wrapping a for-loop around the solver call. Notice that COMSOL Multiphysics has new built-in capabilities for handling selections based on coordinates, which are also supported by the LiveLink for MATLAB.

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 following geometry functions from version 3.5a is not available in version 4.2. Most of this functionality will be available in a later release.

- `f1im2curve`, `f1contour2mesh`, and `f1mesh2spline`
- `geomspline`, `geomsurf`
- `loft` (partially available as Sweep)
- `geom1`, `geom2`, and `geom3`

COMSOL API

See the section “The COMSOL Java API” on page 41 regarding changes in the COMSOL API.

The COMSOL Java API

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

- The solution to eigenvalue and eigenfrequency problems may now appear in a different order.
- The Batteries & Fuel Cells Module's boundary feature `BoundaryReactionCoefficient`, is tagged by `rc` instead of `brc` by default. This means that files that explicitly refer to the old default tag name must be modified to refer to the new tag name.

COMSOL 4.0a Java API Changes

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

- The load face parameter (`Fph`) of the *Phase* feature in `AcousticStructure`, `TransientAcousticStructure`, `ThermallyInducedStresses`, `SolidMechanics`, `JouleHeatingThermalExpansion`, `Poroelasticity`, and `FluidStructureInteraction` has changed interpretation from degrees to radians.
- `physics.field()` of `Beam` and `Truss` are not backward compatible with 4.0. Scalar fields are now vector fields.
- The variables for strain and stress in the `Shell` interface have been renamed. Variable names that began with lowercase `s` now begin with uppercase `S`. For variables names that ended in lowercase `l` (`L`), the letter was moved to the third position. For example, `emXXl` is now `emlXX`.
- Force loads on solids and trusses have been moved from material to spatial frame. This implies that index notation of some variables have changed from uppercase `XYZ` to lowercase `xyz`.
- The acoustics interfaces (`PressureAcoustics`, `TransientPressureAcoustics`, `AcousticStructure`, and `TransientAcousticStructure`) have been moved from material to spatial frame. This implies that index notation of vector variables have change from uppercase `XYZ` to lowercase `xyz`.

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 4.2	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 4.2	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 4.2	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, 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 4.2	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 version 4 it is also possible to study fully parameterized geometries using parametric sweeps.

MATERIAL LIBRARIES

NAME IN 4.2	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, Subsurface Flow Module, Heat Transfer Module, and MEMS Module.

NAME IN 4.2	NAMES IN 3.5A	COMMENTS
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.

