

COMSOL
MULTIPHYSICS®
COMSOL
SERVER™

Model Builder

- forced_air_cooling_with_heat_sink.mph
 - Global Definitions
 - Parameters
 - Geometry Parts
 - Materials
 - Component 1
 - Definitions
 - Geometry 1
 - Materials
 - Heat Transfer
 - Turbulent Flow, Algebraic yPlus
 - Fluid Properties 1
 - Initial Values 1
 - Wall 1
 - Inlet 1
 - Outlet 1
 - Multiphysics
 - Non-Isothermal Flow 1
 - Mesh 1
 - Study 1
 - Results

Settings

Non-Isothermal Flow

Heat Turbulence Property

Heat transport turbulence model:

Kays-Crawford

Flow Heating

Coupled Interfaces

Fluid flow:

Turbulent Flow, Algebraic yPlus

Heat transfer:

Heat Transfer

APPLICATION BUILDER

Input and Results

Geometry

Heat sink

Base Geometry: Fin Geometry

Heat sink type: Airfoil

Height: 1.0 cm

Thickness in the thickest: 1.0 mm

Number of fins: 4

Operating

Inlet velocity: 2.0 cm

Outlet distance: 6.0 cm

Lateral distance: 0.5 cm

Top distance: 0.5 cm

Operating Conditions

Inlet velocity: 1.0 m/s

Inlet temperature: 22 °C

Heat source temperature: 100 °C

Results

Dissipated power: 4.957 W

Pressure loss: 0.8725 Pa

Information

Email Message

APPLICATION

Application Library | COMSOL

Forced Air Cooling with Heat Sink

https://www.comsol.com/comsol-server/2136/app/library/Heat_Transfer_Module_Applications_forced_air_cooling_with_heat_sink.mph

Input and Results

Geometry

Heat sink

Base Geometry: Fin Geometry

Heat sink type: Airfoil

Height: 1.0 cm

Thickness in the thickest: 1.0 mm

Number of fins: 4

Operating

Inlet velocity: 2.0 cm

Outlet distance: 6.0 cm

Lateral distance: 0.5 cm

Top distance: 0.5 cm

Operating Conditions

Inlet velocity: 1.0 m/s

Inlet temperature: 22 °C

Heat source temperature: 100 °C

Results

Dissipated power: 4.957 W

Pressure loss: 0.8725 Pa

Information

Email Message

Graphics

Geometry | Mesh | Temperature Field | Velocity Field

Velocity magnitude (m/s)

1.52

1.4

1.2

1.0

0.8

0.6

0.4

0.2

0

0.01

-0.01

About



From Model to Application. Bring Multiphysics to Everyone.

COMSOL® applications realize the vision of fostering collaboration within organizations by providing simulation specialists with the tools to deploy their simulation work, from design and development to production and testing. The idea is simple: Make it possible to model, build, and run simulation apps.

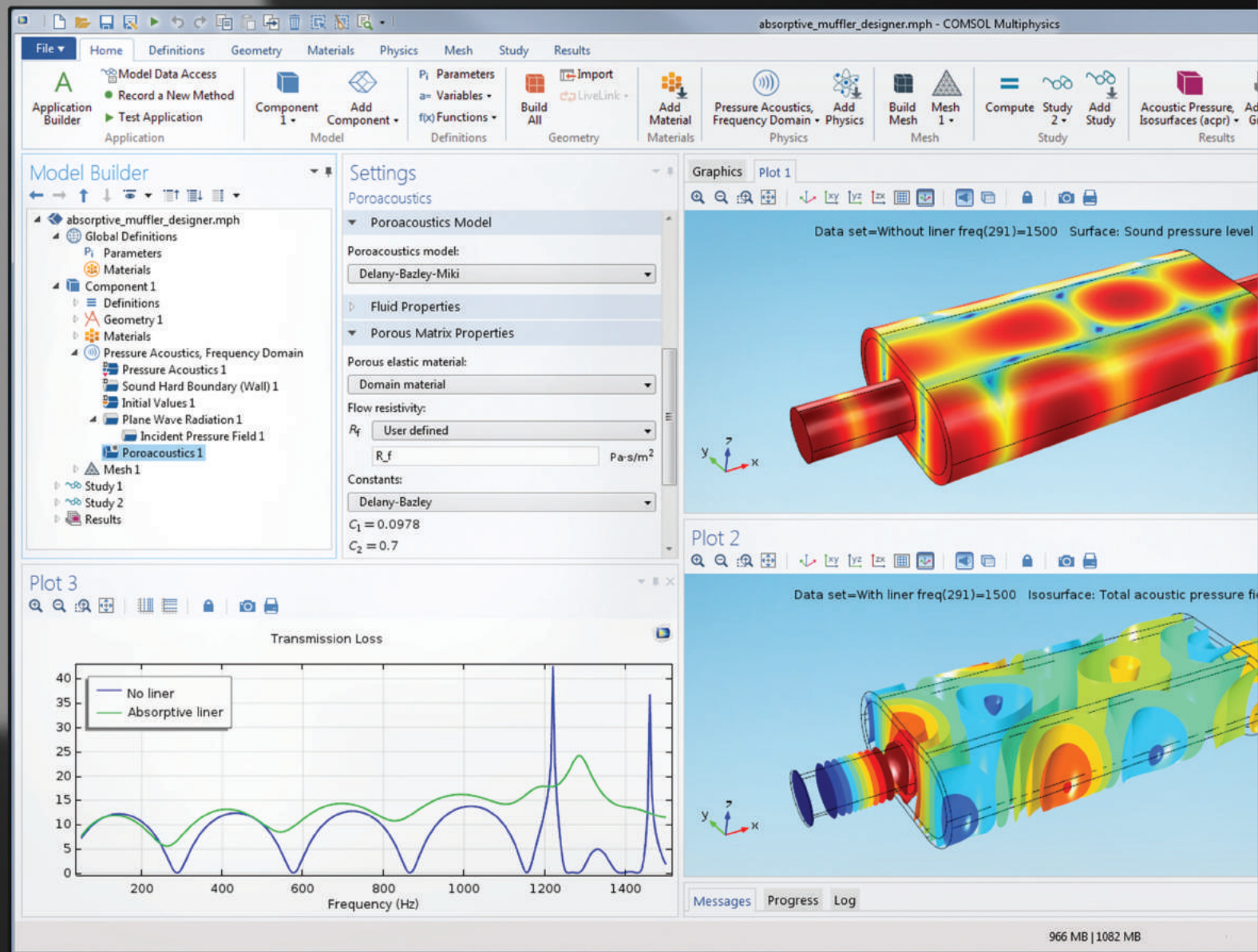
That is exactly what the COMSOL Multiphysics® and COMSOL Server™ products deliver with their fully integrated modeling, simulation, and application design environment.

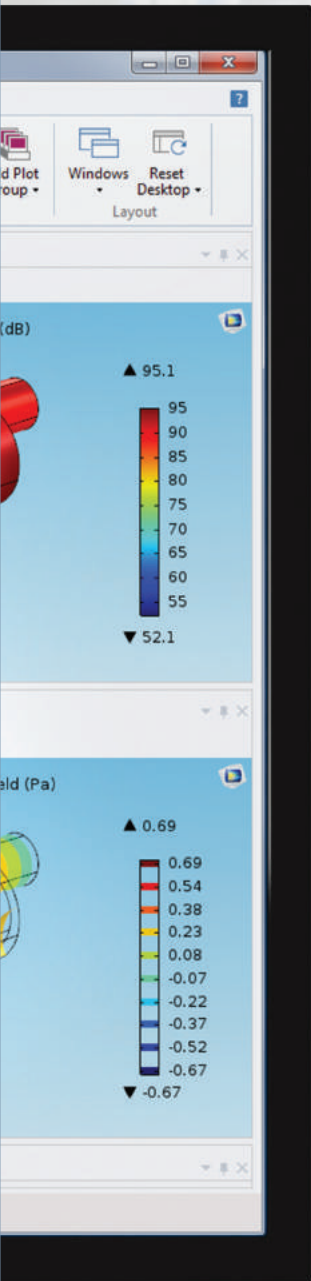
Engineers in all stages of the product design cycle can use the most powerful computational tools and ready-to-use custom applications to drive innovation and boost productivity throughout organizations, from design to production.

Are you ready to build your own applications? COMSOL Multiphysics® and COMSOL Server™ make it easy for you.

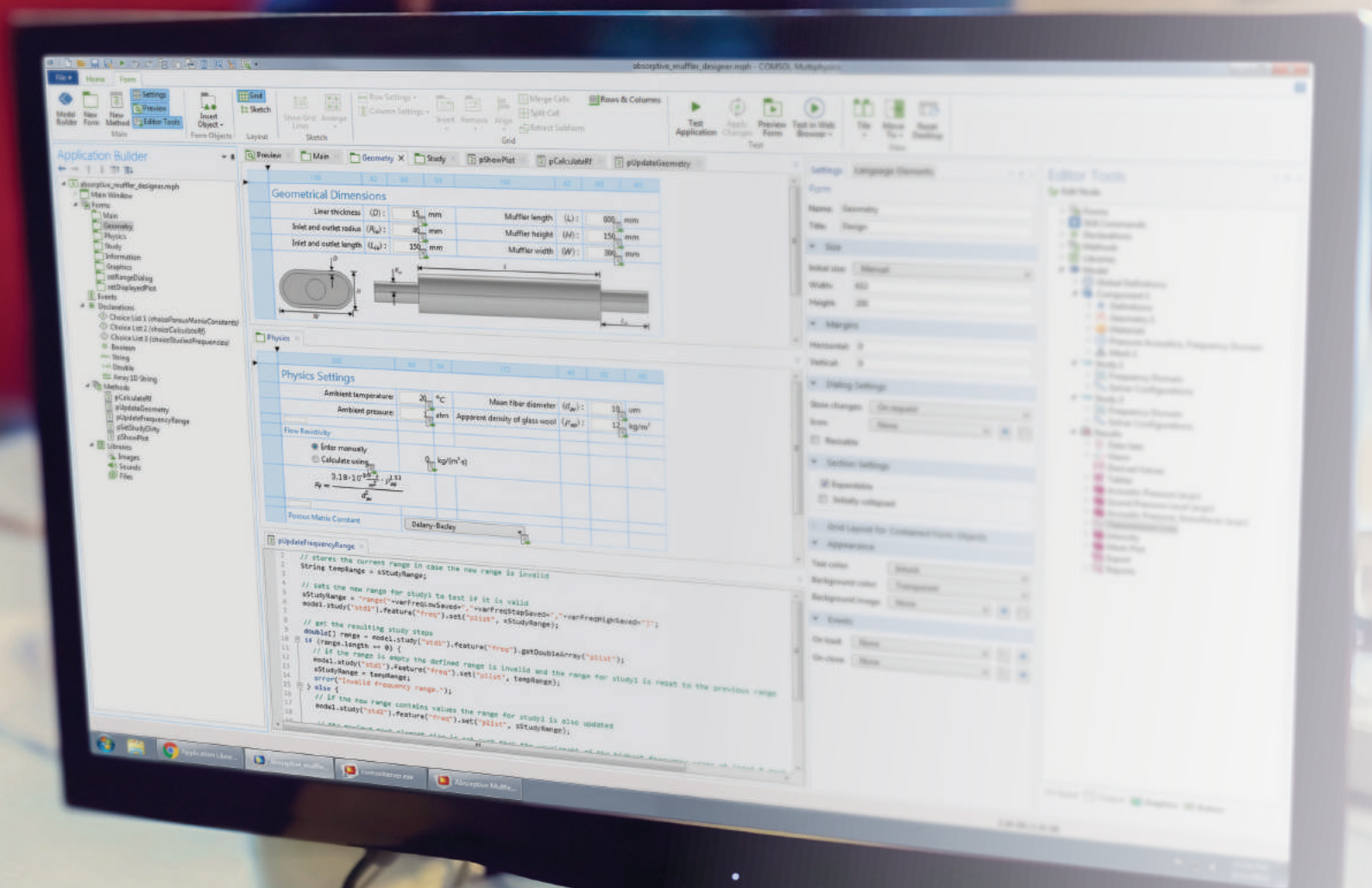
CONTENTS

PRODUCT SUITE	10
COMSOL MULTIPHYSICS	12
COMSOL SERVER	22
APPLICATION-SPECIFIC PRODUCTS	
ELECTROMAGNETICS	26
STRUCTURAL & ACOUSTICS	40
FLUID & HEAT	54
CHEMICAL	68
MULTIPURPOSE	78
INTERFACING	84
USER SUPPORT	97
LICENSE OPTIONS	98

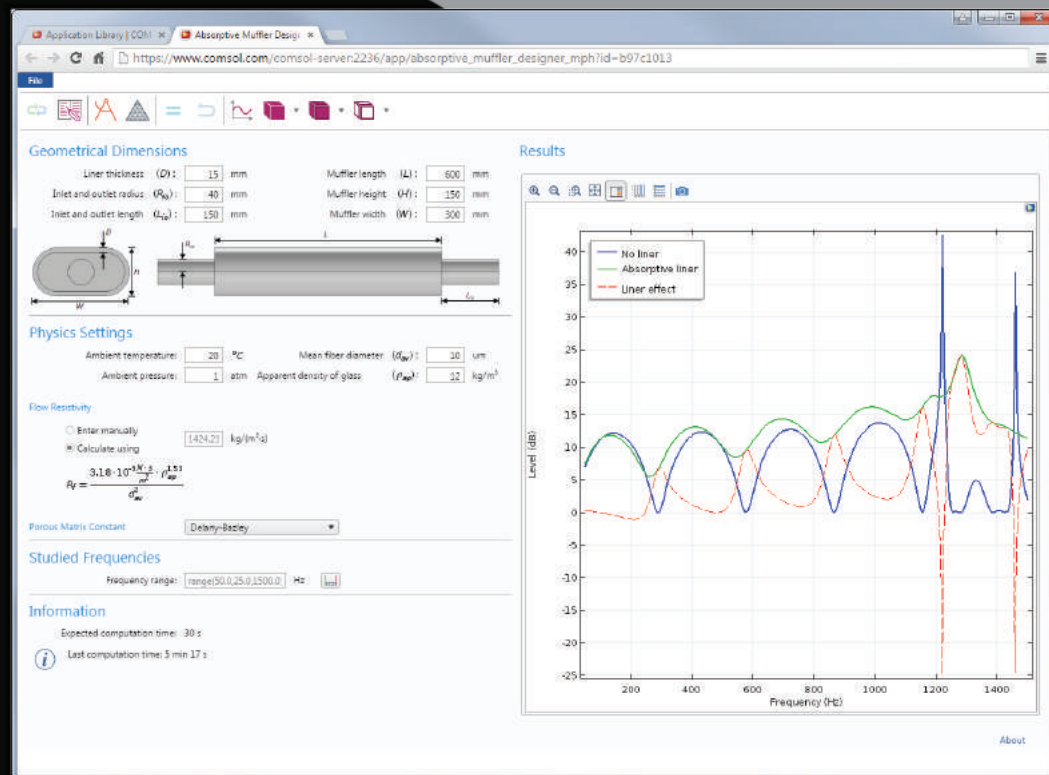




How do you create and communicate innovative designs?



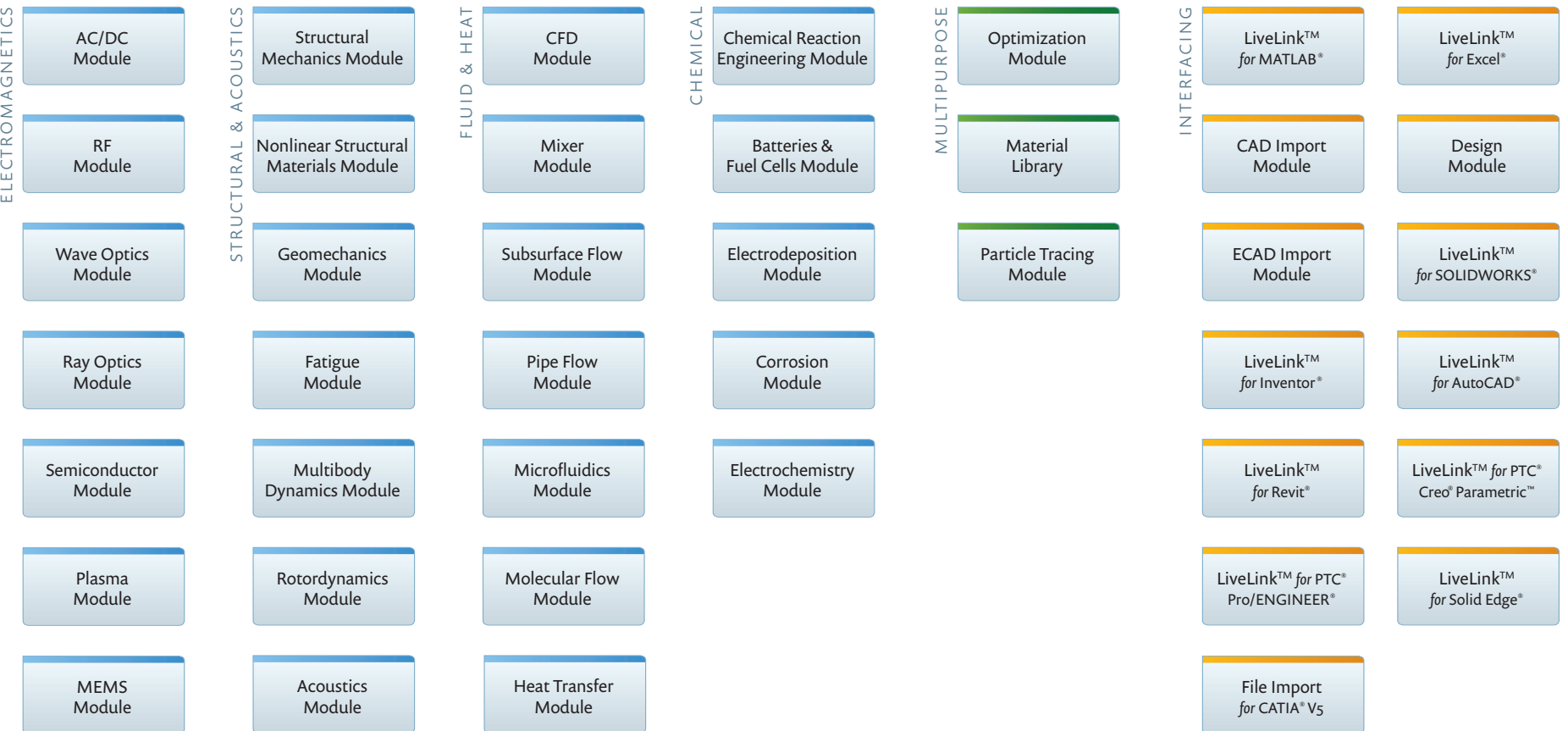
Through powerful computational tools for numerical simulation.



With custom applications
that can be easily deployed.
Worldwide.

COMSOL Multiphysics®

COMSOL Server™



PRODUCT SUITE

COMSOL Multiphysics®	12
COMSOL Server™	22

Electromagnetics

AC/DC Module	26
RF Module	28
Wave Optics Module	30
Ray Optics Module	32
Plasma Module	34
Semiconductor Module	36
MEMS Module	38

Structural & Acoustics

Structural Mechanics Module	40
Nonlinear Structural Materials Module	42
Geomechanics Module	44
Fatigue Module	46
Multibody Dynamics Module	48
Rotordynamics Module	50
Acoustics Module	52

Fluid & Heat

CFD Module	54
Mixer Module	56
Subsurface Flow Module	58
Pipe Flow Module	60
Microfluidics Module	62
Molecular Flow Module	64
Heat Transfer Module	66

Chemical

Chemical Reaction Engineering Module	68
Batteries & Fuel Cells Module	70
Electrodeposition Module	72
Corrosion Module	74
Electrochemistry Module	76

Multipurpose

Optimization Module	78
Material Library	80
Particle Tracing Module	82

Interfacing

CAD Import Module	84
File Import for CATIA® V5	84
Design Module	86
LiveLink™ for SOLIDWORKS®	87
LiveLink™ for Inventor®	88
LiveLink™ for AutoCAD®	89
LiveLink™ for Revit®	90
LiveLink™ for PTC® Creo® Parametric™	91
LiveLink™ for PTC® Pro/ENGINEER®	92
LiveLink™ for Solid Edge®	93
ECAD Import Module	94
LiveLink™ for MATLAB®	95
LiveLink™ for Excel®	96

COMSOL Multiphysics®

The COMSOL Multiphysics® software environment facilitates all of the steps in the modeling process — defining your geometry, meshing, specifying your physics, solving, and then visualizing your results. It also serves as the environment for physics-based modules, which augment the core physics interfaces of COMSOL Multiphysics, and for building simulation applications.

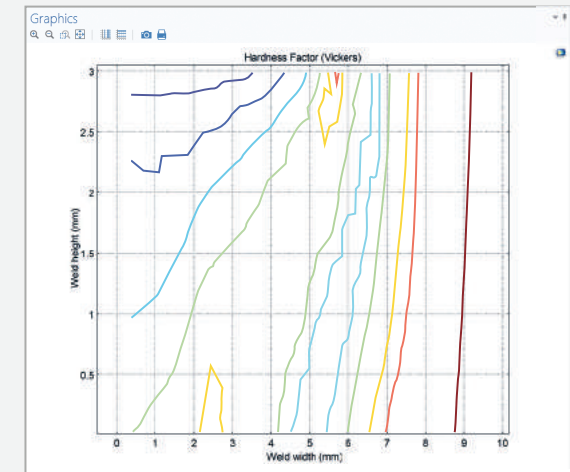
Model setup is quick, thanks to a number of predefined physics interfaces for applications ranging from fluid flow and heat transfer to structural mechanics and electrostatics. Material properties, source terms, and boundary conditions can all be spatially varying, time-dependent, or functions of the dependent variables. You can freely mix physics interfaces into new multiphysics combinations as well as couple them with any application-specific module.

As an alternative to writing your own simulation code, the COMSOL Multiphysics user interface gives you the option to specify your own partial or ordinary differential equations (PDEs or ODEs) and link them with other physics interfaces. When combined with the CAD Import Module or one of the LiveLink™ products, this enables you to run custom simulations on CAD models in many industry-standard formats.

WELDING

Friction stir welding is a solid phase welding process that was developed and patented by The Welding Institute in the 1990s. Since its invention, the process has received worldwide attention and today many companies are using the technology in production, particularly for joining aluminum alloys.

The graph below depicts the temperature distribution in a model that couples fluid flow with friction and heat transfer. The plot shows the hardness factors in the weld as calculated from the model.

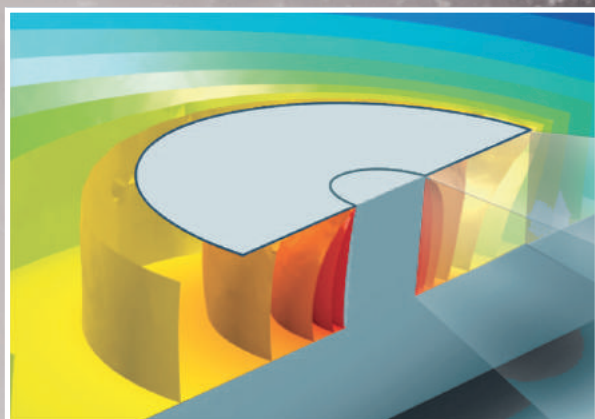


HIGHLIGHTS

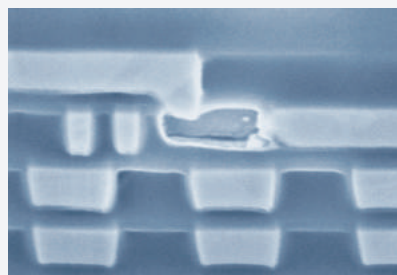
- Model Builder
- Application Builder
- Animation, image, and data export
- Automatic, swept, and boundary layer meshing
- Boundary and volume integration
- Component couplings for linking 1D, 2D, and 3D model components
- Core physics interfaces
- Fast Fourier transform (FFT)
- Interpolation table import for 1D, 2D, and 3D data sets
- Linear and higher-order finite elements
- Mesh import
- Moving and deformable meshes
- Parameterized geometry modeling
- Partial and ordinary differential equations user interfaces
- Probes and probe plots
- Space- and time-dependent expressions for physical properties and boundary conditions
- State-of-the-art multicore solvers
- User-defined linear and nonlinear materials
- Visualization tools, including arrow, contour, line, ribbon, slice, streamline, surface, and volume plots

APPLICATION EXAMPLES

- Beam subjected to traveling load
- Helical static mixer
- Transmission line calculator
- Tubular reactor
- Tuning fork



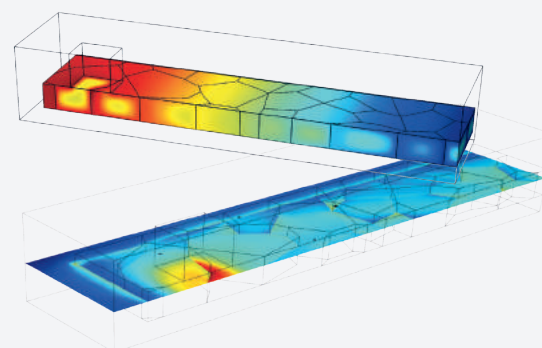
Model courtesy of Dr. Paul Colegrove, Cranfield University, Cranfield, United Kingdom. Pictures courtesy of The Welding Institute (TWI), Cambridge, United Kingdom.



ELECTRONICS

A model predicting a copper line interconnect failure. Simulation results show vacancy concentrations and the resulting von Mises stresses these produce.

Model courtesy of F. Cacho and V. Fiori, STMicroelectronics, Crolles, France.



COMSOL DESKTOP®

Practical simulation software must be straightforward to use regardless of your modeling experience, while also being constructive and powerful enough to achieve your objectives. We have incorporated these qualities into the complete COMSOL Desktop® environment.

This sleek interface gives you full insight into and control over the modeling process, as it concisely reflects the COMSOL Multiphysics architecture. The workflow is integrated and intuitive, guiding you quickly through the building of your simulations and applications.

The COMSOL Desktop structure is uniform throughout, regardless of the physics or application involved. This encourages cross-disciplinary collaboration so that engineering teams can develop better models, faster.

“With its broad internal array of intuitive tools for drawing, meshing, material specification, finite element analysis, postprocessing, and graphical display, COMSOL Multiphysics® version 5 has become one of the favorites of the multiphysics community.”

Bill Vetterling, ZINK Imaging, Bedford, Massachusetts.

MODEL WIZARD

Quick setup of a new model is done in the Model Wizard, where space dimension, physics, and study type are selected.

APPLICATION LIBRARIES

The Application Libraries consist of documented examples and simulation apps from all fields of engineering and science.

MODEL BUILDER

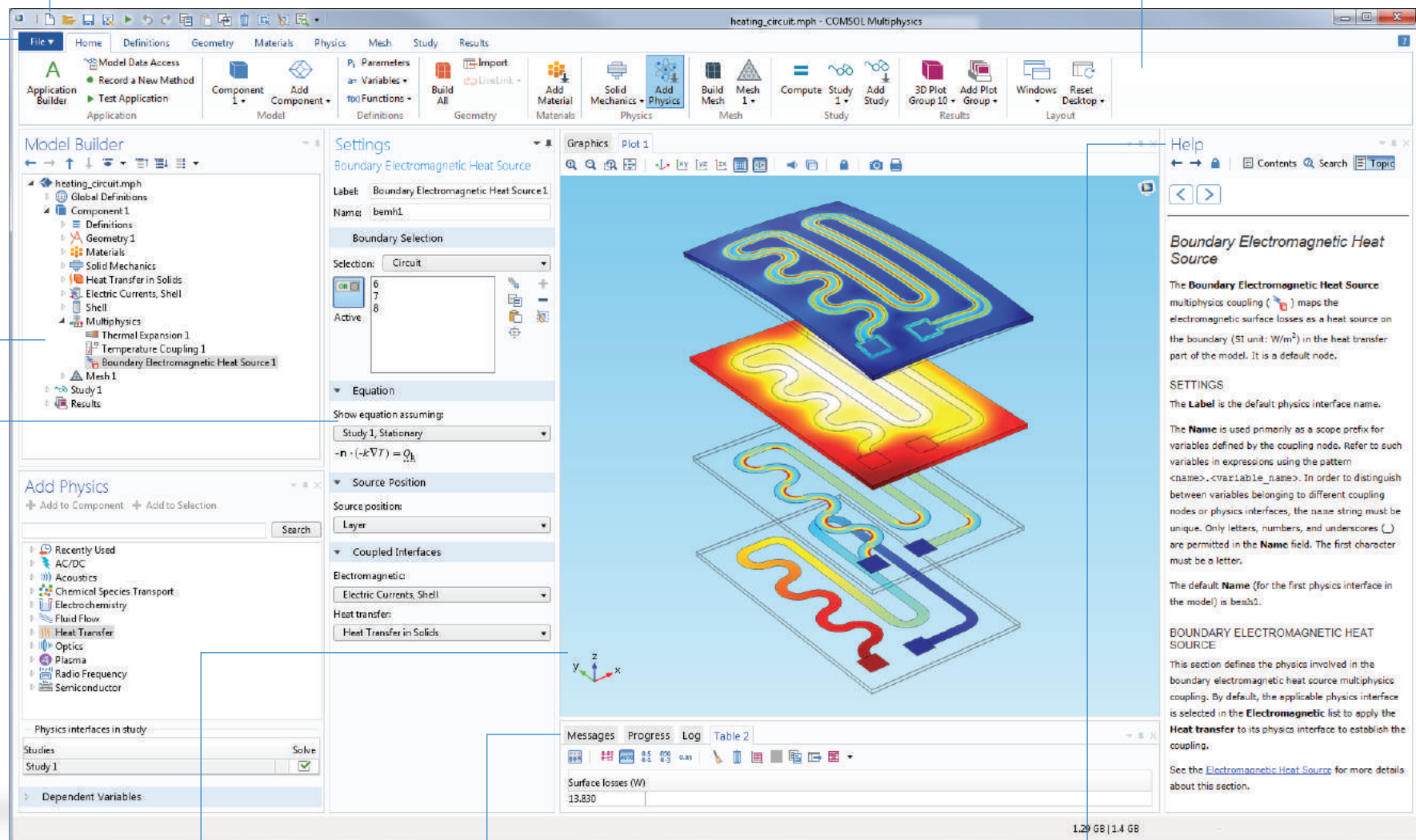
The Model Builder provides access to any part of the model settings.

SETTINGS

Easily enter, verify, and update all specifications and model properties in the Settings window.

RIBBON

The ribbon tabs have buttons and drop-down lists for controlling all steps of the modeling process.



GRAPHICS WINDOW

Ultrafast graphic presentation, stunning visualization, and multiple plots.

INFORMATION WINDOW

The Information windows display vital model information during the simulation, such as the solution time, solution progress, mesh statistics, solver logs, and, when available, results tables.

DYNAMIC HELP

The Help feature enables easy browsing with extended search functionality and is continuously updated with online access to the Knowledge Base and Application Gallery.

MODEL BUILDER

Building a model in the COMSOL® software naturally follows your line of thinking, from concept to realization. The entire model workflow is controlled from the Model Builder, which brings a dynamic, logical structure to your simulations.

A model can be built by following the branches in the model tree structure, from parameter definition and geometry creation to visualization of the simulation results. Nodes can be added to refine and improve your model as it evolves into a true description of your design.

The Model Builder allows instant access to any part of the model settings. You can investigate, refine, and optimize a modeling feature at any stage of your process. This makes the Model Builder a versatile graphical programming tool for parametric analysis, optimization, and customized simulation routines.

“COMSOL allows us to couple mechanisms in a very nice and simple way, allowing us to understand their combined effect. This is the true power of simulation.”

Dr. Roberto Suarez-Rivera, Schlumberger, Salt Lake City, Utah.

DEFINITIONS

Easy control of your parameters, functions, probes, and graphics display from the Definitions branch.

GEOMETRY

Use the built-in drawing tools or import files from a CAD package to create your geometries in the Geometry branch.

MATERIALS

Specify material properties under the Materials branch using expressions and functions, or import a material from one of the libraries.

PHYSICS

Define your physics and choose material models, boundary conditions, loads, and sources.

MESH

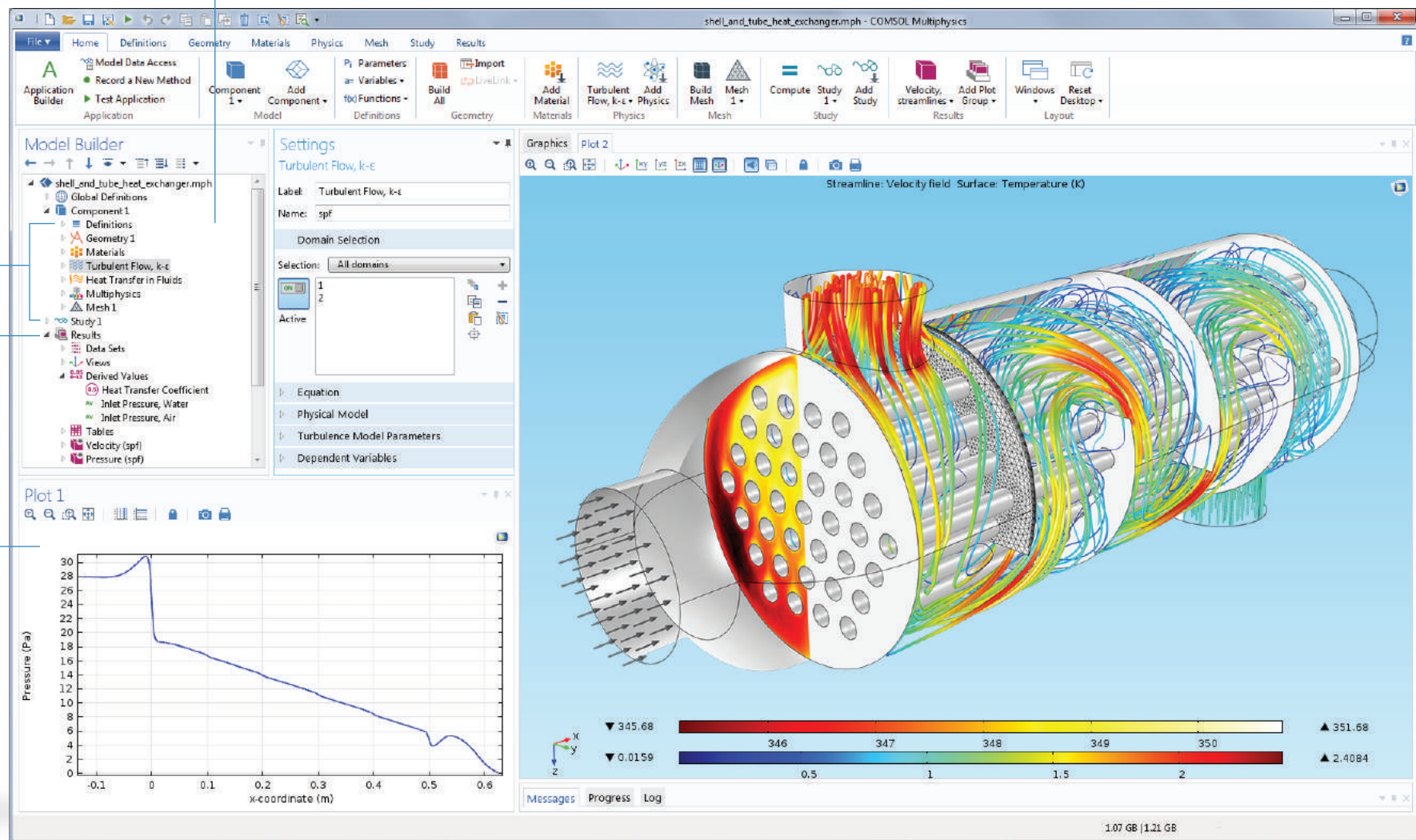
Automatically generate and customize your mesh for optimal resolution and solving.

STUDY

Run a simulation, configure parametric studies, and other sequences of solvers.

MODEL TREE

The model tree gives an overview of the model and provides all of the functionality and operations needed for creating a model as well as postprocessing the results.



RESULTS

Derive and visualize results quantities, probes, and convergence plots. Several Plot windows can be used to show multiple results simultaneously.

APPLICATION BUILDER

The Application Builder, included in the COMSOL Multiphysics® software for the Windows® platform, provides all of the tools needed to build and run simulation applications. Any COMSOL Multiphysics model can be turned into a custom application with its own interface, using the Application Builder desktop environment.

The Application Builder provides two distinct tools for creating applications: The Form Editor and the Method Editor. The Form Editor allows you to build a custom user interface through drag-and-drop of a wide range of form objects such as input fields, graphics windows, and buttons.

The Method Editor is used to implement customized functionality and enhance the user experience of a simulation application. It is a programming environment that allows you to write Java® code to modify the data structure representing a COMSOL Multiphysics® model.

APPLICATION TREE

The application tree gives an overview of forms, events, declarations, methods, and libraries used in the app.

“The Application Builder is the biggest innovation yet to come from COMSOL. We estimate that by building applications for our colleagues, we would save 30-40% of the time we currently spend on calculations for them. This will improve both the way we work and the way our colleagues work as well.”

Massimo Bechis, Prysmian, Milan, Italy.

FORM EDITOR

Use form objects such as input fields, graphics, and buttons to lay out the user interface of a simulation app.

METHOD EDITOR

Write methods for actions not covered by the standard use of the model tree nodes.

The screenshot displays the COMSOL Multiphysics Application Builder interface. The main window is titled 'Corrugated Circular Horn Antenna Simulator'. The interface is divided into several panes:

- Application Builder (Left):** Shows a tree view of the application structure. The 'Main Window' contains 'Forms' (mainForm, sfCommandConsole, sfDesignParam, sfInformation, sfTarCriterion, sfTitle, dfReportFormat, dfProgress) and 'Methods' (fRunAll, mAperturePol, mFeedPol, mGeomInitialization, mGeomView, mLayoutInfo, mOpenDoc, mPrepareReport, mRadPattern2D, mRadPattern3D, mResetParam, mRunReport, mUpdateGeom, pGeomNotUpdated, plnInputValidity). The 'Methods' pane is currently selected.
- Form Editor (Center):** Displays the 'mainForm' design. It includes a 'Design Parameters' section with input fields for:
 - Waveguide radius: 0.183 m
 - TE11 cutoff frequency (fc): 4.8E8 Hz
 - Frequency: fc*1.2 Hz
 - Wavelength (λ): 0.5205 m
 - Flare angle: 17 °
 - Corrugation thickness: 0.105 m
 - Corrugation length: 0.155 m
 - Horn thickness: 0.5 m
 - Horn length: 4 m
 - Waveguide length: 1 m
 - Matching corrugation length: 0.25 m
 Below this is a 'Result Analysis' section with:
 - Output cross-polarization ratio target: 5 %
 - S-parameter (S11): -12.297 dB
- Method Editor (Right):** Shows the 'fRunAll' method code. The code includes a progress dialog, a loop to update the geometry, and a final check for the output cross-polarization ratio.


```

1 dialog("dProgress");
2 long t0 = timeStamp();
3 mUpdateGeom();
4
5 if (validInput) {
6     solution_state = "nosolution";
7     target_state = "nosolution";
8
9     model.study("std1").run();
10    model.result("pg2").run();
11    model.result("pg5").run();
12    model.result("pg6").run();
13    model.result("pg7").run();
14
15    model.result().numerical("gev1").setResult();
16    setFormObjectText("mainForm/labelS11dB", String.format("%.3f", model.result()
17
18    model.result().numerical("int1").setResult();
19    model.result().numerical("int2").setResult();
20    double co_pol_in = model.result().numerical("int1").getReal()[0][0];
21    double x_pol_in = model.result().numerical("int2").getReal()[0][0];
22    inXpolRatio = String.format("%.3f", x_pol_in/co_pol_in*100.0);
23    setFormObjectText("mainForm/labelinXpolRatio", inXpolRatio);
24
25    model.result().numerical("int3").setResult();
26    model.result().numerical("int4").setResult();
27    double co_pol_out = model.result().numerical("int3").getReal()[0][0];
28    double x_pol_out = model.result().numerical("int4").getReal()[0][0];
29    outXpolRatio = String.format("%.3f", x_pol_out/co_pol_out*100.0);
30    setFormObjectText("mainForm/labeloutXpolRatio", outXpolRatio);
31
32    double xpol = model.param().evaluate("xpol");
33    if (x_pol_out/co_pol_out*100.0 < xpol)
34        target_state = "passed";
      
```

Design a custom application user interface by using drag-and-drop to arrange form objects.

The users can write methods that, for example, include for-loops, process inputs and outputs, and messages and alerts sent from the application.

COMSOL APPLICATIONS

A COMSOL application is a model with a customized user interface created with the Application Builder. Simulation apps can be run with a COMSOL Multiphysics or COMSOL Server™ license.

With a COMSOL Multiphysics license, applications can be run from the COMSOL Desktop in different ways. While developing an app, it can be tested directly from the Application Builder to quickly check that forms and methods are performing as expected. Once released, an app can be run by others directly from the File menu available in the COMSOL Desktop or by double-clicking an MPH file icon on the Windows® Desktop. Apps can also be password protected.

Over 50 applications are available to provide users with examples that can be easily inspected, edited, and used as a starting point for their own apps.

With a COMSOL Server license, you can deploy, manage, and run applications in major web browsers on a variety of operating systems and hardware platforms. In addition, apps can be run by connecting to COMSOL Server™ with an easy-to-install COMSOL Client for Windows®.



APPLICATION DISTRIBUTION

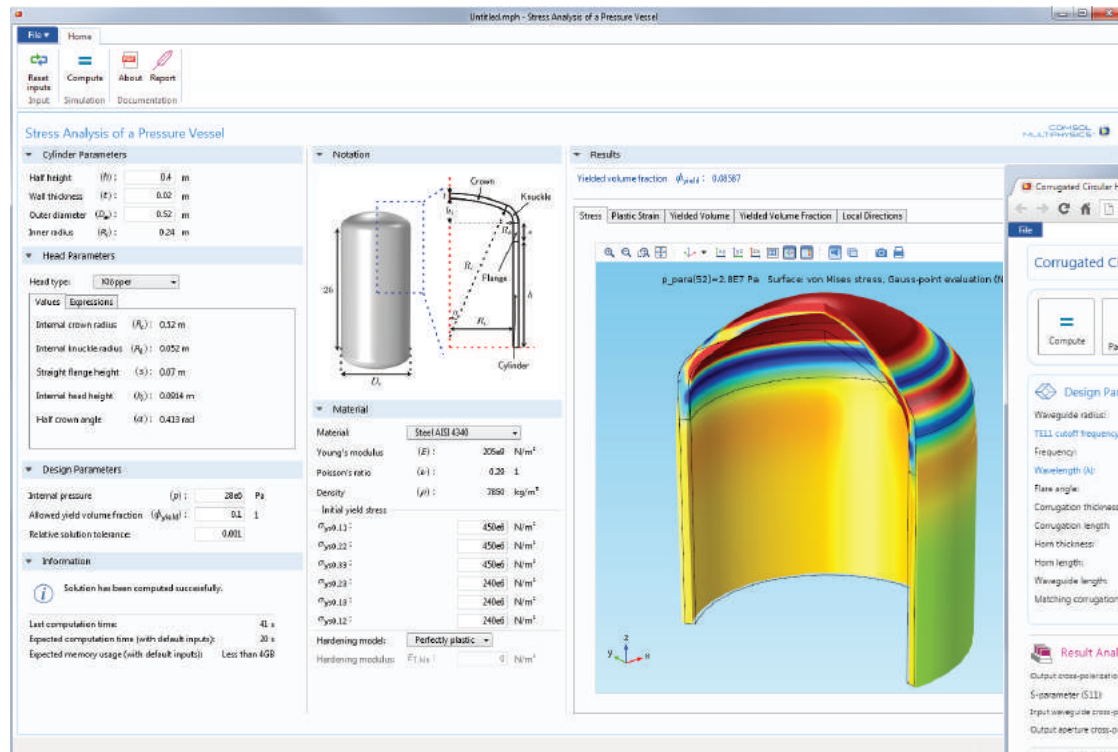
Apps created with the Application Builder can be accessed and used, both within an organization and worldwide, through a local installation of COMSOL Server™.

“Once you create a COMSOL model, it takes little effort to build an application with the Application Builder and distribute it with COMSOL Server™, the benefits will be felt by many Wolfspeed employees.”

Brice McPherson, Wolfspeed, Fayetteville, Arkansas.

App Examples from the Application Libraries

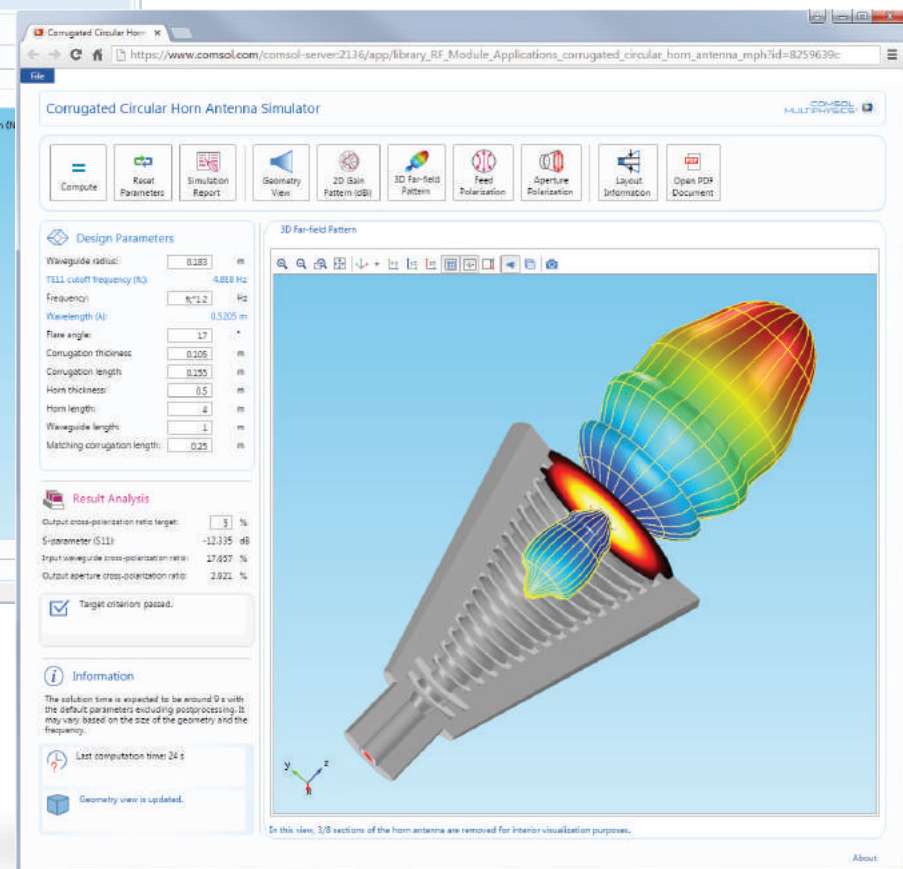
Application run by connecting to COMSOL Server™ with COMSOL Client for Windows®



STRESS ANALYSIS OF A PRESSURE VESSEL

This application analyzes whether a pressure vessel with a given geometry will be able to sustain an applied internal pressure without exceeding a certain volume fraction of the material that has exceeded the yield limit. Geometric parameters and materials may be changed to investigate different vessel designs.

Application run by connecting to COMSOL Server™ from a web browser



CORRUGATED CIRCULAR HORN ANTENNA

App users can modify the geometry of a corrugated circular horn antenna in order to optimize its radiation characteristics and aperture cross-polarization ratio at the frequency of interest.

COMSOL Server™

COMSOL Server™ is the software for running COMSOL applications and controlling their deployment, distribution, and use. After creating simulation apps with COMSOL Multiphysics® and its Application Builder, COMSOL Server™ provides users with a cost-effective solution for managing how the apps are used, either within their organization or externally to a worldwide audience.

COMSOL Server™ can be installed locally in a corporate network or in the cloud, enabling design teams, production departments, and others to use apps. Simulation applications can be run in a COMSOL Client for Windows® or in major web browsers such as Google Chrome™, Firefox®, Internet Explorer®, and Safari®.

COMSOL Server also allows updates to be readily available. As soon as a new version of an application is uploaded to the server, app users will immediately have access to the latest version.

APPLICATION LIBRARY

Upload and display the list of apps that can be run.

MONITOR

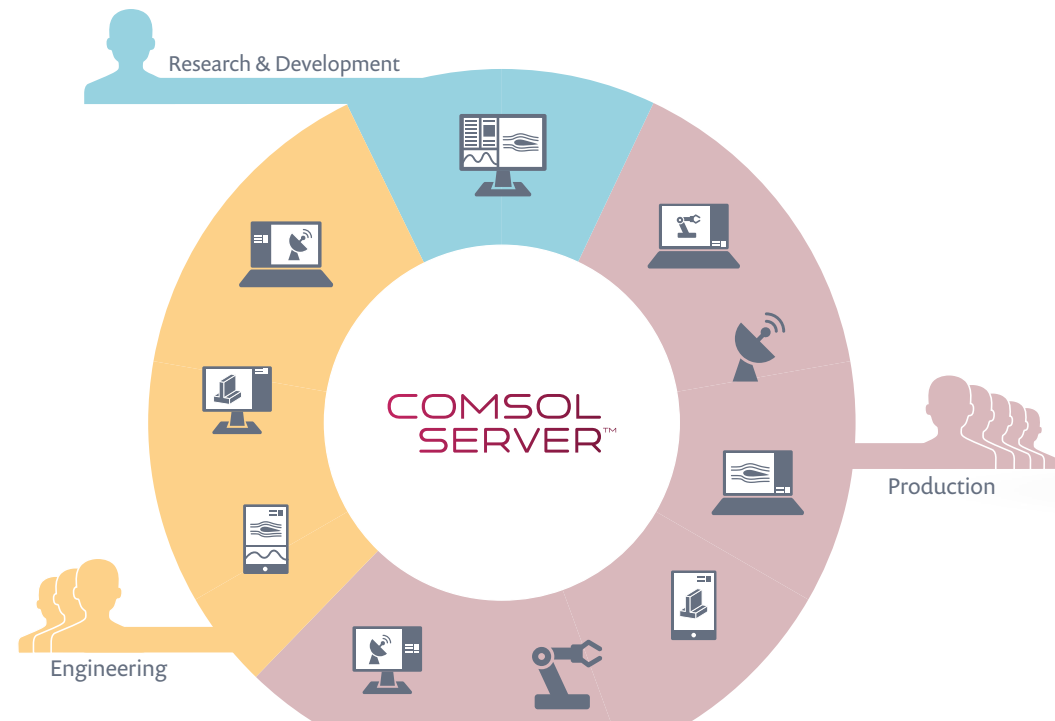
Find information about start time and uptime, CPU and memory use, and an overview of running app sessions.

USER DATABASE

Add users and groups to the server user database.

PREFERENCES

Access server administrative preferences and settings such as language, folders, image rendering, security, sessions, and multicore handling.



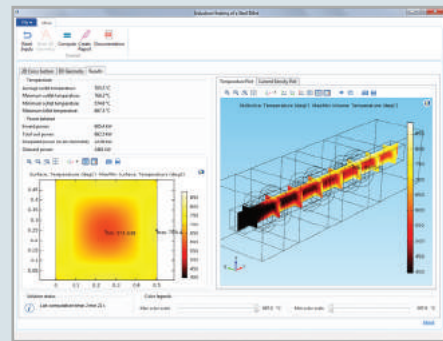
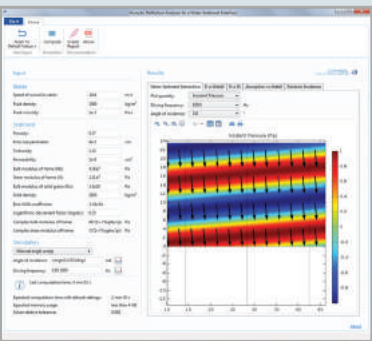
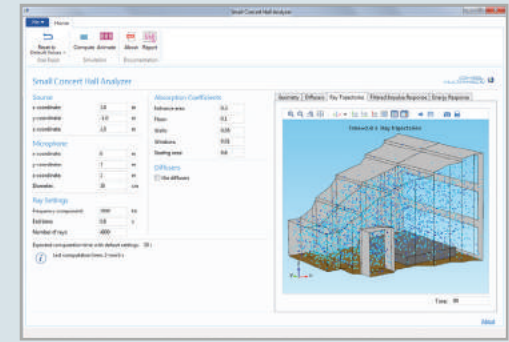
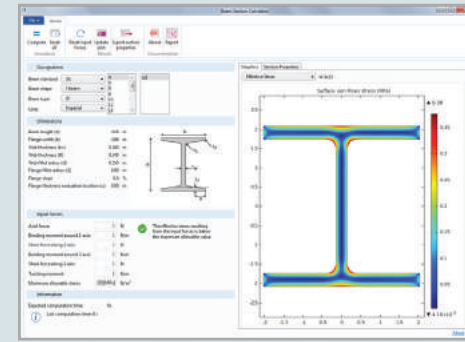
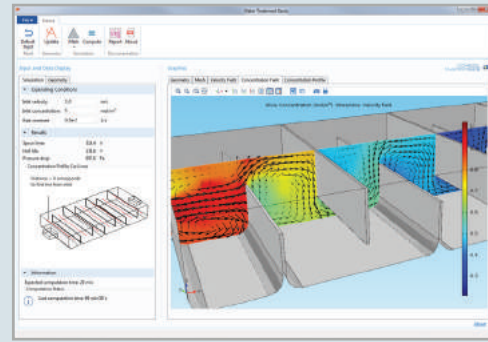
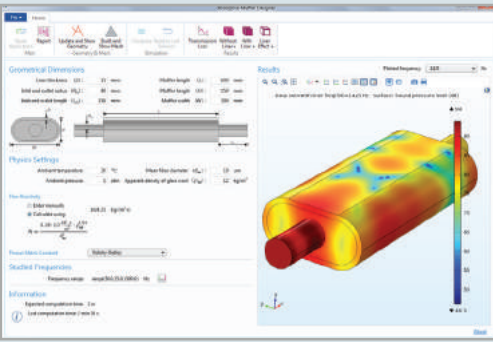
RUNNING APPLICATIONS

Apps can be run in separate tabs and windows. Here, the user is connected to a local installation of COMSOL Server™ from a web browser and is running four apps simultaneously.

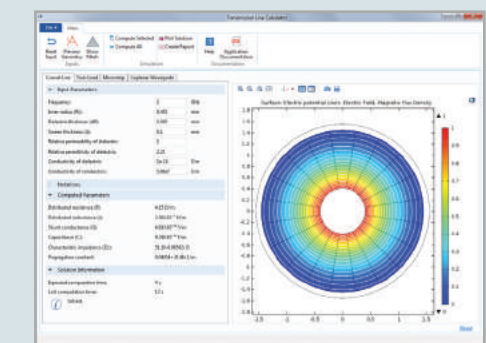
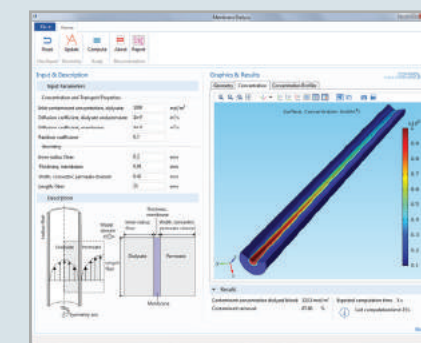
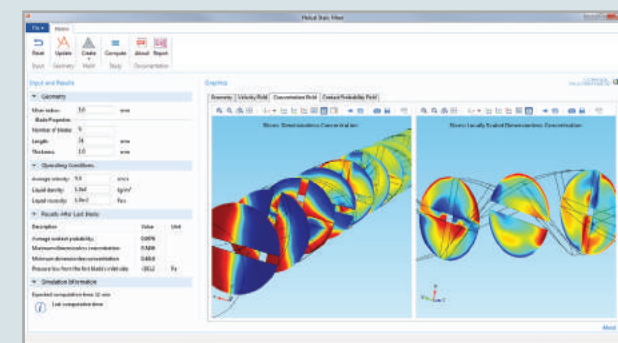
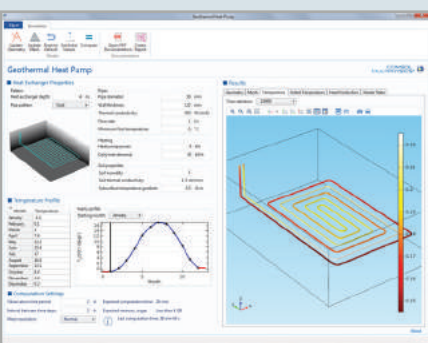
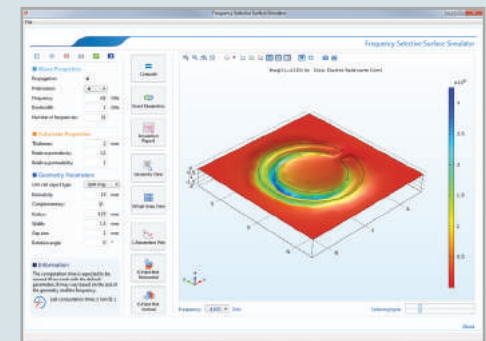
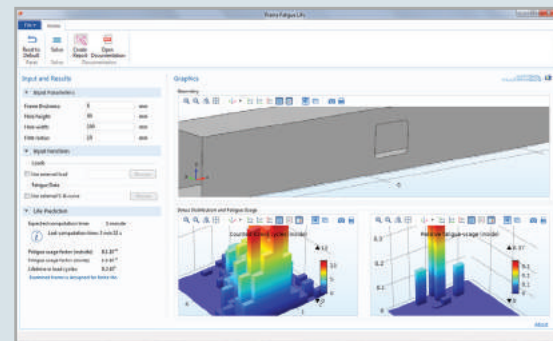
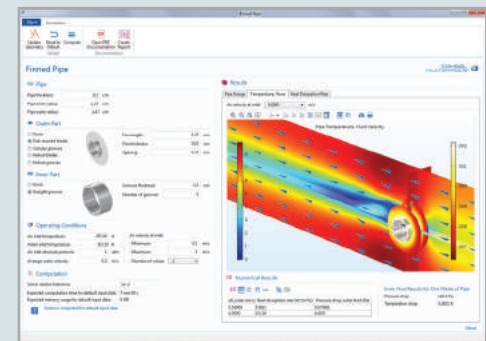
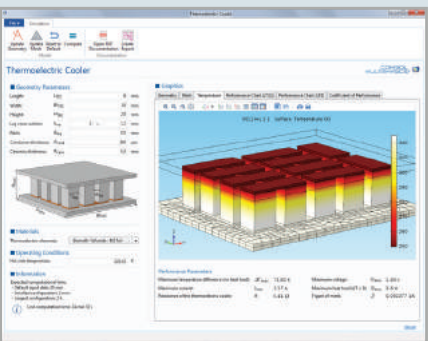
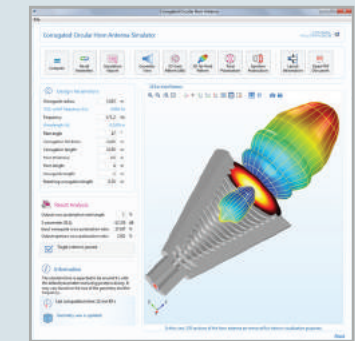
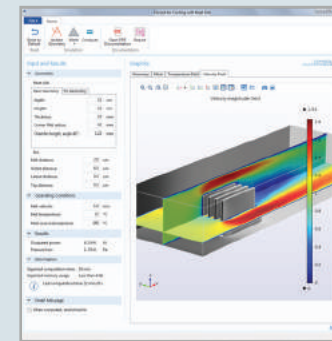
The screenshot displays the COMSOL Server web interface. The browser tabs at the top include 'Application Library | COM...', 'Touchscreen Simulator | C...', 'Mixer | COMSOL Server', 'Finned Pipe | COMSOL Se...', and 'Beam Section Calculator | ...'. The address bar shows 'https://www.comsol.com/comsol-server:2236/app-lib'. The interface features a left sidebar with a user profile for 'valerio administrator' and navigation links: 'Application Library', 'Upload', 'Administration', 'Monitor', 'User Database', 'Preferences', 'Your Settings', 'Change Password', and 'Change Default Email'. The main content area is divided into two sections: 'Running Applications' and 'Library'. The 'Running Applications' section shows four active sessions: 'Beam Section Calculator' (Session ID: 2241), 'Finned Pipe' (Session ID: 2242), 'Mixer' (Session ID: 2250), and 'Touchscreen Simulator' (Session ID: 2254). Each session includes a thumbnail image and a 'Reconnect in Browser' button. The 'Library' section shows four available applications: 'Absorptive Muffler Designer', 'Acoustic Reflection Analyzer for a Water-Sediment Interface', 'Beam Section Calculator', and 'Beam Subjected to Traveling Load'. Each library application includes a thumbnail, a star icon for favorites, and a 'Launch in Browser' button. Red lines from the text annotations point to the 'Application Library' link in the sidebar and the 'Launch in Browser' button for the 'Absorptive Muffler Designer' application.

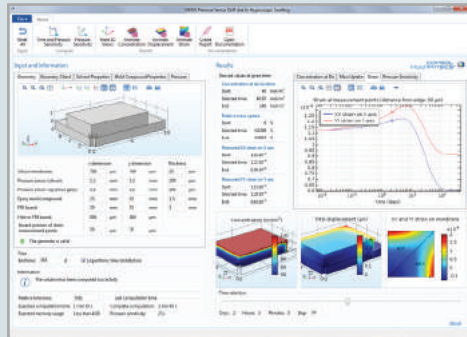
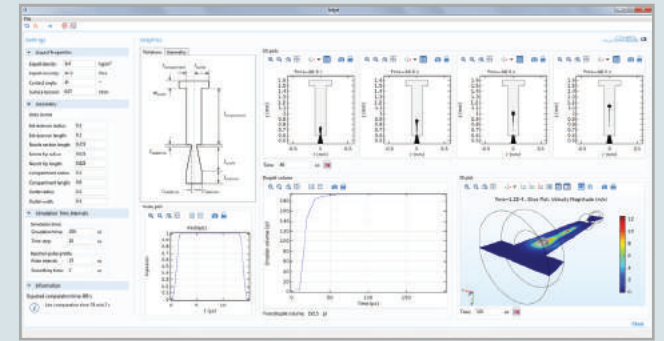
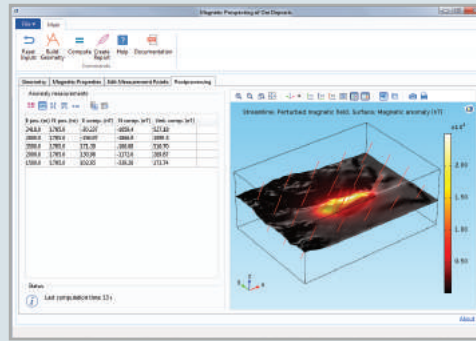
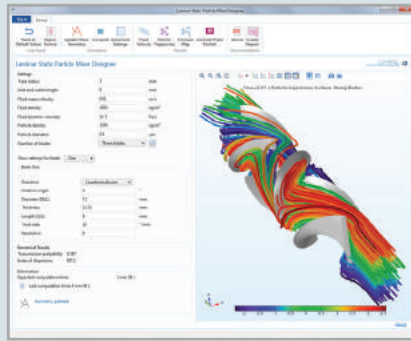
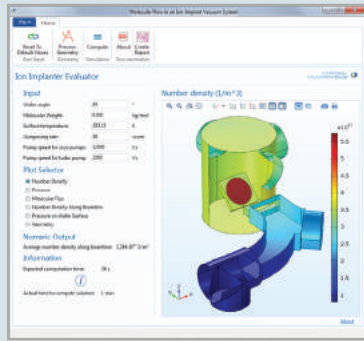
APP CARD

Edit, show general information, favorite, and launch an app.

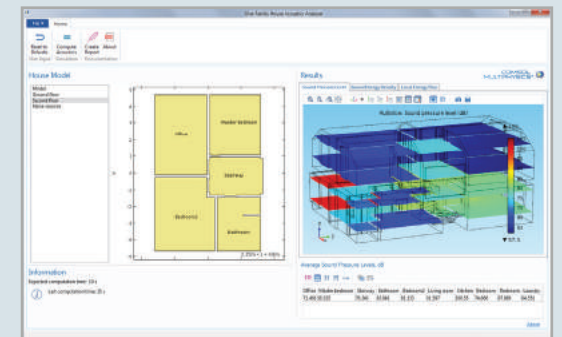
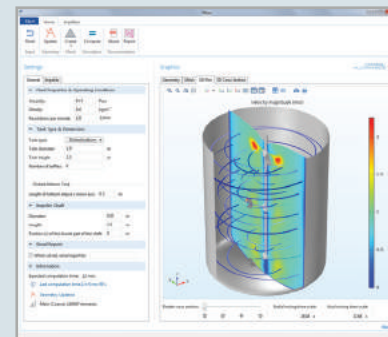
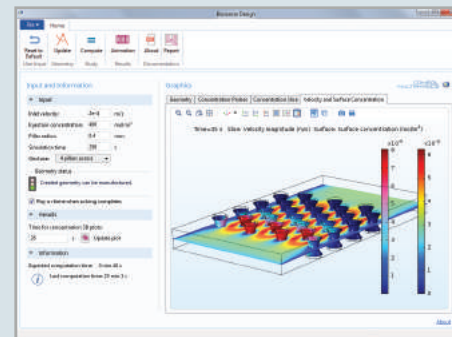
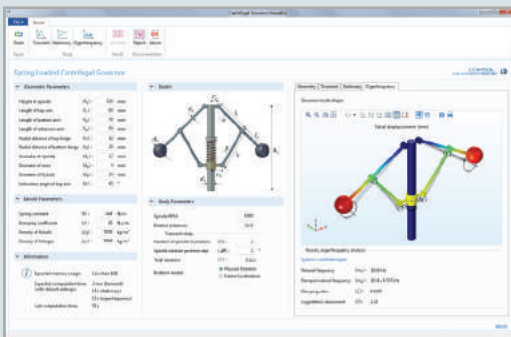
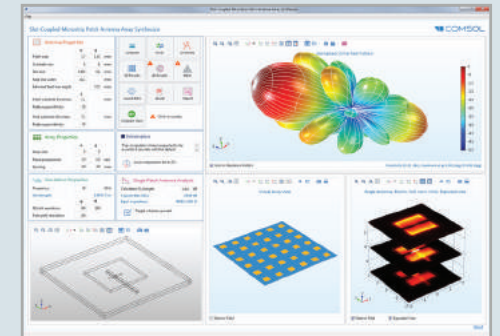
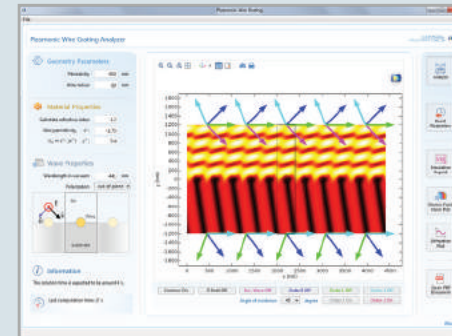
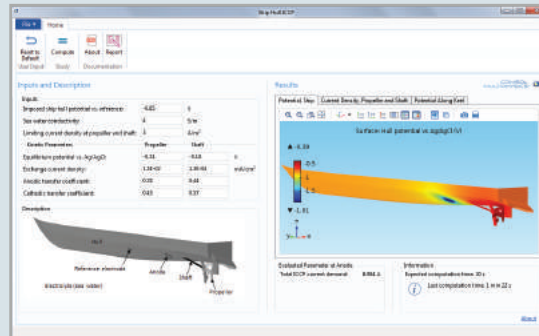
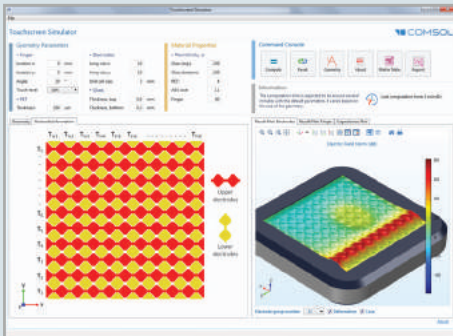
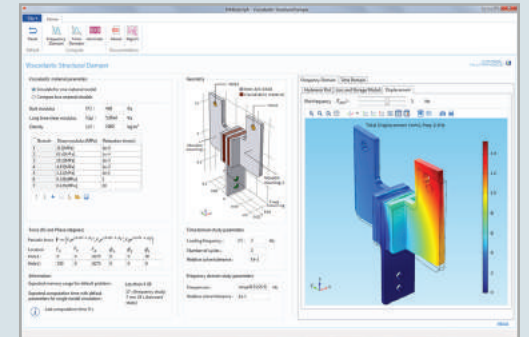
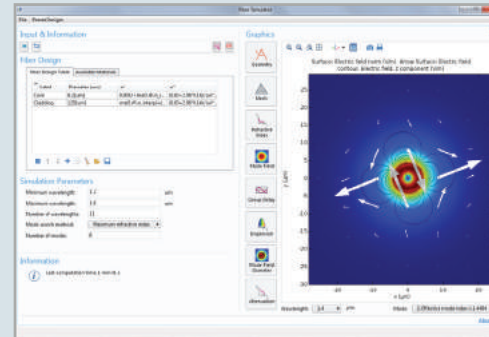


From Model to Application





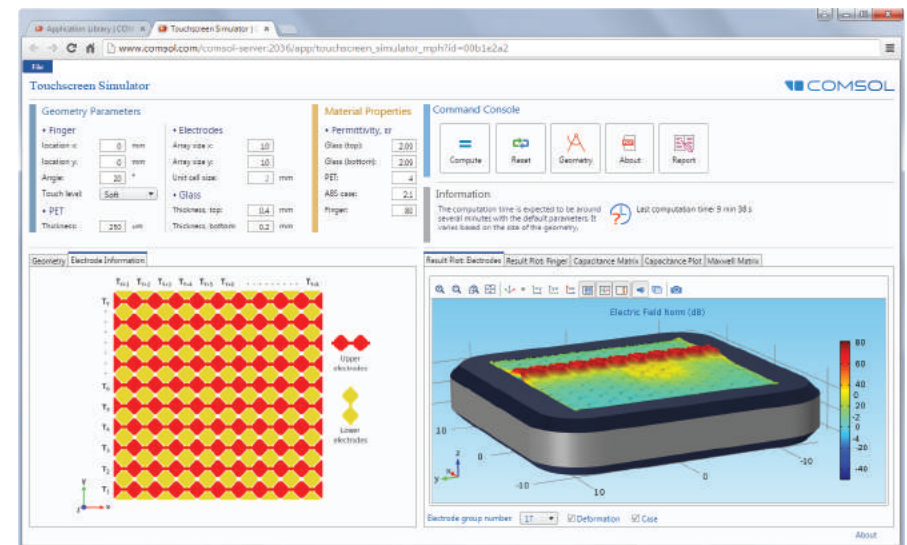
BRING MULTIPHYSICS TO EVERYONE



AC/DC Module

The AC/DC Module sets the stage for modeling the performance of capacitors, inductors, motors, and microsensors. Although these devices are principally characterized by electromagnetics, they are also influenced by other types of physics. Thermal effects, for instance, can change a material's electrical properties, while electromechanical deflections and vibrations in generators need to be fully understood during any design process.

The capabilities of the AC/DC Module span electrostatics, magnetostatics, and electromagnetic quasistatics with access to any derived field quantities and unlimited couplings to other physics. When considering your electrical components as part of a larger system, the AC/DC Module provides an interface with SPICE circuit lists where you choose circuit elements for further modeling. Then, you can take your analysis beyond the conventional by running a single simulation of a mixed system of lumped and high-fidelity component models.



CAPACITIVE TOUCHSCREEN SIMULATOR

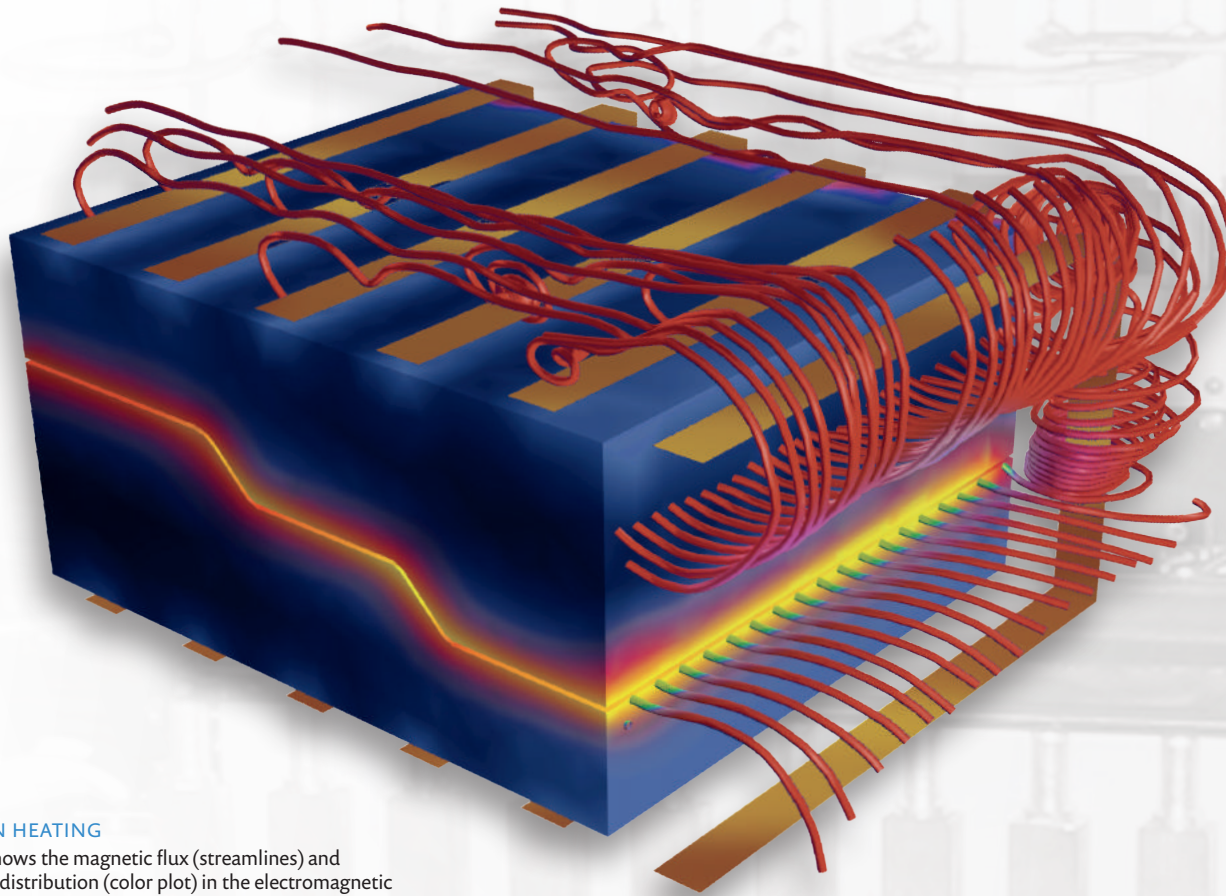
Specialized application based on a model of a capacitive touchscreen. The user can control the position and orientation of a finger pressing the touchscreen, and then compute the capacitance matrix.

HIGHLIGHTS

- AC/DC current and field distribution
- Bioheating
- Coils and solenoids
- Combined SPICE circuit and field simulations
- Contact resistance
- EMC and EMI
- Electromagnetic force and torque
- Electromagnetic shielding
- Electromechanical deformation
- Hall sensors
- Insulators, capacitors, and dielectrics
- Motors, generators, and other electromechanical machinery
- Nonlinear materials
- Parasitic capacitance and inductance
- Permanent magnets and electromagnets
- Porous materials
- Resistive and induction heating
- Sensors
- Superconductors
- Transformers and inductors

APPLICATION EXAMPLES

- Billet induction heating
- Effective nonlinear magnetic curves
- Magnetic prospecting
- Touchscreen simulator



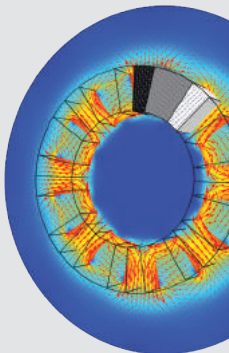
INDUCTION HEATING

The model shows the magnetic flux (streamlines) and temperature distribution (color plot) in the electromagnetic induction molding apparatus and composite material.

Model and pictures courtesy of José Feigenblum, RocTool, Le Bourget Du Lac, France.

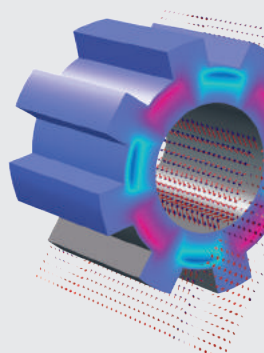
HALBACH ROTOR

Simulation of a 16-segment, 4-pole pair Halbach rotor. Results show the magnetic flux density distribution.



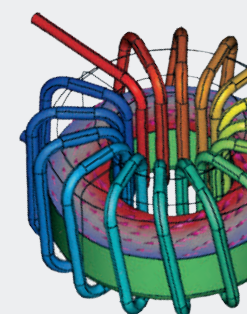
POWER ENGINEERING

Static 3D simulation of a generator with a rotor consisting of annealed medium-carbon steel, which is a nonlinear ferromagnetic material that is saturated at high magnetic flux densities. Shown are the magnetic fields inside and around the generator.



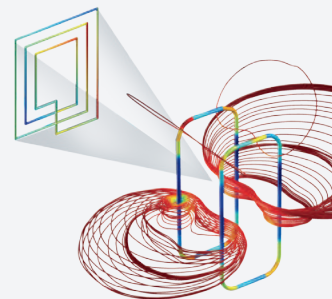
ELECTROMAGNETIC INDUCTOR

An inductor consisting of a wire wrapped around a ferrite core is modeled and the device inductance is calculated. The current flow, due to an applied voltage, induces a magnetic field. The magnetic flux density is shown.



RFID TAGS

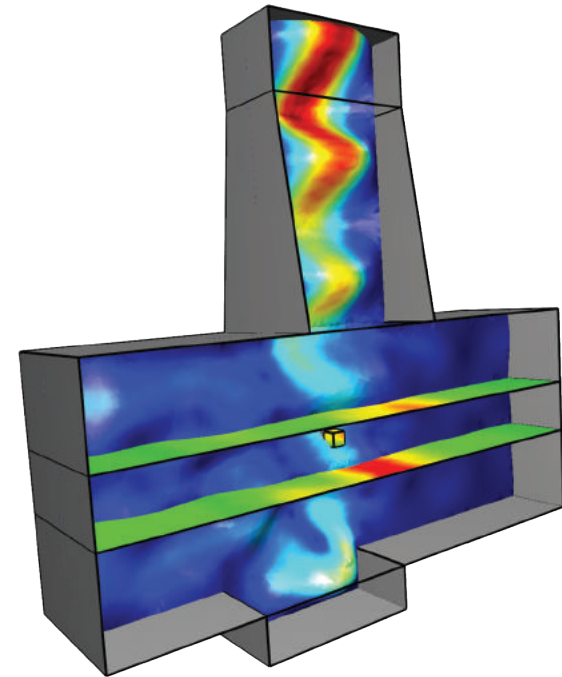
An RFID system consists of a tag or transponder with a PCB antenna (left) and a much larger reader antenna (right). This generates an electromagnetic field to energize the IC-circuit inside the tag. Shown is the magnetic flux density.



RF Module

Modeling in the RF and microwave regimes requires modeling the propagation of electromagnetic waves in and around structures that can be metallic, dielectric, gyromagnetic, or even metamaterials with engineered properties. The RF Module offers you the tools to meet this challenge by including port and scattering boundary conditions; complex-valued, spatially varying, anisotropic material models; perfectly matched layers; and the best solvers available. As a result, you can easily model antennas, waveguides, and microwave components.

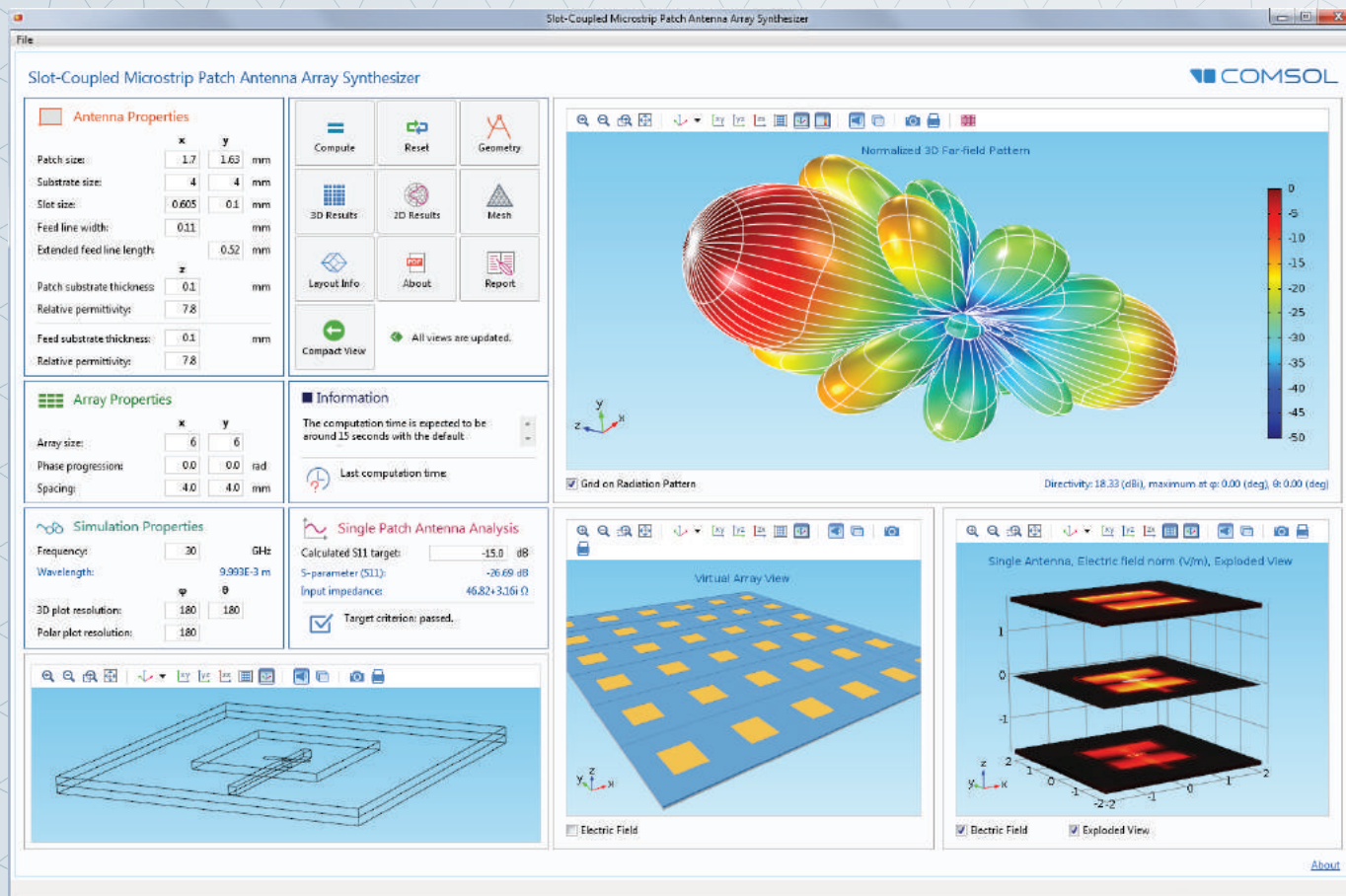
The RF Module completes the modeling experience by providing S-parameter computation and far-field analysis. Taken together with the unsurpassed ability of COMSOL Multiphysics to couple to other physics, such as electromagnetic RF heating, stress, and deformation effects, you have the industry's leading multiphysics solution for electromagnetic waves.



MINERAL PROCESSING

By applying microwave radiation, mineral particles in an ore are selectively heated, which causes local thermal expansion and liberation of the particles. This process requires significantly less energy than the conventional method of crushing the rock and separating the minerals. The model resolves the microwaves in applicators, which are several cubic meters in volume, together with those that heat the mineral particles, which are only tens of microns in size.

Model courtesy of Jan Przybyla, ezv, Chelmsford, UK.

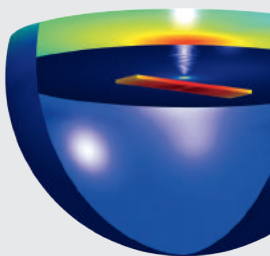


SLOT-COUPLED MICROSTRIP PATCH ANTENNA ARRAY SYNTHESIZER

This application simulates a single slot-coupled microstrip patch antenna fabricated on a multilayered low-temperature cofired ceramic (LTCC) substrate. Users may control inputs such as properties of a single antenna and array geometry. Results show the far-field radiation pattern of the antenna array and its directivity.

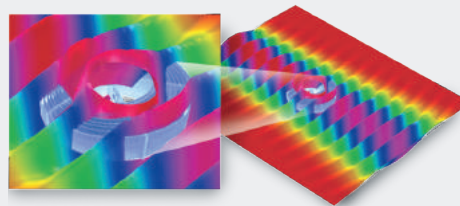
OIL PROSPECTING

The CSEM method has become popular in oil prospecting. Here, a horizontal 1-Hz electric dipole antenna is towed 150 m above the seafloor, where receivers measure the electrical field generated by the source and the reflected or guided transmitted energy resulting from the resistive reservoir.



MATERIALS SCIENCE

Electromagnetic metamaterials have a spatial resolution that is below that of an electromagnetic wavelength, so as to create a form of 'invisibility' over a narrow wave band. In this model, an object located in the center of this structure is invisible to microwave radiation.



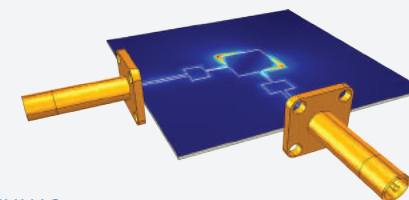
Model courtesy of Cummer and Schurig, Duke University, Durham, North Carolina.

HIGHLIGHTS

- Antennas, waveguides, and cavities
- Bloch-Floquet periodic arrays and structures
- Circulators and directional couplers
- High-speed interconnects
- Metamaterials
- Microwave and RF heating
- Microwave devices
- Microwave sintering
- Oil exploration and controlled-source electromagnetic (CSEM) surveys
- Plasmonics
- Porous materials
- Resonant coil design
- RF and microwave bioheating
- Scattered field formulation for RCS and scattering problems
- S-parameter analyses of antennas
- Thermo-structural effects in antennas and waveguides
- Transmission lines

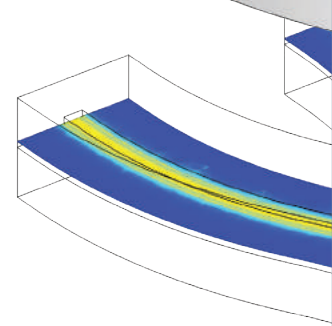
APPLICATION EXAMPLES

- Corrugated circular horn antenna
- Frequency selective surface simulator
- Microstrip patch antenna array synthesizer
- Plasmonic wire grating



ANTENNAS

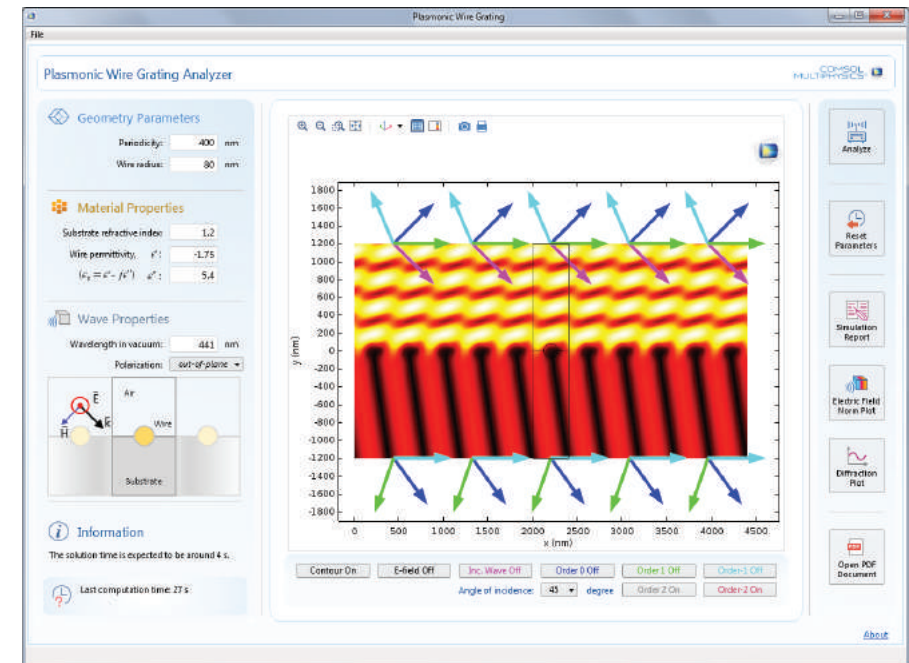
Two coaxial cables work together to create a balanced feed to a patch antenna. The model computes the antenna efficiency to find the optimal operating frequency, 6.3 GHz.



Wave Optics Module

The Wave Optics Module provides dedicated tools for electromagnetic wave propagation in linear and nonlinear optical media for accurate component simulation and design optimization. The beam envelopes method for electromagnetic full-wave propagation overcomes the need for traditional approximations by direct discretization of Maxwell's equations. This allows for accurate simulation of optically large systems where the geometric dimensions can be much larger than the wavelength and where light waves cannot be approximated with rays. Conventional electromagnetic full-wave propagation methods are also featured in the module.

Support for generic anisotropic refractive index, permittivity, or permeability tensors allows for optical media that includes gyromagnetic materials or metamaterials with engineered properties. Several 2D and 3D formulations are available for eigenfrequency mode analysis and frequency- and time-domain electromagnetic simulation.



PLASMONIC WIRE GRATING ANALYZER

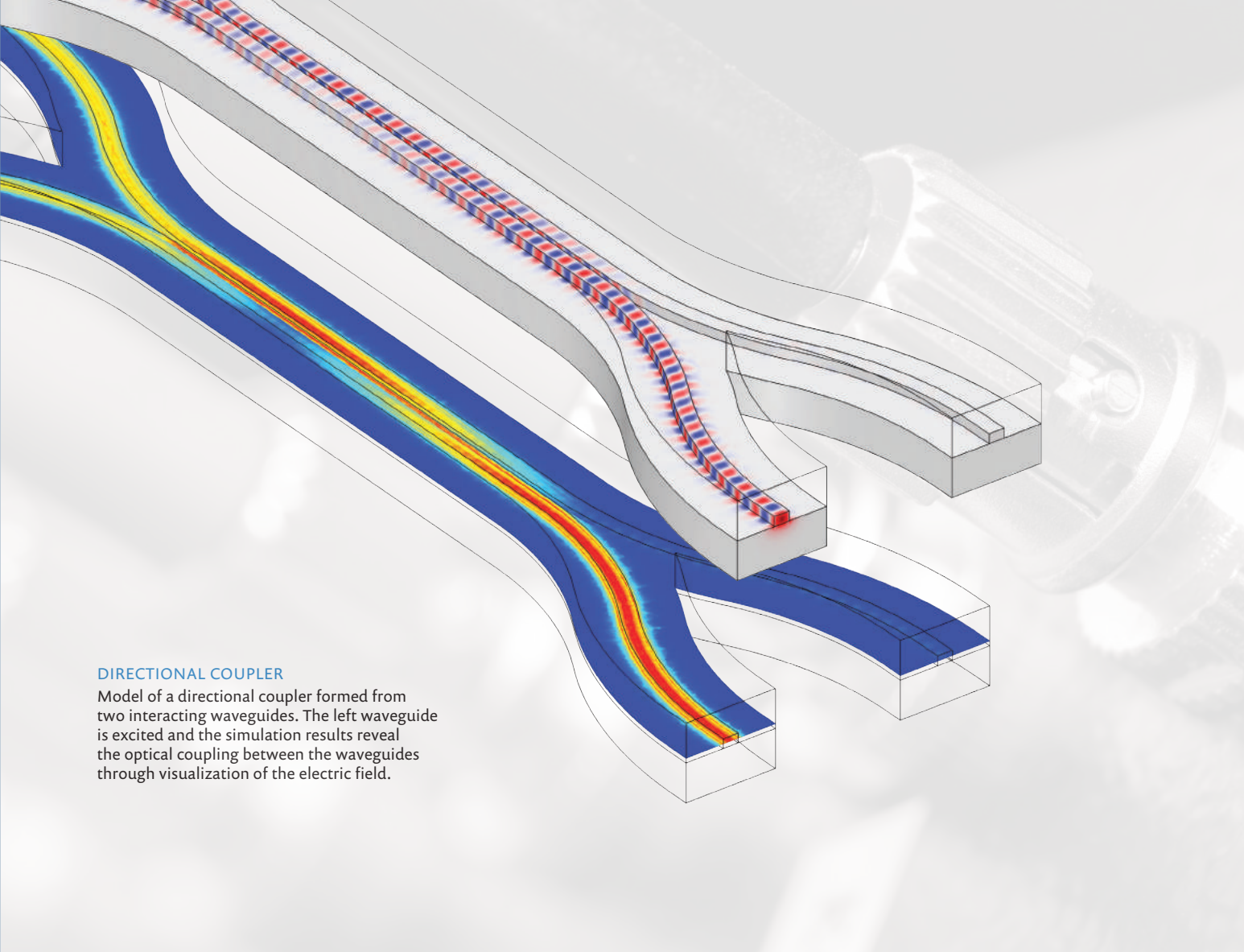
App users can generate a results report including the electric field norm, reflectance, and transmittance for a given angle of incidence. The underlying model computes the diffraction efficiency for reflected and transmitted waves interacting with a wire grating on a dielectric substrate.

HIGHLIGHTS

- Couplers
- Fiber Bragg gratings
- Fiber optics
- Harmonic generation and frequency mixing
- Integrated optics
- Lasers and amplifiers
- Laser heating
- Metamaterials
- Nonlinear optics
- Optical lithography
- Optical scattering
- Optical sensors
- Optoelectronics
- Photonic crystal fibers
- Photonic devices
- Rod, slab, and disk laser design
- Scattering from nanoparticles
- Semiconductor lasers
- Surface scattering
- Stress-optical effects
- Waveguides

APPLICATION EXAMPLES

- Fiber simulator
- Plasmonic wire grating
- Polarizing beam splitter

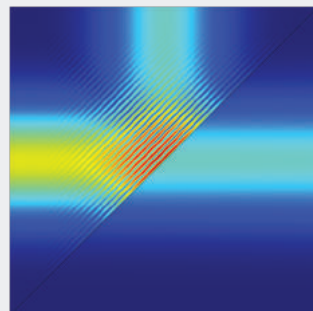


DIRECTIONAL COUPLER

Model of a directional coupler formed from two interacting waveguides. The left waveguide is excited and the simulation results reveal the optical coupling between the waveguides through visualization of the electric field.

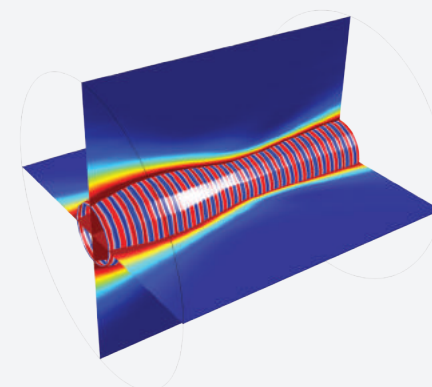
BEAM SPLITTER

A beam splitter is used to split a beam of light in two. One way of making a splitter is to deposit a thin layer of metal between two glass prisms. The beam is slightly attenuated within the layer and split into two paths. The thin metal layer is modeled using a transition boundary condition, which reduces the memory requirements. Losses in the metal layer can be also computed.



SELF-FOCUSING

A Gaussian beam is launched into BK-7 optical glass. The material has an intensity-dependent refractive index. For the center of the beam, the refractive index is the largest. This induced refractive index profile counteracts the diffractive effects and actually focuses the beam. Self-focusing is important in the design of high-power laser systems.

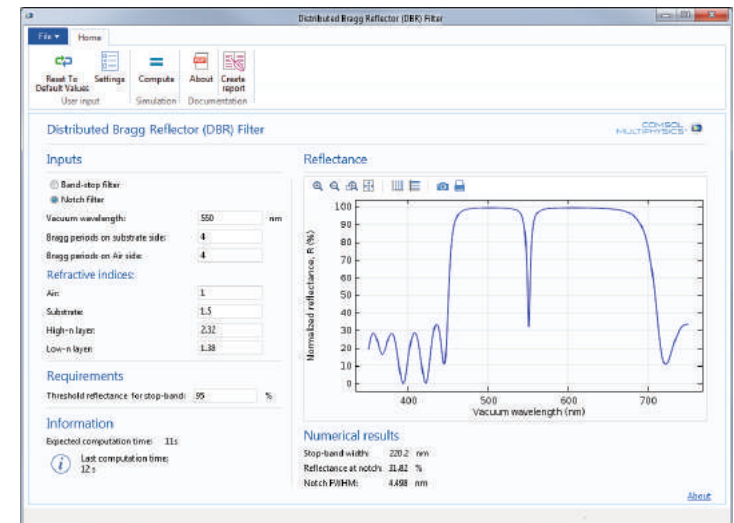
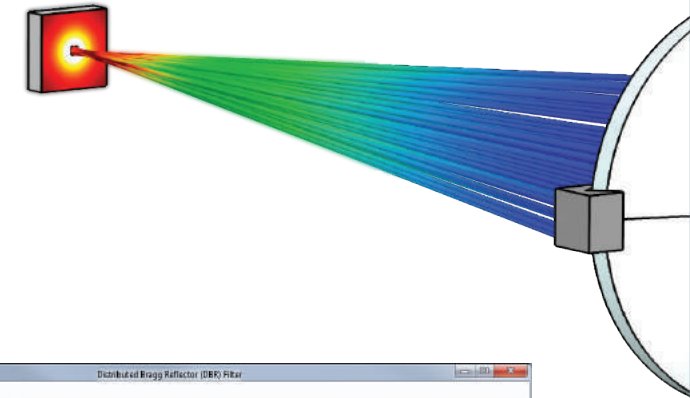


Ray Optics Module

The Ray Optics Module can be used to model electromagnetic wave propagation in systems in which the wavelength is much smaller than the smallest geometric detail in the model. The electromagnetic waves are treated as rays that can propagate through homogeneous or graded media. Because it is not necessary to resolve the wavelength with a finite element mesh, ray trajectories can be computed over long distances at a low computational cost.

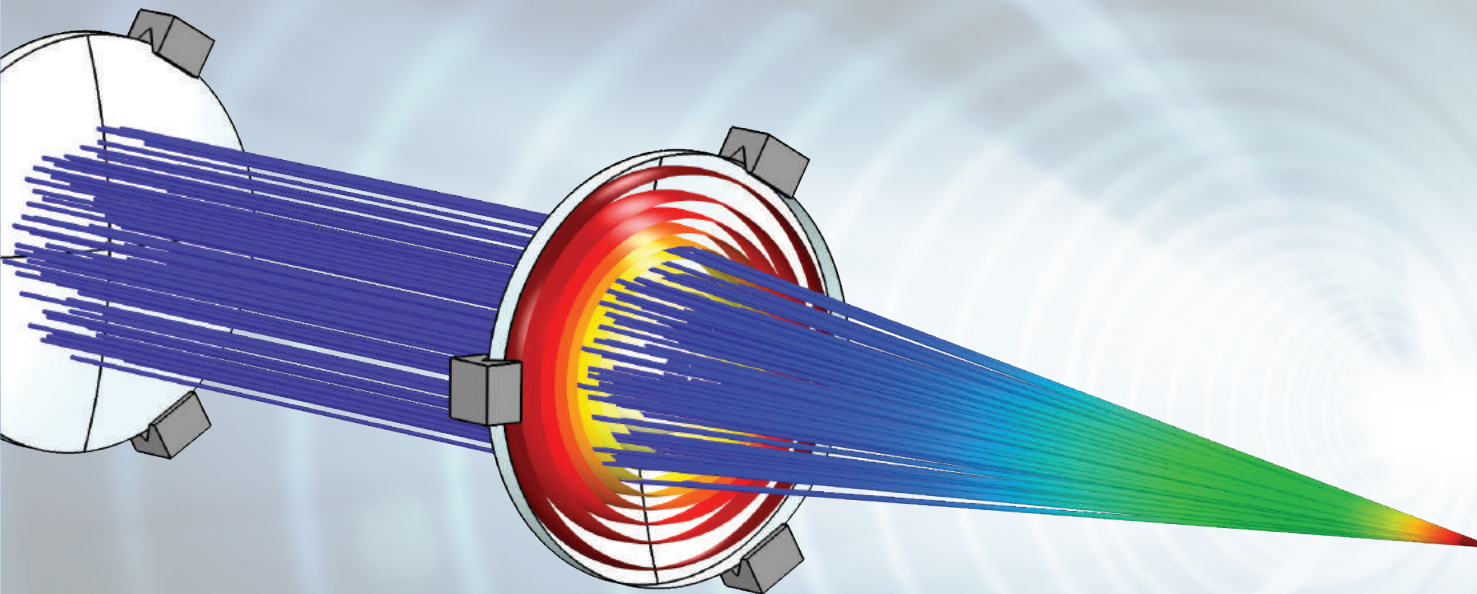
Rays can also undergo reflection and refraction at boundaries between different media. The Ray Optics Module contains a variety of boundary conditions, including combinations of specular and diffuse reflection. Rays can be released from within domains, from boundaries, or at a uniform grid of points. Specialized release features are also available for modeling solar radiation and for releasing reflected or refracted rays from an illuminated surface.

Dedicated postprocessing tools offer you many ways to analyze ray trajectories, evaluate expressions over many rays, and even visualize interference patterns. Built into the Ray Optics Module are a number of specialized physics interface settings including optional variables for computing ray intensity using the Stokes parameters, enabling the modeling of polarized, partially coherent, or unpolarized radiation.



DISTRIBUTED BRAGG REFLECTOR FILTER

This application, created using the Application Builder available in COMSOL Multiphysics, can be used to study the reflectance from an optical notch filter, based on a cavity sandwiched between two distributed Bragg reflectors.



THERMALLY INDUCED FOCAL SHIFT

Model of a high-power industrial fiber laser system. The user can analyze the heat generated in the lens, which can alter ray trajectories by inducing thermal deformation and changing the refractive index of the material.

HIGHLIGHTS

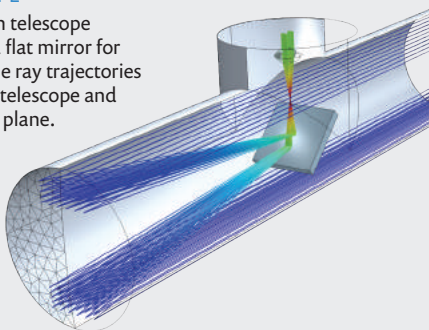
- Absorbing media
- Anti-reflective coatings
- Cameras
- Caustic surface visualization
- Corrections for strongly absorbing media
- Dielectric films
- Diffraction gratings
- Interferometers
- Intensity computation
- Lambertian and specular reflection
- Lasers
- Lens systems
- Monochromators
- Mueller matrices
- Optical path length calculation
- Polarizers and wave retarders
- Polychromatic light
- Ray tracing in graded media
- Self-consistent modeling of ray heating
- Solar energy harvesting
- Spot diagrams
- Stokes parameters
- Thermal lensing
- Unpolarized and partially polarized radiation

APPLICATION EXAMPLES

- Distributed Bragg reflector filter

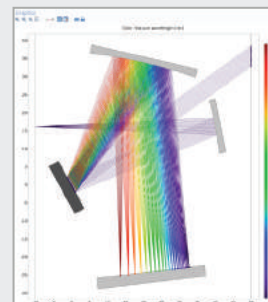
NEWTONIAN TELESCOPE

Simulation of a Newtonian telescope that uses a parabolic and a flat mirror for reflection. Results show the ray trajectories as they travel through the telescope and are reflected into the focal plane.



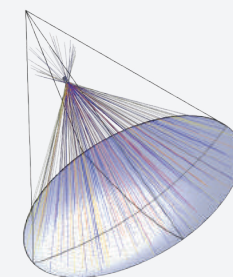
CZERNY-TURNER MONOCHROMATOR

A Czerny-Turner monochromator used for separating polychromatic light into monochromatic rays. Simulation results show the ray trajectories at each wavelength and quantify the resolution of the device.



SOLAR DISH RECEIVER

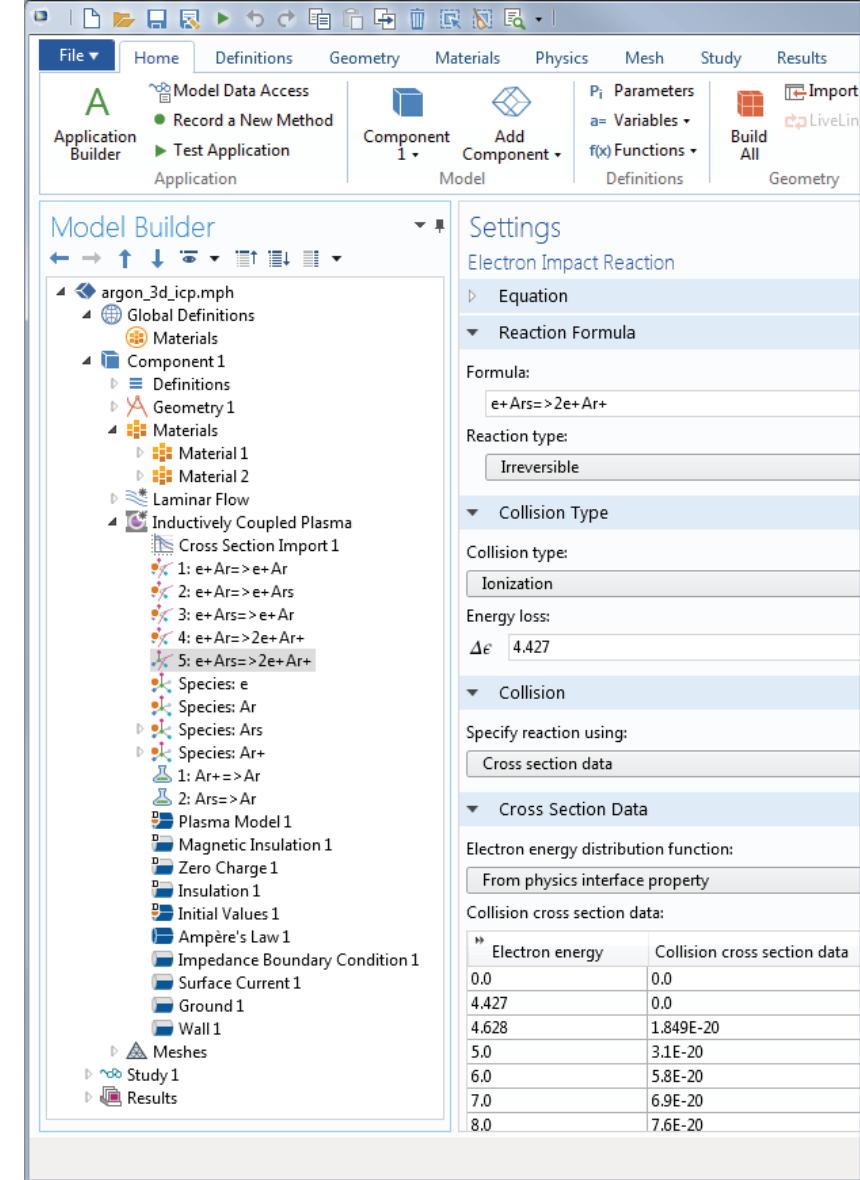
A parabolic solar dish collects solar radiation for generating electrical power. The results of this simulation trace the ray trajectories as they reflect off the dish and are concentrated onto a thermal receiver.



Plasma Module

Low-temperature plasmas represent the amalgamation of fluid mechanics, reaction engineering, physical kinetics, heat transfer, mass transfer, and electromagnetics. The Plasma Module is a specialized tool for modeling equilibrium and nonequilibrium discharges that occur in a wide range of engineering disciplines. There are specialized modeling interfaces for the most common types of plasma reactors including inductively coupled plasmas (ICP), DC discharges, wave-heated discharges (microwave plasmas), and capacitively coupled plasmas (CCP).

Modeling the interaction between the plasma and an external circuit is an important part of understanding the overall characteristics of a discharge. The Plasma Module provides tools to add circuit elements directly or to import an existing SPICE netlist into the model. The plasma chemistry is specified either by loading in sets of collision cross sections from a file or by adding reactions and species to the Model Builder. The complicated coupling between the different physics that constitute a plasma is automatically handled by the physics interfaces. The Plasma Module is designed for researchers, engineers, and experimentalists in the field of plasma science.



HIGHLIGHTS

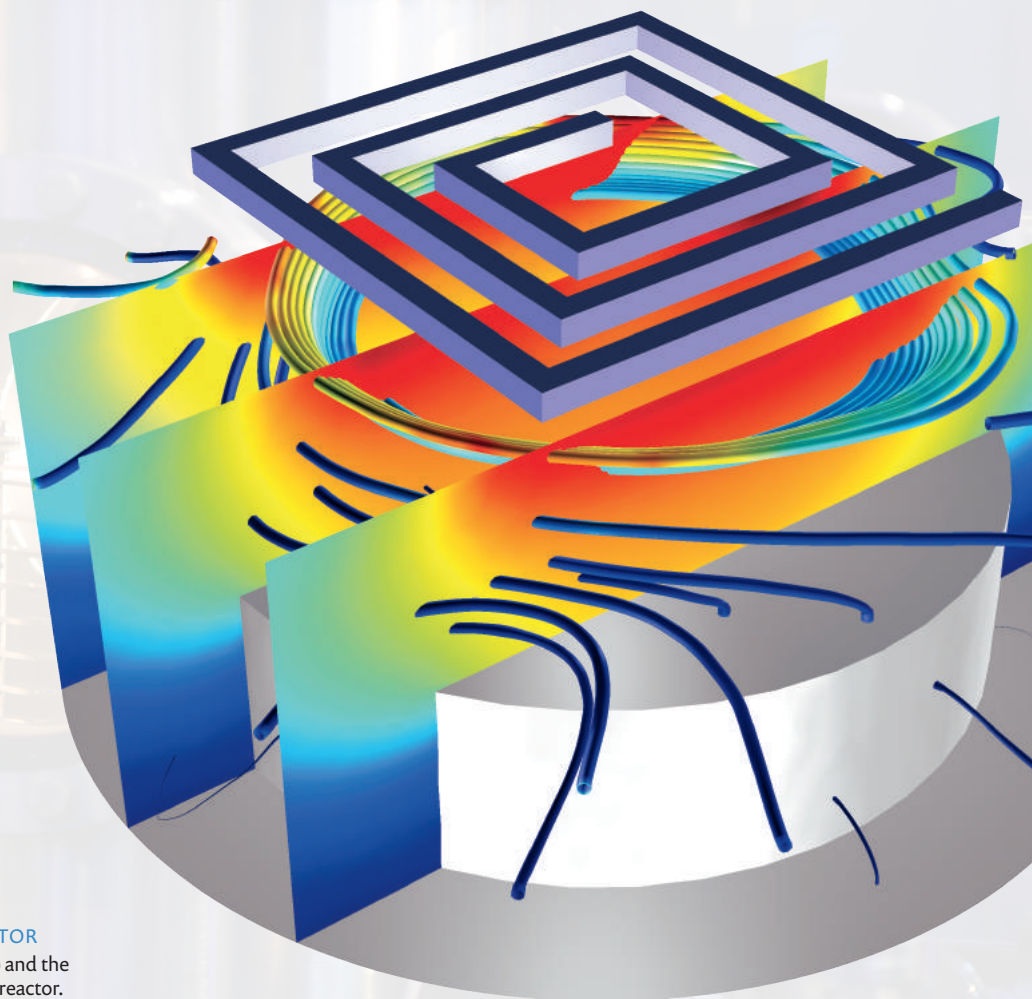
- Capacitively coupled plasmas (CCP)
- CVD and PECVD
- DC discharges
- Dielectric-barrier discharges
- ECR sources
- Etching
- Hazardous gas destruction
- ICP torches*
- Inductively coupled plasmas (ICP)*
- Ion sources
- Materials processing
- Microwave plasmas**
- Ozone generation
- Plasma chemistry
- Plasma display panels
- Plasma sources
- Power systems
- Semiconductor processing
- Thrusters

* Together with the AC/DC Module

** Together with the RF Module

APPLICATION EXAMPLES

- CCP reactor

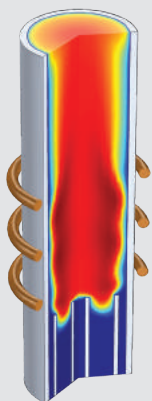


3D MODEL OF AN ASYMMETRIC ICP REACTOR

The plot shows the mean electron energy (slice) and the electron current density (streamlines) in an ICP reactor.

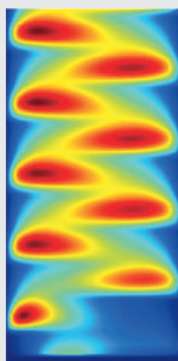
ICP TORCH

Model of an inductively coupled plasma torch under atmospheric pressure. Simulation results reveal electrical and thermal characteristics of the torch.



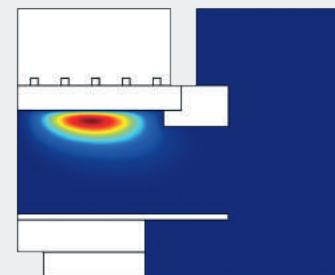
DIELECTRIC BARRIER DISCHARGE

Excited argon atoms can spontaneously decay back to the ground state, releasing a photon in the ultraviolet spectrum. The plot to the right shows the concentration of excited argon atoms in a dielectric-barrier discharge.



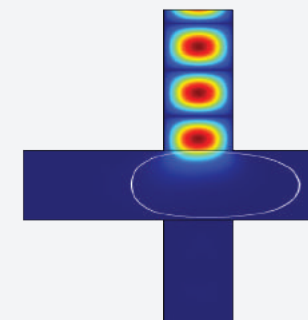
INDUCTIVELY COUPLED PLASMA

This plot displays the power deposition into a GEC ICP reactor with argon chemistry. The power deposition is shielded towards the top of the reactor due to the skin effect.



TE MODE MICROWAVE DISCHARGE

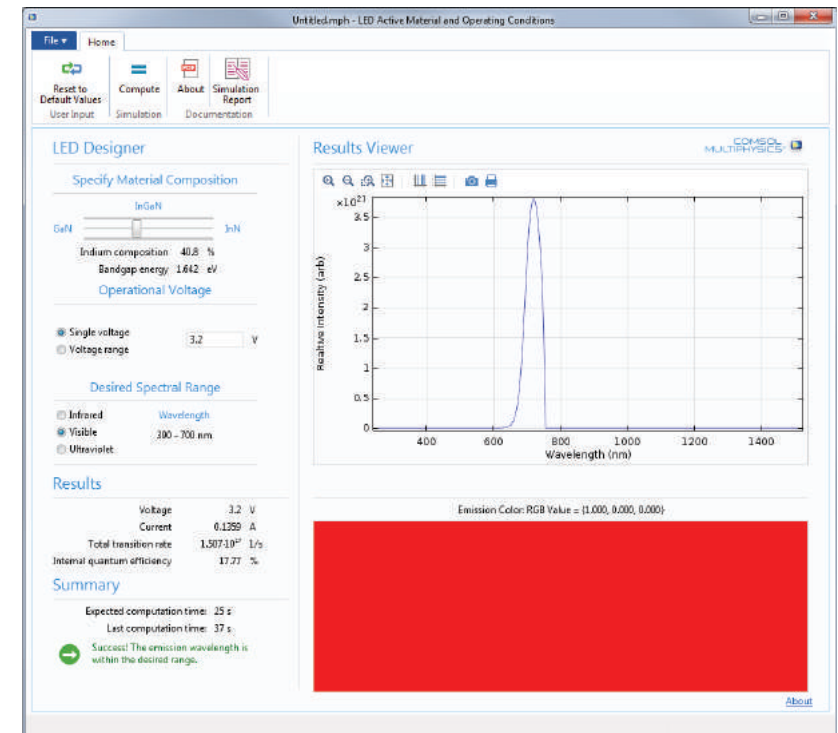
The surface color shows the magnitude of the high-frequency electric field as it propagates down a waveguide and into a plasma. The contour indicates where the electron density is equal to the critical plasma density.



Semiconductor Module

The Semiconductor Module allows for detailed analysis of semiconductor device operation at the fundamental physics level. The module is based on the drift-diffusion equations with both isothermal and nonisothermal transport models. A straightforward but powerful interface makes it easy to set up doping profiles and specify the physics and boundary conditions.

Models for semiconducting and insulating materials as well as a range of recombination mechanisms, including detailed modeling of domain traps, are provided. Several mobility models are included with the product and the predefined mobility models can be combined with each other or even mixed with user-defined mobility models. Optical transitions are handled by a feature that can be coupled with a wave optics simulation of a propagating electromagnetic wave or used in a standalone fashion. Phenomena such as electroluminescence, light absorption, spontaneous and stimulated emission, and carrier photogeneration can be modeled. A range of boundary conditions allow you to model metal contacts (both Schottky and ohmic contacts), gates, surface traps, and Fowler-Nordheim tunneling.



LED DESIGNER

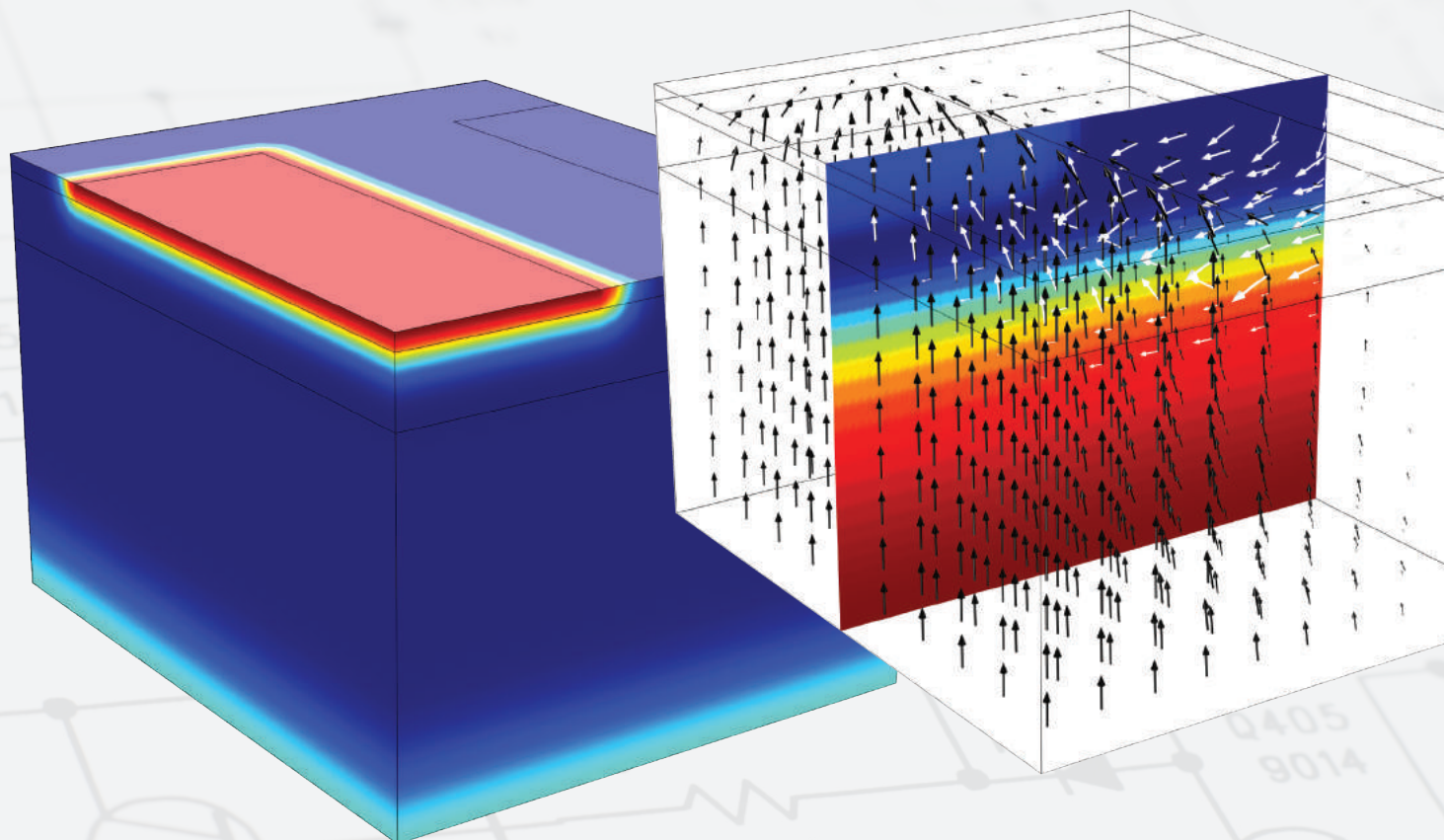
Created using the Application Builder available in COMSOL Multiphysics, this application example investigates light emission from an LED, allowing the user to change the composition of the semiconductor material and generate a report from the results. The current, emission intensity, and internal quantum efficiency are given by the study. The color emitted by the LED is visualized with an RGB plot.

HIGHLIGHTS

- Bipolar transistors
- Bulk lasers
- EEPROM
- LEDs
- Metal-oxide-semiconductor field-effect transistors (MOSFETs)
- Metal semiconductor field-effect transistors (MESFETs)
- Photodiodes
- P-N junctions
- Schottky diodes
- Solar cells
- Surface and bulk traps
- Thyristors

APPLICATION EXAMPLES

- Si solar cell with ray optics
- Wavelength tunable LED

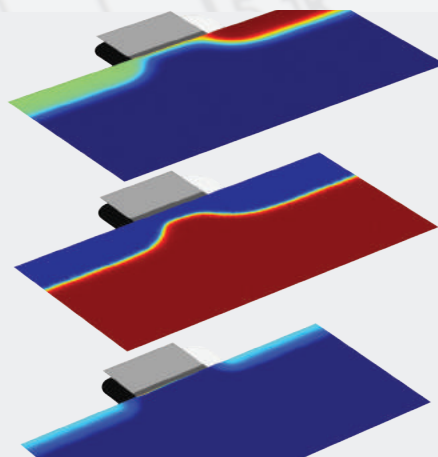


3D ANALYSIS OF A BIPOLAR TRANSISTOR

The device is simulated while operating in the common-emitter regime.

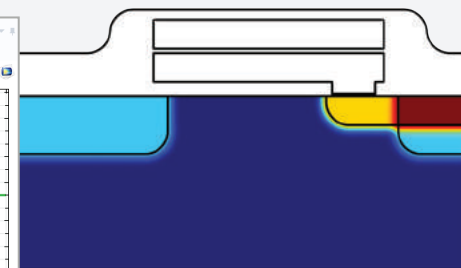
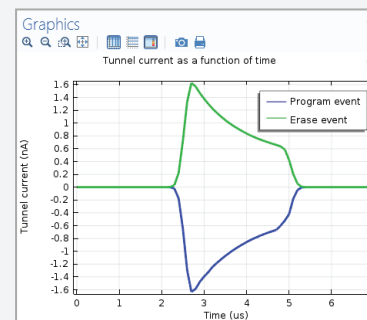
SEMICONDUCTOR DESIGN

2D model of a MOSFET transistor. Simulation results demonstrate the transistor operation where an applied gate voltage turns the device on. The drain saturation current is then determined. From top to bottom: electric potential, hole density, and electron density are shown.



PROGRAMMING OF A FLOATING GATE EEPROM DEVICE

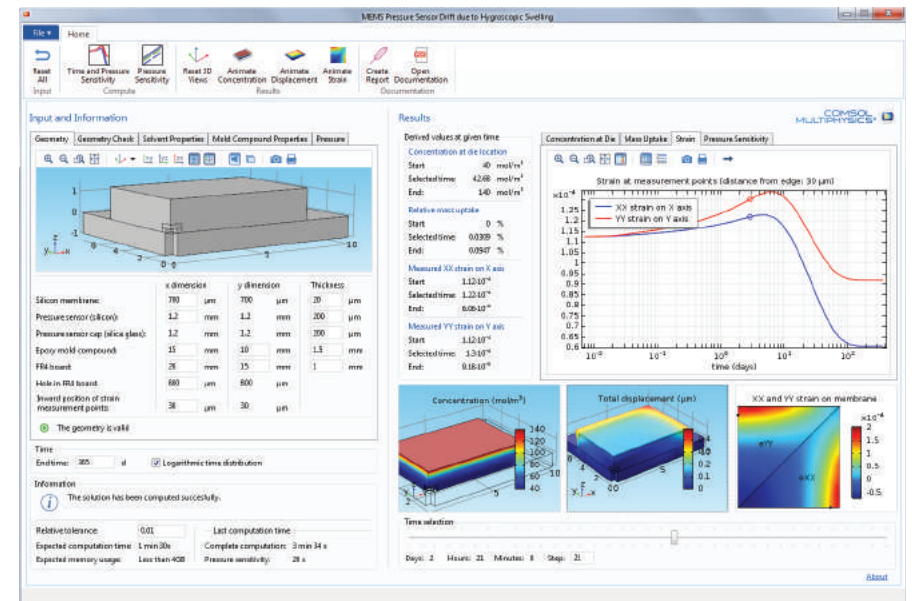
This model calculates the current and charge characteristics of a floating gate electrically erasable programmable read-only memory (EEPROM) device. Shown are the tunnel current as a function of time throughout the program and erase events, and signed dopant concentration.



MEMS Module

The design and modeling of microscale electromechanical systems (MEMS) is a unique engineering discipline. At small length scales, the design of sensors and actuators must consider the effects of several physical phenomena. To this end, the MEMS Module provides physics interfaces for electromagnetic-structure, thermal-structure, or fluid-structure interactions. A variety of damping phenomena can be included in a model: thin-film gas damping, anisotropic loss factors for solid and piezo materials, as well as thermoelastic damping and anchor damping. For elastic vibrations and waves, perfectly matched layers (PMLs) provide state-of-the-art absorption of outgoing elastic energy.

Best-in-class piezoelectric and piezoresistive tools allow for simulations where composite piezo-elastic-dielectric materials can be combined in any imaginable configuration. The module includes analyses in the stationary and transient domains as well as fully coupled eigenfrequency, parametric, quasi-static, and frequency-response analyses. The extraction of capacitance-, impedance-, and admittance-lumped parameters is straightforward, as is connecting a model to an electrical circuit. Circuits can be defined manually using the Electrical Circuit interface or can be imported directly from a SPICE netlist. Built upon the core capabilities of COMSOL Multiphysics, the MEMS Module can be used to address virtually any phenomena related to mechanics at the microscale.

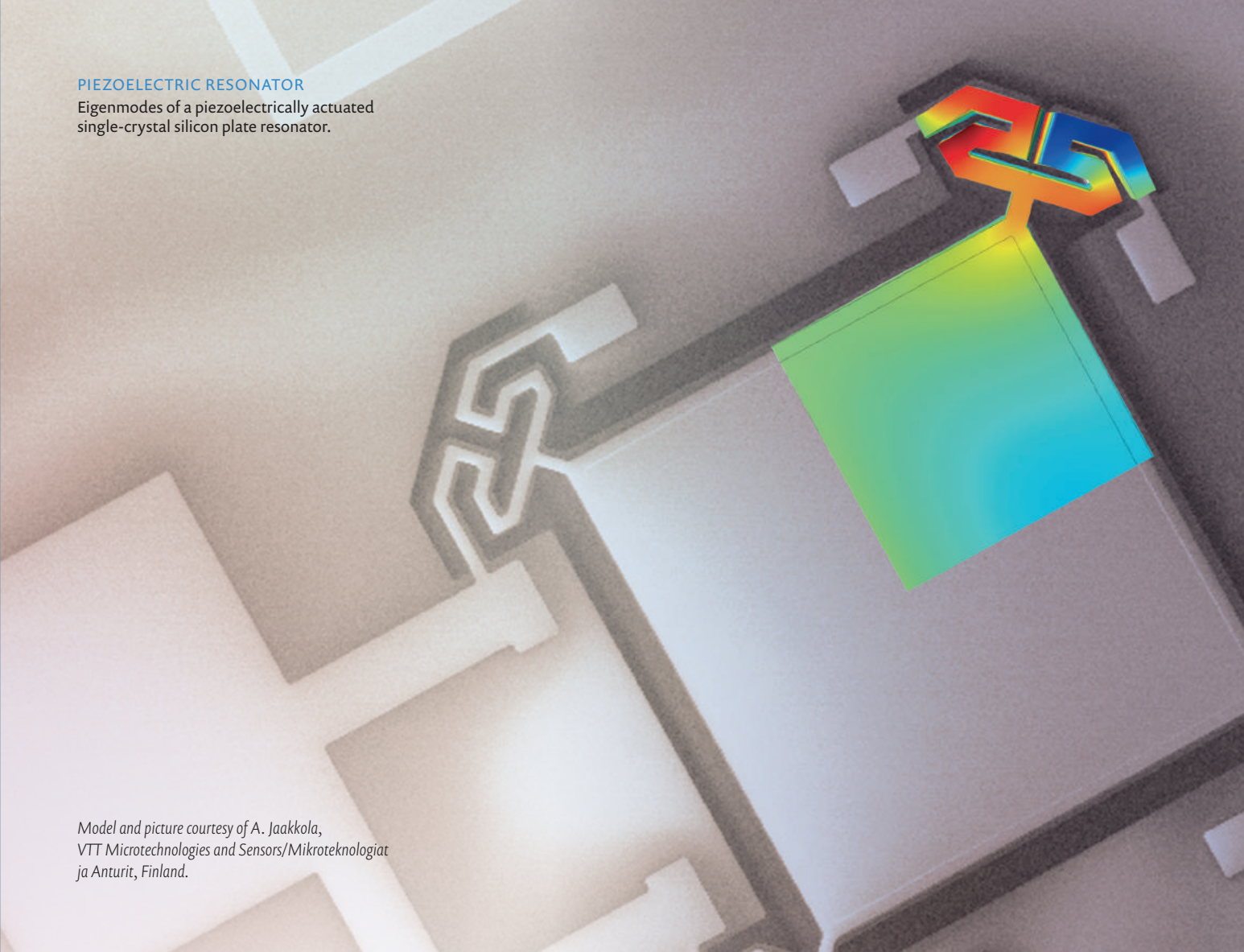


MEMS PRESSURE SENSOR DRIFT DUE TO HYGROSCOPIC SWELLING

To compute the drift of measured strain in a MEMS pressure sensor due to hygroscopic swelling, a specialized app was created. App users can control the geometric design of the sensor, solvent properties, and material properties of an epoxy mold compound (EMC) to calculate drift for different scenarios.

PIEZOELECTRIC RESONATOR

Eigenmodes of a piezoelectrically actuated single-crystal silicon plate resonator.



Model and picture courtesy of A. Jaakkola, VTT Microtechnologies and Sensors/Mikroteknologiat ja Anturit, Finland.

HIGHLIGHTS

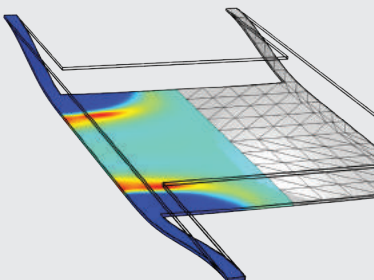
- Accelerometers
- Actuators
- Anchor damping
- Bulk Acoustic Wave (BAW) devices
- Cantilever beams
- Fluid-structure interaction (FSI)
- Loss factors
- MEMS capacitors, gyroscopes, resonators, and thermal devices
- Piezoelectric devices
- Piezoresistive devices
- RF MEMS devices
- Sensors
- Structural contact and friction
- Surface Acoustic Wave (SAW) devices
- Thermoelastic damping
- Thin-film gas damping
- User-defined material models

APPLICATION EXAMPLES

- MEMS pressure sensor swelling
- Microresistor beam

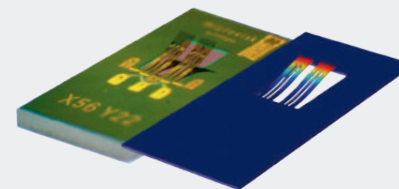
PULL-IN ANALYSIS OF AN RF MEMS SWITCH

Model of a thin micromechanical bridge suspended over a dielectric layer. Results show the contact force and displacement when the device is pulled in.



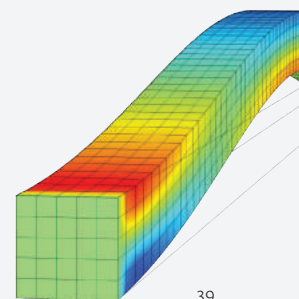
ACTUATORS & SENSORS

A two-arm thermal actuator is activated through thermal expansion, where the temperature profile is plotted on the resulting deformed shape. Relative viscoelasticity changes during the clotting of blood are then measured through deflections of the microcantilevers.



MEMS RESONATOR

This model accounts for the thermoelastic damping arising when a material is subjected to cyclic stress. Results show the mode shape and the corresponding temperature distribution within the beam when the resonator is operating at an eigenfrequency of 63.3 kHz. The computed quality factor is 10,700.

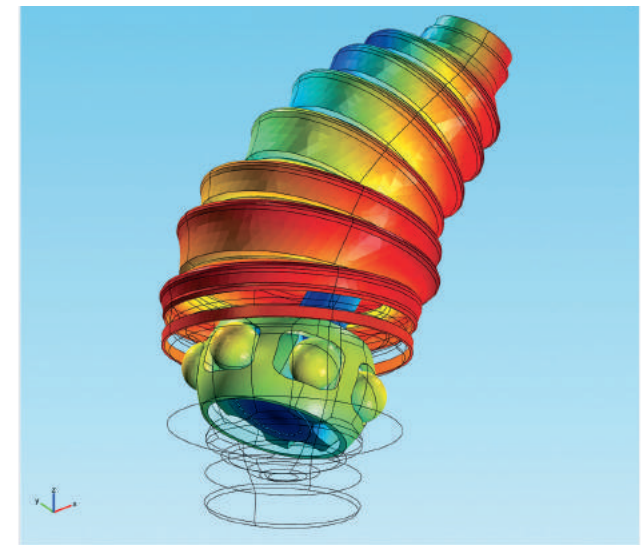


Structural Mechanics Module

The Structural Mechanics Module is dedicated to the analysis of components and subsystems where it is necessary to evaluate deformations under loads. It contains special user interfaces for the modeling of plates, shells, membranes, beams, and trusses.

The Structural Mechanics Module is used for a wide range of analysis types, including stationary, time-dependent, eigenmode, parametric, quasistatic, frequency-response, and buckling analyses. Its user interfaces allow for geometrically nonlinear analysis and mechanical contact. Predefined multiphysics couplings are available for heat transfer, acoustics, fluid flow, and electrostatics. This allows for the easy setup of applications including piezoelectric materials, fluid-structure interaction (FSI), and thermal strain.

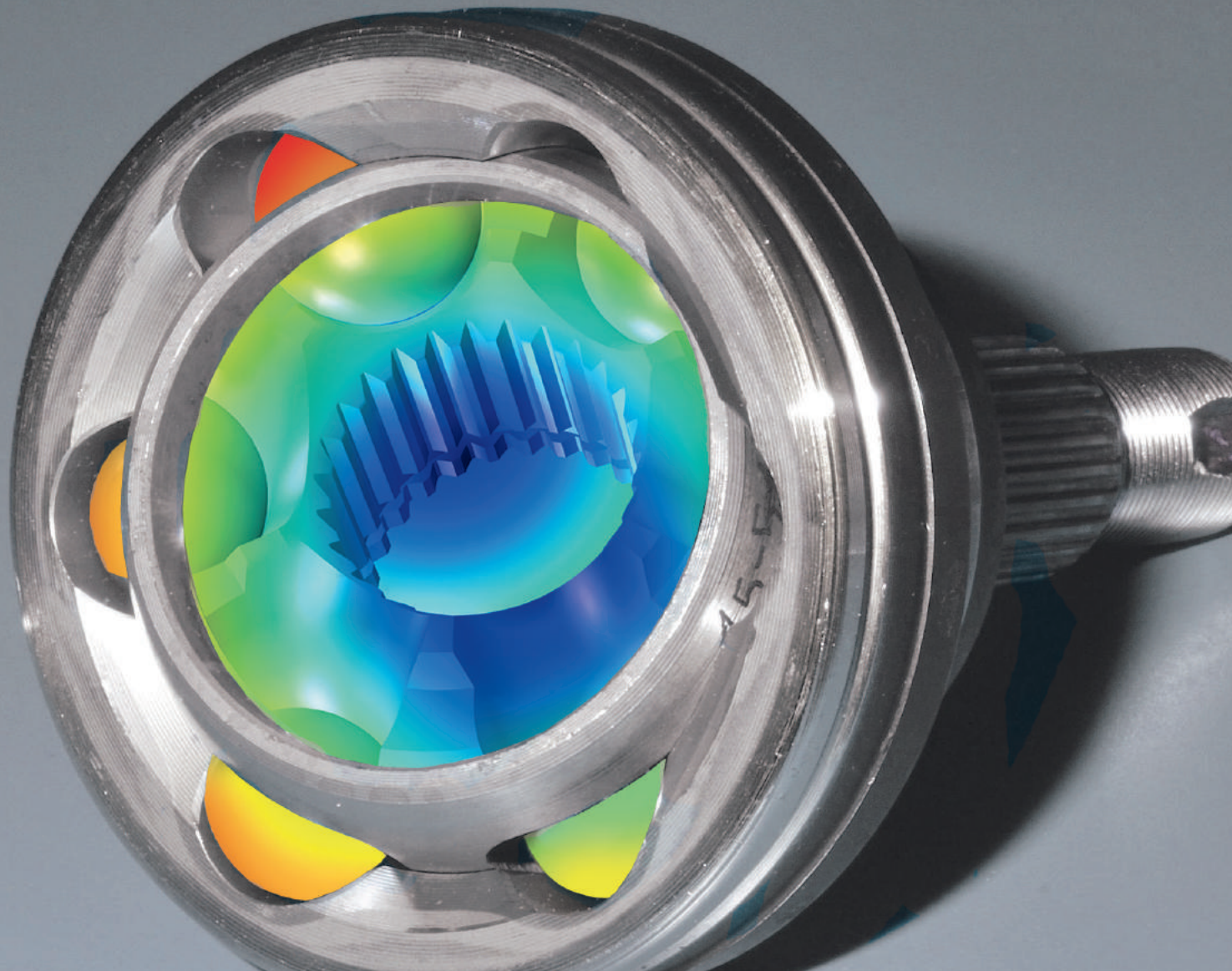
Four add-on modules are available for the Structural Mechanics Module: the Nonlinear Structural Materials Module, Geomechanics Module, Fatigue Module, and Multibody Dynamics Module. The Structural Mechanics Module also works in tandem with COMSOL Multiphysics and the other discipline-specific modules to couple structural analysis with any multiphysics phenomenon.



CV JOINT

Von Mises stresses and deformation in the ball bearings, cage, and rubber seal of a continuous velocity (CV) joint.

Model courtesy of Fabio Gatelli, Metelli S.p.A., Cologne, Italy.



HIGHLIGHTS

- Bolt pretension
- Buckling and postbuckling
- Built-in connections between solids, shells, and beams
- Centrifugal, Coriolis, and Euler forces
- Contact and friction
- Dynamic analysis of prestressed structures
- Elastic waves
- Euler and Timoshenko beams
- Fluid-structure interaction (FSI)
- Geometric nonlinearity
- Hygroscopic swelling
- Infinite elements, PMLs, and low-reflecting boundaries
- Lubrication and elastohydrodynamics*
- Modal analysis
- Periodic boundary conditions
- Piezoelectric devices
- Rigid domains
- Springs and dampers
- Stress-optical effects
- Structural vibrations
- Thermal stress
- User-defined material models
- Viscoelasticity
- Wires and cables

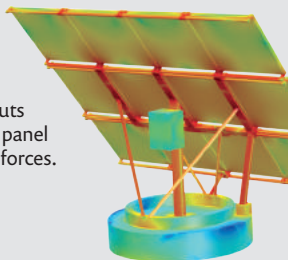
* Together with the CFD Module

APPLICATION EXAMPLES

- Beam section calculator
- Bike frame analyzer
- Interference fit
- MEMS pressure sensor swelling
- Truss bridge designer
- Truss tower buckling
- Viscoelastic structural damper

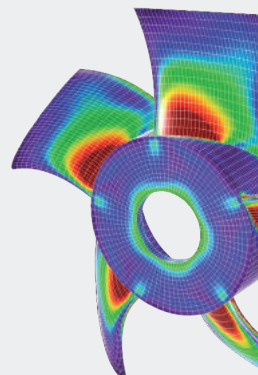
SOLAR PANEL

Stresses on the struts supporting a solar panel subjected to wind forces.



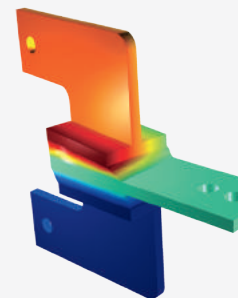
FAN ASSEMBLY

Vibrational analysis of a fan assembly using nonconforming meshes with hanging nodes. Swept meshes are used in all domains, resulting in a mix of hexahedral and prism elements.



DAMPING ELEMENT

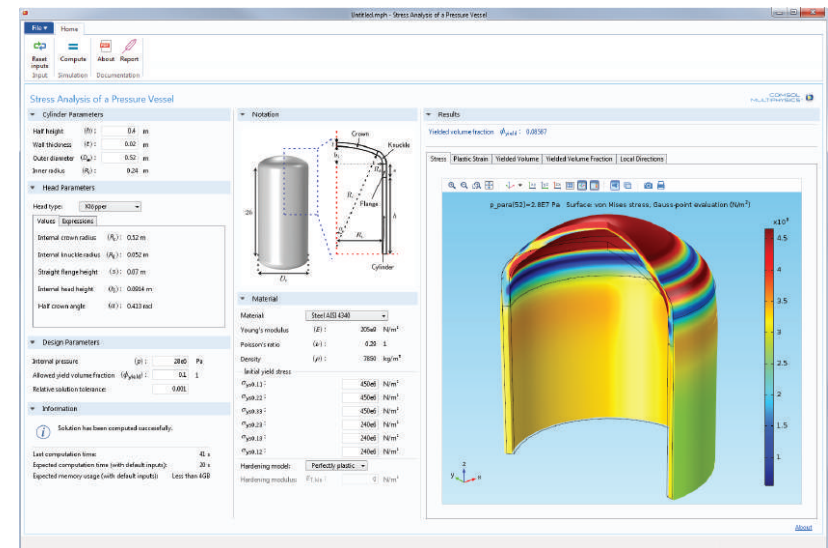
Structural analysis of a viscoelastic damping element used to stabilize tall buildings. Shown is the displacement as both the color and deformed shape plot.



Nonlinear Structural Materials Module

The Nonlinear Structural Materials Module augments the mechanical capabilities of the Structural Mechanics Module and MEMS Module by adding nonlinear material models. When the mechanical stress in a structure becomes large, certain nonlinearities in the material properties force the user to abandon linear material models. This situation also occurs in some operating conditions, such as high temperature.

The module adds elastoplastic, viscoplastic, creep, nonlinear elastic and hyperelastic material models, and large strain plastic deformation. User-defined material models based on strain invariants, flow rules, and creep laws can easily be created directly in the user interface with the built-in constitutive laws as a starting point. Material models can be combined as well as include multiphysics effects. The tutorial models illustrate this by showcasing combined creep and elastoplasticity, thermally induced creep, and viscoplasticity.

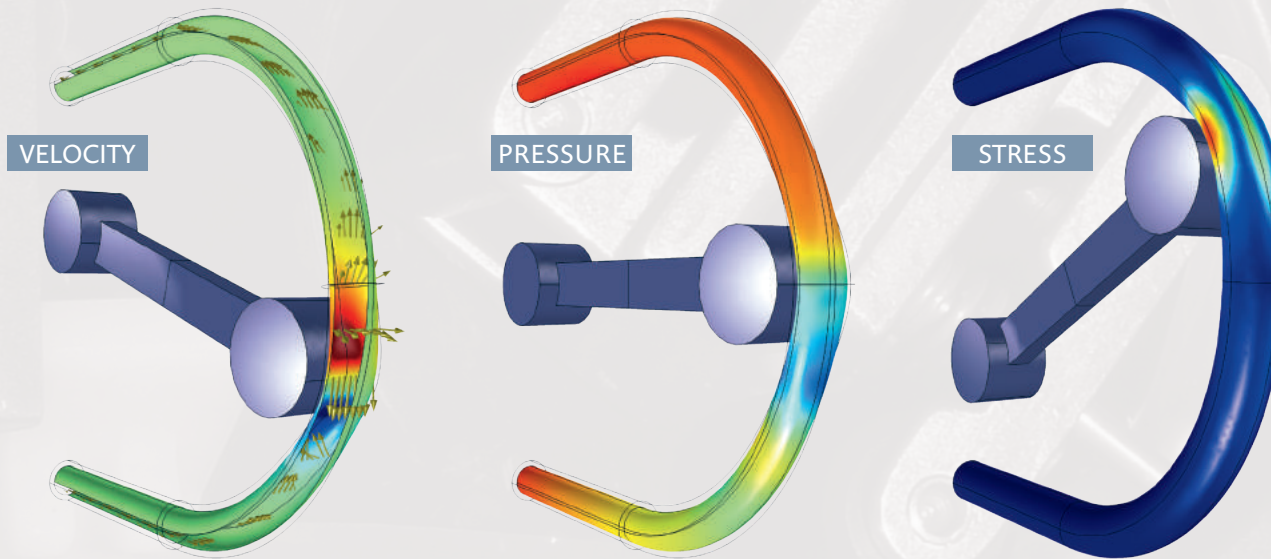


STRESS ANALYSIS OF A PRESSURE VESSEL

This specialized application analyzes whether a pressure vessel with a given geometry will be able to sustain an applied internal pressure without exceeding a certain volume fraction of the material that has exceeded the yield limit. Geometric parameters and materials may be changed to investigate different vessel designs.

PUMPING

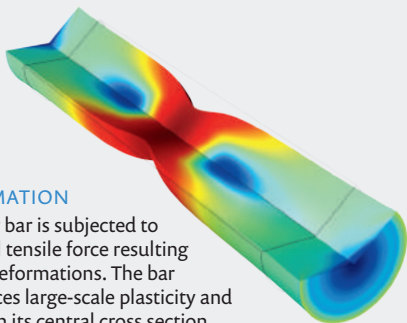
This model of a peristaltic pump accounts for the fluid-structure interaction caused by the roller squeezing the tubing's wall and the pressure resulting from the fluid flow within the tubing. The model considers large deformations, contact, and the hyperelastic behavior of the tubing material, modeled using the Arruda-Boyce material model. Shown are the von Mises stresses, fluid pressure, and velocity direction and magnitude at three different time intervals.



Model images are provided courtesy of Nagi Elabbasi of Veryst Engineering, Needham, MA. To read more, visit: comsol.com/papers/11574. Photograph of peristaltic pump first produced by Andy Dingley.

PLASTIC DEFORMATION

A circular bar is subjected to a uniaxial tensile force resulting in large deformations. The bar experiences large-scale plasticity and necking in its central cross section.



CLAMPING

This model shows the von Mises stresses of a pipe during its flattening. The material model assumes large-strain elastoplastic deformation.

HIGHLIGHTS

- Anand viscoplasticity
- Biomechanics
- Creep material models
 - Coble
 - Deviatoric
 - Garofalo
 - Nabarro-Herring
 - Norton
 - Norton-Bailey
- Elastomers
- Elastoplasticity
- Hyperelastic material models
 - Arruda-Boyce
 - Blatz-Ko
 - Gao
 - Gent
 - Mooney-Rivlin
 - Murnaghan
 - St. Venant-Kirchhoff
 - Storakers
 - Varga
 - Yeoh
- Isotropic, kinematic, and perfectly plastic hardening laws
- Large-deformation and large-strain plasticity
- Nonlinear elastic models
 - Bilinear elastic
 - Uniaxial data
 - Ramberg-Osgood
 - Power law
- Polymers
- Rubber
- Temperature-dependent plasticity
- Tresca, von Mises, and Hill orthotropic plasticity
- User-defined creep, hyperelasticity, nonlinear elasticity, and plasticity

APPLICATION EXAMPLES

- Stress analysis of a pressure vessel

Geomechanics Module

The Geomechanics Module is a specialized add-on to the Structural Mechanics Module for the simulation of geotechnical applications such as tunnels, excavations, slope stability, and retaining structures. The module features tailored material models to study deformation, creep, plasticity, and the failure of soils and rocks, as well as their interaction with concrete and human-made structures. A variety of material models for soils are provided, including Cam-Clay, Drucker-Prager, Mohr-Coulomb, Matsuoka-Nakai, and Lade-Duncan. Other nonlinear elastic material models are available, such as Ramberg-Osgood, as are hyperbolic law models such as the Duncan-Chang, Duncan-Selig, and Hardin-Drnevich soil models.

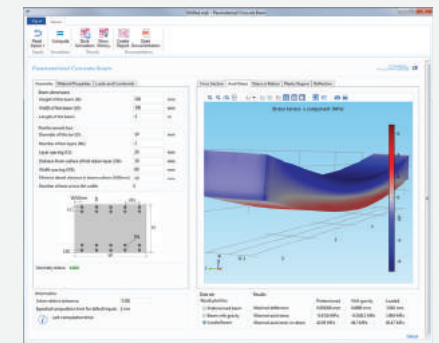
In addition to the built-in plasticity models, user-defined yield functions can be created using the versatile user interface provided by the COMSOL Multiphysics environment. Dependencies of a computed temperature field as well as other field quantities can be blended into these material definitions.

The Geomechanics Module also makes available very powerful tools for modeling concrete and rock materials. The Willam-Warnke, Bresler-Pister, Ottosen, and Hoek-Brown models are available as built-in options and can also be adapted and extended to a more general class of brittle materials. The Geomechanics Module can easily be combined with analysis from other modules, such as the porous media flow, poroelasticity, and solute transport functionality of the Subsurface Flow Module.



SETTLEMENT IN EMBANKMENTS

Long-term settlement in clay constitutes an engineering challenge in, for example, road and railway design and construction, where supporting lime-cement columns are used. A test site was constructed to evaluate, among other things, the deformation properties of the supporting columns and surrounding clay. Shown in the model are the stresses in the vertical direction in the supporting columns and the displacement of the surrounding environment (background boundary surface plot).



HIGHLIGHTS

- Bresler-Pister, Ottosen, and Willam-Warnke concrete models
- Brittle materials
- Creep
- Ductile materials and saturated soils
- Embankments
- Excavations
- Foundations
- Hoek-Brown rock model
- Nuclear waste installations
- Retaining structures and reinforcements
- Roads
- Slabs
- Slope stability
- Soil models
 - Drucker-Prager
 - Duncan-Chang
 - Lade-Duncan
 - Matsuoka-Nakai
 - Modified Cam-Clay
 - Mohr-Coulomb
 - Ramberg-Osgood
- Soil and rock modeling
- Tunnels
- User-defined soil, rock, and concrete materials

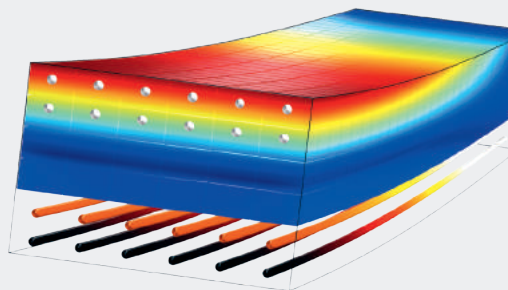
APPLICATION EXAMPLES

- Parameterized concrete beam

Model courtesy of Mats Olsson, Department of Civil and Environmental Engineering, Chalmers University of Technology, Sweden.
Photo courtesy of Torbjörn Edstam.

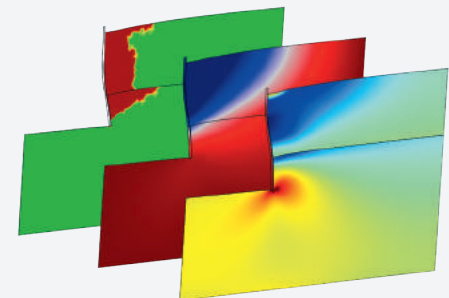
PARAMETERIZED CONCRETE BEAM

This application allows the user to calculate axial stress and deflection for a concrete beam reinforced by steel rebars. Input parameters such as geometry, material properties, loading conditions, and distribution of the rebars may be varied in order to investigate different concrete beam configurations.



EXCAVATION

The horizontal stresses, deformation, and plastic regions are plotted from a model of the excavation of soil. The Drucker-Prager soil model is used in the simulation.

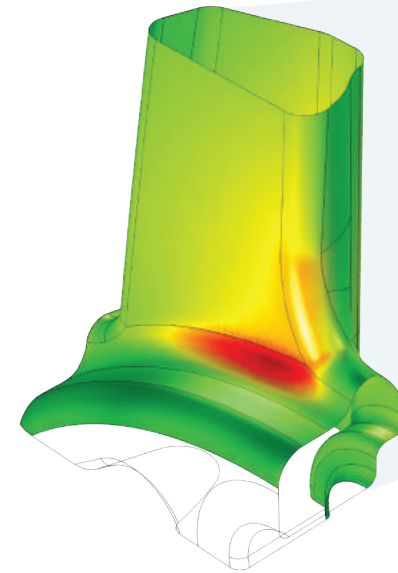


Fatigue Module

When structures are subjected to repeated loading and unloading, material fatigue can cause component failure at loads below the static limit. A virtual fatigue analysis can be performed in the COMSOL Multiphysics environment with the Fatigue Module, an add-on to the Structural Mechanics Module.

The stress-life and strain-life models evaluate fatigue in structures subjected to proportional loading, while the stress-based and strain-based models are suitable in the case of nonproportional loading. With these models, you can evaluate fatigue in the high-cycle and low-cycle regimes. In applications involving nonlinear materials, you can use energy-based and strain-based methods to simulate, for example, thermal fatigue. When considering variable loads, the accumulated damage can be calculated from the load history and the fatigue limit.

The fatigue load cycle can be simulated for solid bodies, plates, shells, multibody dynamics models, applications involving thermal stress and deformation, and even for piezoelectric devices. A fatigue evaluation can be performed on domains, boundaries, lines, and points in order to improve computational efficiency when evaluating subsurface or surface-initiated fatigue.



HIGH-CYCLE FATIGUE ANALYSIS OF A CAR WHEEL

High-cycle stress-based fatigue analysis of a ten-spoke car wheel. The highest stresses occur in the fillet, where the spoke connects to the hub. Results show the von Mises stress distribution for the whole wheel (above on the right) and the fatigue usage factor according to the Findley criterion for the fillet (above).

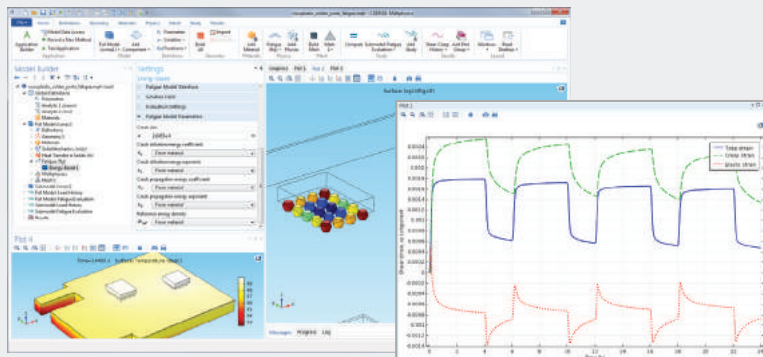


HIGHLIGHTS

- Cumulative damage analysis
 - Palmgren-Miner linear damage rule
 - Rainflow counting
- Energy-based fatigue
 - Darveaux criterion
 - Morrow criterion
- Fatigue usage factor computation
- High-cycle fatigue and low-cycle fatigue
- Strain-based fatigue
 - Fatemi-Socie criterion
 - Mean stress correction
 - Notch approximation
 - Smith-Watson-Topper (SWT) criterion
 - Wang-Brown criterion
- Strain-life models
 - Coffin-Manson criterion
 - Combined Basquin and Coffin-Manson criterion
 - E-N curve criterion
- Stress-based fatigue
 - Findley criterion
 - Matake criterion
 - Normal stress criterion
- Stress-life models
 - Basquin criterion
 - S-N and approximate S-N curve criteria

APPLICATION EXAMPLES

- Frame fatigue life

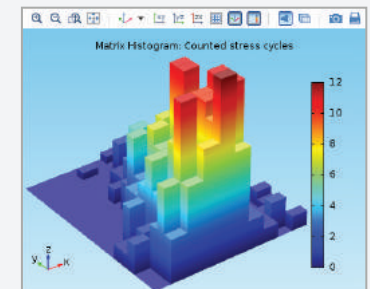


THERMAL FATIGUE EVALUATION

A continuous power switching introduces thermal cycling in a microelectronic component. Fatigue in two ball grid arrays of viscoplastic solder joints is predicted using an energy-based model. The visualizations (left to right) show the temperature field, the lifetime logarithm, as well as the creep total and elastic strain components history.

CUMULATIVE DAMAGE ANALYSIS OF RANDOM LOADS

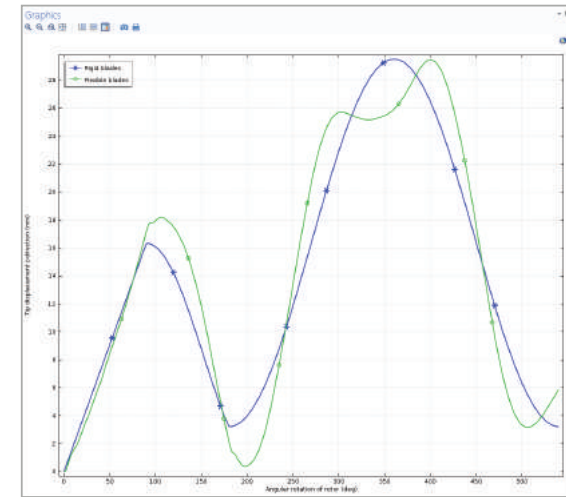
A load-carrying beam with a central cutout is subjected to a random nonproportional load. A fatigue response analysis is carried out and the load distribution at the point of highest fatigue usage is captured and shown in the Rainflow histogram.



Multibody Dynamics Module

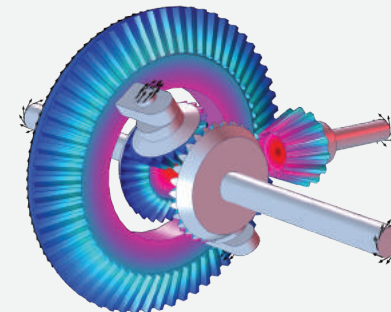
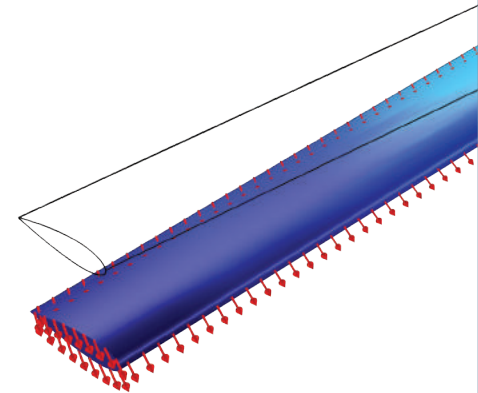
The Multibody Dynamics Module is an add-on to the Structural Mechanics Module that provides an advanced set of tools to design and optimize multibody systems using finite element analysis. The module enables simulation of a mixed system of flexible and rigid bodies, each of which may be subjected to large rotational and translational displacements. The relative motion of the bodies can be restricted by specifying rotational and translational constraints on joints.

All bodies in a multibody structural mechanics model have elastic properties by default and can be made rigid through tagging them selectively with a Rigid Domain property. Boundaries or parts of boundaries of flexible bodies can also be made rigid. You can assign nonlinear material properties to the flexible bodies in a multibody system by combining the Multibody Dynamics Module and the Nonlinear Structural Materials Module. Transient, frequency-domain, eigenfrequency, and stationary multibody dynamics analyses can be performed. Joints can be assigned translational or torsional springs with damping properties, applied forces and moments, and prescribed motion as a function of time. Friction can also be included on joints to model the losses or stabilize the system.



HELICOPTER SWASHPLATE MECHANISM

Transient simulation with both rigid and flexible blade designs provides insight into useful performance metrics such as blade deformation and lift force.



DIFFERENTIAL GEAR

Simulation of the mechanism of a differential gear used in cars and other wheeled vehicles. The model is built using the available functionality for simulating gears and racks, including spur gears, helical gears, bevel gears, worm gears, spur racks, and helical racks. Different types of gears and racks can be connected. A gear mechanism can be built with gear parts imported from the Parts Library.

HIGHLIGHTS

- Applications
 - Automotive and aerospace
 - Biomechanics
 - Biomedical instruments
 - Machinery and robotics
 - Vehicle and engine dynamics
- Joints
 - Ball
 - Cylindrical
 - Distance
 - Fixed
 - Hinge
 - Planar
 - Prismatic
 - Reduced Slot
 - Screw
 - Slot
 - Universal
- Structural elements
 - Rigid, elastic, shell, and beam
 - Nonlinear solid, piezoelectric

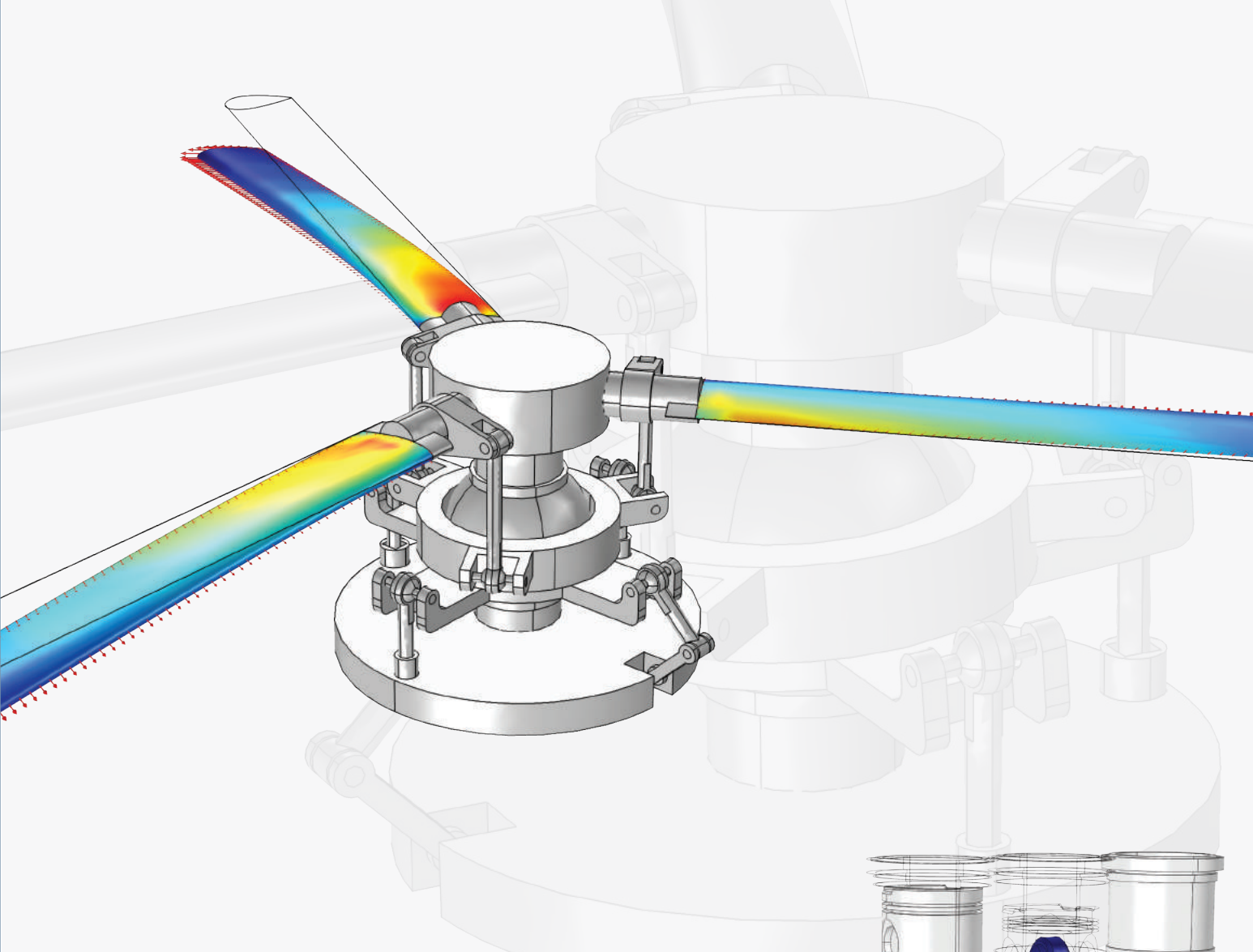
CAPABILITIES

- Consistent initialization for transient analysis
- Coupling multibody systems with other physical phenomena
- Eigenmodes of flexible multibody systems
- Fatigue analysis of critical flexible bodies*
- Reaction forces and moments at a joint
- Relative displacement/rotation between two components and their velocities
- Static, transient, and frequency-response analysis
- Stresses and deformations in flexible bodies

* Together with the Fatigue Module

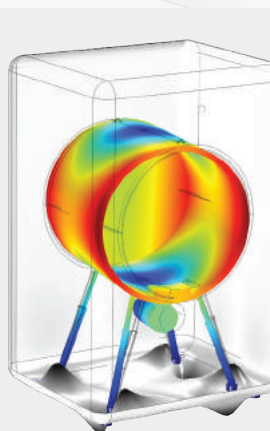
APPLICATION EXAMPLES

- Centrifugal governor simulator
- Truck mounted crane analyzer



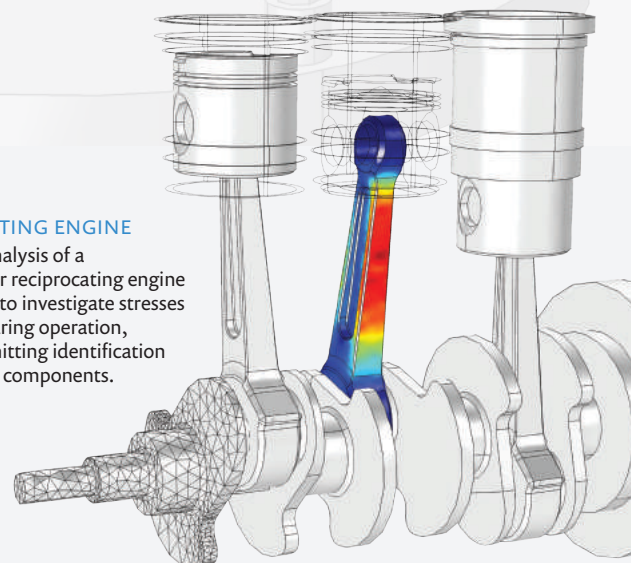
VIBRATION IN A WASHING MACHINE ASSEMBLY

Multibody dynamics simulation of a horizontal-axis portable washing machine. An eigenfrequency analysis is performed to compute the natural frequencies and mode shapes of the entire assembly. The vibrations induced in the housing during the spinning cycle are determined with a transient analysis.



RECIPROCATING ENGINE

A dynamic analysis of a three-cylinder reciprocating engine is performed to investigate stresses generated during operation, thereby permitting identification of the critical components.

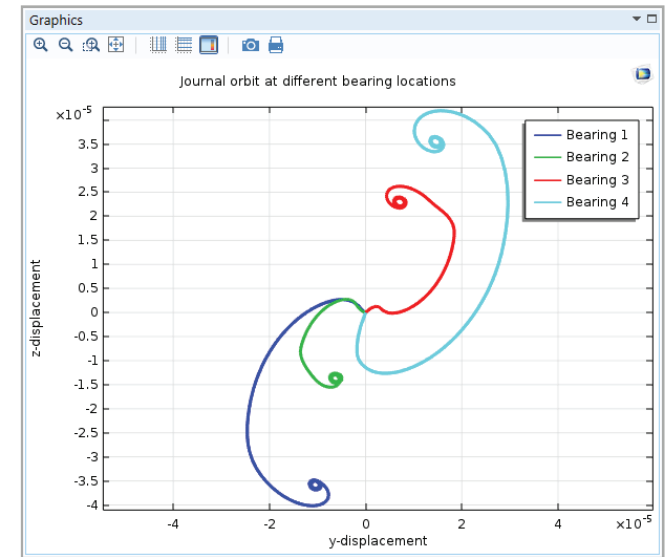


Rotordynamics Module

The study of vibrations is very important in the design of rotating machinery employed in a wide array of industries, such as the automotive, aerospace, power plant, electrical machinery, and household appliance. The Rotordynamics Module, available as an add-on product to the Structural Mechanics Module, provides dedicated tools to predict and analyze vibrations in rotating components using finite element analysis (FEA). Using this module, you can take into account the effects of several stationary and moving rotor components, including disks, bearings, and foundations.

You can also easily postprocess your results, presenting them as Campbell diagrams, orbits, waterfall plots, and whirl plots. A specific physics interface for hydrodynamic bearings allows you to simulate the pressure profile in the lubricant film for a range of standard bearing profiles such as plain, elliptic, multilobe, and tilted pad. You can also model the interaction between the journal motion and the pressure distribution in the lubricant film simultaneously for accurate simulation of the rotor vibration.

Multiple rotors connected through gears can also be simulated by combining the Rotordynamics Module with the Multibody Dynamics Module. All the rotor components have linear elastic properties by default and can be made rigid. You can perform time dependent, frequency domain, eigenfrequency, and stationary analysis for the rotor assembly. A special analysis, Time Dependent with FFT, can be used to generate the waterfall plot.



ROTORDYNAMICS ANALYSIS

Eccentricity of the crankpin and the balance masses induce the self excited vibrations in the reciprocating engine crankshaft. A transient rotordynamics simulation of a three cylinder reciprocating engine crankshaft is performed by coupling its dynamics with the pressure distribution in the bearing. Journal orbits (above) and pressure distribution in the fluid film and the stress distribution in the crankshaft (above right) are shown.

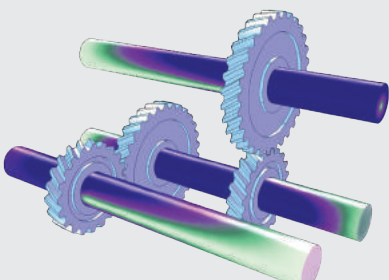
HIGHLIGHTS

- Solid, beam, and rigid elements
- Lateral, torsional, and axial vibrations
- Critical speeds of the rotor
- Damped natural frequencies and stability
- Variable speed analysis
- Hydrodynamic bearings
 - Plain
 - Elliptic
 - Split halves
 - Multilobe
 - Tilted pad
- Gears and gear pairs*
 - Spur gear
 - Helical gear
 - Bevel gear
- Results
 - Whirl plot
 - Waterfall plot
 - Campbell plot
 - Orbits

* Together with the Multibody Dynamics Module

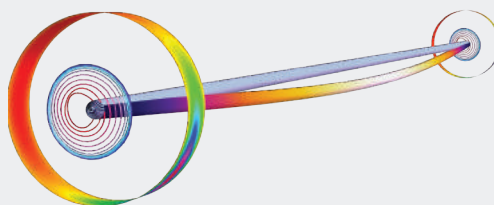
GEARED ROTOR SYSTEM

Rotordynamics analysis of multiple rotors connected through helical gears. The presence of gears in the system induces both lateral and torsional vibrations in the rotors. Simulation results show von Mises stress distribution in the shafts.



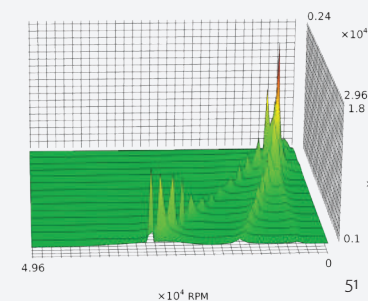
WHIRLING OF A SHAFT

Dynamic analysis of a uniform shaft under gravity and supported by two hydrodynamic journal bearings. Rotor stress profile, bearing fluid pressure, and orbit of the journals at the bearings are shown.



SIMPLY SUPPORTED ROTOR

Time Dependent with FFT (Fast Fourier Transform) analysis of a simply supported rotor. Simulation results show the absolute vertical displacement at the first bearing location with respect to variation in the frequency spectrum of the rotor and change in its angular speed (waterfall plot).



Acoustics Module

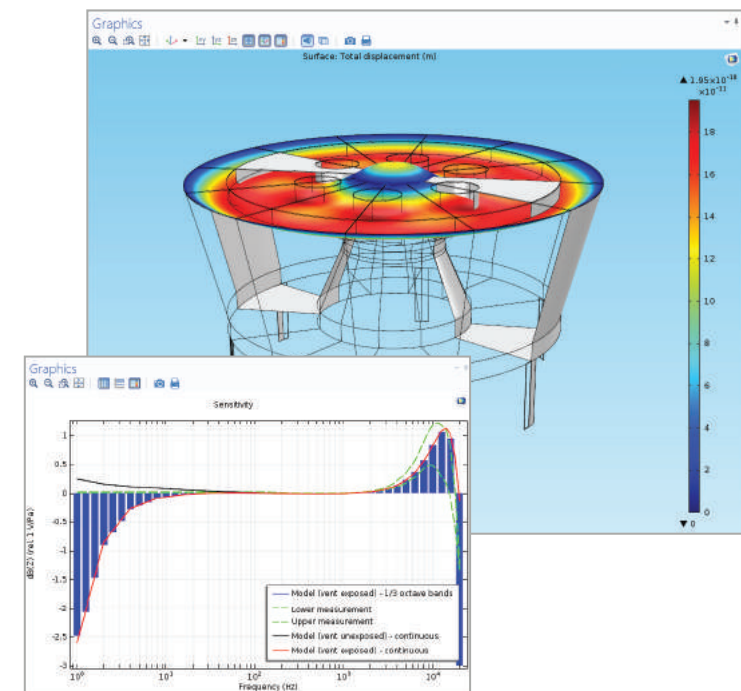
The Acoustics Module is a world-class solution to your acoustics modeling needs. This module is designed specifically for those who work with devices that produce, measure, and utilize acoustic waves. Application areas include speakers, microphones, hearing aids, and sonar devices, as well as noise control that can address muffler design, sound barriers, diffusers and absorbers, and building and room acoustics applications.

Easy-to-use physics interfaces provide the tools to model acoustic pressure wave propagation in air, water, and other fluids. Modeling tools for thermoviscous acoustics enable highly accurate simulation of miniaturized speakers and microphones in handheld devices. Dedicated aeroacoustic interfaces allow the simulation of acoustic waves propagating in moving fluids. Ray tracing and acoustic diffusion interfaces can be used for modeling acoustics in rooms and other large spaces. You can also model vibrations and elastic waves in solids, piezoelectric materials, and poroelastic structures. A wide range of predefined, easy-to-use multiphysics couplings allow the modeling of acoustic-solid, acoustic-shell, acoustic-porous, and piezo phenomena to bring your acoustic simulations to a new level of predictive power.

B&K 4134 CONDENSER MICROPHONE

Electromechanical acoustic simulation of a pressure response type microphone for medium and high audio levels and for coupler measurements. Results show the sensitivity level and membrane deformation.

Geometry, material parameters, and pictures courtesy of Brüel & Kjær Sound & Vibration Measurement A/S, Nærum, Denmark.

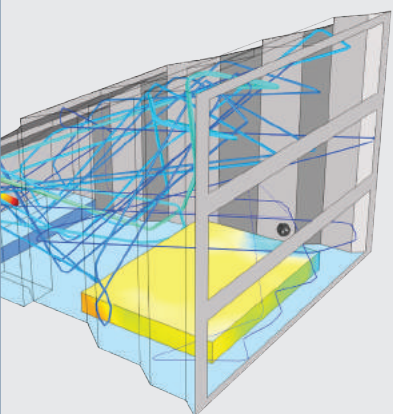


HIGHLIGHTS

- Acoustic diffusion equation
- Acoustic-structure interaction
- Biot's equations and parameters
- Elastic waves
- Electroacoustic transducers and speakers
- Hearing aids
- Linearized Euler equations
- Linearized Navier-Stokes equations
- Linearized potential flow
- Loudspeakers and microphones
- MEMS acoustics sensors and microphones
- Mufflers and particulate filters
- Noise and vibration of machinery
- Noise-reducing materials and insulation
- Piezoacoustics
- Poroelastic waves
- Ray tracing
- Room acoustics
- SONAR devices
- Structural vibrations
- Thermoacoustics and viscothermal losses

APPLICATION EXAMPLES

- Absorptive muffler designer
- Acoustic reflection analyzer
- One family house analyzer
- Organ pipe design
- Small concert hall analyzer

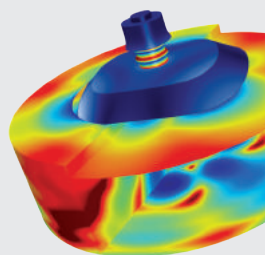


CONCERT HALL

This simulation of a small concert hall uses ray acoustics to determine sound pressure and impulse response.

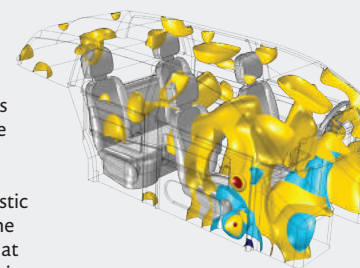
SONAR

The tonpiz piezo transducer is used for low-frequency, high-power sound emission. The model shows the voltage distribution in the piezoceramic rings, the deformation in the massive ends, and the pressure field underneath the transducer.



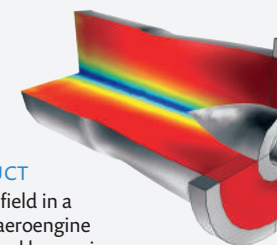
SEDAN INTERIOR

This model simulates the acoustics inside a sedan and includes sound sources at the typical loudspeaker locations. Results show the total acoustic pressure field and the frequency response at points inside the cabin.



ENGINE DUCT

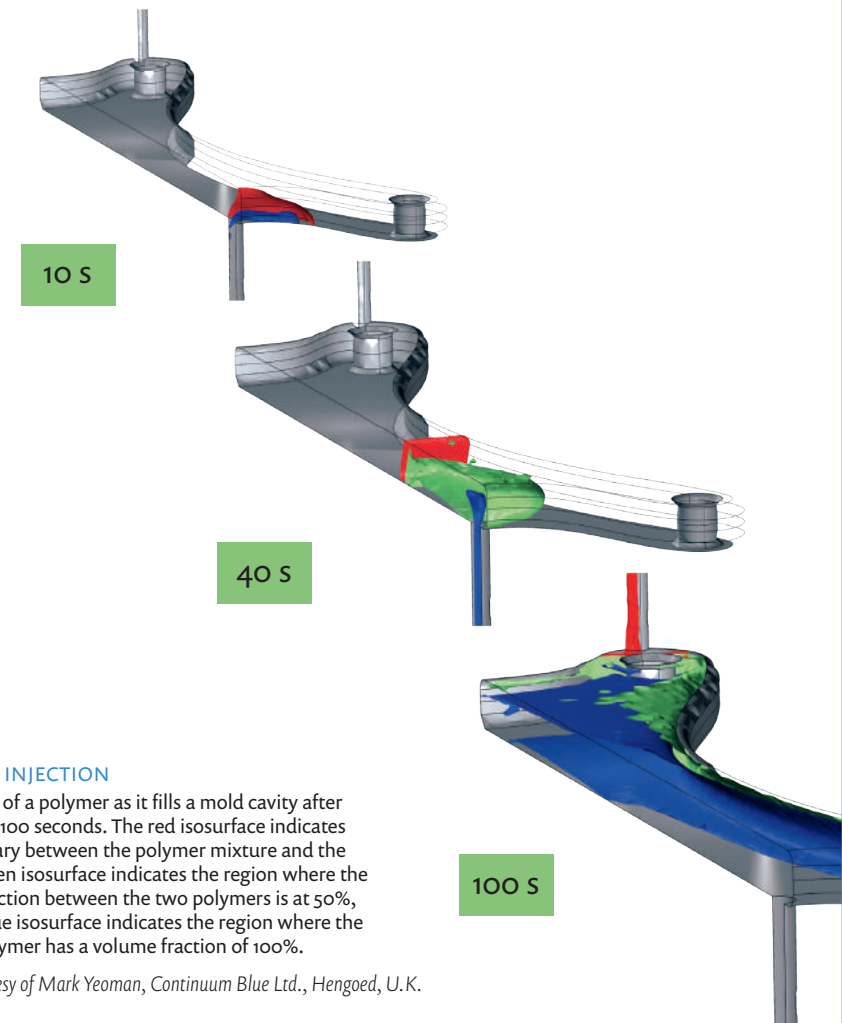
The acoustic field in a model of an aeroengine duct, generated by a noise source at the boundary, is computed and visualized.



CFD Module

The CFD Module is the premier tool for sophisticated fluid flow simulations in the COMSOL product suite. Incompressible and compressible formulations can be combined with advanced turbulence models for simulations of flow with heat and mass transfer. An important characteristic of the CFD Module is its ability to accurately capture multiphysics phenomena such as nonisothermal flow with conjugate heat transfer and viscous dissipation, fluid-structure interaction, non-Newtonian flow, and flow with concentration-dependent viscosity. The porous media flow physics interfaces allow for simulations in isotropic or anisotropic media with automatically coupled porous and free flow domains. Tools for modeling stirred vessels with rotating parts are also available.

Specific physics interfaces for reacting flow allows for simulations of species transport and chemical reactions in dilute and concentrated solutions. The module's interfaces for disperse two-phase flow include the mixture and Euler-Euler models for fine particle suspensions, and a bubbly flow model for macroscopic gas bubble flow. For interface tracking of two-phase flows, formulations are provided using the level set and phase field methods. A three-phase flow interface based on the phase field method is also available.



HIGHLIGHTS

- Conjugate heat transfer
- Cyclones, filters, and separation units
- Electronic cooling
- Fans, grills, screens, and pumps
- Flow around vehicles and structures
- Flows in pipes, valves, joints, and nozzles
- Fluidized beds and sprays
- Fluid-structure interaction (FSI)*
- Gas bubble flow and multiphase flow
- Heat exchangers and cooling flanges
- High Mach number flow
- Lubrication and elastohydrodynamics
- Medical/biophysical applications
- Mixers and stirred vessels
- Nonisothermal flow
- Non-Newtonian flow
- Polymer flow and viscoelastic flow
- Porous media flow
- Reacting flow**
- Sedimentation, emulsions, and suspensions
- Turbulent flow

* Together with the Structural Mechanics Module or MEMS Module

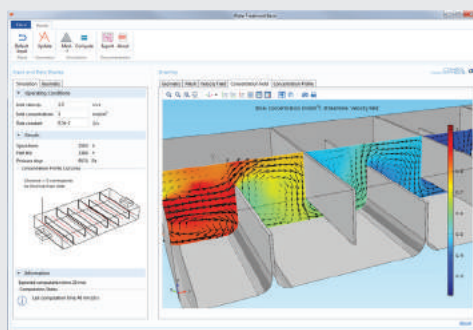
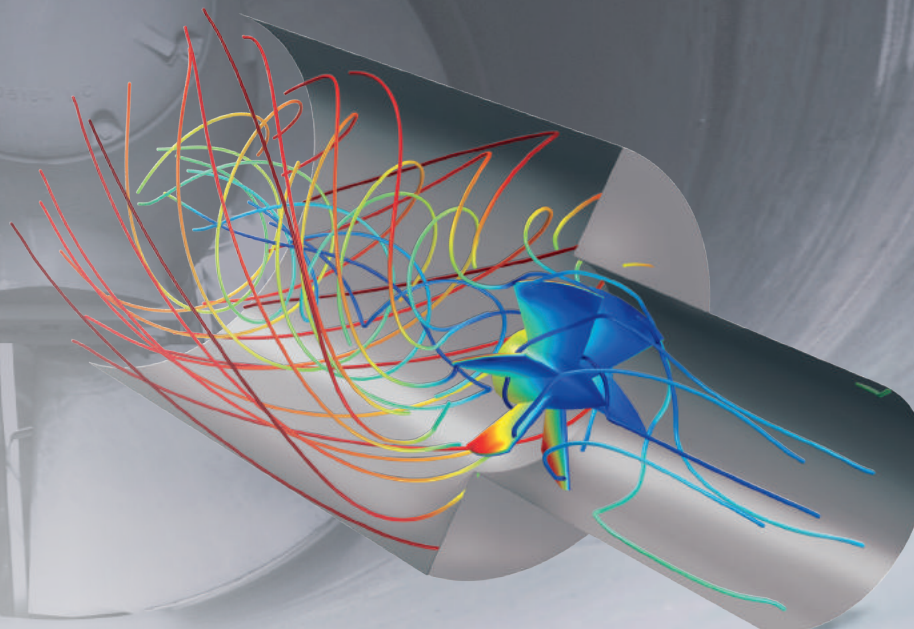
** Together with the Chemical Reaction Engineering Module

APPLICATION EXAMPLES

- Inkjet
- NACA airfoil optimization
- Water treatment basin

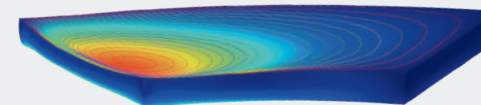
HVAC

Flow past a propeller in a duct. The streamlines show the direction of the flow and its magnitude, while the color plot shows the pressure on the propeller vanes.



WATER TREATMENT BASIN

This application can be used for modeling turbulent flow and chemical reactions in a water treatment process. The user may change inputs for the desired operating conditions and the geometry of the basin in order to test different designs.

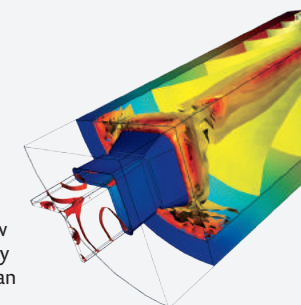


LUBRICATING OIL FILM

Tilted pad thrust bearings are used in rotating machines with high thrust loading. The picture shows pressure distribution contours for the lubricating engine oil together with elastic deformation (exaggerated) and effective stress.

METAL FORMING

Fluid-structure interaction (FSI) in the cast and mold of an aluminum extrusion process. The isosurfaces show the dynamic viscosity in the non-Newtonian aluminum flow.

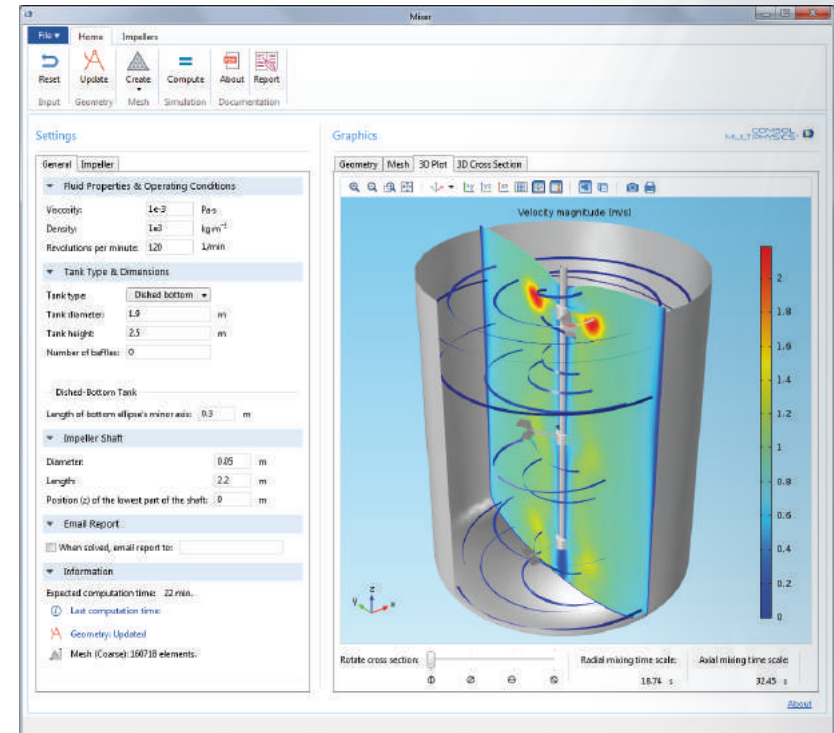


Mixer Module

An add-on to the CFD Module, the Mixer Module allows you to analyze fluid mixers and stirred reactors. The Mixer Module includes a series of physics interfaces that are based on the conservation laws for momentum, mass, and energy, in addition to species material balances in fluids. They are applicable to laminar and turbulent flow, incompressible and weakly compressible flow, and non-Newtonian flow. Various physics interfaces are available for describing the effects of temperature, reacting species, and free surface deformation in rotating and stationary domains.

The most accurate way to simulate mixing processes is a fully time-dependent study for fluid flow affected by rotating machinery, which accounts for the movement of geometric parts relative to each other. COMSOL defines one modeling domain that encompasses the impeller or rotor and another outside of this, where walls and obstacles such as baffles are found. Sliding mesh technology is then used to couple the flow in the two domains.

The Mixer Module also supplies a Frozen Rotor study type for simulating rotating flow. This study type significantly reduces the computational resources required to simulate pseudosteady-state conditions. The frozen rotor approximation is appropriate for mixers without baffles, pipes, or other geometric entities for which the topology relative to the rotating part does not change or when the entire system rotates, such as in centrifugal separation.



MIXER

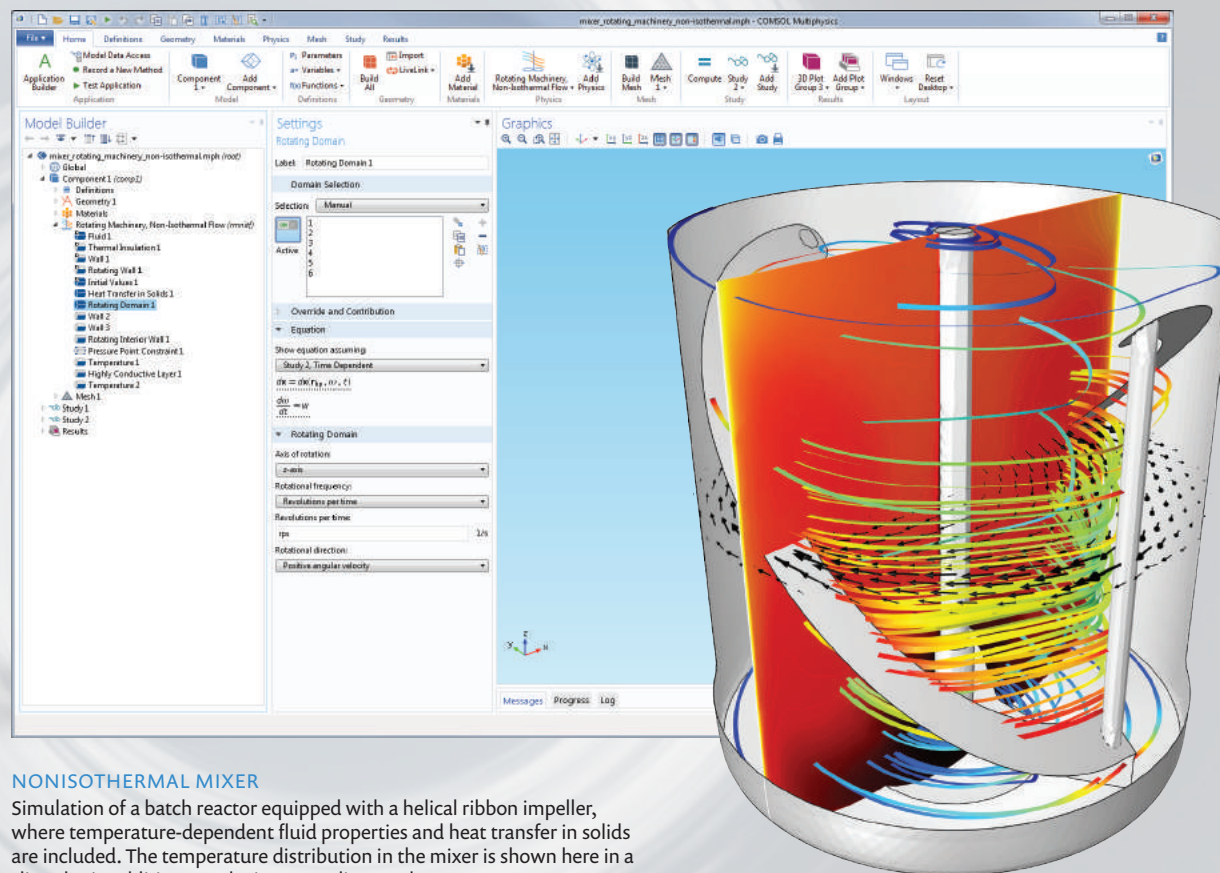
The analysis of mixers equipped with axial or radial impellers is performed with this custom application. For a given geometry, the app computes the efficiency of the mixing process. The vessels, which are available in three different shapes, can also be equipped with baffles and any of eleven different types of impellers.

HIGHLIGHTS

- 2D and 3D stationary and time-dependent studies
- Batch stirred reactors
- Continuous stirred reactors
- Fluid and chemical mixing
- Free fluid surfaces including the effects of surface tension forces and contact angles
- Frozen rotor approximation to simulate a rotating impeller
- Laminar, turbulent, incompressible, and weakly compressible flow
- Mixing of concentrated species
- Nonisothermal flow
- Non-Newtonian flow
- Optimization of vessel, baffle, and impeller designs
- Predefined libraries of surface tension coefficients
- Reacting flow
- Rotating machinery-based mixers
- Sliding mesh technology for a full rotation description
- Turbulent flow simulations with the algebraic $yPlus$ model, LVEL model, k -epsilon model, k -omega model, SST model, low Reynolds number k -epsilon model, and Spalart-Allmaras model

APPLICATION EXAMPLES

- Mixer



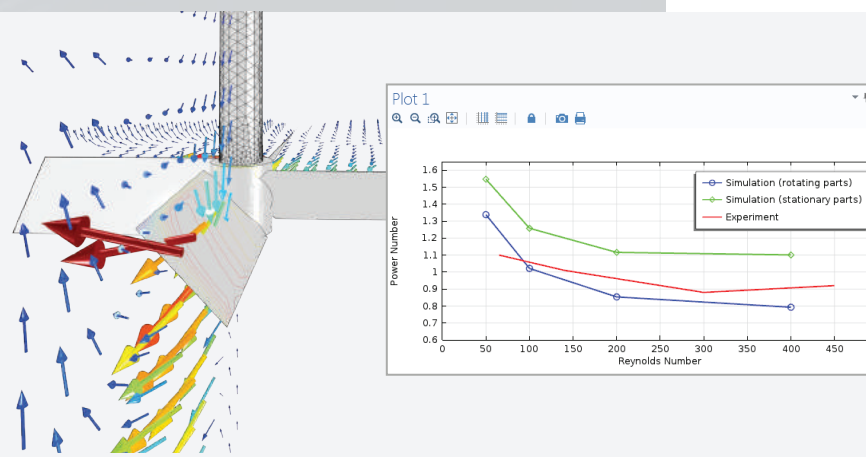
NONISOTHERMAL MIXER

Simulation of a batch reactor equipped with a helical ribbon impeller, where temperature-dependent fluid properties and heat transfer in solids are included. The temperature distribution in the mixer is shown here in slice plot in addition to velocity streamlines and arrows.

MIXING NON-NEWTONIAN FLUIDS

The frozen rotor feature is used to simulate the mixing of a non-Newtonian fluid. The arrows represent the direction and magnitude of flow around the impeller (left).

The power number used to predict the power draw of an impeller design has been calculated for four different Reynolds numbers (right).



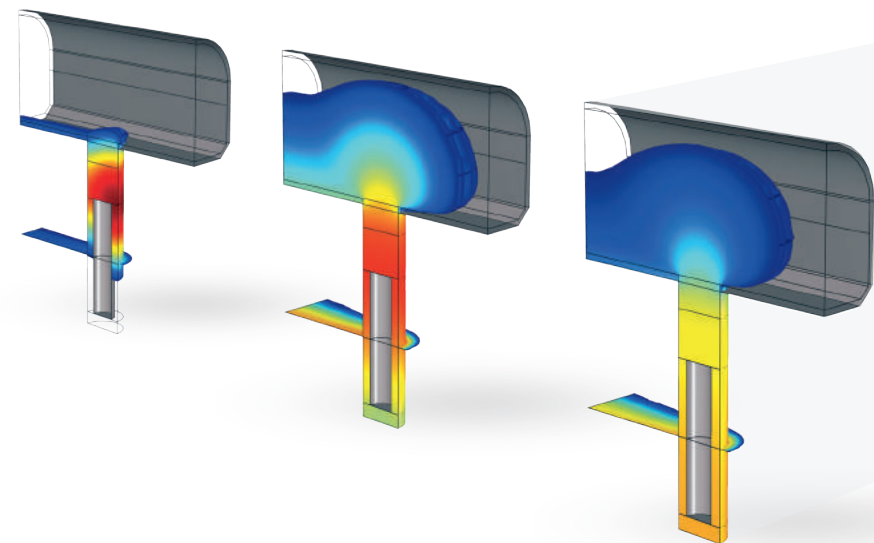
Subsurface Flow Module

The Subsurface Flow Module is designed for geophysical and environmental phenomena studies, such as the modeling of groundwater flow, the spread of pollution through soil, and oil and gas flow in porous media. The Richards' Equation interface describes nonlinear flow in variably saturated porous media, while the options for saturated porous media include the Darcy's Law interface for slow flow and the Brinkman Equations interface, where shear is nonnegligible. Free laminar Navier-Stokes flow can easily be combined with porous media flow and even include flow in thin fractures. The module also handles solute transport in free, saturated, and partially saturated porous media.

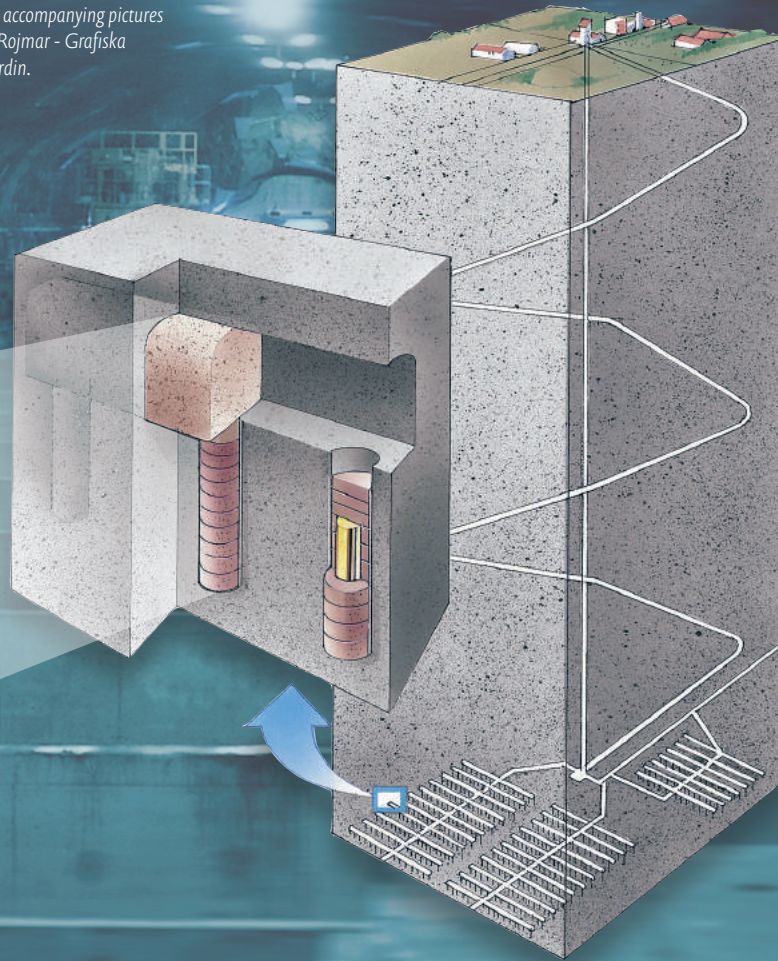
For heat transfer simulations, background geotherms are available as well as automated calculation of effective thermal properties for multicomponent systems, including heat transfer in solids, liquids, porous media, and fractures. Compaction and subsidence modeling is enabled by a very powerful user interface for poroelasticity. To apply multiphysics modeling to geophysical and environmental applications, the Subsurface Flow Module also allows arbitrary couplings to other physics interfaces in COMSOL Multiphysics, such as chemical reaction kinetics and electromagnetics.

NUCLEAR WASTE DISPOSAL

Nuclear waste repositories are now being built to store spent fuel rods for the next one hundred thousand years or so, and modeling has been used extensively to investigate them. This model shows a hypothetical case where a breach in the fuel bundle canister leads to leakage through a fracture in the surrounding rock and backfill in the tunnel above. The concentration distribution after 20, 200, and 2000 years is shown.



The model has been produced by Veli-Matti Pulkkanen, VTT Technical Research Centre of Finland. The accompanying pictures are Copyright © SKB. Illustrator: Jan M Rojmar - Grafiska Illustrationer. Photographer: Bengt O Nordin.



HIGHLIGHTS

- Estuary and riparian analyses—flow, advection, and diffusion
- Gas storage, remediation, and sequestration
- Groundwater and geothermal system modeling
- Heat transfer in fluids, solids, and porous media
- Import of the digital elevation model (DEM) file format
- Mechanical and gravity dewatering of porous and fibrous materials
- Petroleum extraction analysis
- Pollutant plume analyses in subsurface, surface, and atmospheric flows
- Poroelastic compaction and subsidence
- Poroelastic stress and failure analysis
- Radionuclide transport through bedrock
- Saturated and unsaturated porous media flow
- Shallow water flows and sediment transport
- Single-phase and two-phase flow through porous media
- Water table analyses and saline intrusion into groundwater
- Well head analyses

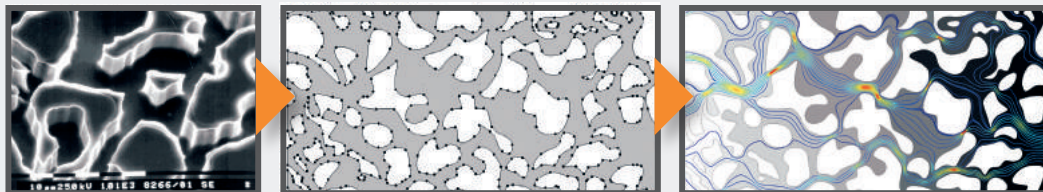
APPLICATION EXAMPLES

- Perforated well
- Two-phase flow column

SUBSURFACE FLOW

Produced by scanning electron microscope images, the geometry can be imported to COMSOL where the velocity and pressure distributions are calculated.

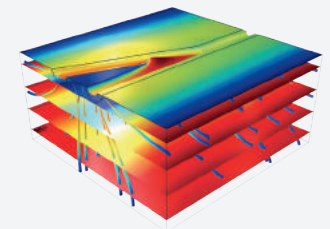
Model courtesy of Arturo Keller, University of California, Santa Barbara.



RESERVOIR MECHANICS

This model analyzes 3D compaction of an oil reservoir brought about by pumping, and the possible failure at the junction of an “open” multilateral well.

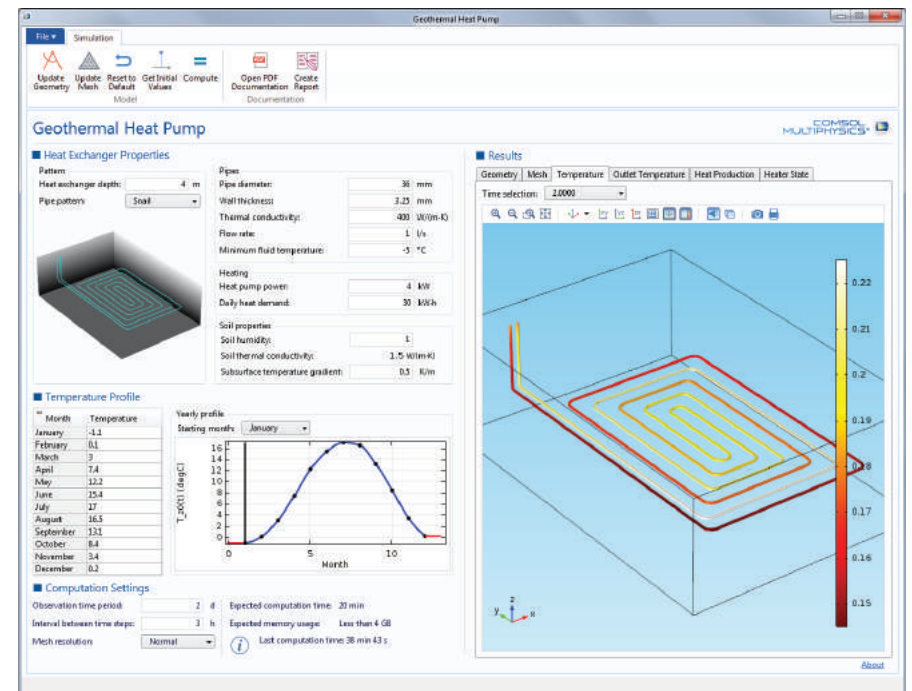
Model idea: A simulation performed by Roberto Suarez-Rivera, Schlumberger, Salt Lake City, Utah.



Pipe Flow Module

The Pipe Flow Module is used for simulations of fluid flow, heat and mass transfer, hydraulic transients, and acoustics in pipe and channel networks. Pipe flow simulations yield the velocity, pressure variation, concentrations, chemical reactions, and temperature along pipes and channels. The module is suitable for pipes and channels that have lengths large enough that the flow can be considered to be fully developed and represented by a 1D approximation. Automatic tools are available to connect pipe flow sections to 3D flow domains.

The module can be used to design and optimize complex cooling systems in turbines; ventilation systems in buildings; geothermal heating systems; heat exchangers; pipe systems in chemical processes; and pipelines in the oil, gas, and mining industries. Preset piping components such as bends, valves, T-junctions, contractions/expansions, and pumps are available. A dedicated user interface is included for transient simulations of the water hammer effect. Multispecies mass transport is available when the Pipe Flow Module is combined with the CFD Module or any of the other modules that feature multispecies transport. Acoustic wave propagation in pipes is available when the Pipe Flow Module is combined with the Acoustics Module.



GEO THERMAL HEAT PUMP

In this application, you can evaluate an underground heat pump that supplies heat to a house. The user can control the properties of the heat exchanger, surrounding soil, and climate conditions in order to analyze designs for different environments.

HIGHLIGHTS

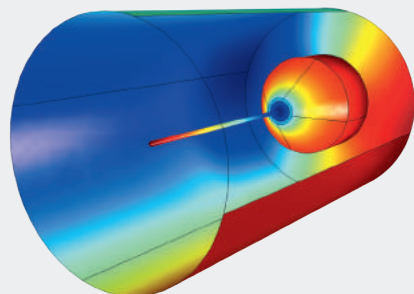
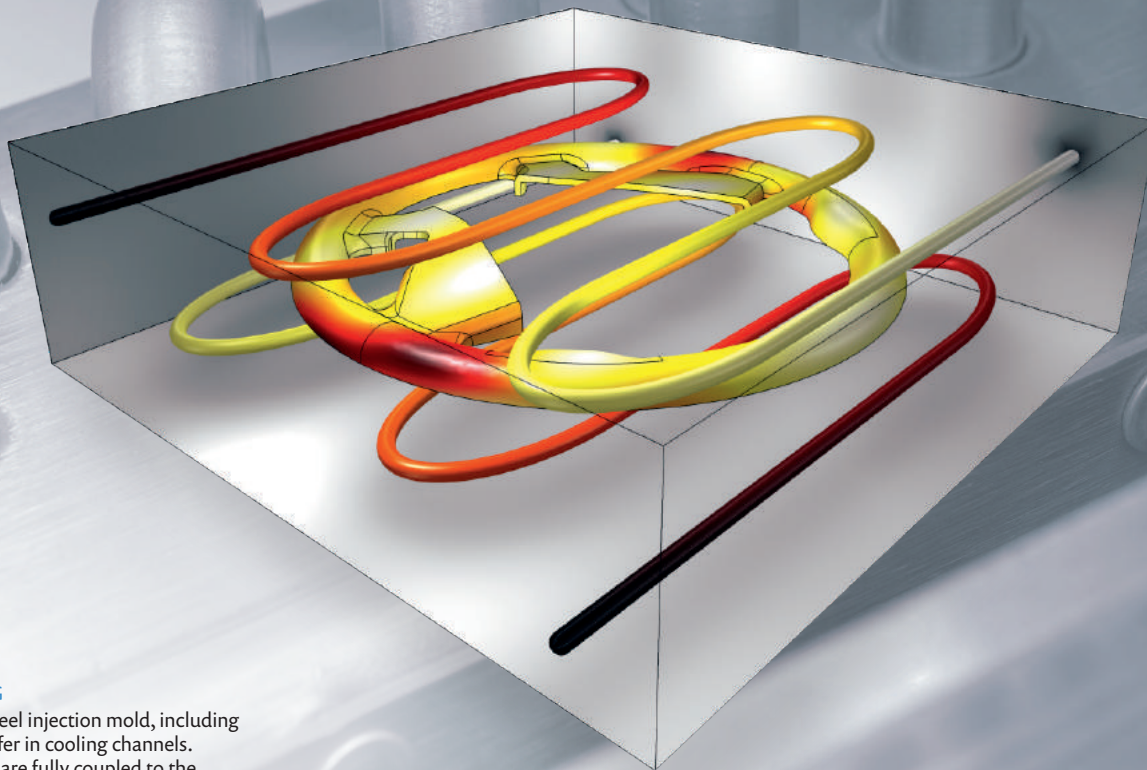
- Cooling systems
- Geothermal systems
- Heat exchangers and cooling flanges
- Heat transfer to and from pipes
- Hydraulics and water hammer
- Lubrication
- Mass transfer and chemical reactions in pipes
- Nonisothermal pipe flow
- Oil refinery pipe systems
- Pressure drop calculation in piping and channels
- Radiant floor heating
- Ventilation and air conditioning systems

APPLICATION EXAMPLES

- Geothermal heat pump
- Organ pipe design

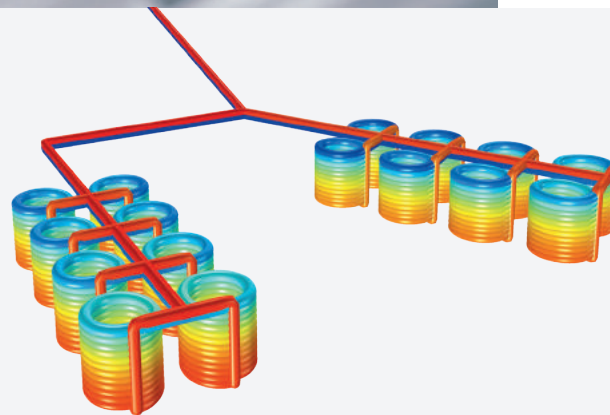
INJECTION MOLDING

Cooling of a steering wheel injection mold, including pipe flow and heat transfer in cooling channels. The 1D cooling channels are fully coupled to the heat transfer simulation of the 3D mold and the polyurethane part on an imported CAD geometry.



MICROPHONES

A probe tube is attached to a microphone for a transient analysis that couples two pressure acoustics domains. Results indicate the pressure at the probe tip as well as at the microphone diaphragm.



GEOTHERMAL HEATING

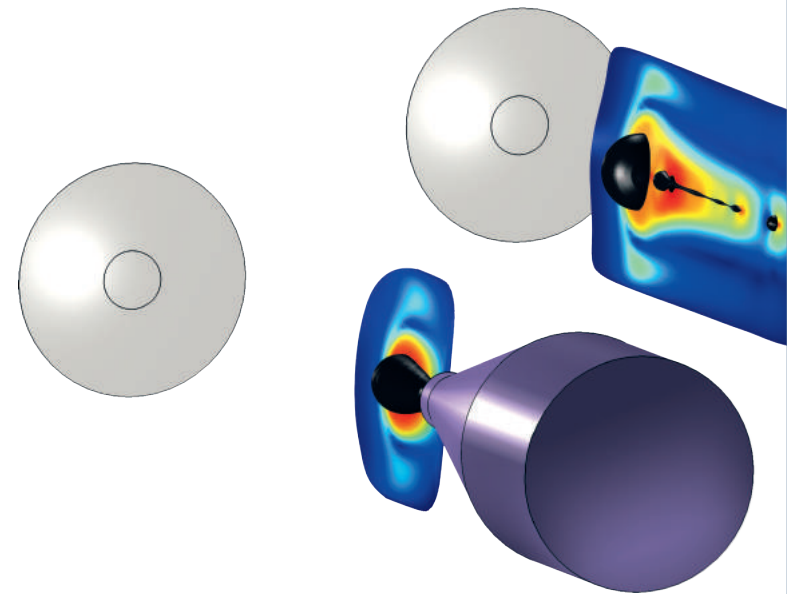
Ponds and lakes can serve as thermal reservoirs in geothermal heating applications. In this example, fluid circulates underwater through polyethylene piping in a closed system.

Microfluidics Module

The Microfluidics Module provides easy-to-use tools for the study of microfluidic devices. Important applications include simulations of lab-on-chip devices, micro total analysis systems, digital microfluidics, electrokinetic and magnetokinetic devices, and inkjets.

In addition to enhanced interfaces for single-phase flow, Microfluidics Module users have dedicated interfaces for two-phase flow using the level set, phase field, and moving mesh methods. These interfaces enable the modeling of surface tension forces, capillary forces, and Marangoni effects. A three-phase flow interface based on the phase field method is also available.

The general-purpose multiphysics features of COMSOL make it easy to set up coupled electrokinetic and magnetodynamic simulations, including electrophoresis, magnetophoresis, dielectrophoresis, electroosmosis, and electrowetting. The chemical diffusion and reactions for dilute species interfaces included in the module enable the simulation of processes occurring in lab-on-chip devices. Gas flows in microstructures can be modeled with the slip flow interface.



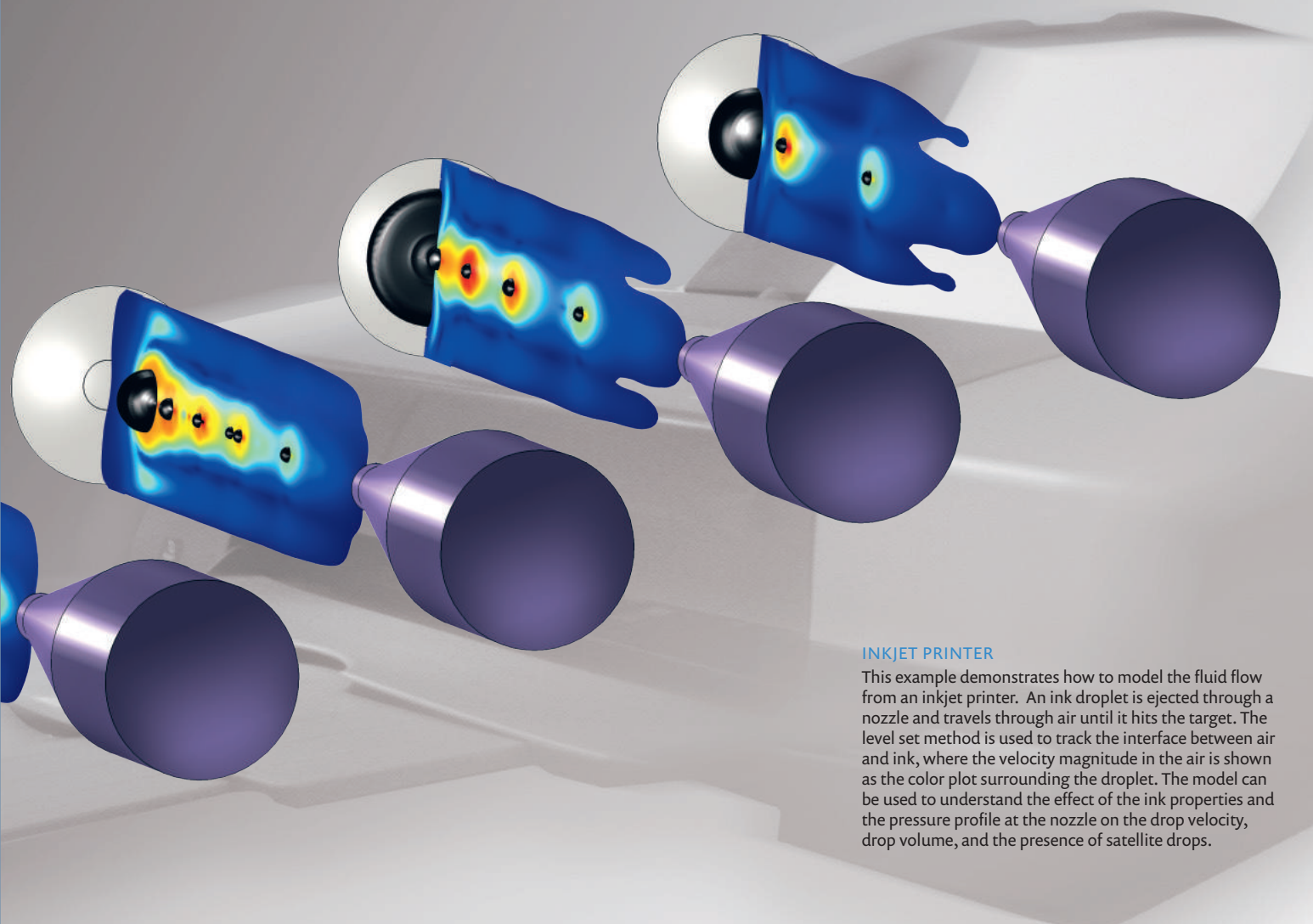
HIGHLIGHTS

- Capillary forces
- Chemical and biochemical sensors
- Dielectrophoresis (DEP)
- DNA chips and lab-on-chips
- Electrocoalescence
- Electrokinetic flow
- Electroosmosis
- Electrophoresis
- Electrowetting
- Fluid-structure interaction (FSI)*
- Inkjets
- Magnetophoresis
- Marangoni effects
- Micro total analysis systems (micro TAS)
- Microreactors, micropumps, and micromixers
- Porous media flow
- Slip flow
- Static mixers
- Surface tension effects
- Two- and three-phase flow

* Together with the Structural Mechanics Module or MEMS Module

APPLICATION EXAMPLES

- Inkjet
- Red blood cell separation

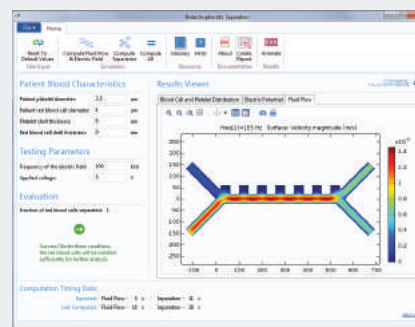


INKJET PRINTER

This example demonstrates how to model the fluid flow from an inkjet printer. An ink droplet is ejected through a nozzle and travels through air until it hits the target. The level set method is used to track the interface between air and ink, where the velocity magnitude in the air is shown as the color plot surrounding the droplet. The model can be used to understand the effect of the ink properties and the pressure profile at the nozzle on the drop velocity, drop volume, and the presence of satellite drops.

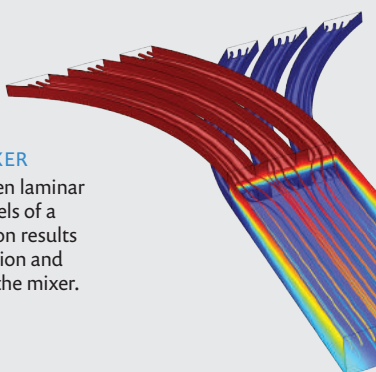
RED BLOOD CELL SEPARATION

This application allows users to calculate the trajectories of platelets and red blood cells in a blood sample as they travel through a filtration device. Results show the electric potential, fluid flow distribution in the filtration system, and the particle locations.



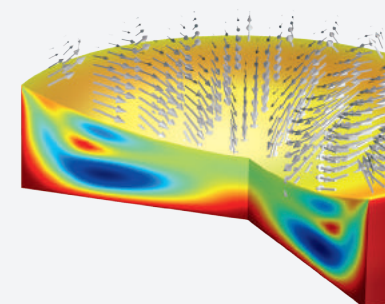
MICROFLUIDICS MIXER

Model of pressure-driven laminar flow in the microchannels of a lamella mixer. Simulation results show solute concentration and velocity streamlines in the mixer.



ELECTROWETTING

Simulation of a variable-focus liquid lens for a miniature camera during the transition between focal lengths. The lens is made from the interface between two fluids.



Molecular Flow Module

The Molecular Flow Module enables the design and simulation of low-pressure gas flow in vacuum systems. Kinetic effects become important as the mean free path of the gas molecules becomes comparable to the length scale of the flow. Under these conditions, conventional fluid dynamics tools cannot produce an accurate model.

The Molecular Flow Module uses a fast angular coefficient method to simulate free molecular flows. Isothermal and nonisothermal flows can be modeled and the heat flux contribution from the gas molecules can also be computed. For transitional flows, an interface based on the discrete velocity method is also included.

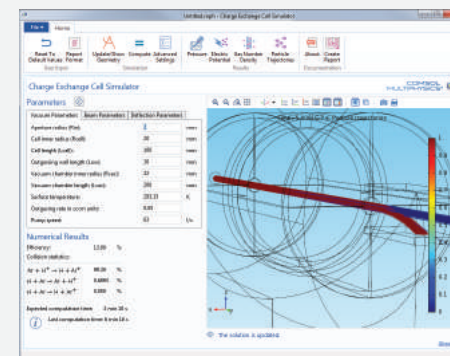
This module is ideal for the simulation of vacuum systems, including those used in semiconductor processing, particle accelerators, and mass spectrometers. Small-channel applications can also be addressed.

Processes such as chamber pump-down and film growth can be designed and optimized through simulation by utilizing the features for defining adsorption/desorption and deposition. Mesh generation for complex CAD geometries is greatly simplified by the possibility of meshing only surfaces. The functionality to recover the number density anywhere within the flow geometry is also available.

FLOW TYPE	KNUDSEN NUMBER
Continuum flow	$Kn < 0.01$
Slip flow	$0.01 < Kn < 0.1$
Transitional flow	$0.1 < Kn < 10$
Free molecular flow	$Kn > 10$

Flow regimes categorized quantitatively via the Knudsen number (Kn) representing the ratio of the molecular mean free path to the flow geometry size.

While the Microfluidics Module and the CFD Module are used to model slip and continuum flows, the Molecular Flow Module is designed for accurately simulating flows in the free molecular flow and transitional flow regimes.



CHARGE EXCHANGE CELL

This application computes the high-energy collisions of ionized hydrogen atoms as they pass through a charge exchange cell filled with argon gas. The user can control parameters such as geometry and operating conditions to evaluate pressure in the device, particle trajectories, electric potential distribution, efficiency, and collision statistics.

HIGHLIGHTS

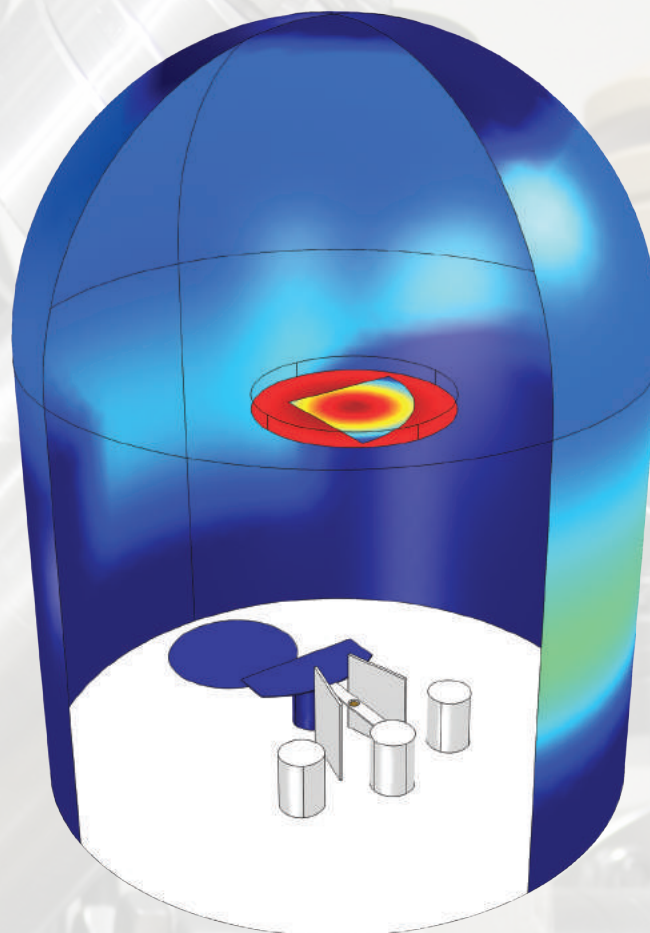
- Gas cells
- Highly rarefied flows
- Mass spectrometers
- Materials processing equipment
- Nanopore flow
- Particle accelerators
- Semiconductor processing equipment
- Vacuum systems

APPLICATION EXAMPLES

- Charge exchange cell simulator
- Ion implanter evaluator

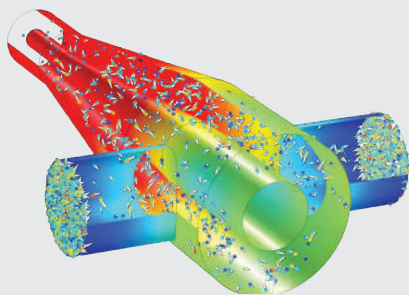
FILM DEPOSITION

Thermally evaporated gold film thickness on both a sample and the interior surfaces of the evaporator chamber is shown. The thickness is computed as a function of time with a transient simulation.



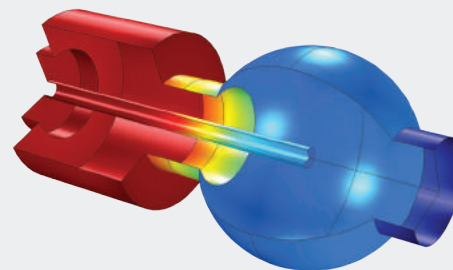
RF COUPLER

This model computes the transmission probability through an RF coupler.



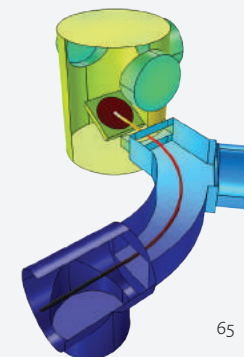
LOAD LOCK VACUUM SYSTEM

Time-dependent simulation of adsorption and desorption of water in a vacuum system at low pressures. The water is introduced into the system when a gate valve to a load lock is opened, and the subsequent migration and pumping of the water are modeled.



ION-IMPLANT VACUUM SYSTEM

In ion implantation, outgassing molecules interact with the ion beam to produce undesirable species. The average number density of outgassing molecules along the beam path is simulated and used as a figure of merit to evaluate the design.

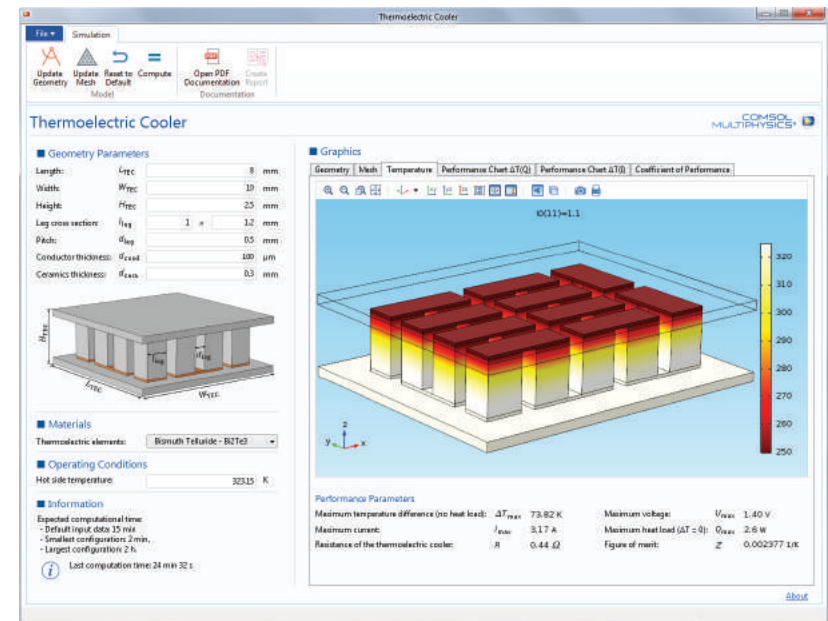


Heat Transfer Module

Almost every manufacturing process and product design must consider thermal effects. The Heat Transfer Module provides a combination of capabilities to model heat transfer via conduction, convection, and radiation, as well as the ability to couple these to other physics.

The Heat Transfer Module has physics interfaces that are specifically written for the user interested in free and forced convection, thermal conduction, porous media, phase change modeling, radiative heat transfer through both transparent and semitransparent participating media, as well as couplings between all of these effects. With dedicated features for describing convective and radiative effects, contact resistance, and heat transfer in thin structures (shells, films, fractures, and rods), you can simulate anything from a simple “back of the envelope” model to a full model with all effects explicitly described. Specialized formulations are also included for users who are interested in the heating of living tissue.

Since all material properties are functions of temperature, you can conceivably couple a thermal model to any other physical model. Moreover, COMSOL allows you to include heat generation from any other physics into a thermal model.



THERMOELECTRIC COOLER

The design of a single-stage thermoelectric cooler can be explored using this custom application. The user may test different geometries, thermocouple configurations, and material selection in order to determine the ideal cooler option for a specific application or an optimized design for best performance.

HIGHLIGHTS

- Anisotropic heat transfer in composite structures
- Bioheat treatment and tissue necrosis
- Casting and thermal processing
- Conjugate heat transfer
- Disc brakes
- Drying and freeze drying
- Food processing, cooking, and sterilization
- Furnace and burner design
- Heat and moisture transport
- Heat exchangers and cooling flanges
- Heat transfer in porous media
- Laser and friction stir welding
- Material heat treatment
- Power electronics and electronic cooling
- Radiative heat transfer
- Resistive, induction, and laser heating
- Thermal contact and friction
- Thermal performance of constructions
- Thermoelectric effect (Seebeck, Peltier, Thomson)

APPLICATION EXAMPLES

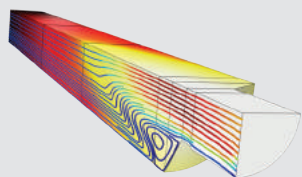
- Concentric tube heat exchanger
- Equivalent properties of periodic microstructures
- Finned pipe
- Flash method
- Forced air cooling with heat sink
- Inline induction heater
- Parasol and solar irradiation
- Thermoelectric cooler

TEMPERATURE SENSOR

Top: X-ray imaging of the device.
Right: Photograph and model plot of the temperature distribution in a temperature sensor and on the sensing element inside the cap.

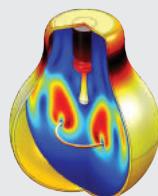


Model courtesy of Martin Sás, Continental Corporation, Frenstat, Czech Republic.



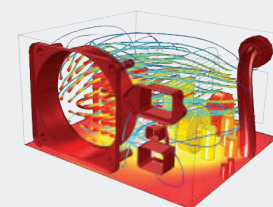
CONTINUOUS CASTING

Simulation of heat transport and the fluid-to-solid flow in a continuous casting process, including the phase transfer from melt to solid.



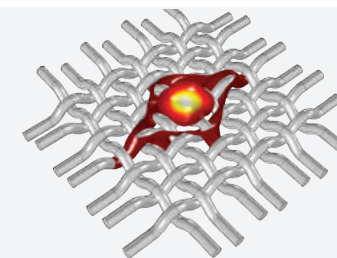
LIGHT BULB

Temperature distribution as influenced by the radiation and the natural convection created within a light bulb.



POWER SUPPLY COOLING

Simulation of the thermal behavior of an enclosed computer power supply unit. The forced convection cooling is driven by a fan extracting air from the device.



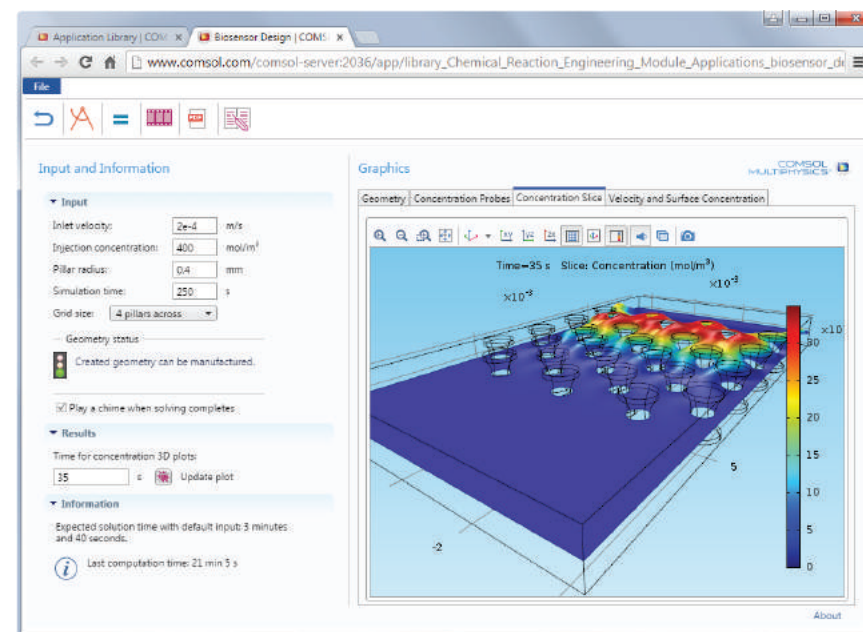
CARBON FIBERS

Heat transfer analysis of thermally anisotropic carbon fibers.

Chemical Reaction Engineering Module

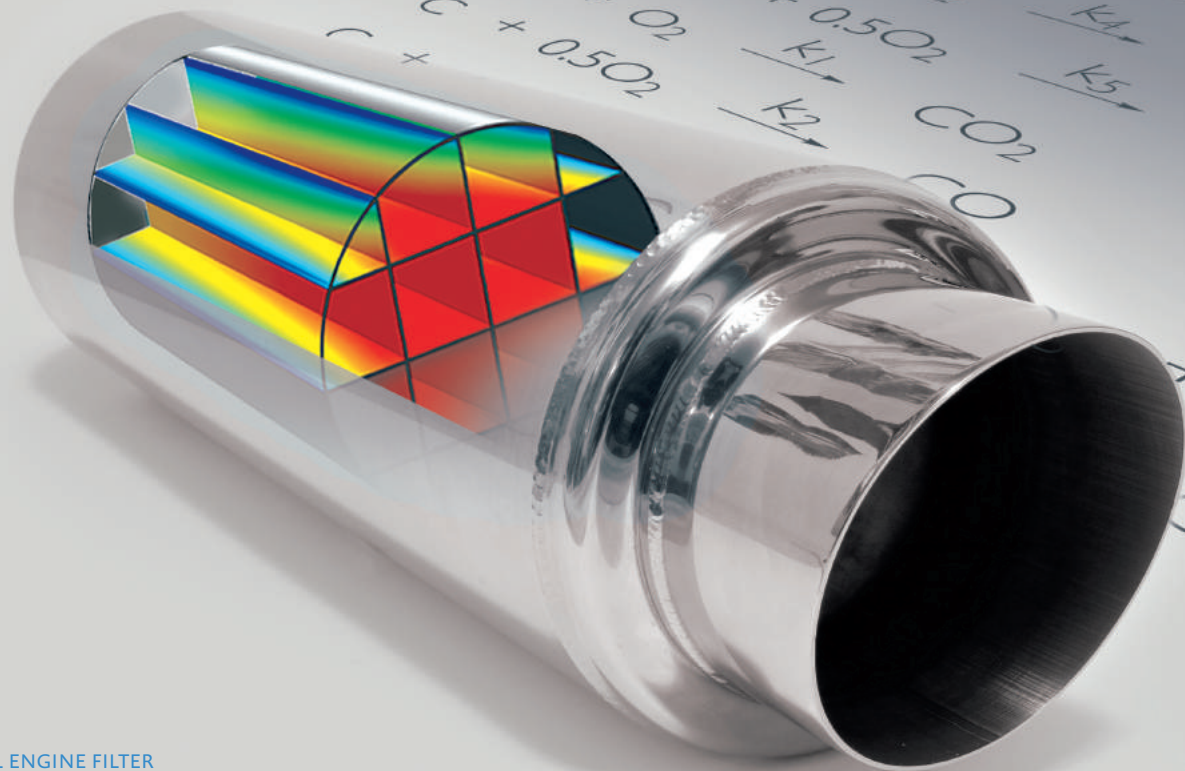
The Chemical Reaction Engineering Module is intended for the modeling of reactors, filtration and separation units, and other equipment common in the chemical and similar industries. It is specifically designed to easily couple fluid flow and mass and energy transport to chemical reaction kinetics. Initially, the Chemical Reaction Engineering Module uses reaction formulas to create models of reacting systems. It can then solve the material and energy balances for such systems, including the reaction kinetics, where the composition and temperature vary with time, space, or both.

The Chemical Reaction Engineering Module combines seamlessly with the power of COMSOL Multiphysics for coupled and equation-based modeling. This allows for the inclusion of arbitrary expressions, functions, and source terms in the material property, transport, and reaction kinetics equations. You can also access a variety of thermodynamic and physical property data through the CHEMKIN® file import feature and Thermodynamics interface available in the Chemical Reaction Engineering Module.



BIOSENSOR

This application allows the user to change parameters such as grid spacing, inlet velocity, and pillar diameter in a biosensor in order to investigate how different designs affect the detection results.



DIESEL ENGINE FILTER

A filter system for a diesel engine includes a soot layer that builds up at the filter walls and is subsequently oxidized by both catalytic and noncatalytic reactions in the reactor volume and on a moving surface. All reaction rates are temperature dependent and transport limited. The colors represent the conversion of nitric oxide along the catalytic unit.

HIGHLIGHTS

- Adsorption, absorption, and deposition
- Batch, plug-flow/tubular, and tank reactors
- Biochemistry and food science
- Catalytic converters and reformers
- Chromatography and electrophoresis
- Cyclones, separators, scrubbers, and leaching units
- Exhaust after-treatment and emission control
- Fermentation and crystallization devices
- Filtration and sedimentation
- Laminar and turbulent flow patterns
- Microfluidics and lab-on-chip devices
- Multicomponent and membrane transport
- Pharmaceutical synthesis
- Plastics and polymer manufacturing
- Preburners and internal combustion engines
- Packed bed reactors
- Reaction kinetics modeling
- Reactor sizing and optimization
- Semiconductor processing and CVD
- Surface kinetics
- Species transfer through diffusion, convection, and ionic migration

APPLICATION EXAMPLES

- Biosensor design
- Liquid chromatography
- Membrane dialysis

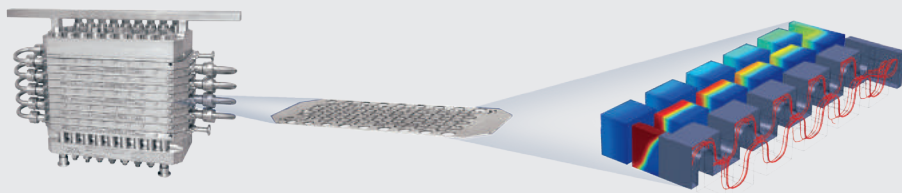


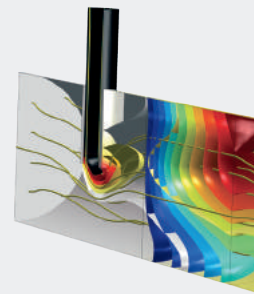
PLATE REACTOR

Flow streamlines and concentration in a plate reactor. One species is included at the beginning of the reactor, while another is introduced halfway through it.

The photographs of the Plate Reactor are Copyright © Alfa Laval AB, Tumba, Sweden.

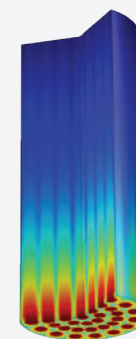
POROUS REACTOR

Two species enter a reactor from different inlets and then undergo a reaction in a porous part of the reactor. The streamlines show flow and the isosurfaces show the concentration for one of the reactants and the product species.



A MULTISCALE 3D PACKED BED REACTOR

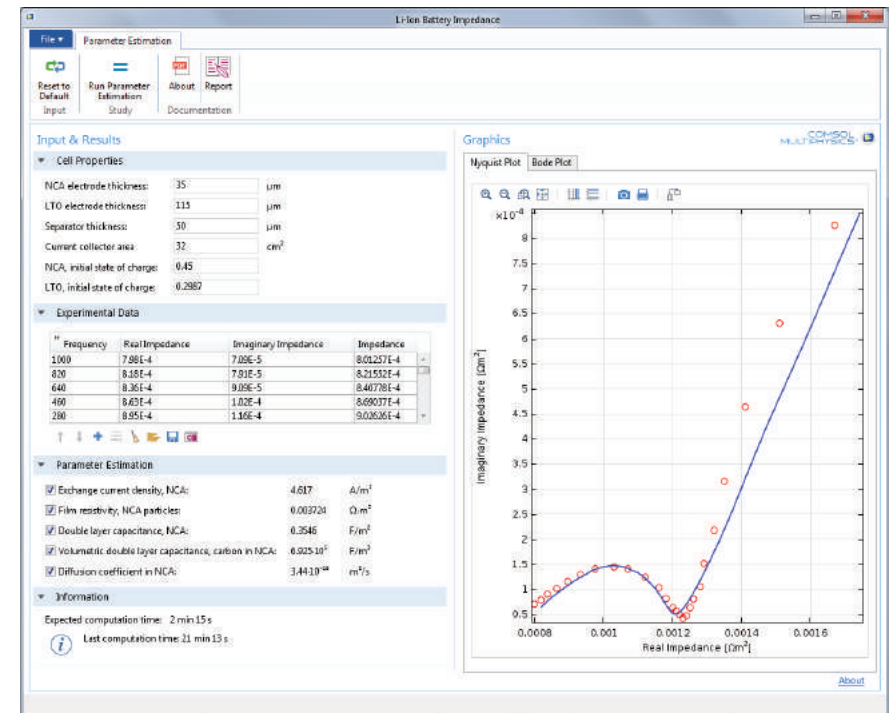
This model efficiently solves a combined macroscale and microscale (multiscale) problem of a heterogeneous system with bimodal pore distribution. Simulation results include both the concentration distribution in the fluid surrounding and within the pellets.



Batteries & Fuel Cells Module

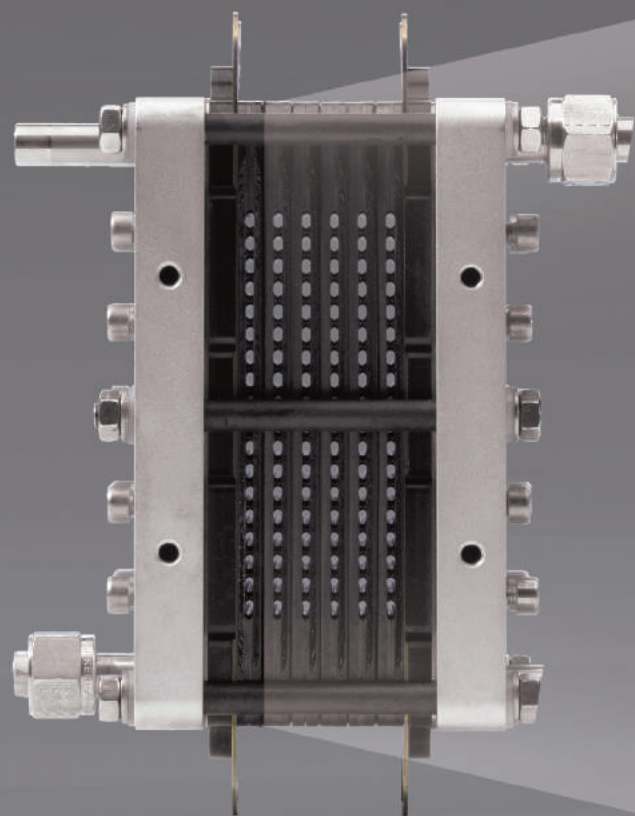
The Batteries & Fuel Cells Module provides a full set of easy-to-use tools for the simulation of fundamental processes in the electrodes and electrolytes of lithium-ion batteries, nickel metal-hydride batteries, flow batteries, solid oxide fuel cells, and proton exchange membrane fuel cells. With this software, you can quickly and accurately investigate the performance impact of different materials, geometric configurations, and operating conditions.

The module features tailored interfaces to study primary, secondary, and tertiary current distributions in electrochemical cells. Electrode reactions, which are fully coupled to transport phenomena in the cell, provide full descriptions of the electrode kinetics, including activation and concentration overpotential. The cell can contain solid or porous electrodes with dilute or concentrated electrolytes. Furthermore, couplings of the electrochemical reactions and mass transfer in batteries and fuel cells to other phenomena, such as heat transfer and convective flow, can be performed through the powerful capabilities of COMSOL Multiphysics.



LI-ION BATTERY IMPEDANCE

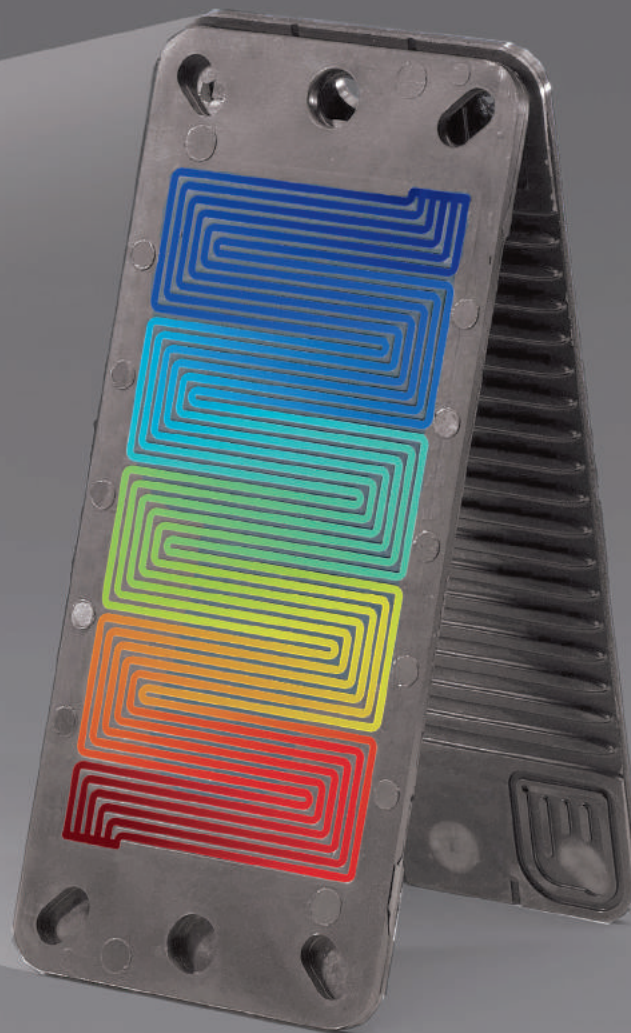
App users can estimate the characteristics of a lithium-ion battery. Results such as the exchange current density and properties of the resistive layer, determined through electrochemical impedance spectroscopy (EIS), are computed.



PEMFC

The hydrogen concentration through the channels of a bipolar plate considering the effects of the edges is shown.

Model courtesy of Christian Siegel, Center for Fuel Cell Technology (ZBT GmbH), Duisburg, Germany.



HIGHLIGHTS

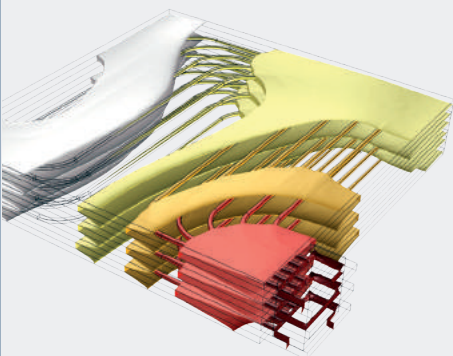
- Fuel Cells
 - Alkaline
 - Direct methanol
 - Molten carbonate
 - Proton exchange membrane
 - Solid oxide
- Batteries
 - Lead-acid
 - Lithium-air
 - Lithium-ion
 - Nickel-hydride
 - Solid-state lithium
- Flow Batteries
 - Soluble lead
 - Vanadium

CAPABILITIES

- Aging mechanisms due to structural, thermal, and chemical effects
- Battery load cycling, internal resistance, and rate capability simulations
- Concentrated and supporting electrolytes
- Detailed electrochemical analysis of cells
- Predefined impedance spectroscopy study for electrode kinetics and aging investigations
- Primary, secondary, and tertiary current distributions
- Short circuits in batteries
- Thermal management
- Thermal runaway in batteries

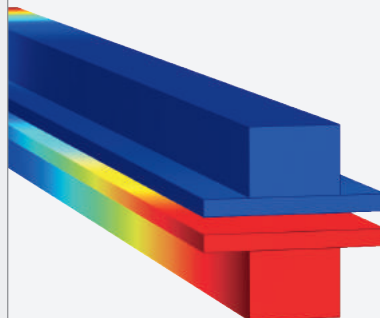
APPLICATION EXAMPLES

- Cyclic voltammetry
- Electrochemical impedance spectroscopy
- Li-ion battery impedance



BATTERY PACK

The picture shows the temperature field in the cooling channels and the batteries in a battery pack for automotive applications. The model includes detailed electrochemistry of the batteries coupled to a thermal analysis for the batteries and components in the battery pack, as well as the fluid flow in the cooling channels.



SOFC

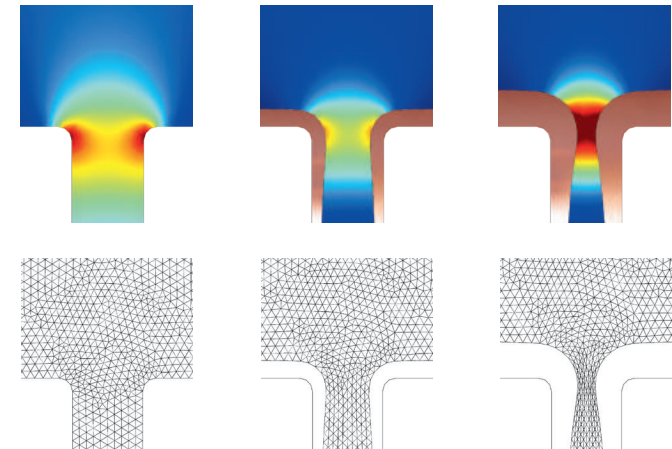
Reactant concentrations in a solid oxide fuel cell. The model includes full coupling between porous electrode reactions and the mass balances at the anode and cathode, the momentum balances in the gas channels, the gas flow in the porous electrodes, the balance of the ionic current carried by the oxide ion, and a balance of electronic current.

Electrodeposition Module

Modeling and simulation are cost-effective methods for understanding, optimizing, and controlling electrodeposition processes. A typical simulation yields the current distribution at the surface of the electrodes and the thickness and composition of the deposited layer. They are used to study important parameters such as cell geometry, electrolyte composition, electrode kinetics, operating voltages and currents, as well as temperature effects.

The Electrodeposition Module brings the power of COMSOL Multiphysics to simulations of electrodeposition processes. Easy-to-use physics interfaces are provided for primary, secondary, and tertiary current distribution models, while very accurate geometric representations of deposited layer buildup are included as model parameters. The shape evolution of the electrode can also be simulated with deforming geometries.

The Electrodeposition Module is applicable to a variety of diverse applications, including metal deposition for electronics and electrical parts, corrosion and wear protection, decorative electroplating, electroforming of parts with thin and complex structures, and metal electrowinning.



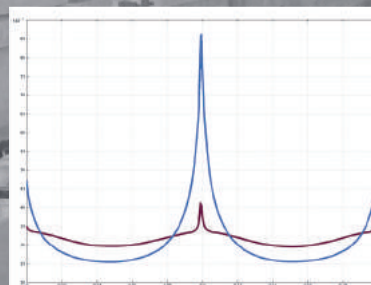
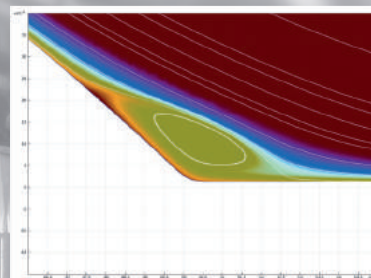
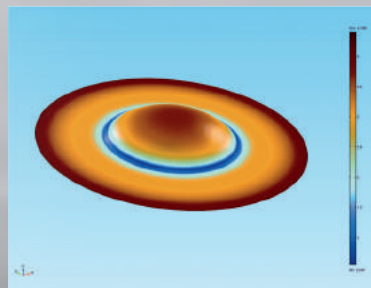
ELECTRODEPOSITION

This model demonstrates the effect of a deforming geometry in the application of copper electrodeposition on circuit boards. The model is time-dependent and results clearly show that the mouth of the trench narrows due to the nonuniform deposition of the copper.

INDUSTRIAL ELECTROPLATING

Model of a gold electroplated contact for the automotive industry, which would sit on the tape moving through the electroplating unit. The 3D figure shows the extent of gold deposition on the contact, which is a maximum at the top of the contact, and a minimum where the contact is bent. Zooming in on the region of the bend, the 2D figure indicates flow recirculation where deposition is least, an area where secondary reactions can take place. The graph shows a different application: the distribution of tin thickness across a copper strip when the process uses screens (red line) and when it does not (blue line).

Model and picture courtesy of Philippe Gendre, PEM, Siaugues, France.

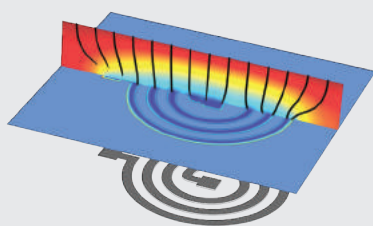


HIGHLIGHTS

- Anodizing
- Chrome plating
- Chroming
- Electrochemical machining
- Electrocoating
- Electrocoloring
- Electrodeposition for mining applications
- Electrodeposition for PCB manufacturing
- Electroforming
- Electroplating
- Functional electroplating
- Wear resistance coatings

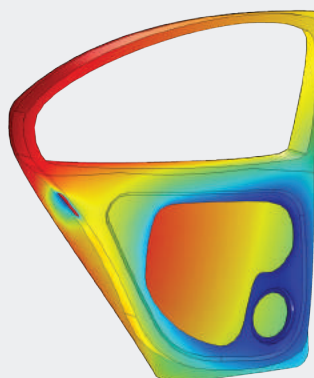
APPLICATION EXAMPLES

- Cyclic voltammetry
- Electrochemical impedance spectroscopy



INDUCTOR COIL

Simulation of the electrodeposition of an inductor coil using a thin photoresist mask.

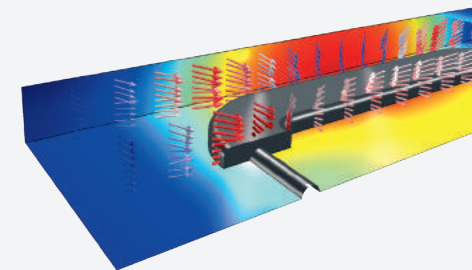


CAR DOOR

Simulation of deposited paint thickness on an electrocoated car door.

DECORATIVE ELECTROPLATING

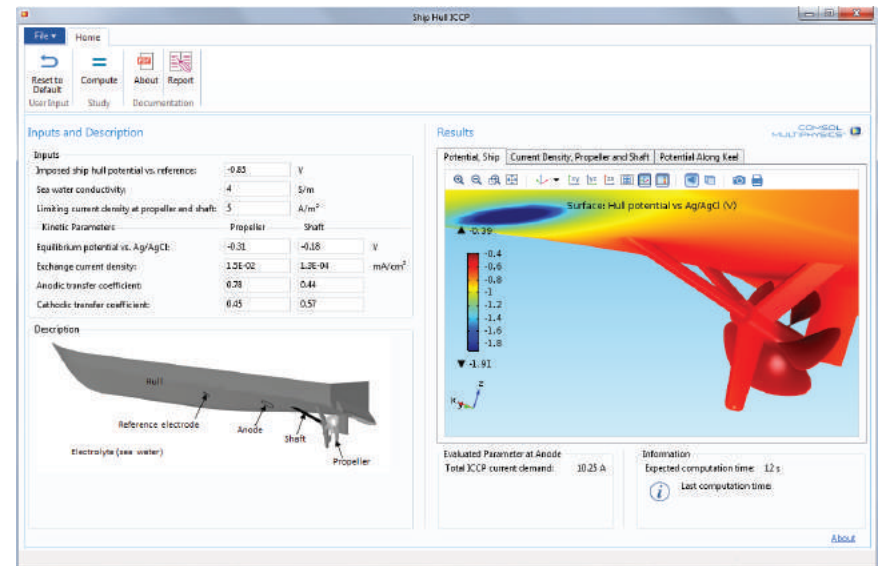
In this example, the anode is planar and dissolves, while the cathode is a furniture fitting that is to be decorated by the dissolved metal. The model assumes secondary current distribution with full Butler-Volmer kinetics for both the anode and the cathode.



Corrosion Module

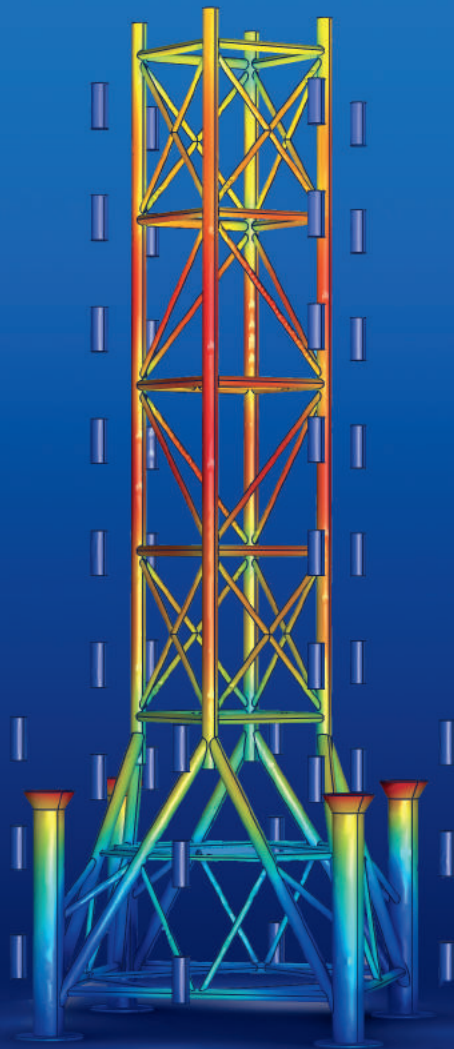
The Corrosion Module empowers engineers to simulate the electrochemistry of corrosion and corrosion protection of metal structures with finite element or boundary element as a method of choice. Models in 1D, 2D, and 3D are set up to include the relevant corrosion and other reactions within the electrolyte and at the metal surface interface using a series of predefined physics interfaces. These models are solved while considering the transport of ions and neutral species in the solution, the current conduction in the metal structure, and other phenomena such as fluid flow and heat transfer.

Simulations using the Corrosion Module can be used to understand and avoid corrosion situations as well as to design and optimize corrosion protection. This can be done at the microscopic scales, such as in crevice and pitting corrosion, where the localized concentrations can be significant parameters in the charge-transfer reactions, or at larger scales, where the placement of sacrificial anodes around a structure is the goal. In some cases, linking the simulations at both of these scales is necessary and also achievable with the Corrosion Module.



SHIP HULL ICCP

This specialized application computes the impressed current cathodic protection demand of a ship hull with a coated propeller. Results show the electrolyte potential distribution.



CORROSION PROTECTION

An oil rig structure immersed in seawater is protected by 40 sacrificial aluminum anodes. Before deploying the anodes, simulations are used to optimize their positions for the best possible corrosion protection. The visualization shows the electrolyte potential on the surface of the structure.

HIGHLIGHTS

- Anodic protection
- Atmospheric corrosion
- Butler-Volmer equation
- Cathodic protection
- Corrosion
- Corrosion protection
- Crevice corrosion
- Galvanic corrosion
- Impressed Current Cathodic Protection (ICCP)
- Nernst-Planck equation
- Passivation
- Pitting corrosion
- Primary current distribution
- Secondary current distribution
- Tafel equation
- Tertiary current distribution

CAPABILITIES

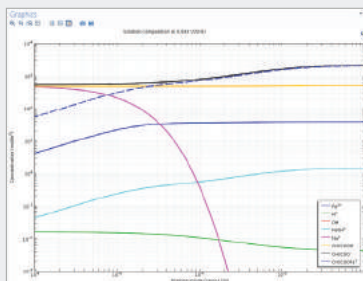
- Boundary Element Method (BEM)
- Finite Element Method (FEM)

APPLICATION EXAMPLES

- Cyclic voltammetry
- Electrochemical impedance spectroscopy
- Ship hull ICCP

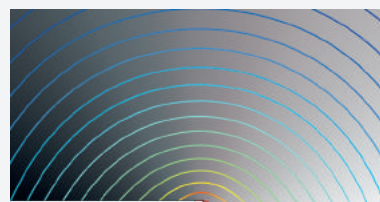
CREVICE CORROSION

Concentration distribution in the crevice for iron in an acetic acid/sodium acetate solution at 0.844 V(SHE).

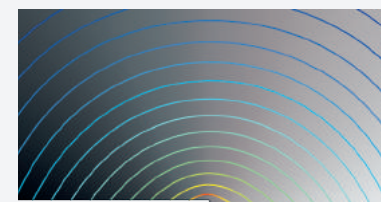


GALVANIC CORROSION

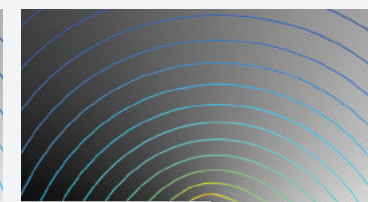
The corrosion of a magnesium alloy (AE44) connected to mild steel in a saltwater solution. The electrode material removal is also modeled, as it is an important variable to be considered when performing such simulations.



t = 0 hours



t = 28 hours



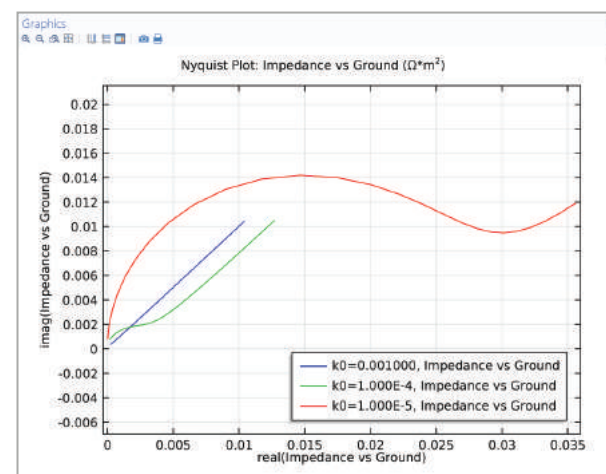
t = 72 hours

Electrochemistry Module

The Electrochemistry Module expands the possibilities in designing, understanding, and optimizing electrochemical systems through accurate simulation. This module offers significant benefits to researchers in the lab and industrial chemical engineers. Capabilities such as modeling current density distributions, electrochemical reactions, and mass transport enable efficient simulation for applications including electrolysis, electrochemical sensors, electrodialysis, electroanalysis, and electrobiochemistry.

Dedicated interfaces in the Electrochemistry Module enable the definition of voltammetry, amperometry, potentiometry, electrochemical impedance, and coulometry studies. Exchange current densities and activation overpotentials can then be determined from the combined experiment and simulation results.

The Electrochemistry Module permits modeling systems assuming primary, secondary, or tertiary current distributions, as implemented via the Nernst-Planck and Butler-Volmer equations. The Electrochemistry Module covers a wide range of applications. This is accomplished through interfaces for electric currents, flow in free and porous media, heat transfer, surface and homogeneous chemical reactions, and material transport in dilute and porous media.



IMPEDANCE SPECTROSCOPY

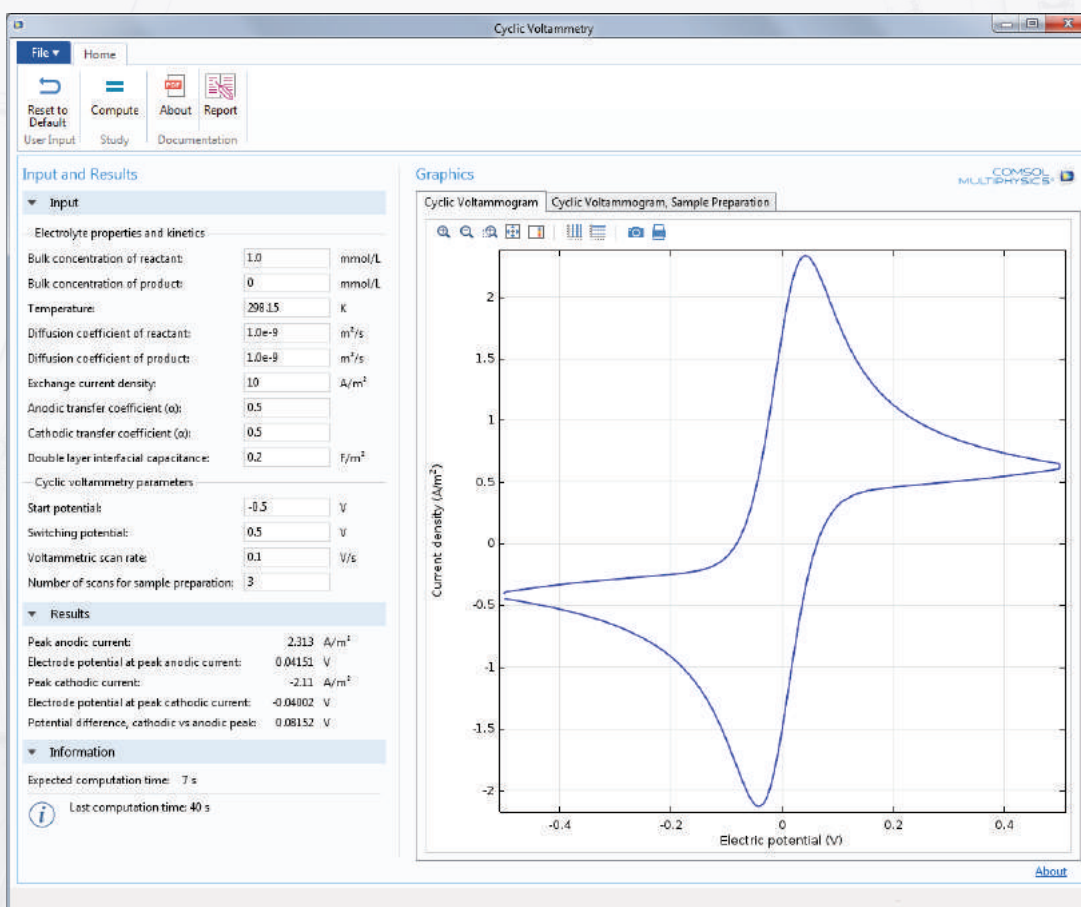
Electrochemical impedance spectroscopy (EIS) is a common technique for electrochemical analysis. During the simulation, the frequency of the alternating part of the electrode potential is varied, while the electrode current density is recorded. Results provide information about the electrode-electrolyte interface and the mass transport properties of the electrolyte.

HIGHLIGHTS

- Bioelectrochemistry
- Chlor-alkali electrolysis
- Control of electrochemical reactions in biomedical implants
- Desalination of seawater
- Electroanalysis
- Electrochemical sensors
- Electrodialysis
- Electrolysis
- Electrolyte waste treatment
- Gas sensors
- Glucose sensors
- Hydrogen and oxygen production
- pH control of liquid foods
- Ultra-pure water production

APPLICATION EXAMPLES

- Cyclic voltammetry
- Electrochemical impedance spectroscopy

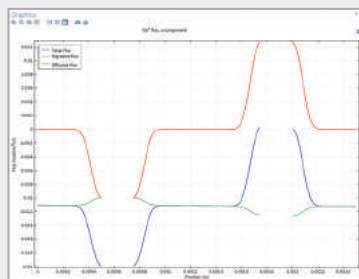


CYCLIC VOLTAMMETRY

This application simulates the cyclic voltammetry technique used to determine information about chemical processes.

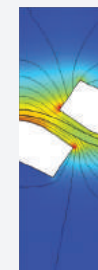
ELECTRODIALYSIS

Simulation of the removal of sodium chloride from a water solution (diluate) into another stream (concentrate). Such an application can be found in desalination of process streams, food and juice products, and biomedical engineering.



CHLOR-ALKALI MEMBRANE CELL

Advances in membrane cell design by increased internal convection, decreased ohmic losses, and better membranes have allowed for large increases in current density with small increases in cell voltage. One of the important parameters in the design of modern membrane cells is the current-density distribution on the electrode surfaces. This model describes the current-density distribution in a realistic structure for the anodes and cathodes in a chlor-alkali membrane cell.

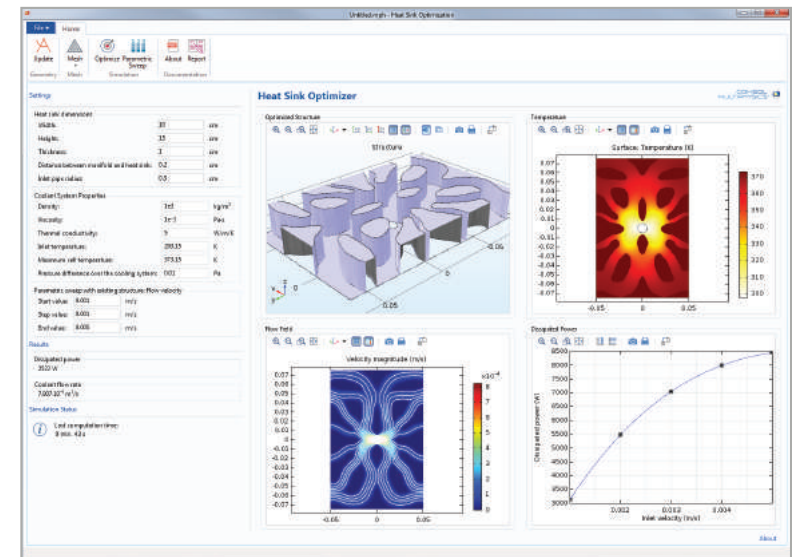


Optimization Module

No matter the engineering discipline, once you have a model of a product or process, you want to improve upon it. The Optimization Module can be used throughout the COMSOL Multiphysics product family; it is a general interface for computing optimal solutions to engineering problems. Any model input, be it geometric dimensions, part shapes, material properties, or material distribution, can be treated as a design variable, and any model output can be an objective function.

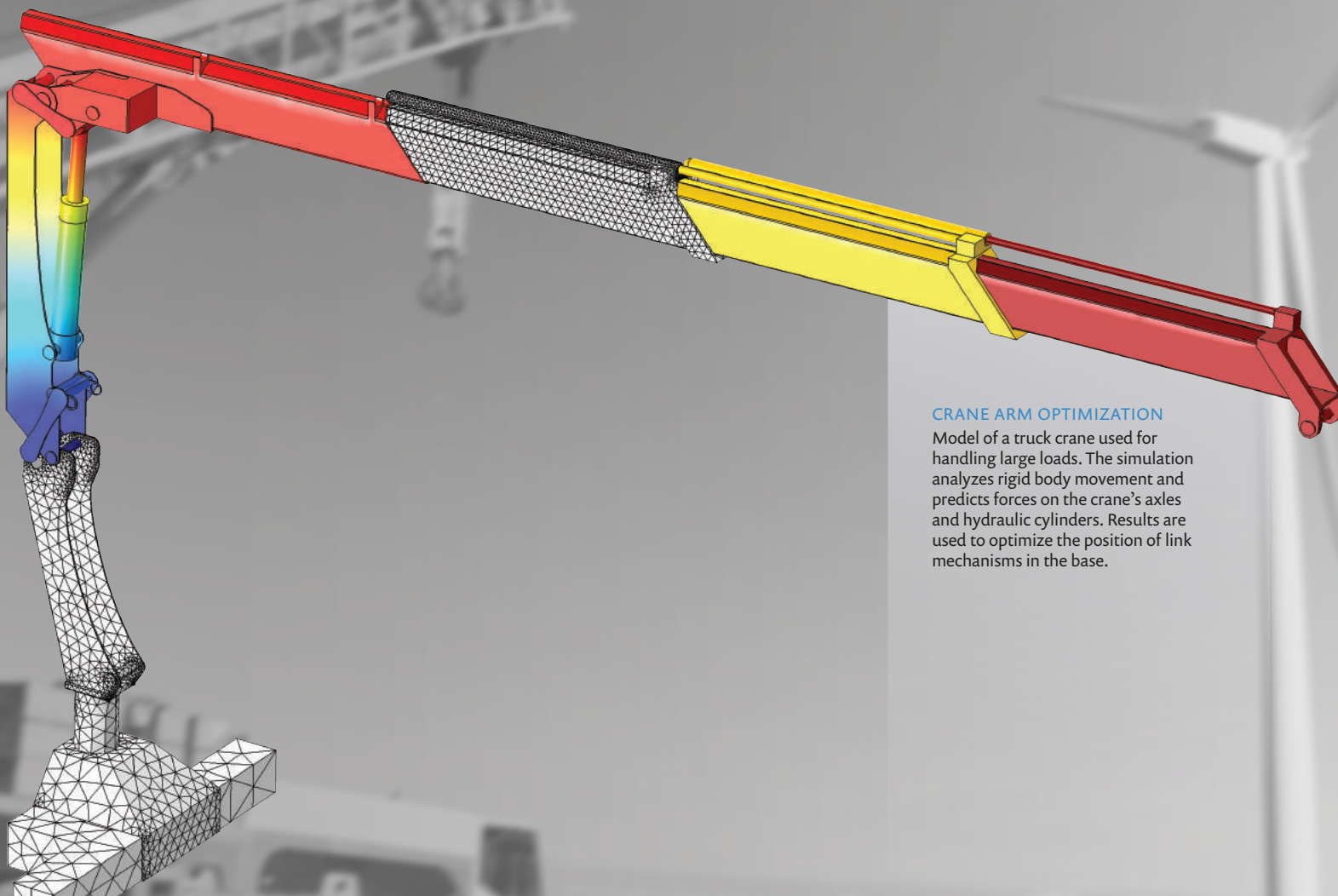
Both derivative-based and derivative-free methods are available in order to cover a wide range of optimization problems. Derivative-based methods include SNOPT, MMA, and Levenberg-Marquardt, while derivative-free methods include coordinate search, Nelder-Mead, COBYLA, and BOBYQA.

The Optimization Module can be used to solve shape, size, and topology optimization problems; inverse problems such as parameter estimation; and time-dependent and stationary optimization problems. With a very general interface, the capabilities of the Optimization Module can be used in conjunction with any combination of other COMSOL products.



HEAT SINK OPTIMIZER

The channel structure in a planar heat sink, equipped with a centrally positioned inlet pipe for coolant, is optimized with this specialized application. The channel topology is optimized for a given pressure drop and the maximum allowed temperature for the device being cooled.

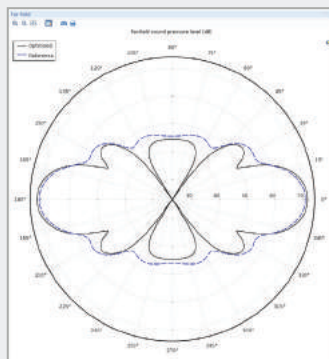
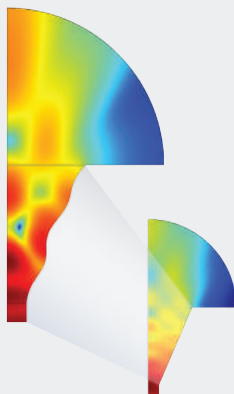


CRANE ARM OPTIMIZATION

Model of a truck crane used for handling large loads. The simulation analyzes rigid body movement and predicts forces on the crane's axles and hydraulic cylinders. Results are used to optimize the position of link mechanisms in the base.

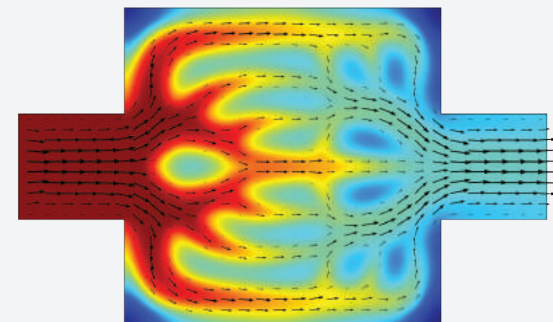
SHAPE OPTIMIZATION OF A HORN

A horn has the initial shape of an axisymmetric cone with a straight boundary. This design is optimized with respect to the far-field sound pressure level. Shown is the new shape of the internal boundary and the optimized far-field sound pressure level in comparison to the initial reference design.



REACTOR OPTIMIZATION

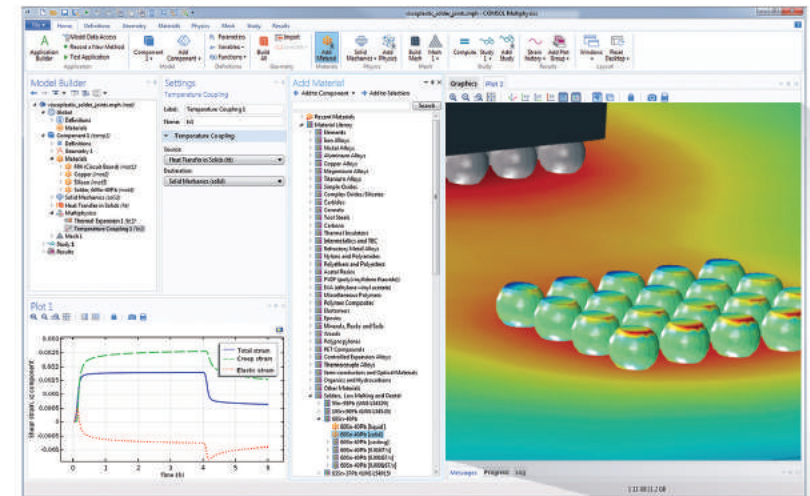
In this model, the reaction rate for a given total pressure difference across the bed is maximized by finding an optimal catalyst distribution.



Material Library

COMSOL Multiphysics gives you complete control over definitions and use of your material properties through the Model Builder and Material Browser. The Material Browser allows you to manage all of your model's materials in one place and can be complemented by the Material Library.

The Material Library contains data for more than 2700 materials, including the elements, minerals, metal alloys, thermal insulators, semiconductors, and piezoelectric materials. Each material is represented by referenced property functions for as many as 31 key properties, dependent on some variable, typically temperature. You can plot and inspect these function definitions, as well as change and add to them. They can then be used in any coupling to other physics simulations that also depend on the property function variable in your multiphysics modeling.



SOLDER JOINTS

This example studies viscoplastic creep in solder joints under thermal loading used to mount two chips to a cellular phone circuit board. The circuit board consists of two layers: a thin layer of copper and a thicker layer of FR4 material, while the chips are made of silicon. The Material Library is utilized to provide temperature-dependent material properties for the model. The plot shows that plastic flow clearly appears after about 40 seconds of loading. The model results show a close-up of the temperature distribution in the circuit board and the plastic strain in the solder joints.

AVAILABLE MATERIALS

- Acetal, PVDF, and EVA
- Al and Cu alloys
- Carbides, cermets, and tool steels
- Carbons and thermal insulation
- Cast irons and mold materials
- Controlled expansion and thermocouple alloys
- Elastomers and epoxies
- Elements
- Fe and Ni alloys
- Glasses, metallic glasses, nitrides, and beryllides
- Intermetallics, TBC, and refractory metals
- Metal and ceramic matrix composites
- Mg and Ti alloys
- Minerals, rock, soil, and wood
- Miscellaneous polymers and polymer composites
- Oxides
- Polyamides and polyesters
- Polypropylenes and PET
- Resistance and magnetic alloys
- Salts, fuel cells, batteries, and electro-ceramics
- Semiconductors, optical, and related materials
- Silicides and borides
- Solders, dental, and Co alloys

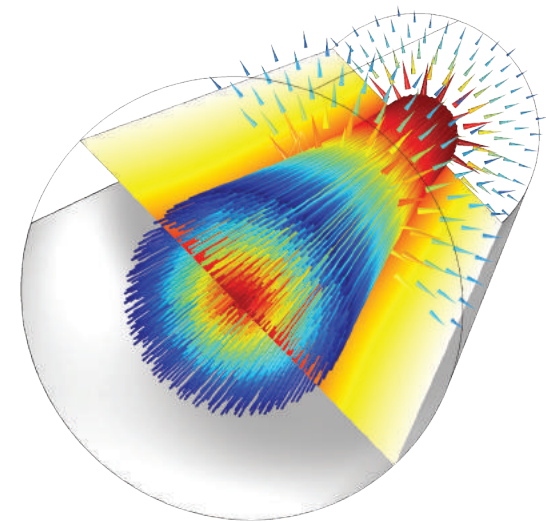
Particle Tracing Module

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-particle and particle-field interactions. Any application-specific module combines seamlessly with the Particle Tracing Module and gives you access to additional modeling tools and fields to drive the particle motion.

Built-in equations for particle mass and temperature can be added to allow particles to exchange mass or energy with their surroundings. Other quantities, such as particle spin, can be defined using auxiliary variables. A variety of wall conditions can be used to make particles stop moving, disappear, reflect specularly or diffusely, or change trajectory according to a user-defined expression when they hit a wall. Particle-wall interactions can be conditional, specified using a logical expression or a sticking probability. Secondary emission of particles is included as an option, where the emission may depend on incident velocity and energy. Particles can be massless or have mass, where the movement is governed by Newtonian, Lagrangian, or Hamiltonian formulations from classical mechanics. Particles can be released within each mesh element with a uniform distribution via a grid or using a user-defined expression. Low-level access to the mathematical formalism is available for highly customized simulations.

CVD CHAMBER

The Particle Tracing Module offers many ways to specify the initial position and velocity of particles. They can be released from selected domains, surfaces, or edges, and their distribution can be based on a user-defined expression or on the underlying finite element mesh. Particles can be released at specific locations by selecting points in the geometry, entering a list of coordinates, or importing initial positions from a file. The particle velocity can be initialized with a user-defined expression, by sampling from a Maxwellian distribution, or by releasing particles uniformly in a sphere, hemisphere, or cone. In this example, sprays of particles entering a CVD chamber are modeled by releasing the particles in cones from a set of points. The color of the particle trajectory represents the particle velocity. While the particles initially move in nearly linear trajectories according to their initial velocity vectors, they gradually lose momentum, causing their motion to be determined by the drag force exerted by the surrounding fluid.



PARTICLE-FIELD INTERACTIONS

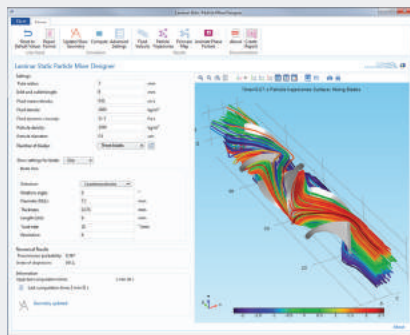
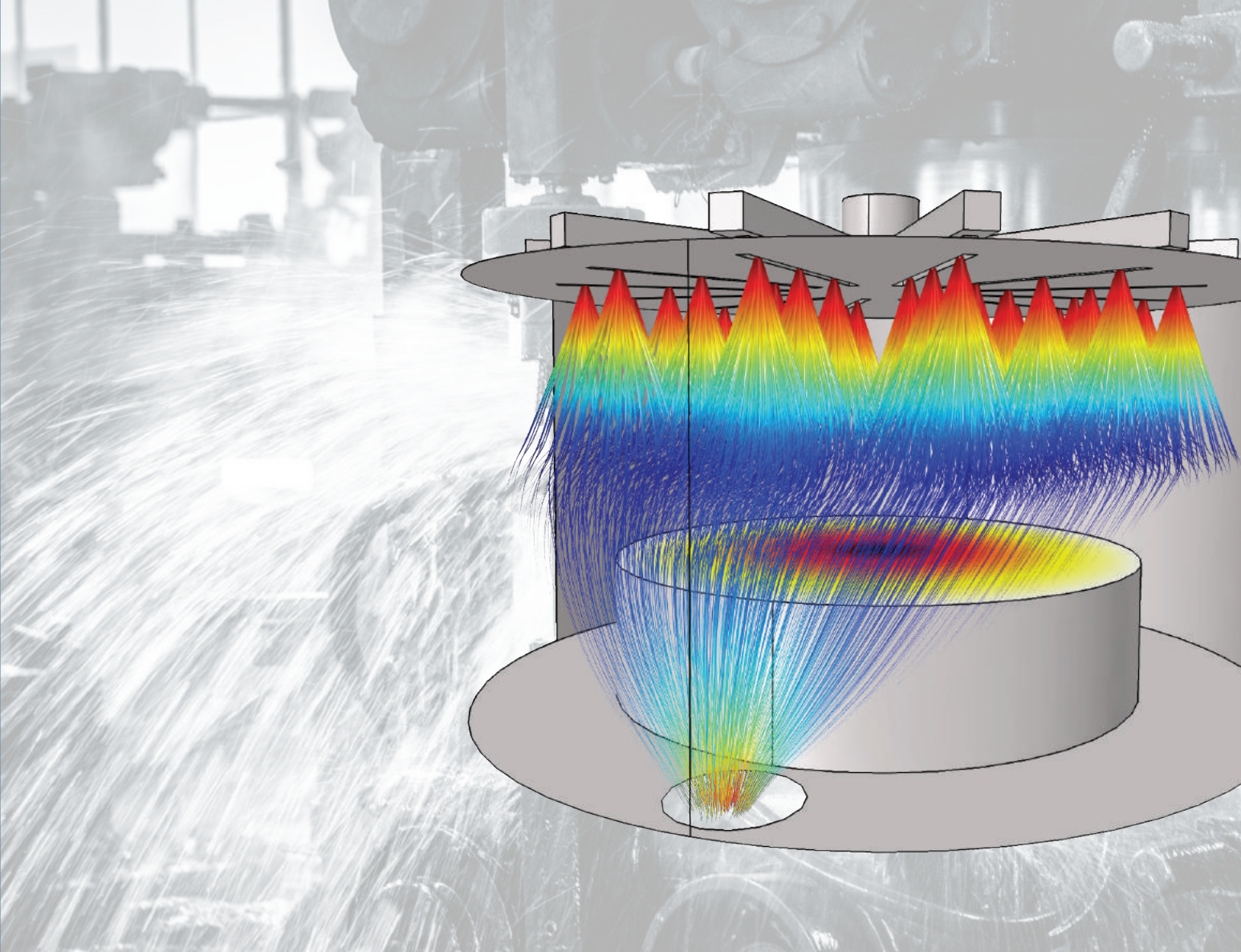
The Charged Particle Tracing interface can be used to model a beam of electrons diverging due to its own space charge. An iterative procedure allows the particle trajectories and the beam potential to be computed in a self-consistent manner. In this image, the particle trajectories are colored according to each particle's radial displacement from its initial position. The slice plot represents the beam potential and the vector field represents the force acting on an electron due to the beam potential.

HIGHLIGHTS

- Acoustophoresis
- Aerosol dynamics
- Beam physics
- Brownian motion
- Classical mechanics
- Dielectrophoresis and magnetophoresis
- Electric propulsion
- Erosion
- Etching
- Fluid flow visualization
- Fluid-particle interactions
- Geometrical optics in continuous media
- Ion energy distribution function visualization
- Ion mobility spectrometry
- Ion optics
- Mass spectrometry
- Mixers
- Multipaction
- Particle accelerators
- Rarefied gas dynamics
- Secondary emission
- Sprays
- Separation and filtration

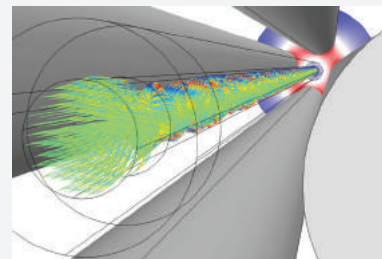
APPLICATION EXAMPLES

- Charge exchange cell simulator
- Laminar static particle mixer designer
- Red blood cell separation



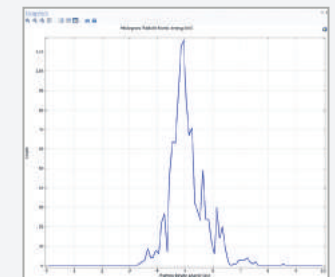
LAMINAR STATIC PARTICLE MIXER DESIGNER

The performance of a static mixer is studied in this specialized application. The user can change particle properties and mixer geometry to analyze the effects of different configurations on the mixing process.



MASS SPECTROMETRY

Trajectories of argon ions are modeled in a quadrupole mass spectrometer. The electric fields, which exert forces on the ions, have both AC and DC components. The combination of the two is essential for the function of the spectrometer. The graph shows the ion energy distribution function at the spectrometer's collector.



CAD Import Module

Collaboration within design teams is made easy with the CAD interoperability tools for COMSOL. The CAD Import Module allows for all major CAD file formats to be brought directly into the COMSOL Desktop, where you can simulate your design accurately using real-world multiphysics simulations. By including the Parasolid® software geometry engine, the CAD Import Module allows more advanced geometry operations to be performed on complex CAD models within the COMSOL Desktop®. The interactive repair feature assures that imported geometries are mathematically correct for simulation and includes defeaturing tools that remove fillets, small faces, sliver faces, as well as spikes and short edges.

LiveLink™ Products for CAD

All of the capabilities of the CAD Import Module are also available in the LiveLink products for SOLIDWORKS®, Autodesk® Inventor®, Autodesk® AutoCAD®, Autodesk® Revit®, PTC® Creo® Parametric™, PTC® Pro/ENGINEER®, and Solid Edge® CAD systems, all of which deliver further seamless integration between your CAD designs and multiphysics simulations. With these products, changing a feature in the CAD and BIM design automatically updates the geometry in COMSOL Multiphysics, while retaining the physics settings. This enables simulations that involve sweeps of the geometric parameters, and allows you to optimize your designs directly from within COMSOL Multiphysics.

Powered by
PARASOLID



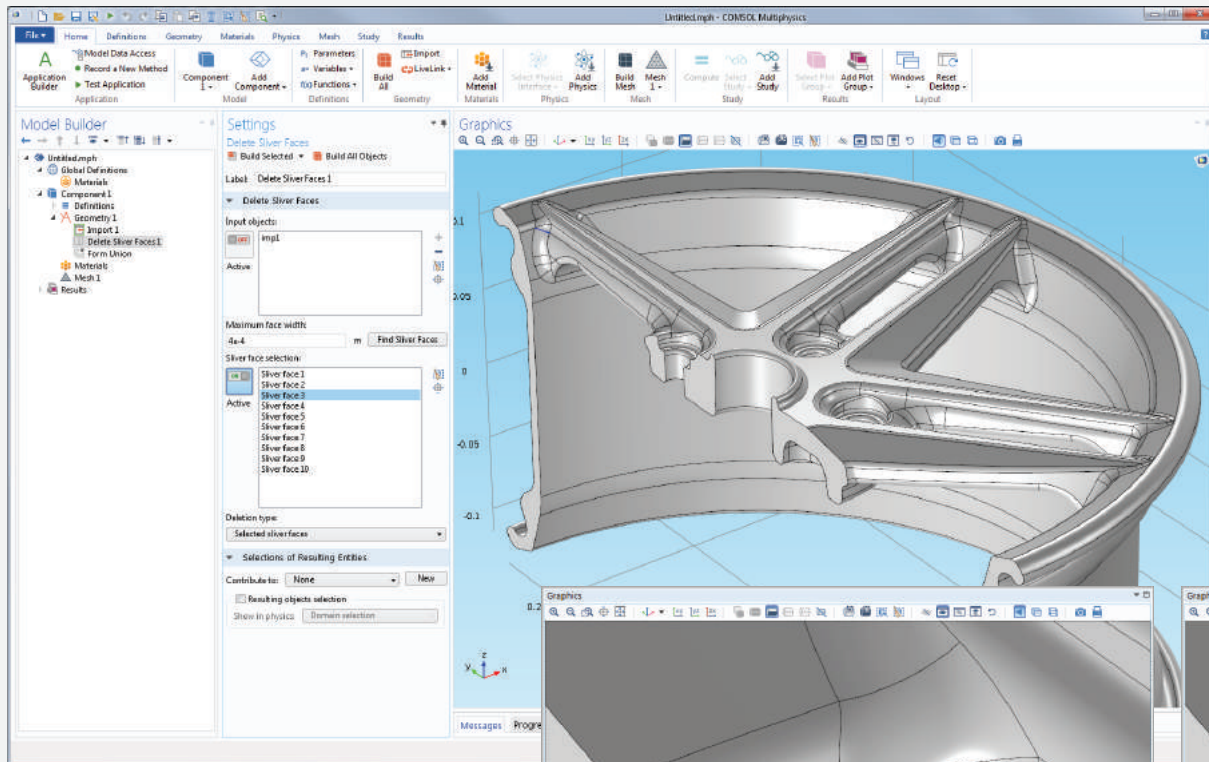
SUPPORTED CAD FILE FORMATS

ACIS® (.sat, .sab, .asat, .asab)
AutoCAD® (.dwg, .dxf)
CATIA® V5 (.CATPart, .CATProduct) *
IGES (.iges, .igs)
Inventor® (.ipt, .iam)
NX® (.prt)
Parasolid® (.x_t, .xmt_txt, .x_b, .xmt_bin)
PTC® Creo® Parametric™ (.prt, .asm)
PTC® Creo® Elements/Pro® (.prt, .asm)
PTC® Pro/ENGINEER® (.prt, .asm)
Revit® and Revit® Architecture (.rvt)
SOLIDWORKS® (.sldprt, .sldasm)
STEP (.step, .stp)
STL (.stl)
VRML, v1 (.vrm, .wrl)

* Requires File Import for CATIA® V5.

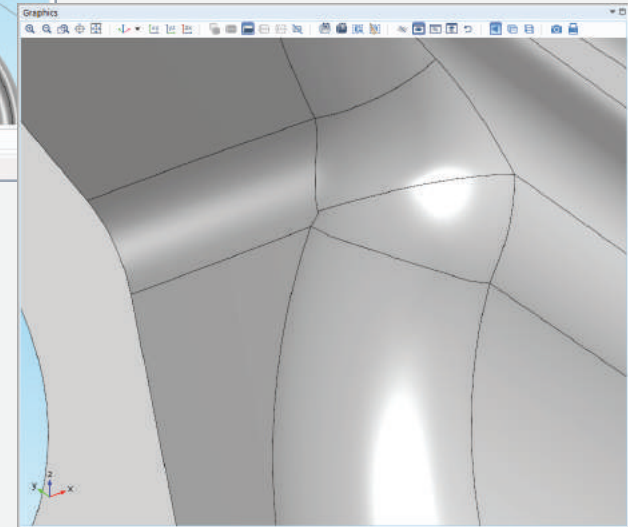
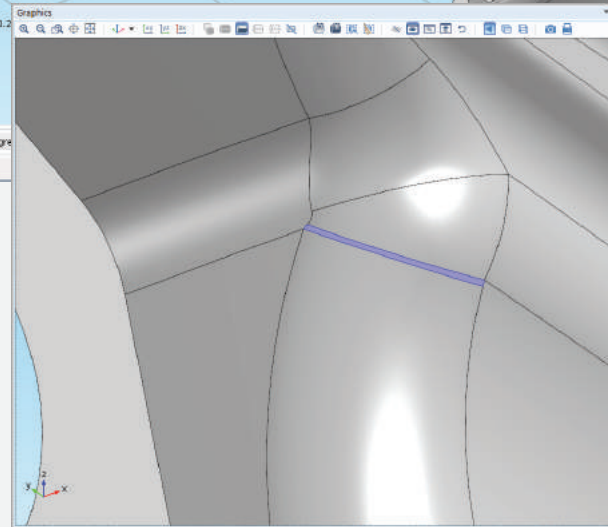
SUPPORTED OPERATIONS

Repair and defeaturing
Cap faces



DEFEATURING

Zooming in on the area around the spoke reveals a sliver face. Such a geometric entity can be defeated by collapsing it into an edge. Defeated geometry is shown to the right.



Design Module

The preparation of geometry for simulation is made easy by the Design Module, which provides tools for creating geometry and importing a variety of CAD file formats. You can create lofted objects based on cross-sectional profiles and use operations such as fillet, chamfer, midsurface, and thicken. For example, by using the loft tool, it is possible to create geometries from slices or contours, such as from an MRI scan.

The Design Module is built on the proven parametric framework for generating geometries in COMSOL Multiphysics® and on the industry standard Parasolid® software geometric kernel. Thereby, the module enables modifying or constructing designs from the ground up and sharing them with other tools.

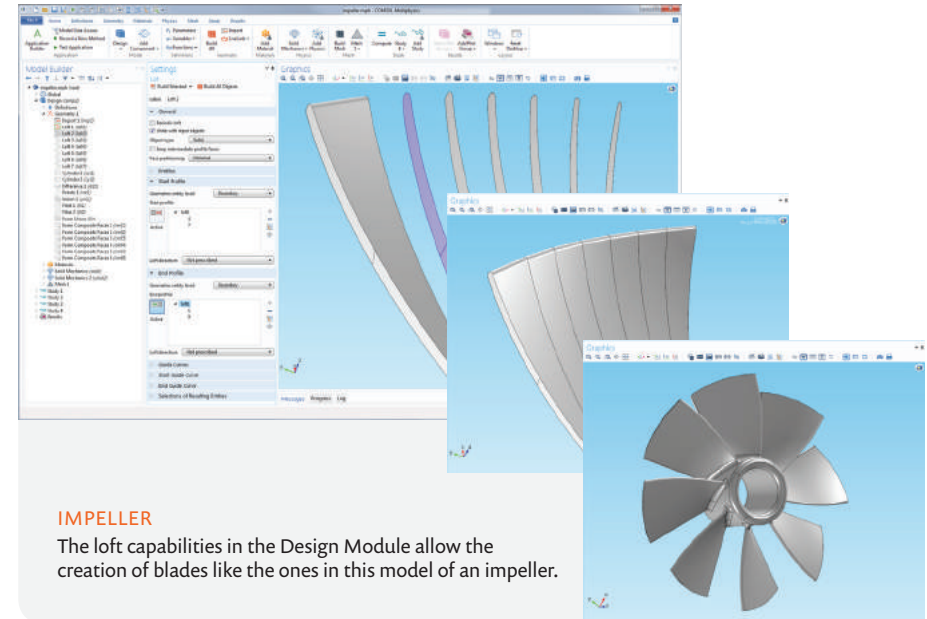
All of the capabilities of the CAD Import Module are also available with the Design Module.

SUPPORTED CAD FILE FORMATS

See CAD Import Module

SUPPORTED OPERATIONS

Repair and defeaturing	3D chamfer
Cap faces	Thicken
Loft	Midsurface
3D fillet	



Powered by
PARASOLID

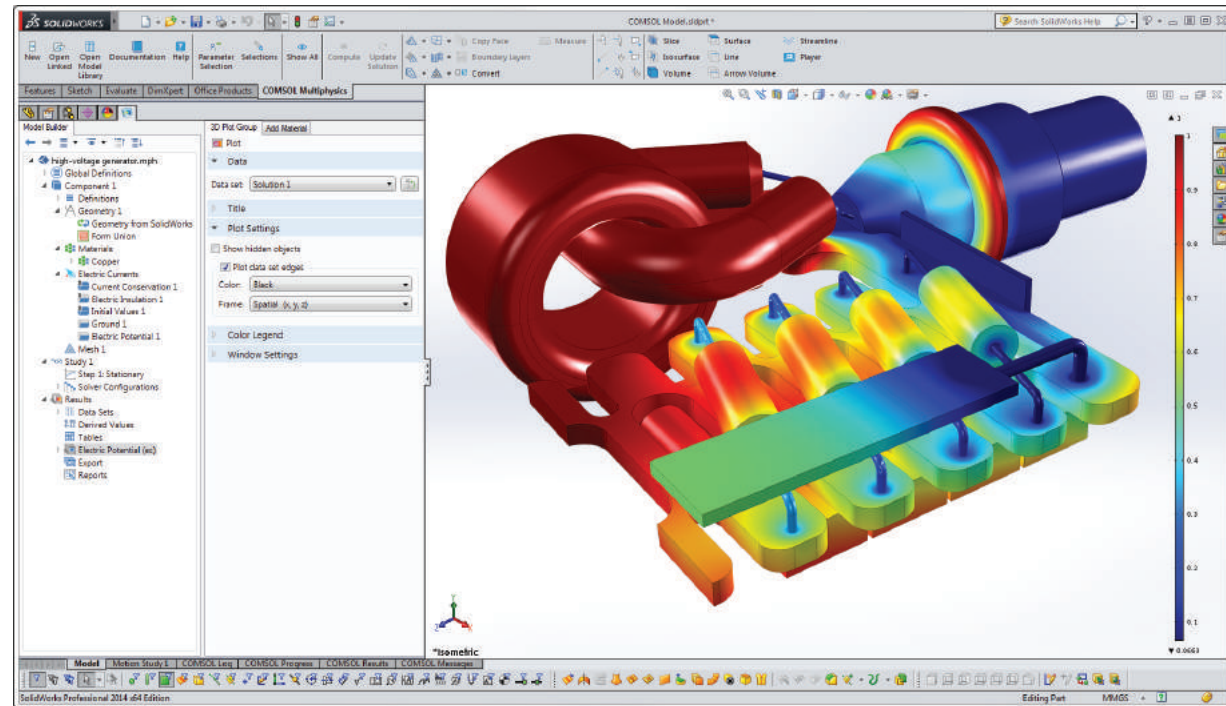


Parasolid is a trademark or registered trademark of Siemens Product Lifecycle Management Software Inc. or its subsidiaries in the United States and in other countries.

LiveLink™ for SOLIDWORKS®

ONE WINDOW INTERFACE

The LiveLink™ interface for the SOLIDWORKS® software CAD system has been extended with a One Window interface. SOLIDWORKS® software users can stay inside its environment and perform their multiphysics analysis directly from it. All of the COMSOL Multiphysics® modeling tools are available through the One Window interface, while changes in geometry are synchronously updated between the two packages.



HIGH-VOLTAGE GENERATOR

Simulation of the electromagnetic field in a high-voltage generator in an X-ray device. The design is created in the SOLIDWORKS® software CAD system and brought into COMSOL Multiphysics® using the LiveLink™ interface. From here, you can select different parts, define physical properties, and solve the model.

Model courtesy of Comet AG, Flamatt, Switzerland.

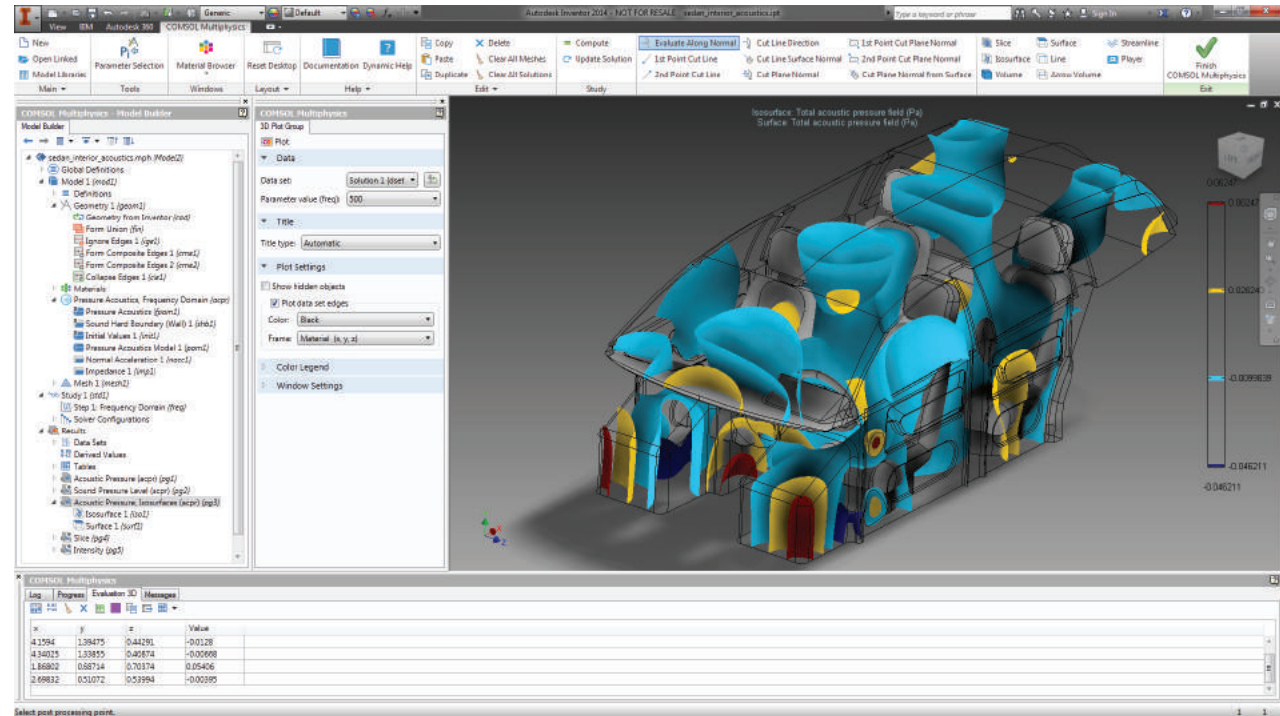


SOLIDWORKS is a registered trademark of Dassault Systèmes SolidWorks Corp.

LiveLink™ for Inventor®

ONE WINDOW INTERFACE

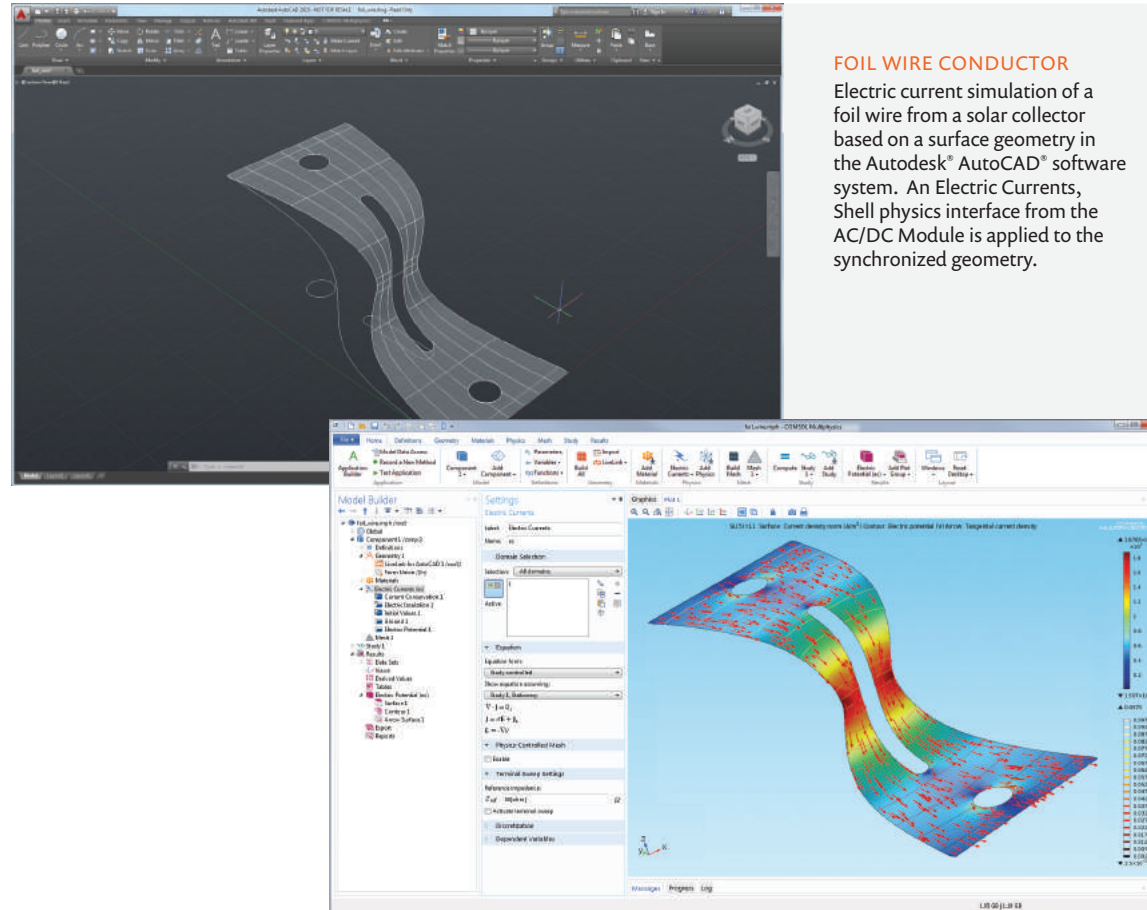
The LiveLink™ interface for the Autodesk® Inventor® software CAD system has been extended with a One Window interface. Autodesk® Inventor® software users can stay inside its environment and perform their multiphysics analysis directly from it. All of the COMSOL Multiphysics modeling tools are available through the One Window interface, while changes in geometry are synchronously updated between the two packages.



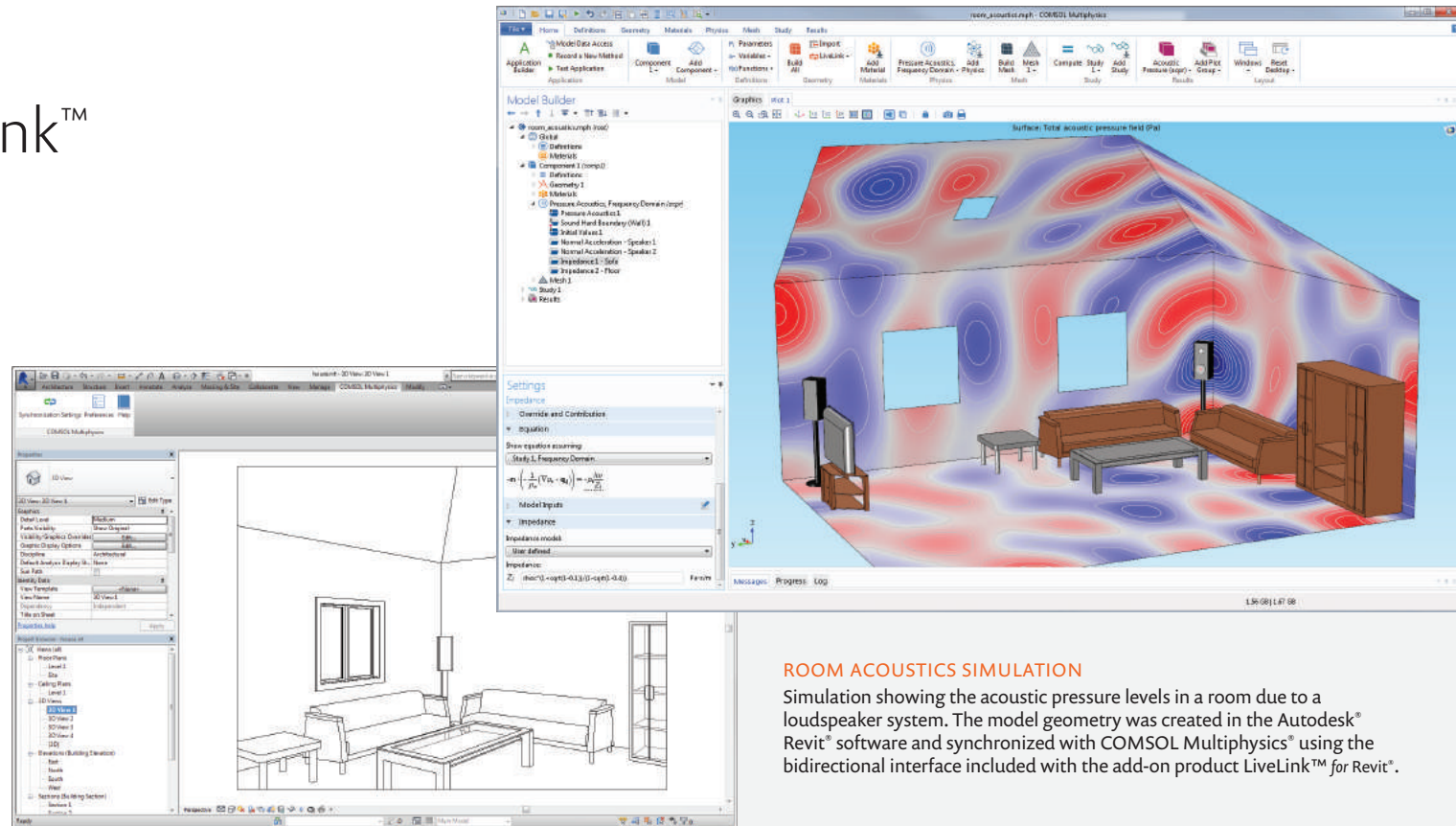
ACOUSTICS ANALYSIS

Simulation results show the isosurfaces of the acoustic pressure in a car interior. The geometry is created in the Autodesk® Inventor® software CAD system, while the simulation is set up and performed using COMSOL Multiphysics®.

LiveLink™ for AutoCAD®



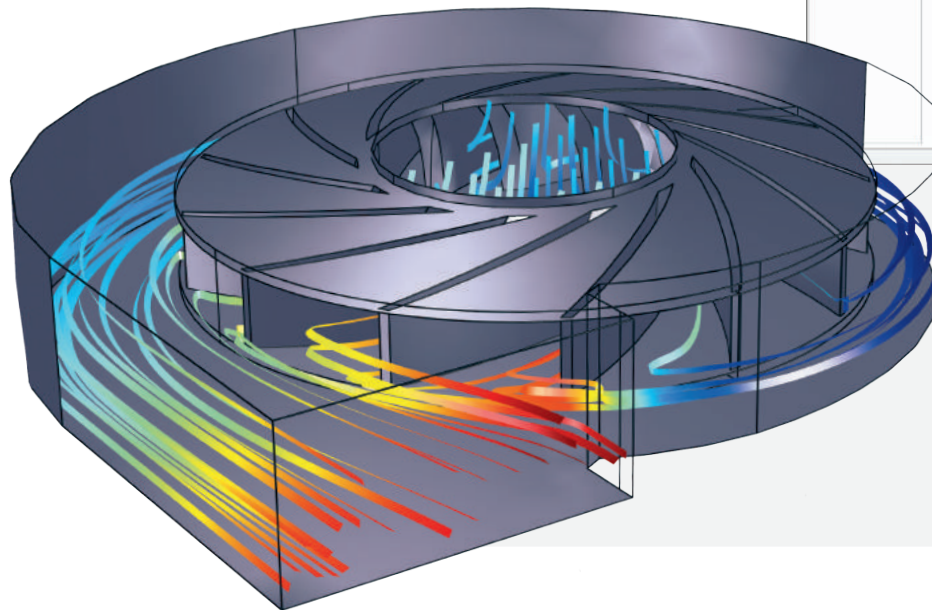
LiveLink™ for Revit®



ROOM ACOUSTICS SIMULATION

Simulation showing the acoustic pressure levels in a room due to a loudspeaker system. The model geometry was created in the Autodesk® Revit® software and synchronized with COMSOL Multiphysics® using the bidirectional interface included with the add-on product LiveLink™ for Revit®.

LiveLink™ for PTC® Creo® Parametric™



BURNER VENTILATION FAN

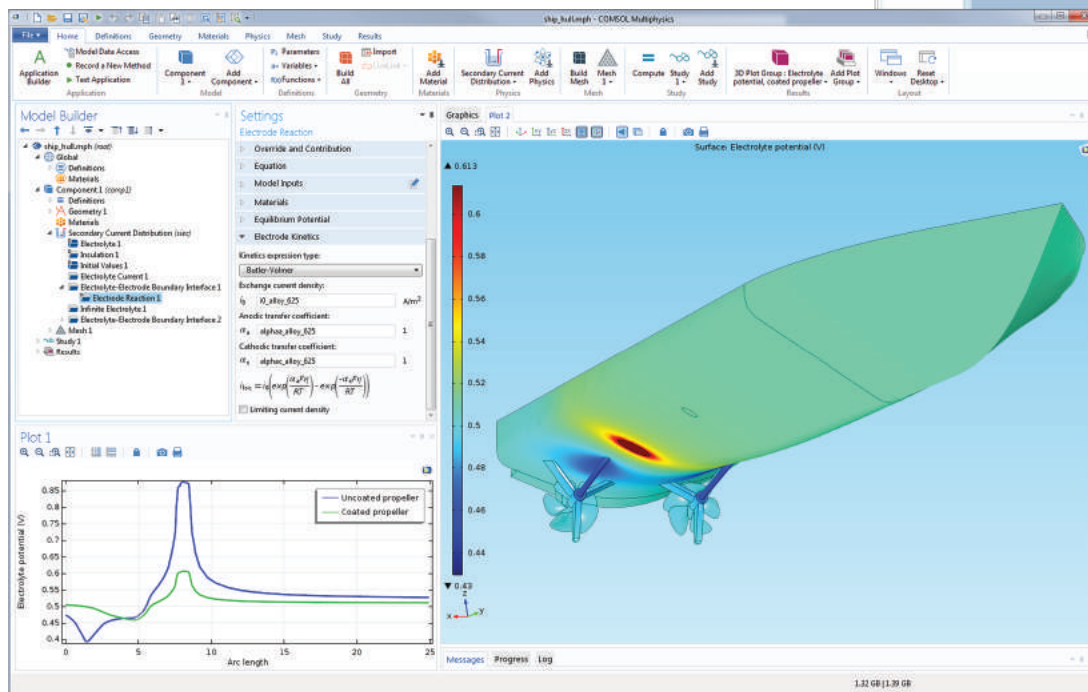
In a high-power burner, up to 40% of the energy required to run the system is consumed by the fan. This model shows the velocity vector (streamline ribbons) and pressure drop (color scale) of the flow into the impeller and housing of a burner ventilation fan. The positioning and size of the vanes can be updated through the LiveLink™ interface, while associativity between the simulation geometry and the CAD design assures that physics settings are retained.

Model courtesy of Gianluca Argentini, Riello S.p.A, Italy.



PTC and Creo are trademarks or registered trademarks of PTC Inc. or its subsidiaries in the U.S. and in other countries.

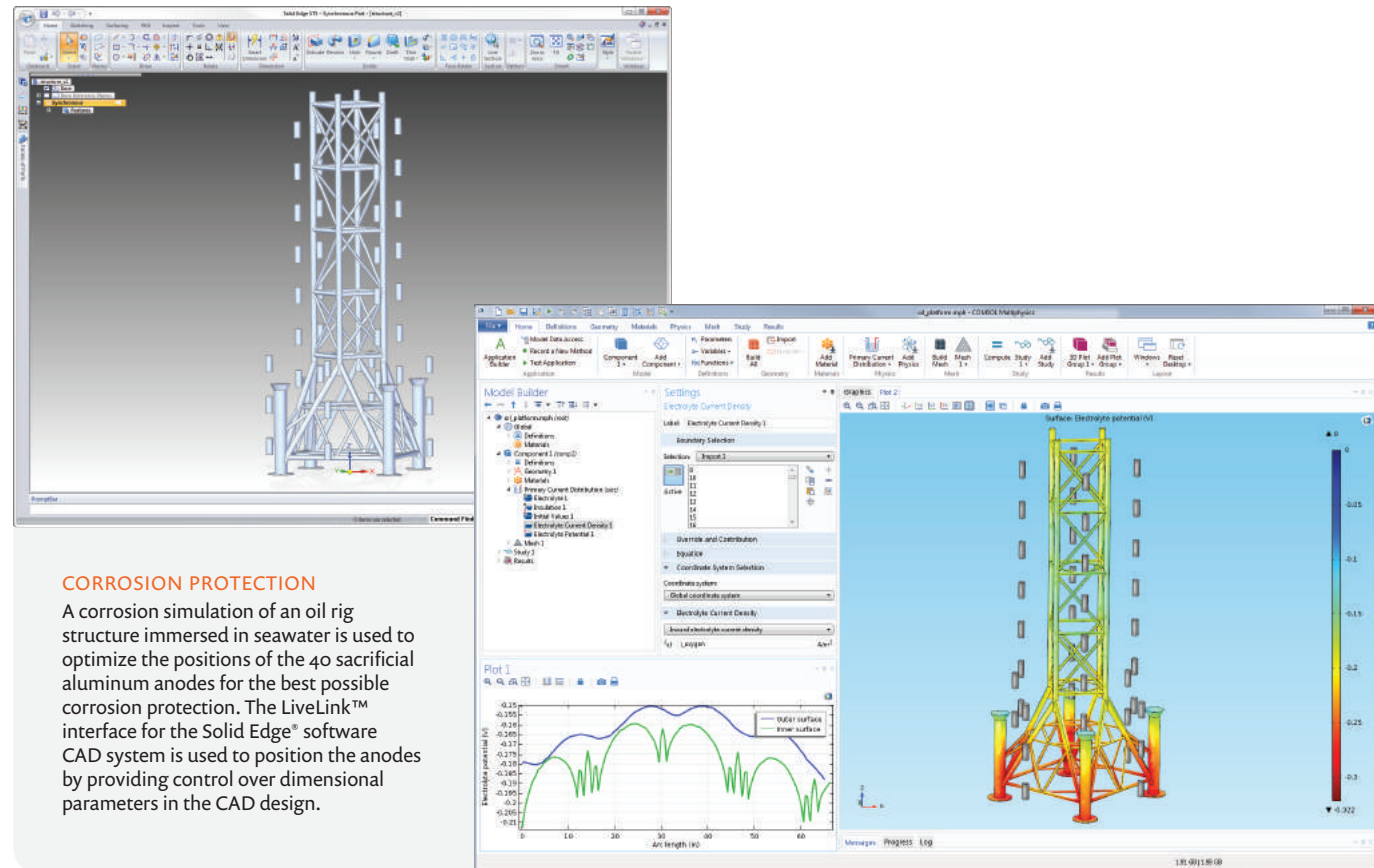
LiveLink™ for PTC® Pro/ENGINEER®



CORROSION MITIGATION

Using the LiveLink™ interface for the PTC Pro/ENGINEER® software CAD system, engineers are able to include geometries, such as the one shown here, in the COMSOL Multiphysics workspace. From here, meshing and solving a model of impressed current cathodic protection on a ship hull is straightforward. Results show electrolyte potential for a coated propeller.

LiveLink™ for Solid Edge®



Powered by
PARASOLID

DS
SPATIAL

Solution
Partner
SIEMENS
PLM

Solid Edge is a trademark or registered trademark of Siemens Product Lifecycle Management Software Inc. or its subsidiaries in the USA and in other countries. Siemens and the Siemens Logo are registered trademarks of Siemens AG.

ECAD Import Module

Using the ECAD Import Module, you can bring your ECAD files into COMSOL Multiphysics, where the 2D layouts are automatically converted to 3D CAD models. This opens up the world of modeling for simulating, among other applications, the components in integrated circuits, systems of connected MEMS devices, and electronics cooling.

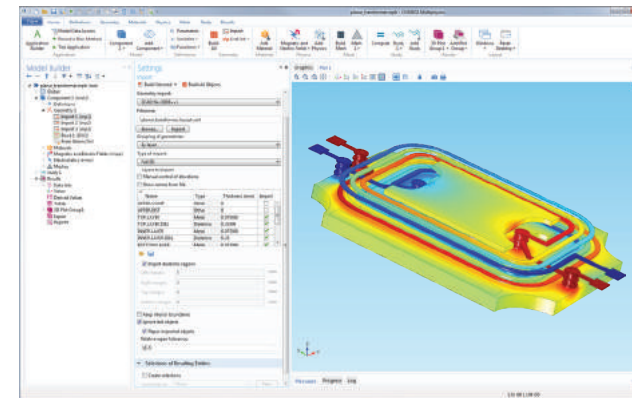
The ECAD Import Module supports the popular GDSII, ODB++, ODB++(X), and NETEX-G file formats. As part of the import process, you can select which subset of cells, nets, and layers to import. You can also edit layer thicknesses, control the geometric representation of bond wires, and include selected dielectric regions prior to or after importing.

The layout is automatically extruded and converted to a 3D CAD model for use in any kind of COMSOL Multiphysics simulations with any combination of add-on products. The 3D geometry model can further take part in general solid modeling operations in COMSOL Multiphysics. When combined with the CAD Import Module, Design Module, or one of the LiveLink™ products, the 3D model can be exported to the Parasolid® (.x_b, .x_t) or ACIS® (.sat) file formats, for use in other software.

SUPPORTED ECAD FILE FORMATS

DXF (.dxf)*
GDSII (.gds)
ODB++ (.zip, .tar, .tgz, .tar.gz)
ODB++(X) (.xml)
NETEX-G (.asc)

*Included in COMSOL Multiphysics; extended functionality is available with the CAD Import Module



POWER ELECTRONICS

The simulation results show the electric potential on the surface of a planar transformer. The layout is imported from an ODB++(X) file and converted to a 3D geometry model.

Support for implementation of the ODB++ Format was provided by Mentor Graphics Corporation pursuant to the ODB++ Solutions Development Partnership General Terms and Conditions (www.odb-sa.com).

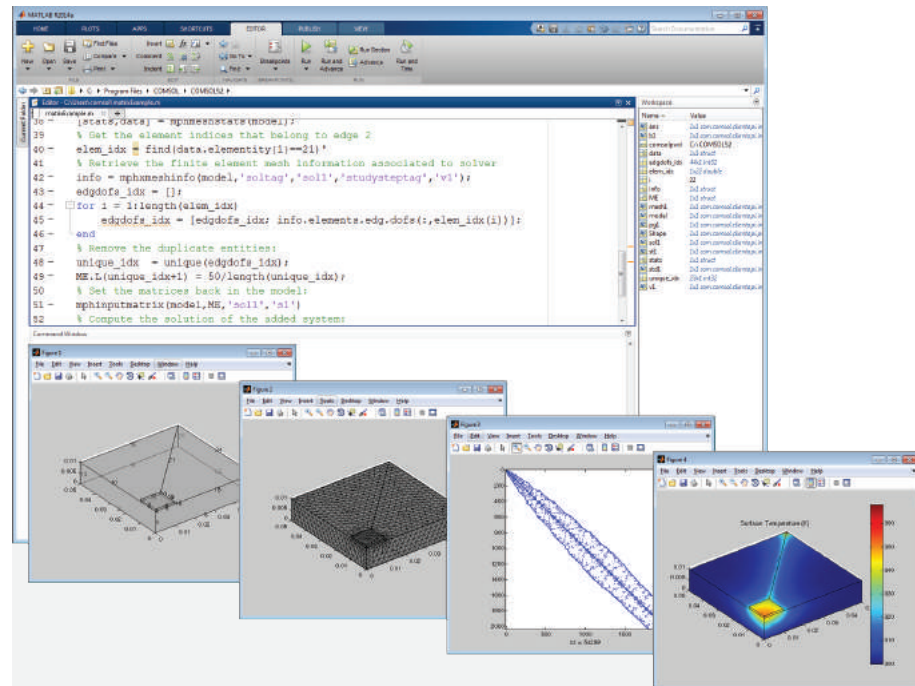
LiveLink™ for MATLAB®

LiveLink™ for MATLAB® allows the user to control a model from MATLAB® software code to streamline workflow. You can use MATLAB® software as a scripting language for making changes to parameters and other model settings, solve the model, and extract graphical or numerical results from the model.

- Enhance your in-house MATLAB® software code with powerful multiphysics simulations
- Create geometries using MATLAB® software scripts and extract properties of existing geometries, which may be imported from a CAD program
- Perform arbitrary statistical analysis on simulation results
- Use multiphysics models together with your own optimization procedures, such as genetic or simulated annealing algorithms
- Export COMSOL models as state-space matrices
- Call MATLAB® software functions from the COMSOL Desktop®
- Interface to external libraries and use MATLAB® software to read and write data in external sources such as the web, files, and databases



MathWorks and MATLAB are registered trademarks of The MathWorks, Inc.



HEAT TRANSFER

A transient heat transfer model is modified using the LiveLink™ interface for the MATLAB® computation environment. State-space matrices, mesh, and solution data from COMSOL Multiphysics® are made available in the MATLAB® software workspace, where the state-space model has been run in the MATLAB® software environment. The figures show temperature change over time, the finite element mesh, and the sparsity pattern of the stiffness matrix.

LiveLink™ for Excel®

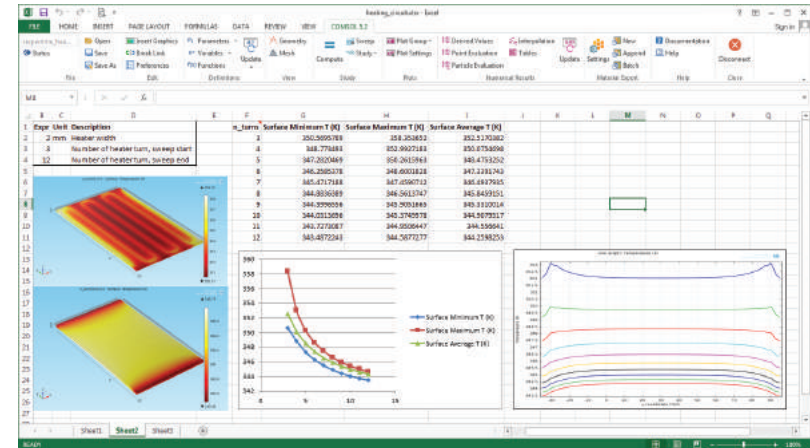
Extend your modeling capacity by running COMSOL Multiphysics® simulations from a spreadsheet with LiveLink™ for Excel®.

LiveLink™ for Excel® adds a COMSOL Multiphysics® tab to the Excel® ribbon for controlling parameters, variables, and other model settings. After having solved a model, results can easily be embedded as images or numerical data into an Excel® spreadsheet for reporting and further analysis.

The COMSOL API (Application Programming Interface) gives access to all aspects of modeling with COMSOL Multiphysics. The COMSOL API is available for use with Visual Basic for Applications (VBA), which is the programming language built into Excel® spreadsheet software. Using VBA you can change parameters and settings, solve the model, and extract numerical data from the solved model.

LiveLink™ for Excel® adds the capability to import/export Excel® files for parameters, variables, and more. This is available on all platforms (Windows®, Linux®, and OS X) in COMSOL Multiphysics® and COMSOL Server™.

The COMSOL Multiphysics® tab and support for VBA is only available on the Windows® operating system and requires Excel® 2007, 2010, 2013, or 2016.



PARAMETERIZED STUDY

LiveLink™ for Excel® is used for a parameterized multiphysics simulation of a heating circuit including DC-induced Joule heating, heat transfer, and structural behavior of a thin resistive layer covered on a solid glass plate. The parameters controlling the geometric dimensions as well as the boundary conditions are edited in an Excel® spreadsheet and synchronized to the underlying COMSOL Multiphysics® model. A dedicated ribbon tab is added to the Excel® program for easy access to parameters, variables, functions, geometry, mesh, solvers, and results.

SUPPORTED FILE FORMATS

Microsoft Excel® workbook (.xlsx)

Microsoft Excel® workbook code (.xlsm)

User Support

✓ Software Subscription

With your purchase of a perpetual COMSOL software license, you will automatically receive access to software updates and technical support for 12 months. Product updates ensure that your software stays current with new functionality and operating systems. Additionally, the subscription program provides license administration services to install COMSOL software on a new computer, change named users, and more.

✓ Rapid Response Technical Support

COMSOL® technical support is available directly from our highly skilled support engineers. We are easy to reach by telephone or email. Plus, our highly respected online Knowledge Base provides a comprehensive database of answers to just about any multiphysics modeling question you may have, 24/7.

✓ Specification Charts

The specification charts will guide you in finding the product combination that best suits your engineering and scientific applications. You can use the charts to find products for a specific application area, discover the simulation capabilities available through combining multiple modules (comsol.com/products/specifications), or integrate with CAD software (comsol.com/products/specifications/cad).

✓ Continuous Training

Training courses are offered frequently at locations worldwide. Available training covers everything from an introduction to COMSOL software to specific focus areas covering electrical, mechanical, fluid, chemical, optimization, particle tracing, and interfacing applications.

✓ Video Tutorials and Webinars

The COMSOL® Video Gallery provides a convenient way of learning from worked models, where how-to videos will guide you step-by-step through the modeling process. Webinars offer live online demonstrations on how to apply COMSOL software to specific physics or applications. Videos and archived webinars are available at comsol.com/videos.

✓ Online Community and COMSOL Blog

The online COMSOL Community lets you network and share modeling tips and tricks with thousands of users across all scientific and engineering disciplines via the Discussion Forum (comsol.com/community/forums). Check out the COMSOL Blog (comsol.com/blogs) for posts on simulation topics including the electrical, mechanical, fluid, and chemical disciplines.

✓ Certified Consultants

COMSOL® Certified Consultants provide analysis projects and custom applications. COMSOL® Certified Consultants are renowned for their domain expertise and for their extensive experience using COMSOL Multiphysics.

✓ COMSOL Conference and COMSOL Days

Industry analysts have described the COMSOL® Conference as the premier multiphysics event. Held in leading centers of science and technology worldwide, the annual user conference is a must-attend event for new as well as experienced COMSOL users. Here, you will learn about the latest multiphysics tools and solvers, receive training, and learn from hundreds of user presentations, papers, and posters. COMSOL Days offers a one-day digest of everything new and inspiring with multiphysics modeling and app design. Attendees can learn from minicourses, invited speakers, and opportunities to meet with the COMSOL community.

License Options - COMSOL Multiphysics®

Named Single User License (NSL): One physical person designated to COMSOL by name can run the software. Not for use over a network.

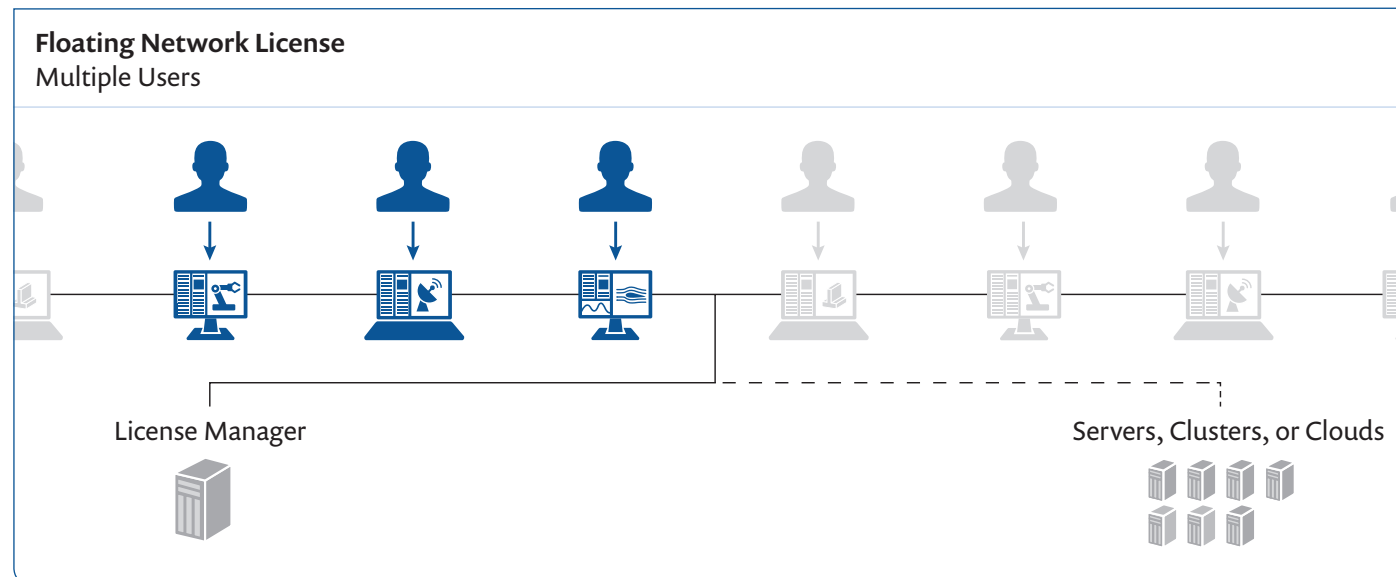
CPU-Locked Single User License (CPU): You can install the software on one computer and different users can take turns using COMSOL on that computer, one person at a time. Not for use over a network.

Floating Network License (FNL): Licensed per concurrent user, you can install the software on as many machines on your network as you want. COMSOL can run on local computers with the network being used only for license authentication. Alternatively, you can run COMSOL on a remote computer over your network.

Class Kit License (CKL): As many as 30 students can simultaneously use the software for a class over a network. Teachers can also use it to prepare class materials.

SUPPORTED PLATFORMS: Windows®, Linux®, OS X.

Usage	NSL	CPU	FNL
Multiple Computers	✓		✓
Multiple Platforms	✓		✓
Multiple Users		✓	✓
Client/Server			✓
Clusters			✓
Cloud Computing			✓



*Diagram depicts a Floating Network License enabled for 3 concurrent users.

High-Performance Computing (HPC) support is included for all license options with no limit on the number of cores.

License Options - COMSOL Server™

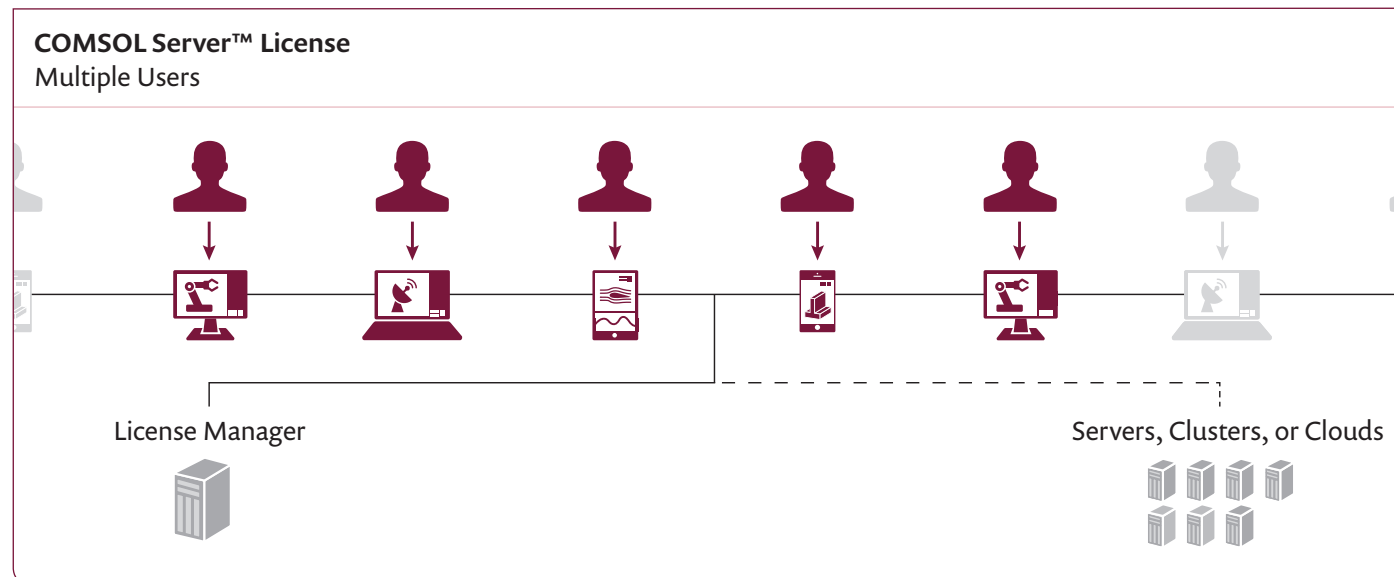
COMSOL Server License (CSL): Licensed on a concurrent user basis. Each concurrent user may run up to four simultaneous applications. The CSL version of the software enables you to run apps built with COMSOL Multiphysics. The CSL allows you to locally host and run apps within your organization and also to those outside your organization who time-share your concurrent user slots, worldwide.

Academic Server License (ASL): Licensed on a 300-concurrent-user basis. Each concurrent user may run up to four simultaneous applications. The ASL version of the software enables you to run apps built with COMSOL Multiphysics. The ASL allows you to locally host and run apps for students, faculty, and staff of your academic institution or of any other academic institution, worldwide.

Applications can be accessed through a major web browser on a variety of operating systems or a COMSOL Client for the Windows® operating system.

SUPPORTED PLATFORMS: Windows®, Linux®, OS X, iOS, Android™.

Usage	CSL
Multiple Computers	✓
Multiple Platforms	✓
Multiple Users	✓
Client/Server	✓
Clusters	✓
Cloud Computing	✓
Worldwide Use	✓



*Diagram depicts a COMSOL Server™ License enabled for 5 concurrent users; each concurrent user can run up to 4 simultaneous simulation applications.

For detailed license conditions and more information, please refer to the **COMSOL Software License Agreement**.

