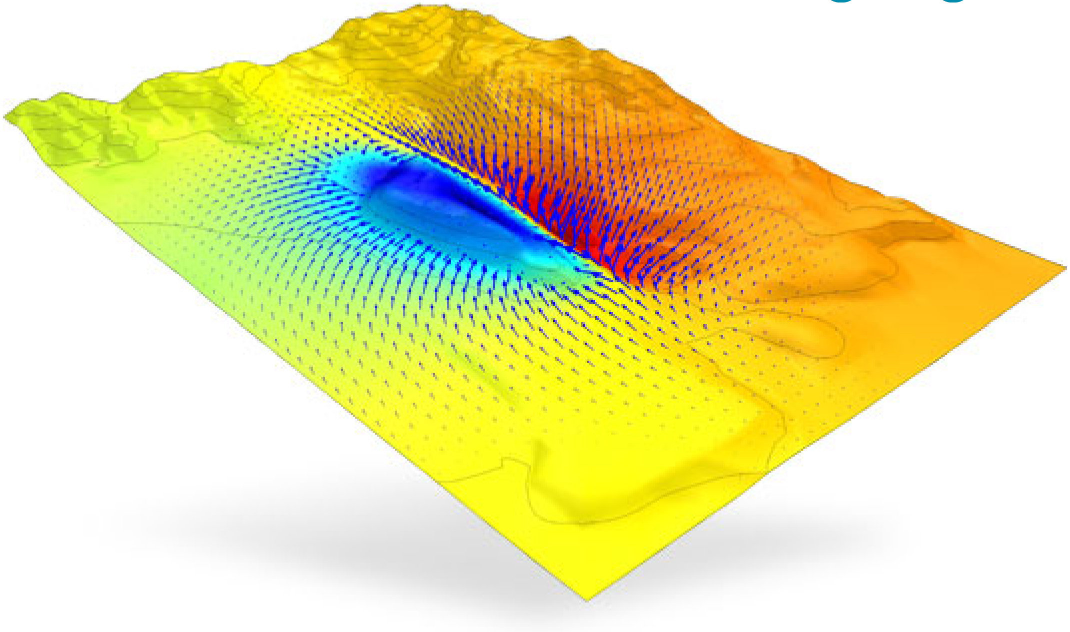
COMSOL 4.2a Release Highlights



COMSOL has built a solid reputation of fast-paced innovation for multiphysics simulation and analysis. The new Version 4.2a adds to the long history of successful releases of the flagship COMSOL Multiphysics product suite. By including features that reach new communities of engineers and scientists, COMSOL is creating a tightly-integrated platform for analysis whose breadth and depth is unmatched. Major news in the Version 4.2a release:

Particle Tracing Module Track particles with particle-field interactions for CFD, electromagnetics, acoustics and other applications.

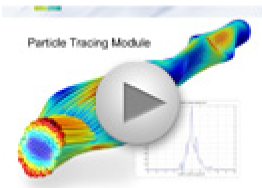
LiveLink™ for Creo™ Parametric Seamless bidirectional CAD integration with the latest design software from PTC™.

Faster and more efficient parametric sweeps Take control over memory usage for large parametric sweep and quickly create response surface plots.

Digital Elevation Map (DEM) import Import topographical surface data and combine with solids for any application such as fluid flow, structural, or electromagnetics.

Image import Import 2D images for material table-lookup based on photographic or scanned data.

Interactive slice and isosurface plots Quick interactive positioning of plot surfaces with slider control.



Introducing
COMSOL Multiphysics 4.2a
Duration: 2:18

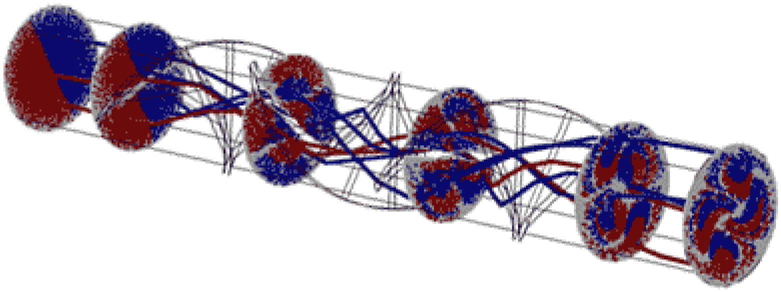
Magnetic prospecting is a method for geological exploration of iron ore deposits. The picture shows a simulation where imported terrain data was used to represent the underlying geometry. Passive magnetic prospecting relies on accurate mapping of local geomagnetic anomalies. This model estimates the magnetic anomaly for both surface and aerial prospecting by solving for the induced magnetization in the iron ore due to the earth's magnetic field.

- New Products
  - Particle Tracing Module
  - LiveLink for Creo Parametric

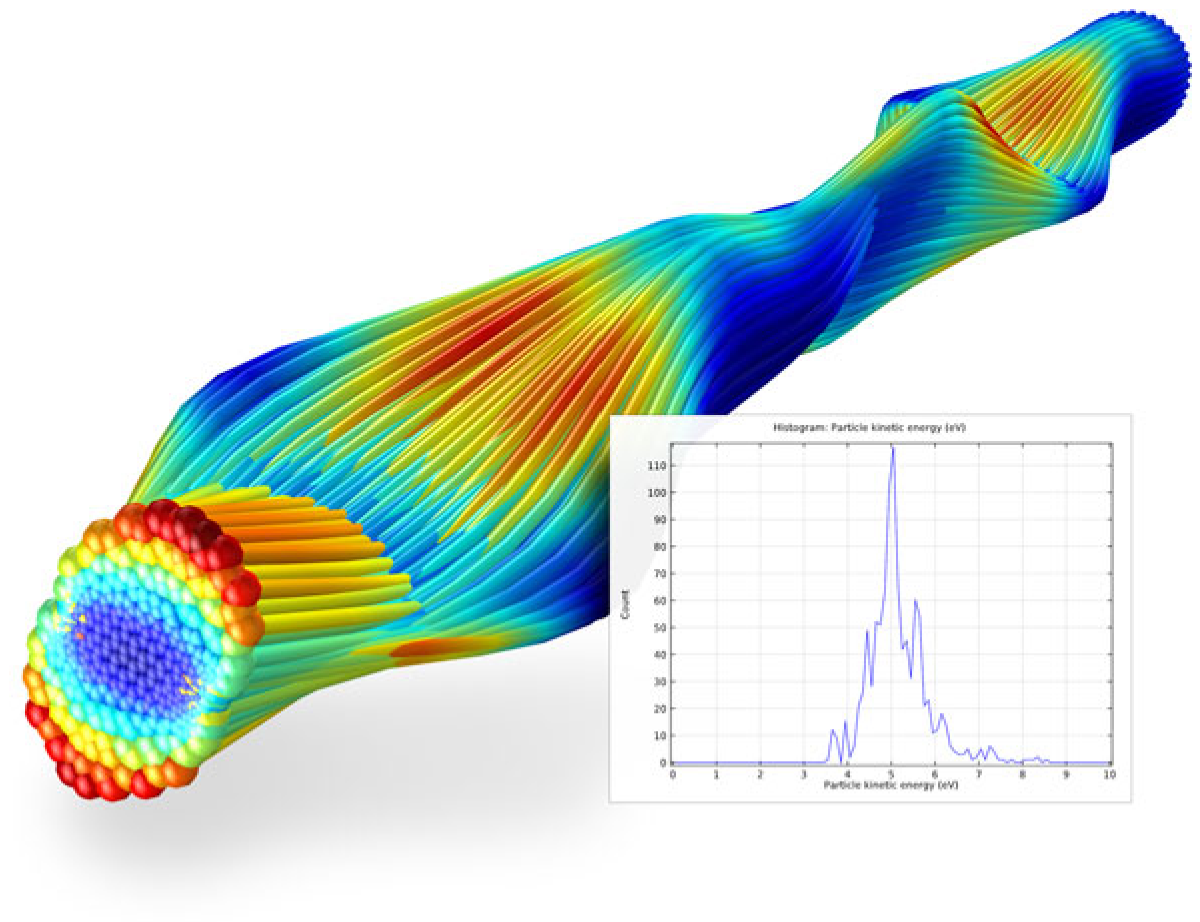
Particle Tracing Module

The Particle Tracing Module extends the functionality of the COMSOL Multiphysics environment for computing the trajectory of particles in a fluid or electromagnetic field, including particle-field interactions. Any add-on module combines seamlessly with the Particle Tracing Module and gives you access to additional modeling tools and fields to drive the particle motion.

LEARN MORE



This flow simulation computes the trajectories of quartz particles through a static mixing device. Due to the fact that the particles have mass, only a certain fraction make it to the outlet. This fraction, the transmission probability, is computed during postprocessing.



A mass spectrometer is used to separate and identify different substances from a sample. Applications are numerous including materials engineering and environmental science. The picture shows a particle tracing simulation with trajectories of ions of various molecular weights in a quadrupole mass spectrometer. The electric fields has both AC and DC components and the combination of the two is essential for the function of the spectrometer.

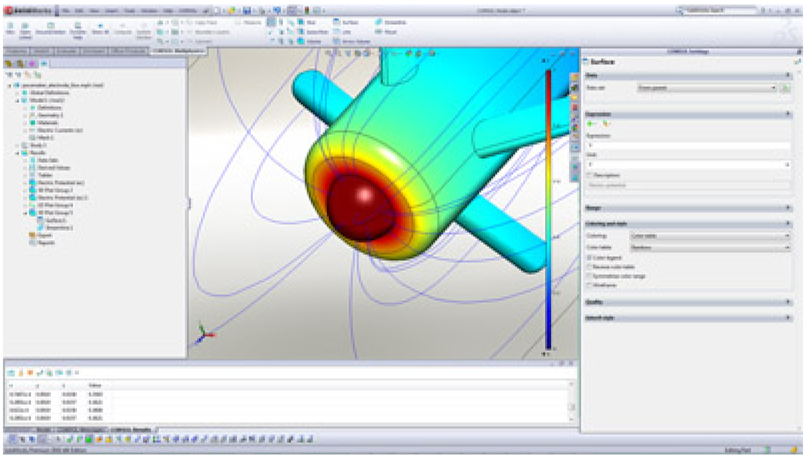
LiveLink™ for Creo™ Parametric

With the new LiveLink for Creo Parametric, COMSOL Multiphysics can be seamlessly integrated with the latest design software from PTC®. By establishing an associative connection between the two applications, a change of a feature in the CAD model automatically updates the geometry in COMSOL Multiphysics, while retaining physics settings. All parameters specified in Creo Parametric can be interactively linked with your simulation geometry. This enables multiphysics simulation involving parametric sweeps and design optimization in sync with the CAD program. The LiveLink for Creo Parametric includes all the capabilities of the CAD Import Module and enables import and defeaturing of CAD files from all major CAD packages.

LEARN MORE

Model Library, Animations, and Images

COMSOL Multiphysics' extensive Model Library is now accessible from within the One Window Interface that is included with the LiveLink for SolidWorks. Animations and images can now be created from the One Window Interface and a series of performance enhancements make for quicker synchronization of large models.

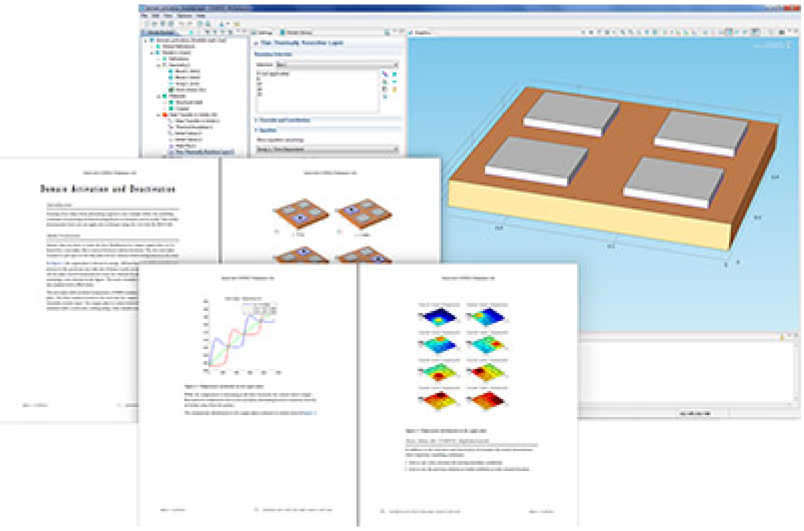


The LiveLink for SolidWorks now makes available animations and image generation from its One Window Interface.

LiveLink for MATLAB

The new version of the LiveLink for MATLAB includes numerous optimizations for improved performance and memory handling as well as new and updated functions, including a user interface for navigating COMSOL's Model Library.

VIEW SCREENSHOT



The new tutorials for the LiveLink for MATLAB help you get up to speed with combining MATLAB with COMSOL. Detailed step-by-step instructions are available on HTML and PDF formats

New Tutorials

Five new Model Library tutorials demonstrate how to efficiently combine MATLAB scripting with COMSOL Multiphysics simulations. These models show capabilities that are unique to the LiveLink for MATLAB such as extracting data at the MATLAB prompt, running models in nested MATLAB for-loops, using previous solution data within MATLAB, and calling external MATLAB functions from the COMSOL Desktop:

- **Domain Activation and Deactivation** This model of a time-dependent heat-transfer problem implements heating from alternating regions by using domain activation and deactivation.
- **Homogenization in a Chemical Reactor** This model illustrates how to simulate a periodic homogenization process in a space-dependent chemical reactor model. This homogenization removes concentration gradients in the reactor at a set time interval.
- **Convective Heat Transfer with Pseudo-Periodicity** This model simulates convective heat transfer in a channel filled with water. To reduce memory requirements, the model is solved repeatedly on a pseudo-periodic section of the channel. Each solution corresponds to a different section, and before each solution step the temperature at the outlet boundary from the previous solution is mapped to the inlet boundary.
- **Temperature Distribution in a Thermos** This example solves for the temperature distribution inside a thermos holding hot coffee. The main purpose is to illustrate how to use MATLAB functions to define material properties and boundary conditions.

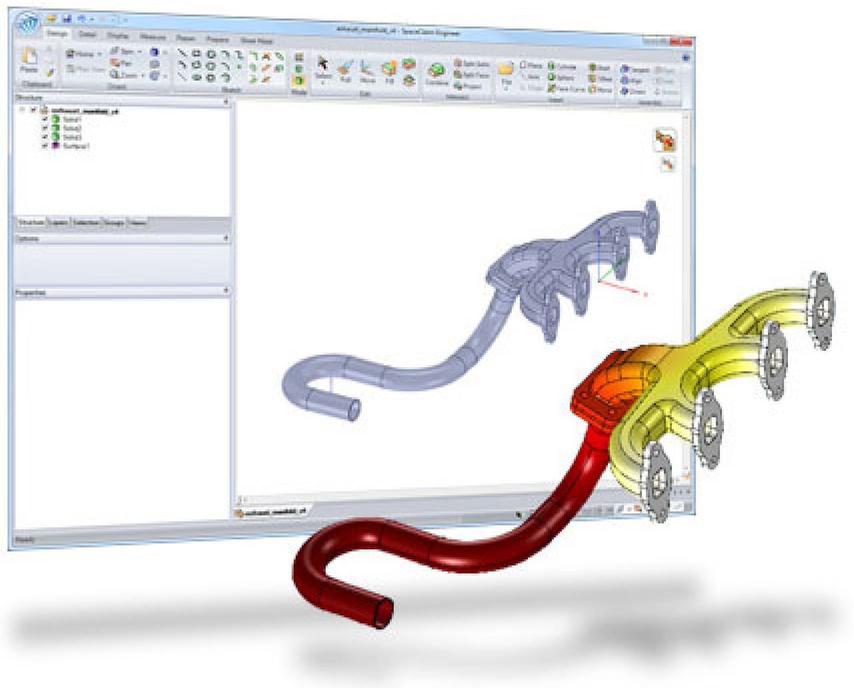
- **Geometry Parametrization using the LiveLink for Solidworks** This example shows geometry parametrization using both the LiveLink for Solidworks and the LiveLink for MATLAB. MATLAB is used to created nested loops that change geometry parameters and update the geometry using the LiveLink for Solidworks. The same modeling approach also works with the LiveLink for AutoCAD, LiveLink for Creo Parametric, LiveLink for Pro/ENGINEER, LiveLink for Inventor, and the LiveLink foir SpaceClaim.

LiveLink™ for SpaceClaim™

The LiveLink for SpaceClaim brings you the fusion of direct modeling and multiphysics simulation in a tightly integrated environment, enabling optimal designs and collaboration across CAD and CAE teams.

The LiveLink for SpaceClaim interface allows you to transfer a 3D geometry from SpaceClaim to COMSOL Multiphysics. The synchronized geometry in the COMSOL model stays associative with the geometry in SpaceClaim. This means that settings applied to the geometry, like physics or mesh settings, are retained after subsequent synchronizations. The LiveLink interface is also bidirectional to allow you to initiate a change of the SpaceClaim geometry from the COMSOL model.

The latest version has increased performance for synchronizing larger CAD models.



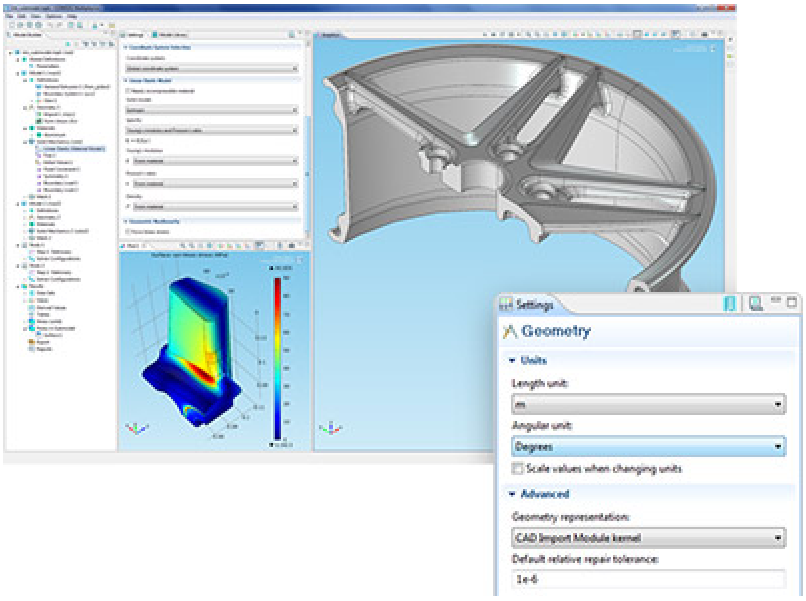
The latest version of the LiveLink for SpaceClaim comes with increased performance for synchronizing larger CAD models.

CAD Import Module

The Parasolid® geometry kernel from Siemens PLM is now the default geometry kernel for users of any of the following products: CAD Import Module, LiveLink for AutoCAD, LiveLink for Inventor, LiveLink for Pro/ENGINEER, LiveLink for Creo Parametric, LiveLink for SolidWorks, LiveLink for SpaceClaim.

The Parasolid kernel enables more advanced geometry operations and allows for creation and handling of complex CAD models within the native COMSOL Multiphysics geometry modeling environment. Without any add-on products, users can still create geometry models in the native COMSOL Multiphysics environment but with the functionality of COMSOL's native geometry kernel.

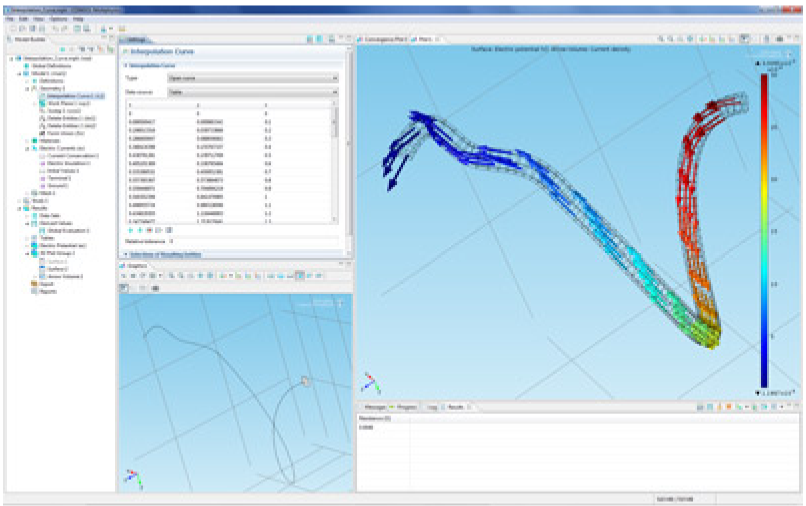
Automatic scaling is now enabled for handling CAD models of vastly different length scales ranging from nanodevices to mountains and beyond.



This structural simulation tutorial uses the submodeling technique to accurately resolve the stress concentrations in a wheel rim. First a global model is solved to obtain the displacements, which are then used as boundary conditions in a local model of the region where the stress concentrations occur. The CAD Import Module and certain LiveLink products enable import and geometry repair of the original CAD model. With these products added, the geometry representation now defaults to the Parasolid kernel from Siemens PLM, which is also used when creating geometries from scratch. This enables handling of more advanced geometry objects.

Interpolation Curves

Interpolation curves can be created from tabulated x,y or x,y,z data in both 2D and 3D. A cubic spline interpolates or approximates the given points – controlled by a user-defined tolerance. Data can be loaded from file or directly typed in the Interpolation Curve settings window. Curves can be open, closed, or automatically turned into a solid object. Such objects can be used for 2D analysis or be extruded, revolved, and combined to form 3D objects. Interpolation curves in 3D can be used as the spine of a geometric sweep or to guide and control mesh density as well as for postprocessing purposes.



The screenshot shows an electrical resistance calculation for a copper wire created with a geometric sweep along an interpolation curve with x,y,z data imported from file.

Cut-and-Paste Geometry Objects

You can now cut-and-paste or duplicate one or multiple geometry objects and operations in the Model Builder tree's Geometry node. This avoids reproducing complicated geometry objects or sequences of geometry operations and allows for faster geometry creation and parameterization.

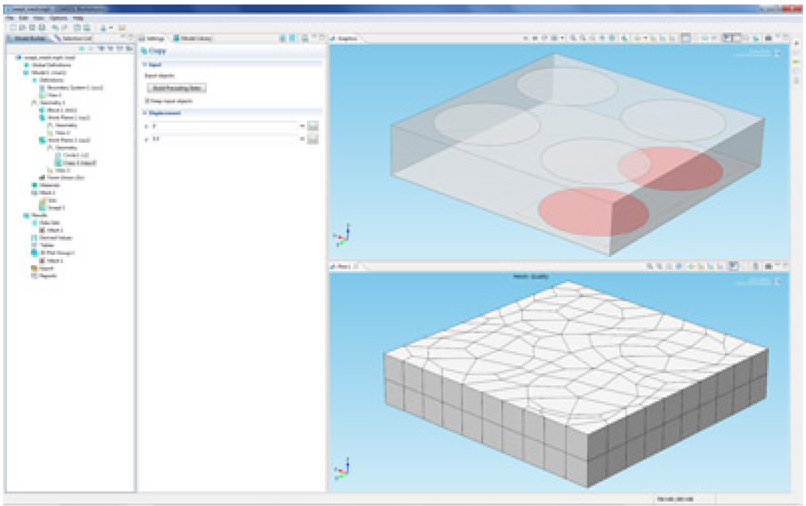
VIEW SCREENSHOT

Extended Swept Meshing

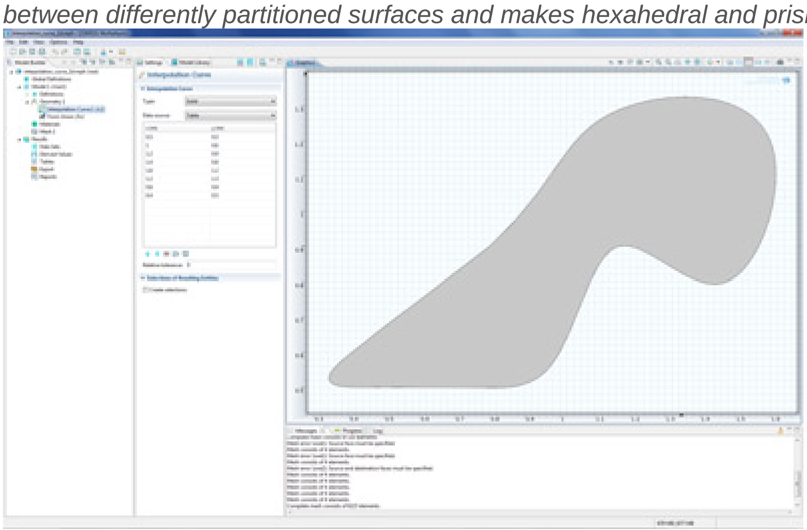
Swept meshing can now be used between partitioned surfaces. A surface partitioned in N segments can be swept into a surface of M segments, where N ≥ M. In general, it is required that the partitioning of the source (into faces) is a refinement of the partitioning of the destination.

The virtual geometry functionality has been generalized to cover swept meshes for geometry objects with surfaces where virtual geometry operations have been made.

VIEW SCREENSHOT



A swept mesh going from a surface partitioned in 5 segments to a surface of 2 segments. This new feature allows for mesh sweeps between differently partitioned surfaces and makes hexahedral and prismatic meshes available for a larger class of thin objects.



Interpolation curves are available in both 2D and 3D. Curves can be open, closed, or automatically be turned into a solid object. Such objects can be used for 2D analysis or be extruded, revolved, and combined to form 3D objects.

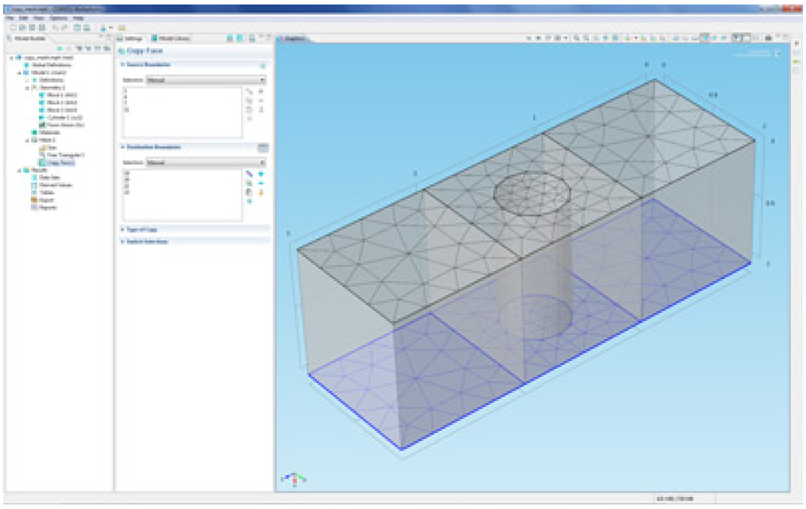
Reuse Parameterized Geometry Objects

You can now reuse parameterized geometry objects between simulations by inserting a geometry sequence from another model file. The geometry sequence in the Model Builder tree defines the geometric objects and the sequence of operations used to combine them into composite shapes. If the geometry sequence contains references to functions or parameters, those functions and parameters are also inserted into the model.

VIEW SCREENSHOT

Extended Mesh Copying

New copy mesh functionality makes it possible to copy a mesh from a partitioned surface to a similar surface using an automatic rigid-body transformation. This functionality is important for periodic boundary condition applications with high-accuracy requirements such as cyclic symmetry for structural analysis and Floquet boundary conditions for electromagnetic wave propagation. The new features are available as Copy Domain, Copy Face, and Copy Edge.



A triangular surface mesh is copied to its opposing side. Meshes that are copied using rigid body translations are important for periodic boundary condition applications with high-accuracy requirements such as cyclic symmetry for structural analysis and Floquet boundary conditions for electromagnetic wave propagation.

Sketch on Work Planes in 3D

It is now possible to interactively sketch 2D primitives on work planes directly in 3D allowing for easier geometry object positioning. Activate by selecting the checkbox Draw on work plane in 3D. The feature requires a graphics card with support for texture rendering. The default is still 2D work plane sketching but you can permanently switch the new work plane behavior on by changing a preference entry. Two new toolbar buttons provide Work Plane Clipping and Align with Work Plane functionality for simplifying geometry creation using work planes.

VIEW SCREENSHOT

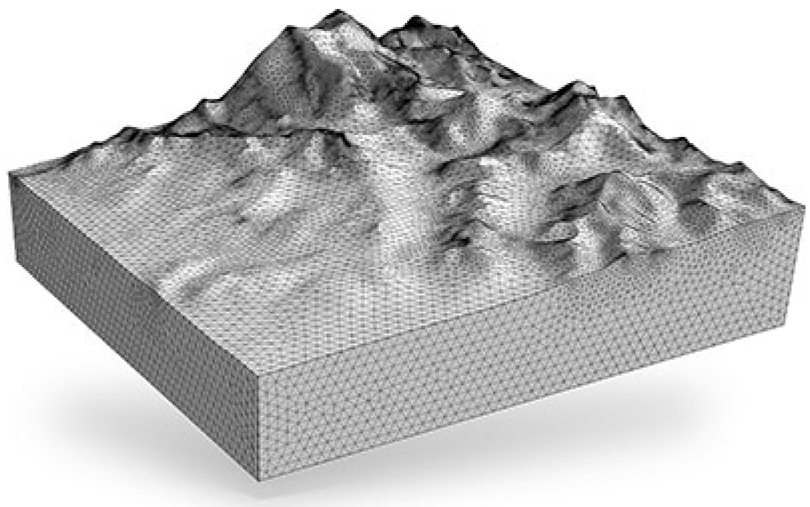
Image Import

You can now use image data to represent 2D material distributions or to identify regions with different materials by their color or gray scale. Images used in this way can have many origins such as scanning electron microscope (SEM), computed tomography (CT), or magnetic resonance imaging (MRI).

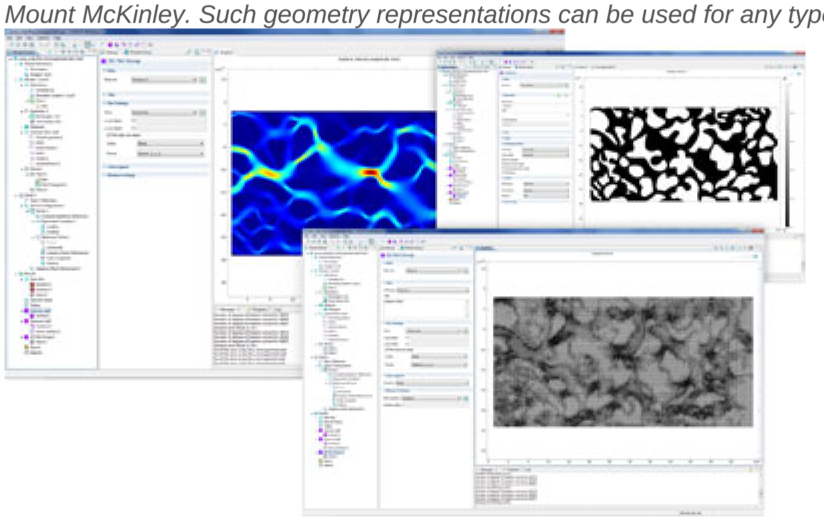
An important application of image import is for easy computation of equivalent volume-averaged material properties for highly inhomogeneous or porous materials. This includes properties such as conductivity, permittivity, elasticity, or porosity and allows for converting spatially distributed values to a single representative averaged value. Such equivalent material properties can then be used for simulations of larger structures avoiding detailed microscopic information. This modeling approach has several advantages such as avoiding the often difficult operations of image segmenting and image-to-geometry conversion. It also brings greatly simplified meshing, less memory usage, and shorter computation times--this can be particularly important when the same type of analysis needs to be repeated many times for different images.

An imported image is made available as a general COMSOL interpolation function that can be used for any modeling purposes.

VIEW SCREENSHOT



A tetrahedral volumetric mesh created on a geometry that combines rectangular solids with imported DEM data of the topography of Mount McKinley. Such geometry representations can be used for any type of simulations in COMSOL Multiphysics.



The pictures shows a simulation where the pore structure is represented implicitly by a gray-scale picture instead of explicitly by a CAD geometry. A flow simulation of the structure is run against a mesh generated on a single rectangle. Such meshes can be generated very quickly and without human intervention. An adaptive mesh was used to increase accuracy. The level of detail of the simulation result can be controlled very easily by coarsening or refining the mesh and be set to meet specified solution time goals where accuracy is traded against speed.

Digital Elevation Map (DEM) Import

Topographic data from geographic information system (GIS) applications can now be imported with a new Digital Elevation Map interpolation function feature with direct support for the DEM file format from U.S. Geological Survey (USGS). You can freely combine DEM surfaces with other surfaces and solids to form a volumetric representation of both geometry and mesh. Multiple DEM surfaces can be combined and intersected as well as embedded inside of other geometrical objects in order to form composite structures. This function utilizes the parametric surface geometry primitive to enable resolution control by varying the number of “knots” of an underlying approximation surface. This way you can start with a rough approximation of the DEM data for quicker computations and when you are satisfied with your simulation setup successively increase the level of detail until enough geometric detail is achieved. The benefit is precise control over memory usage and computation time.

Geometric structures resulting from DEM import are generic in the COMSOL environment and handled in the same way as mechanical CAD. This means that the full power of COMSOL Multiphysics is available for DEM geometry representations and can applied to any single physics or multiphysics simulation such as subsurface flow, electromagnetics, acoustics, and structural mechanics.

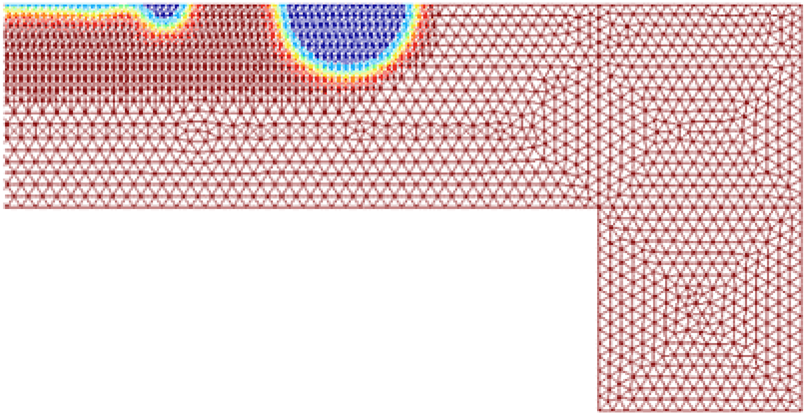
Parametric Sweeps: Accumulated Probe Tables and Response Surfaces

Parametric sweeps can now create Accumulated Probe Tables which enables a Probe to write multiparameter data to tables. For example, the table can include the results from a nested parametric sweep with two independent parameters. From the table you can create a new Table Surface plot for plotting 2D response surfaces and a new Table Graph for a 1D graph plot of the results versus a parameter.

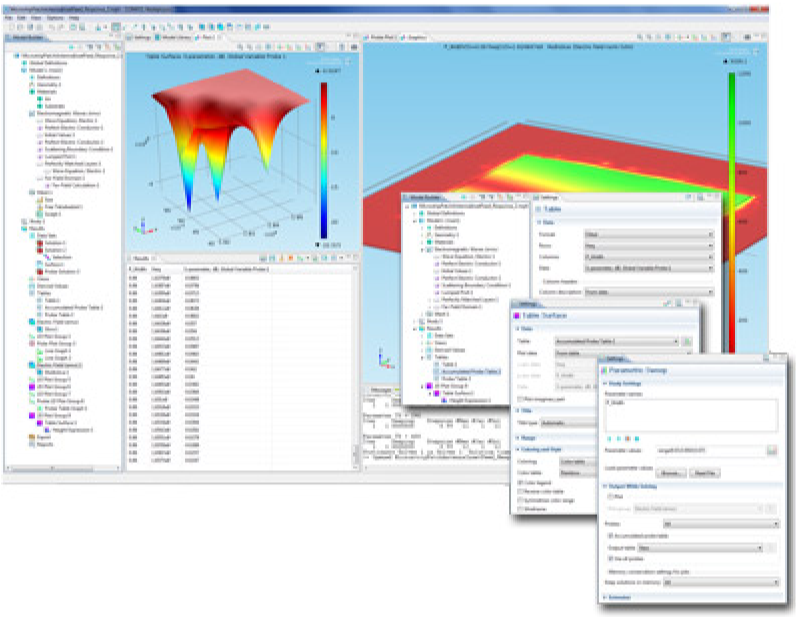
A new user interface for memory conservative parametric sweeps makes it easier to run large parametric sweeps where only a few derived scalar values, and not the entire solution, need to be saved per parametric step.

Time-Dependent Mesh Adaption and Automatic Remeshing

The time-dependent mesh adaption and automatic remeshing capabilities have been enhanced and generalized. The time-dependent mesh adaption algorithm now predicts the next mesh refinement by pre-solving on a coarse mesh. For two-phase flow simulations this results in an adaptive mesh that more closely follows the phase interface and gives more accurate results.



This example demonstrates how to model the fluid flow of an inkjet nozzle, for instance, in a printer. An ink droplet is ejected through the nozzle and travels through air until it hits the target. The fluid flow is modeled by the incompressible Navier-Stokes equations with surface tension. This model uses the level set method and also makes use of adaptive meshing.



The new Table Surface feature makes it easy to plot results as functions of multivariate parametric sweeps. The pictures show the electric field of a microstrip patch antenna together with settings windows and a response surface plot of the S11 parameter vs. geometric width and frequency.

Combined Stationary and Time-Dependent Solutions

A new Study option gives you complete control over combined stationary and transient simulations involving different physics phenomena. For each time-step of a transient simulation you can automatically use a stationary solution of a different study and physics. This has important applications for particle tracing, where the particle trajectory simulation is transient but where particle forces are taken from a stationary solution field. The new tools are available at the bottom of the Time Dependent Study Step settings window in the section called Values of variables not solved for, and is used in combination with the Physics Selection, which is available in the same settings window.

VIEW SCREENSHOT

Compare Solutions on Different Meshes by the Join Data Set

The new Join Data Set is used to compare solutions corresponding to different meshes, time steps, or parameter values. You can form combinations of solutions using the operations difference, sum, product, quotient, and more general and explicit expressions. An important application for the Join Data Set is to plot and evaluate the difference between two solutions in a mesh convergence study.

VIEW SCREENSHOT

Multislice Plots

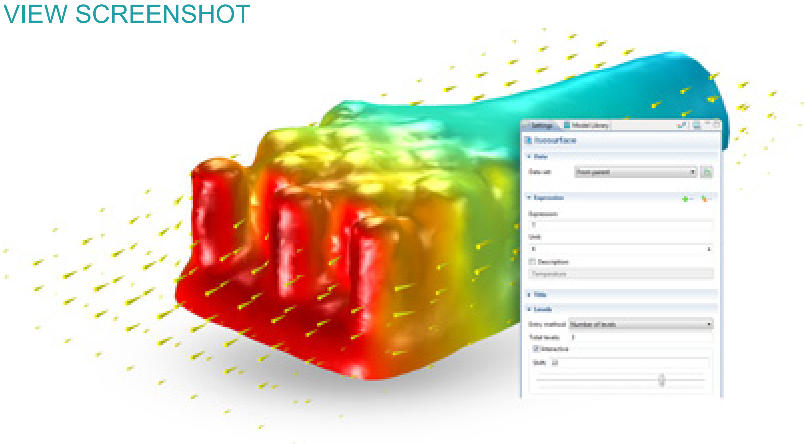
Multislice plots provide a shortcut for generating multiple slices in different directions. The default option is to create three slice planes parallel to the x, y, and z coordinate planes. The Multislice plot type is one of the quickest ways to probe the inside of the computed domain and is available in the More Plots section of any 3D Plot Group.

VIEW SCREENSHOT

Import of External Data to Tables

Import of external data is now available for tables. The imported data can be from a spreadsheet or text-file and used for analyzing and plotting experimental data against simulated results.

VIEW SCREENSHOT

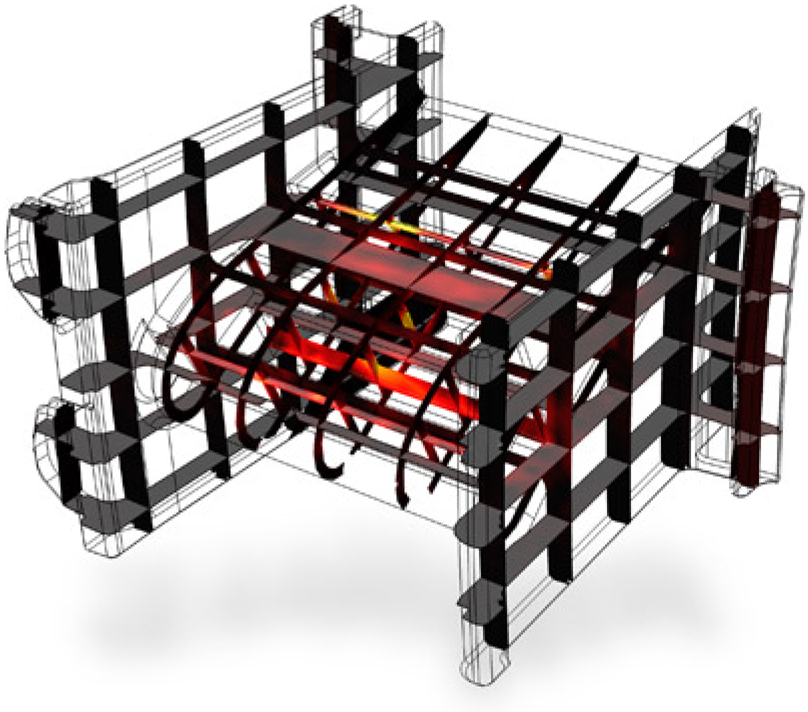


Isosurface levels can now be interactively changed using a slide controller. Multiple isosurfaces can be simultaneously positioned.

Custom Plot Titles

The Title section for plots now provides a Custom setting for creating a customized plot title. When you select Custom you get a number of options for the typical components of a plot title: the data set, its phase and solution when applicable, and the type, description, expression, and unit for the plotted quantity. You can also add a user-defined prefix and suffix.

VIEW SCREENSHOT



This visualization shows the difference in temperature between solutions corresponding to two different mesh densities for a thermal stress simulation. The new Join Data Set is used to compare solutions corresponding to different meshes, time steps, or parameter values.

Interactive Slice and Isosurface Plots

Any scalar quantity of interest can be visualized by slice plots or isosurface plots. Quantities visualized can be one of many predefined expressions or be typed in as a user-defined expression. New in version 4.2a is that slice plots and isosurface plots can be interactively positioned using a slide controller. Slices may be created by giving the total number of evenly distributed slice planes or by exact positioning using coordinate values. Similarly, isosurfaces may be created giving the total number of evenly distributed isosurface levels or the exact value of the levels. Isosurfaces may in addition be colored using a completely different field quantity as a Color Expression. A non-interactive slice or isosurface plot can be turned into an interactive by just selecting a checkbox.

VIEW SCREENSHOT

Data Operations on Results

Several new operators are available for postprocessing. For time-dependent simulations the `timeint()` operator enables time integration of already computed time-dependent solutions. The `timeavg()` operator similarly computes the time-averaged value of any expression.

For small-signal and prestressed analysis, the operator `lintotalavg()` evaluates the average of an expression over all phases for a linearized solution. The operator `lintotalrms()` evaluates the root mean square (RMS) of an expression over all phases for a linearized solution. The operator `lintotalpeak()` evaluates the maximum of an expression over all phases for a linearized solution.

VIEW SCREENSHOT

Model Builder Tree Updates

You can now select multiple nodes of the Model Builder tree simultaneously to quickly delete entire chunks of model definitions. New Previous Node and Next Node arrow buttons helps quick navigation between modeling steps.

VIEW SCREENSHOT

Default Nodes

The Model Builder tree of Version 4.2a comes with clearly indicated default nodes for all physics interfaces. In the Model Tree, a D in the upper-left corner indicates that the node is a default node.

VIEW SCREENSHOT

Automatic Inverse of Interpolation Data

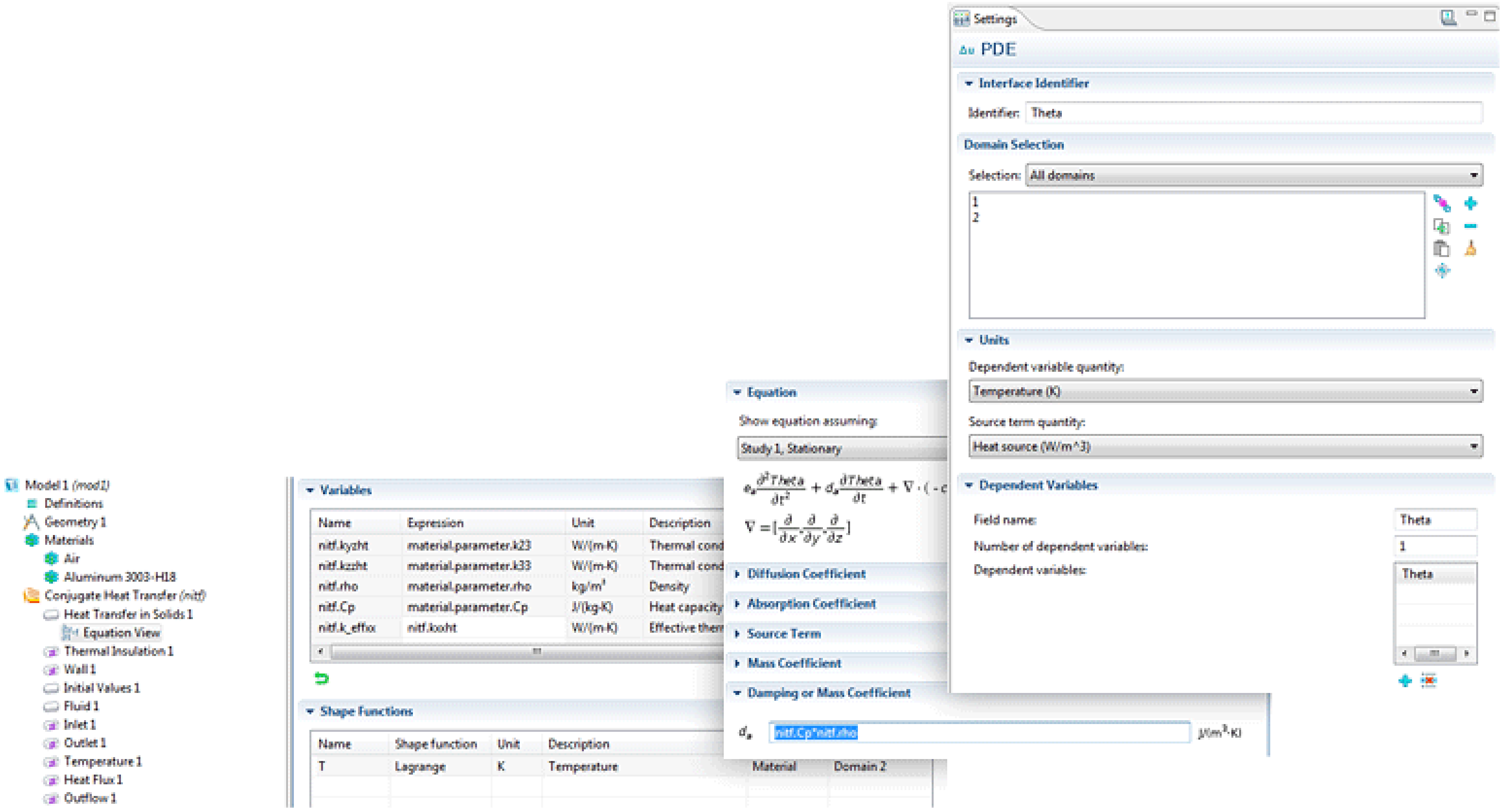
The Interpolation table feature has been extended with an automatic function inverse. This option is available in the Interpolation settings window for 1D interpolation tables. If the original function has the name int1(x), then its inverse is by default made available as int1_inv(x). The name of both functions can be edited. Interpolation table functions and function inverses are made available in most text fields including those for initial conditions, material settings, boundary conditions, and results.

VIEW SCREENSHOT

Units and Material Properties in Equation-Based Models

The equation-based interfaces for Partial Differential Equations (PDEs), Ordinary Differential Equations (ODEs) and Differential Algebraic Equations (DAEs) now support units. By declaring quantities for the dependent variables and the source terms, the equation interfaces define and display units for all equation terms and quantities. This makes it possible to mix equation-based modeling with other physics interfaces and at the same time make full use of the unit system in the model. You can switch off the unit handling for working with dimensionless quantities.

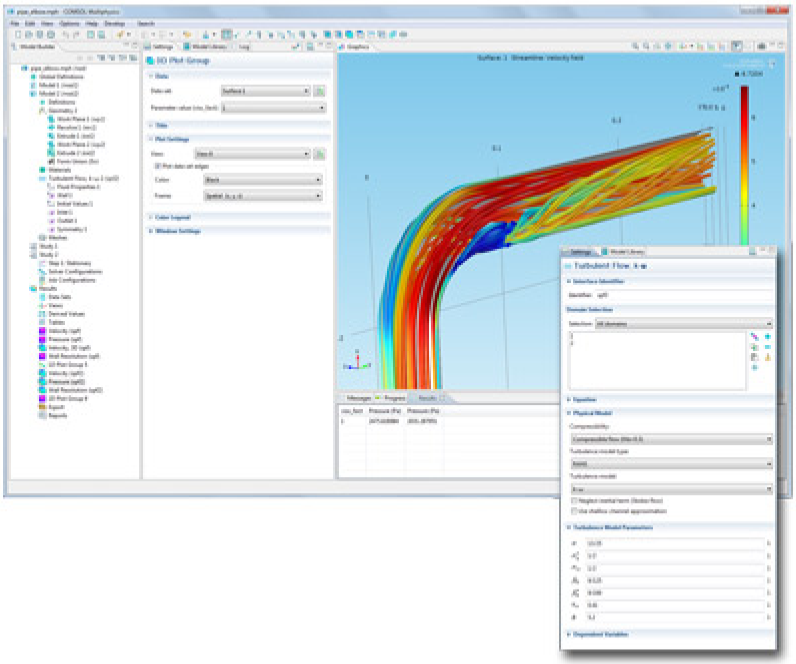
For equation-based modeling you can now access material property variables of library materials when defining your own expressions or equations. A New material container variable root.material simplifies access to material data. For example, root.material.rho is the density rho as defined by the materials in each domain in the geometry. For visualization, you can type the expression material.rho to create a plot that shows the density of all materials.



Unit handling is now available for partial differential equation modeling. In this example, the expression `nitf.Cp*nitf.rho` *is automatically identified as having the correct unit* `J/(m^3*K)`*. Expressions with wrong units are highlighted in orange.*

New k-ω Turbulence Model

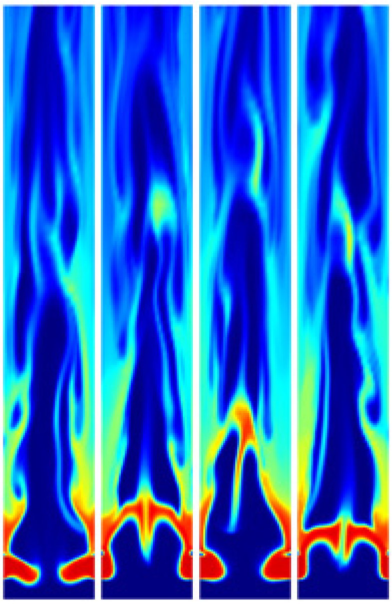
The well-known k-ω turbulence model is now available in the CFD Module Version 4.2a. and corresponds to the so called revised Wilcox model. Even though it can be more demanding to apply than the standard k-ε model, it can often give more accurate results. The turbulence modeling user interfaces of the CFD Module use the Reynolds-averaged Navier-Stokes (RANS) equations and solve for the averaged velocity field and averaged pressure. In addition to the new k-ω turbulence model, different models for the turbulent viscosity are available since earlier versions: a standard k-ε model, a Low Reynolds number k-ε model, and a Spalart-Allmaras model.



A simulation of water flow in a 90 degree pipe elbow. The flow is simulated using the newly added k-omega turbulence model. The result is compared to engineering correlations and this tutorial is available on the web through COMSOL's Model Library Update.

Laminar Euler-Euler Two-Phase Flow

The new Euler-Euler Model user interface for two-phase flow is able to handle similar types of simulations as the Bubbly Flow and Mixture Model user interface but is not limited to low concentrations of the dispersed phase. In addition, the Euler-Euler Model interface can handle large differences in density between the phases, such as the case of solid particles in air. This makes the model suitable for simulations of fluidized beds.



Snapshots of the solid phase volume fraction inside a two-dimensional fluidized bed, taken at four different times; t = 10s, 13s, 16s, and 19s after the start of the simulation. Air is injected at the bottom of the bed, while the solid phase and air is injected through two vertical slots just above the air inlet. The solid inlet mass flux is kept at a rate matching the outlet flux at the top of the bed.

Interior Wall

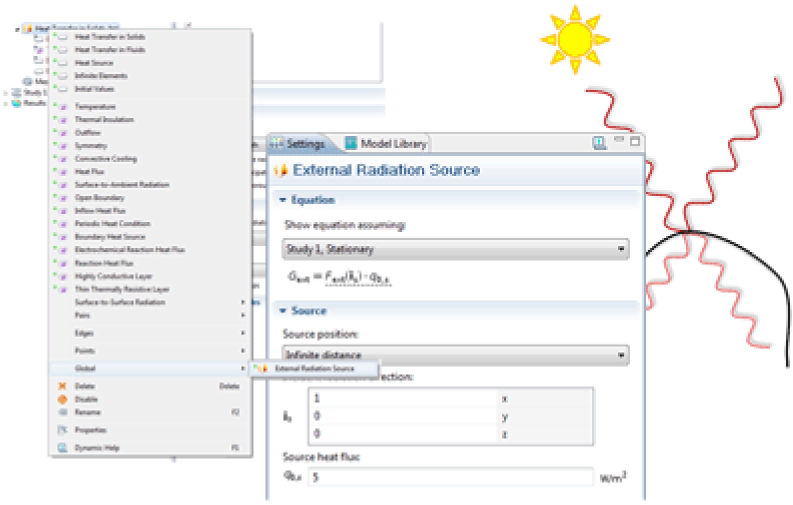
The new Interior Wall boundary condition for single-phase flow makes it easy to define a thin-wall condition between two fluid domains. You no longer need to define a solid domain with a wall boundary condition on both sides, which can result in a dense mesh. This new boundary condition is available in both the CFD Module and the Heat Transfer Module and can also be used together with the Fluid-Structure Interaction multiphysics interface of the Structural Mechanics Module and the MEMS Module.

VIEW SCREENSHOT

External Radiation Sources

External radiation sources can now be defined in the Heat Transfer Module as sources at infinity or as point sources at finite distance. This option is available in the Heat Transfer physics interface and any physics interface that supports surface-to-surface radiation. When defining a source at infinity, the power per unit area is input. This is typically used for incident sun radiation. When defining a point source at finite distance, the total power input is given.

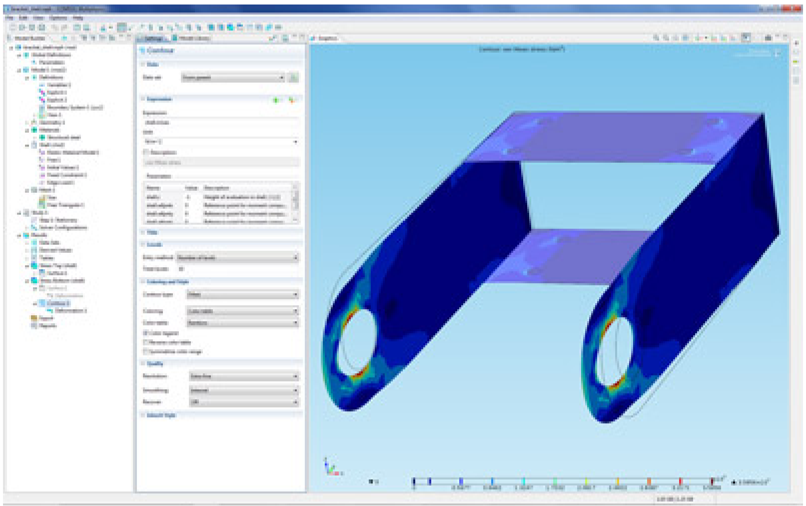
Another new important feature of the Heat Transfer Module is that it you can define radiation on both sides of a boundary when surface-to-surface radiation is used. This new option is available in the Heat Transfer physics interface and any physics interface that supports surface-to-surface radiation.



Expanded Structural Shell Capabilities

The Structural Mechanics Module now supports offsets for shells. This shell property makes is possible to model thin structures where the midsurface is offset from the location of the boundary of the original COMSOL geometry. It also applies to imported CAD models.

Prestressed modal and frequency-response analysis has been available for solids since the previous release and is now also made available for shells. When used for geometrically nonlinear analysis, a shell can be predeformed or prestressed and the modified modal frequency is automatically computed with aid of a very sophisticated and general linearization algorithm. Applications include vibration analysis of any type of prestressed shell structure.



The new version of the Structural Mechanics Module has expanded shell modeling functionality with surface offsets and prestressed vibration analysis. The picture shows surface contours of the von Mises stress at the bottom evaluation level of a bracket shell structure.

New Ways to Specify Isotropic, Orthotropic, and Anisotropic Materials

The Structural Mechanics Module, the Acoustics Module, and the MEMS Module have general support for isotropic, orthotropic or generally anisotropic materials. Voigt material data order is now supported in addition to the previously available standard material data order. The Elastic Waves and Poroelastic Waves interfaces of the Acoustics Module now use Voigt notation by default.

A total of nine different ways of specifying elastic data are now available. The latest addition is that elastic data can be given by the combination of Young's modulus (E) and shear modulus (G).

VIEW SCREENSHOT

New Tutorial Models

The new version of the Structural Mechanics Module includes five new tutorials for important applications:

- **Postbuckling Analysis of a Hinged Cylindrical Shell** Tracing of a postbuckling path where neither the load nor the displacement increases monotonously.
- **Polynomial Hyperelastic Model** This model shows how to implement a Mooney-Rivlin constitutive material model using a user-defined strain energy density.
- **Sheet Metal Forming** Demonstration of plastic metal forming using a rigid punch with elastoplastic deformation, contact, and friction. The results are compared with experimental data.
- **Nonlinear Magnetostrictive Transducer** The magnetic field and displacement as functions of the applied current are computed for a magnetostrictive transducer where the BH curve is nonlinear. This model considers the case when the material is sufficiently prestressed so as to obtain the maximum magnetostriction.
- **Vibration of an Impeller** A tutorial model that demonstrates the use of dynamic cyclic symmetry with postprocessing on the full geometry. This Model can be downloaded from the Model Library Update feature.

Fluid Models for Pressure Acoustics

The Pressure Acoustics interface of the Acoustics Module includes a number of new fluid models. Losses can be accounted for in several different ways in the Acoustics Module. The most advanced user interface covers full Thermoviscoacoustics phenomena. Another way to introduce losses are by using so-called equivalent fluid models directly in the Pressure Acoustics interfaces. This introduces attenuation properties to the bulk fluid in contrast to the thermoacoustic model. The models include losses due to thermal conduction and viscosity, models for simulating the damping in certain porous materials, and macroscopic empirical models for certain fibrous materials. When applicable, the equivalent fluid models are computationally much less heavy than, for example, solving a corresponding full poroelastic model.

VIEW SCREENSHOT

Additional Perfectly Matched Layers

Perfectly Matched Layers (PMLs) for absorbing outgoing acoustic and elastic waves are now also available for Poroelastic Waves, Thermoacoustics, and Structural-Acoustic Interaction. Since earlier versions, PMLs have been available for Elastic Waves, Piezoelectric Waves, Pressure Acoustic Waves and Electromagnetic Waves. PMLs are artificial materials that very efficiently dampen waves and are used to represent infinite computational domains. They give very little or no reflection for a wide range of frequencies and angles of incidence and generalize the concept of non-reflective boundary conditions.

Thermoacoustic-Solid Interaction

The new version of the Acoustics Module has new multiphysics interfaces for thermoacoustic-solid couplings in the frequency domain for 2D, 2D axisymmetric, and 3D models. The Thermoacoustic-Solid Interaction interfaces combine features from the Thermoacoustics and Solid Mechanics interfaces.

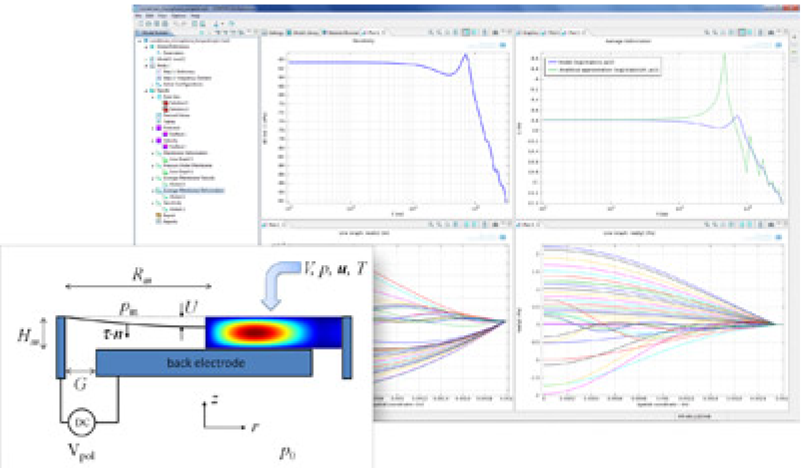
VIEW SCREENSHOT

New Tutorial Models

The new version of the Acoustics Module includes two new tutorials for important applications:

- **Axisymmetric Condenser Microphone with Electrical Lumping** This model is that of a simple axisymmetric condenser microphone. The model includes all the relevant physics and determines the sensitivity of the specific microphone geometry and material parameters. The model uses a lumped approximation for the electric small-signal problem but solves a full finite-element model for the acoustic-mechanical system. The quiescent (zero-point) problem is solved fully using electrostatics and a membrane model. This model requires both the Acoustics Module and the AC/DC Module.
- **Acoustic Levitator** This model is that of a simplified 2D acoustic levitator geometry driven at a constant frequency. Small elastic particles are released uniformly in the standing acoustic field and their path is determined when influenced by the acoustic radiation force, viscous drag, and gravity. This model requires both the Acoustics Module and the Particle Tracing Module.

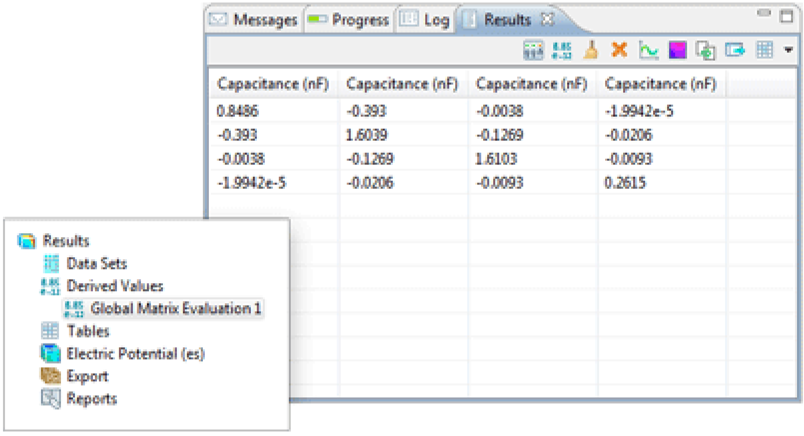
VIEW SCREENSHOT



A new tutorial of a condenser microphone shows how to setup a multiphysics model combining electrical, mechanical, and thermoacoustics effects. It is used to very accurately determine the sensitivity to changes in the microphone geometry and material parameters. This model combines the Acoustics Module and the AC/DC Module.

Capacitance and Lumped Parameter Matrices

A new Global Matrix Evaluation tool computes and displays an entire lumped parameter matrix in one single step. The resulting matrices are displayed directly in a table and they are also available for parametric or frequency sweeps. This functionality is available for all lumped parameters: capacitance, inductance, impedance, and admittance.



Capacitance (nF)	Capacitance (nF)	Capacitance (nF)	Capacitance (nF)
0.8486	-0.393	-0.0038	-1.9942e-5
-0.393	1.6039	-0.1269	-0.0206
-0.0038	-0.1269	1.6103	-0.0093
-1.9942e-5	-0.0206	-0.0093	0.2615

Results node and table output from a capacitance matrix computation using the new Global Matrix Evaluation feature. A four-port electrostatics simulation results in a 4-by-4 capacitance matrix which is displayed in table form.

Automatic Differential Inductance Computation

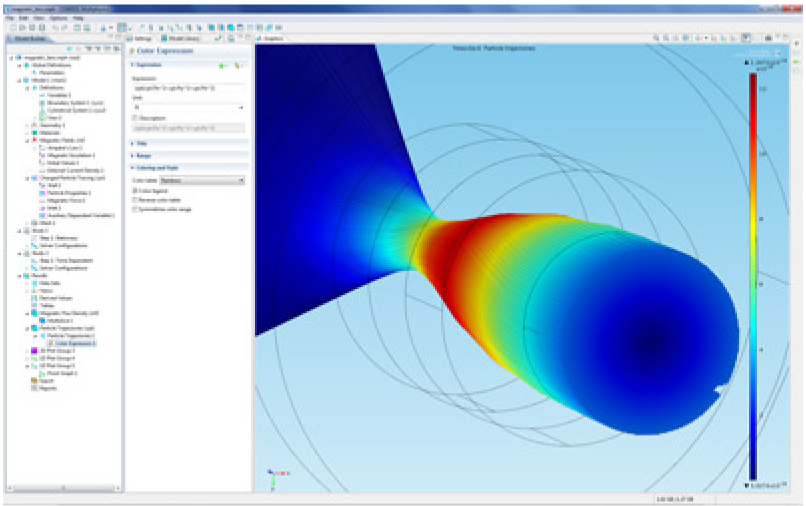
Small-signal analysis, which was introduced in Version 4.2, is now available with automated differential inductance computations. This feature is also available for other lumped parameters such as capacitance and impedance.

VIEW SCREENSHOT

Particle Tracing with the AC/DC Module

The AC/DC Module can be easily be combined with the new Particle Tracing Module for computing charged particle trajectories in electromagnetic fields. Two new examples are available:

- **Magnetic Lens** This model uses the new Charged Particle Tracing user interface to compute the trajectories of electrons in a spatially varying magnetic field. This model requires both the Particle Tracing Module and the AC/DC Module.
- **Quadrupole Mass Spectrometer** This model computes the trajectories of ions of various molecular weights in a quadrupole. There are both AC and DC components of the electric field. This model requires both the Particle Tracing Module and the AC/DC Module.

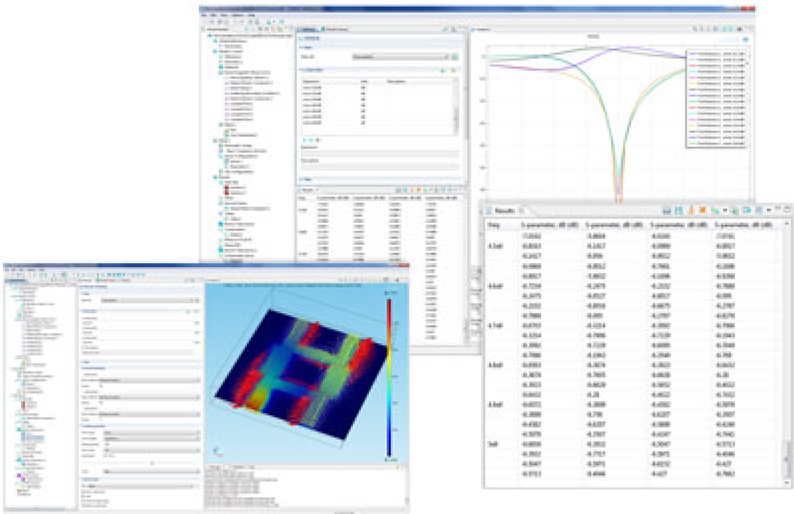


Charged particle trajectories in a magnetic lens using a combination of the AC/DC Module and the Particle Tracing Module.

S-Parameter Matrices

A new Global Matrix Evaluation tool computes and displays the entire S-parameter matrix in one single step. For a frequency or geometric sweep it computes and displays the entire matrix in a table - which can used for a response graph or surface visualization using new table graph and table surface features.

NEW MODEL TUTORIAL



An S-parameter matrix frequency sweep for a branch line coupler. The device can be used for a single antenna TX/RX system or I/Q signal splitter/combiner.

Electromechanics Multiphysics Interface

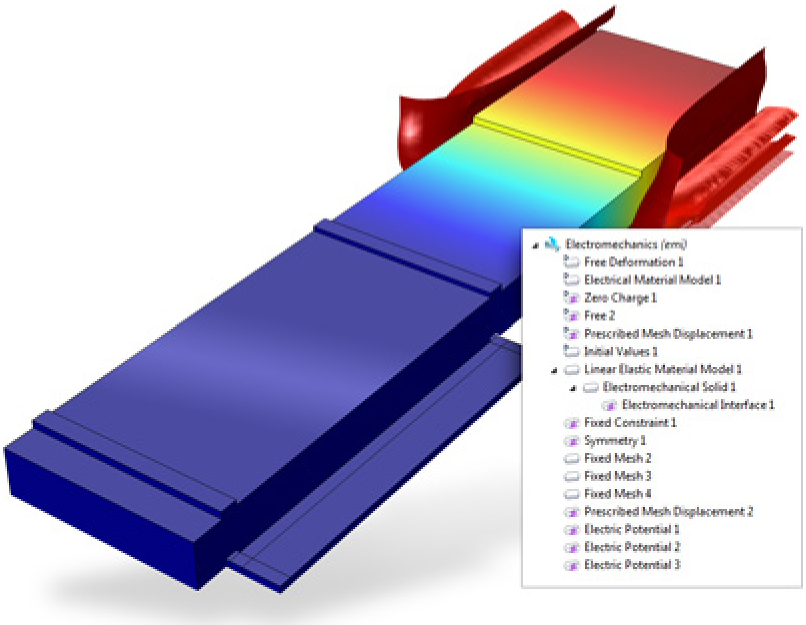
A new Electromechanics multiphysics interface combines solid mechanics and electrostatics with a moving mesh to model the deformation of electrostatically actuated structures. Applications include biased resonator computations with modal and frequency-response analysis as well as pull-in voltage computations.

Several new electromechanical tutorials are available: a suite of 2D and 3D models of a biased resonator showing how to model a stationary analysis, the frequency response, the normal modes, the pull-in voltage, and the transient response. The 3D versions of this suite of models are available from the Model Library Update.

Thin-Film Damping

The thin-film damping user interface has been greatly simplified. You can now add thin-film damping to a boundary directly in the Solid Mechanics interface. In a Fluid-Film Properties subnode you define the fluid properties, gas properties, and rarefaction effects. In a Border subnode you define the border condition: a pressure or a border flow. The Film-Damping Shell interface still exists for coupling film-damping and solid mechanics in the same way as in earlier versions using a multiphysics coupling.

VIEW SCREENSHOT



Biased resonator simulation using the new Electromechanics interface which tightly integrates electrostatics, solid mechanics and moving mesh.

Slip Flow Interface

A new Slip Flow interface is available for modeling thermal and isothermal flows within the slip flow regime. The Slip Flow interface makes it possible to model the flow of the gas, including a thin layer of gas adjacent to the walls (Knudsen layer), where the gas is significantly rarefied. The Slip Flow interface is available in 2D and 3D.

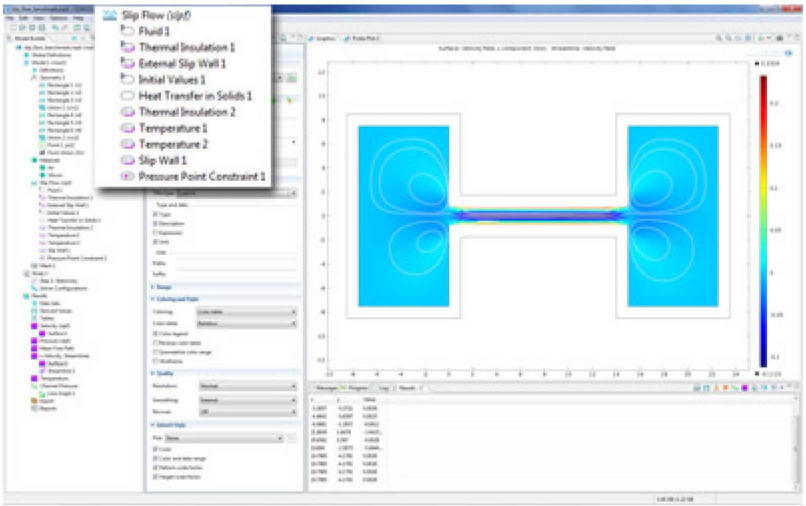
A new Slip Flow Benchmark example model shows the flow between two sealed chambers connected by a microchannel with conducting walls. This model uses the new Slip Flow interface.

Transitional Flow Interface

A new Transitional Flow interface makes it possible to model isothermal flow across the full range of Knudsen numbers from the laminar flow limit to the molecular flow limit. The Transitional Flow interface is available in 2D.

A new Knudsen's Minimum example model, using the Transitional Flow interface, shows that the flow rate of a rarefied gas between parallel plates exhibits a minimum (Knudsen's minimum) at a Knudsen number of about 1.

VIEW SCREENSHOT



A new Slip Flow Benchmark example model shows the flow between two sealed chambers connected by a microchannel with conducting walls. This model uses the new Slip Flow interface.

Ion Energy Distribution Function and Angular Distribution Function

By combining the Particle Tracing Module and the Plasma Module, it is now possible to compute the ion energy distribution function and the angular distribution function. The Ion Energy Distribution Function is important in semiconductor fabrication and surface treatment because it can be manipulated to give precise control over the aspect ratio of nanometer size structures.

VIEW SCREENSHOT

Capacitively Coupled Plasmas (CCP)

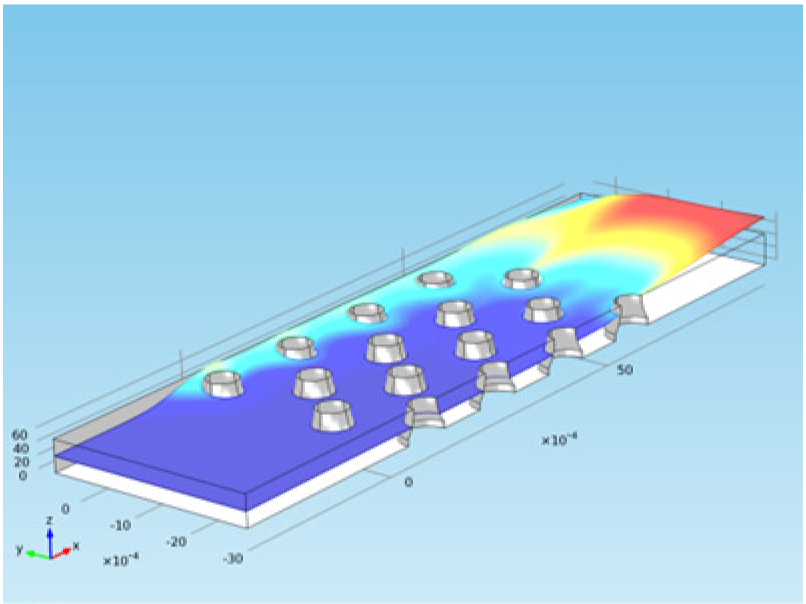
You can now plot cycle-averaged quantities for capacitively coupled plasmas (CCP). A new CCP benchmark model is available that reproduced benchmark results for a one-dimensional capacitively coupled plasma. The model is driven by a constant current rather than a constant voltage. The ion current, power deposition, electron density, ion density and ion flux are all compared to published data.

VIEW SCREENSHOT

CHEMKIN® Import and Parameter Estimation

The new version of the Chemical Reaction Engineering Module features improved performance for CHEMKIN import and improved parameter estimation. In addition, three new model tutorials are available:

- **Parameter Estimation for Nonideal Reactor Models** In this example two ideal CSTRs with interchange are used to model a real reactor with one highly agitated region and another region with less agitation. Two parameters, relating the volume and exchange rate of the two regions, are found by parameter.
- **Microchannel H-cell** This model treats a microchannel H-cell for separation through diffusion. The cell puts two different laminar streams in contact for a controlled period of time. The contact surface is well defined, and by controlling the flow rate it is possible to control the amounts of species that are transported from one stream to the other through diffusion. This highly-requested model was available in Version 3.5a and is now reintroduced in Version 4.2a.
- **Stefan Tube** This example shows 1D steady-state multicomponent gas diffusion. The diffusion of three gases in a Stefan tube is modeled in 1D using the Maxwell-Stefan Diffusion interface. The steady-state mass fraction profiles are calculated. This highly-requested model was available in Version 3.5a and is now reintroduced in Version 4.2a.



A flow cell in a biosensor contains an array of micropillars. The curved side of the pillars are coated with an active material that allows for the selective adsorption of analyte species in the sample stream. The adsorbed species produce a signal that is dependent upon the local concentration at the pillar surfaces. This example investigates the surface concentration distribution in the cell while an analyte pulse is transported through it. It also studies the effect of a quenching surface reaction where adsorbed species are converted into an inactive state. The model illustrates how to use the Surface Reactions interface while coupling mass transport in a fluid stream with chemical reactions that occur on a surface.

Infinite Elements for Current Balance

New Infinite Elements allow the current balance of electrodes and electrolyte to account for unbounded, or infinite, domains. The Infinite Elements are artificial modeling domains added to the outside of the main model and automatically scales the equations to infinity. Using this technique makes it possible to shrink the simulation domain and yet increase the accuracy of a simulation, lowering the computational cost. In the Model Builder Tree, the Infinite Element Domain node is added directly under Model Definitions

VIEW SCREENSHOT

New Tutorials

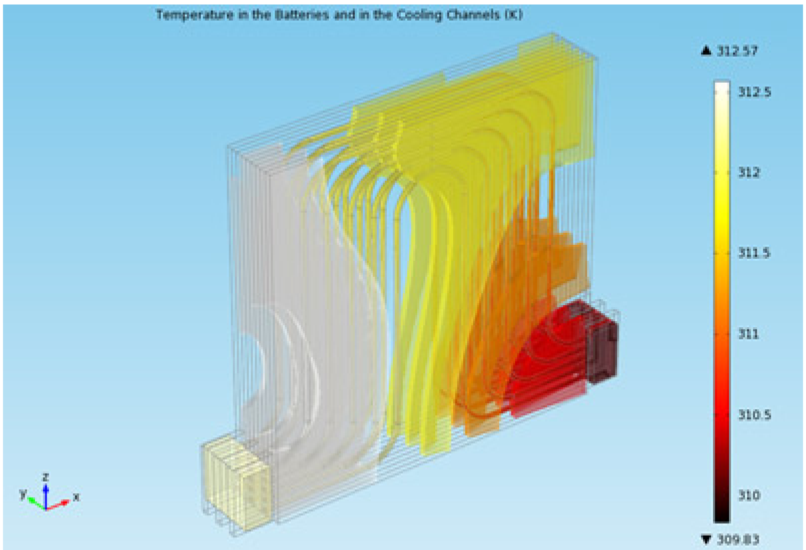
In the new version of the Batteries & Fuel Cells Module, two new model tutorials are available:

- **Liquid-Cooled Li-Ion Battery Pack** This model simulates the temperature profile in a liquid-cooled battery pack. The fluid flow and temperature model are in 3D whereas a lumped 1D model of the batteries is used to calculate the heat source. The model requires the Batteries & Fuel Cells Module and the Heat Transfer Module.
- **Electrochemical Impedance Spectroscopy in a Fuel Cell** This model demonstrates how to perform an electrochemical AC impedance simulation of a fuel cell. It applies a dedicated study type which automatically linearizes the nonlinear simulation and superimposes a given AC signal.

Electrode Power Boundary Conditions

The Batteries & Fuel Cells Module now features an Electrode Power input boundary condition for its Lithium-Ion Battery user interface as well as for several additional user interfaces. You can choose between Average power density or Total power.

VIEW SCREENSHOT

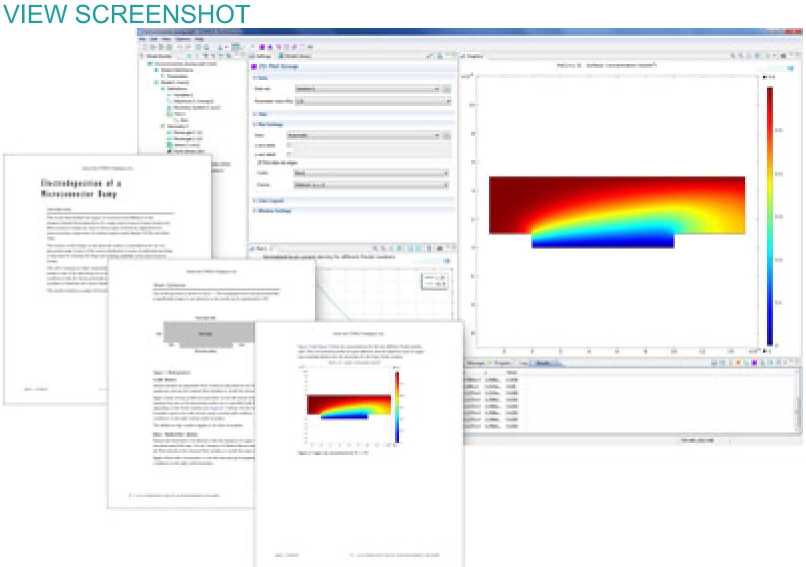


Cooling of a lithium-ion battery pack for automotive applications. This model simulates the temperature profile in a liquid-cooled battery pack. The fluid flow and temperature model are in 3D whereas a lumped 1D model of the batteries is used to calculate the heat source. This model tutorial is now available in the Model Library of the Batteries & Fuel Cells Module.

Infinite Elements for Electrodeposition

Electrodeposition simulations sometimes include large surrounding domains with little geometric detail that influences the electrodeposition process. Such domains can then be approximated as being infinitely large to save computational requirements. New Infinite Elements allow for finite-sized representation of such domains and include the current balance of large parts of electrodes and electrolytes. In the Model Builder Tree, the Infinite Element Domain node is added directly under Model Definitions.

VIEW SCREENSHOT



New Tutorial: Electrodeposition of a Microconnector Bump

A new tutorial shows electrodeposition of a microconnector bump. The deposition process is mass-transport limited and the impact of varied fluid velocities on the current density distribution on the electrode is investigated. Microconnector bumps are used in various types of electronic applications for interconnecting components, for instance liquid crystal displays (LCDs) and driver chips. Detailed step-by-step instructions are available on HTML and PDF format.

The location of the bumps on the electrode surface is controlled by the use of a photoresist mask. Control of the current distribution in terms of uniformity and shape is important for ensuring the shape and resulting reliability of the interconnector bumps.

- COMSOL Version 4.2a Release Notes
- Product Information
- Webinars and Workshops
- 4.2 Release Highlights