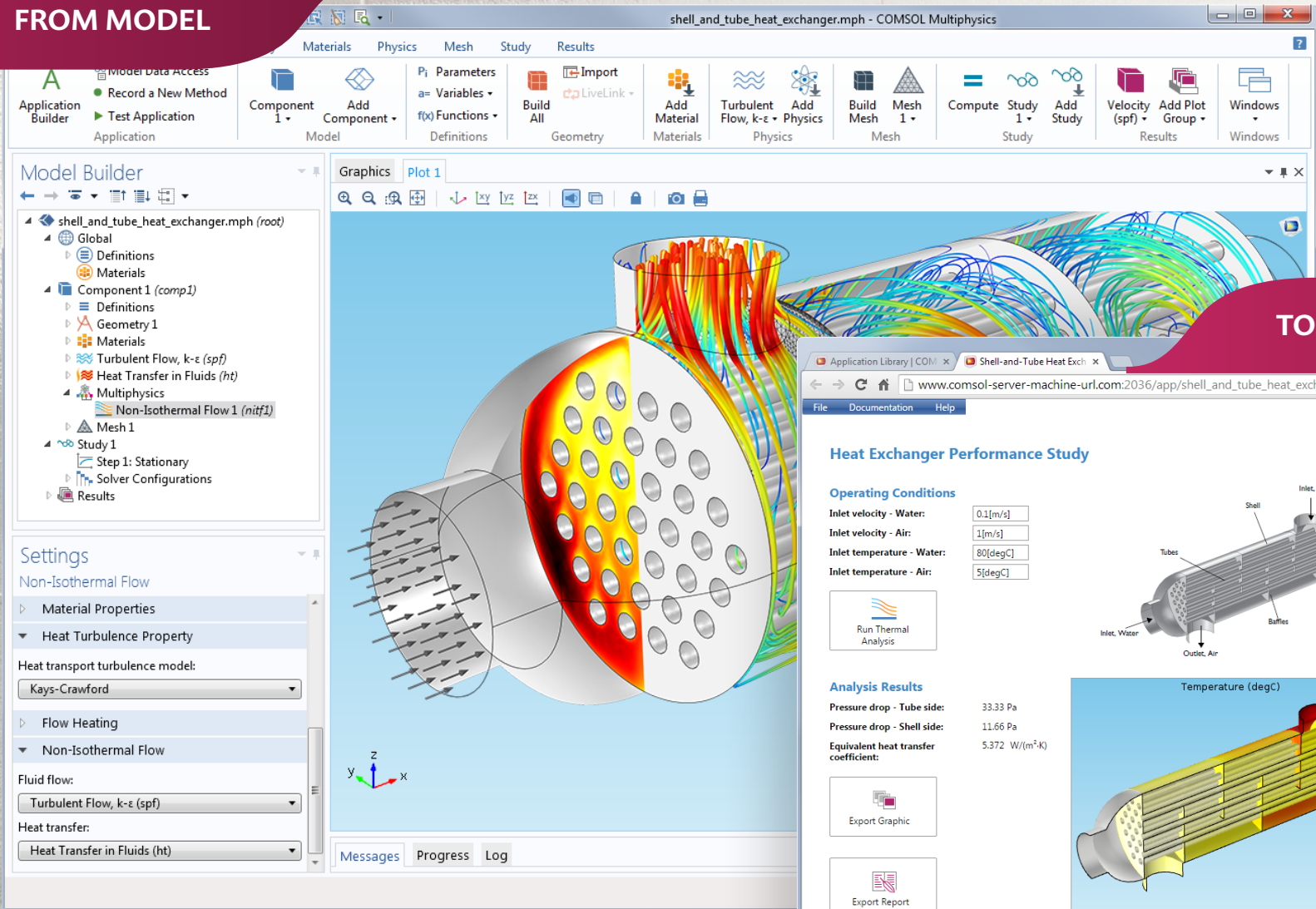
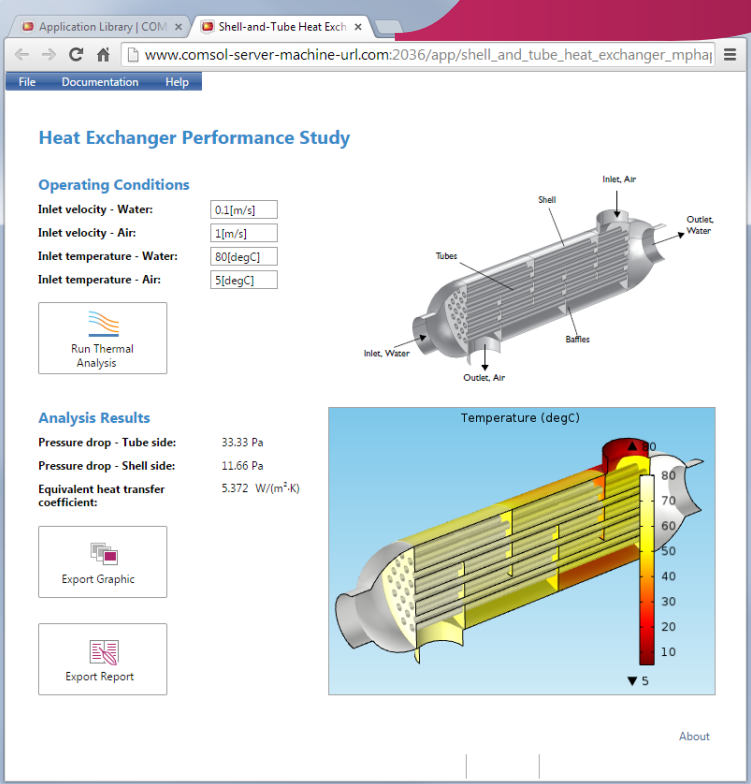


COMSOL
MULTIPHYSICS®
COMSOL
SERVER™

FROM MODEL



TO APP



Modeling and Simulation for Everyone.

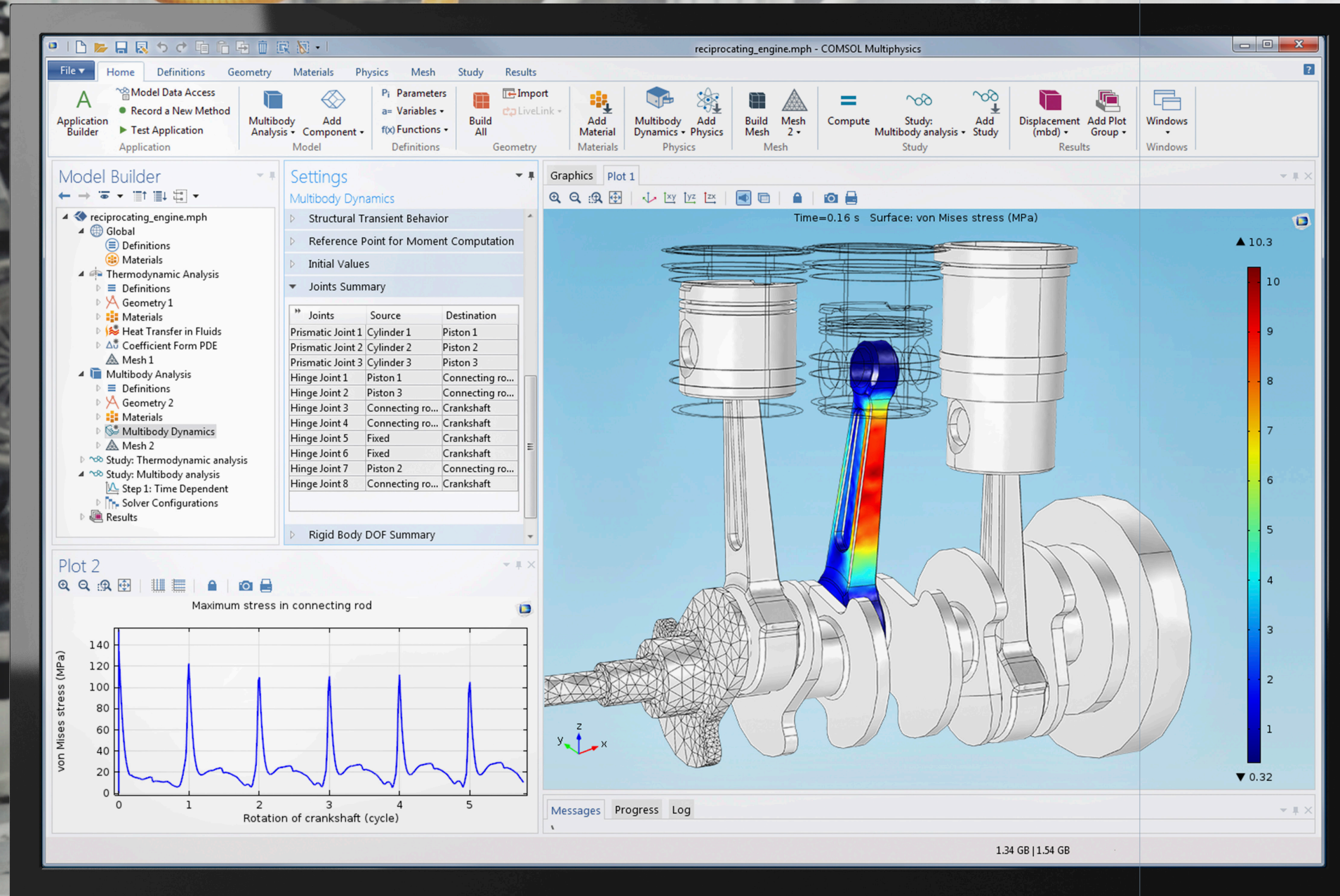
COMSOL® software applications realize the vision of making simulation available for everyone to use. The idea is simple — make it possible to build and run simulation applications.

That is exactly what the COMSOL Multiphysics® software seeks to deliver with its leading simulation and application design environment. It’s an inclusive approach to simulation that provides engineers in all stages of the product design cycle with the most powerful computational tools and ready-to-use applications. This expanded scope of simulation helps drive innovation and boost productivity throughout your organization; from design to production.

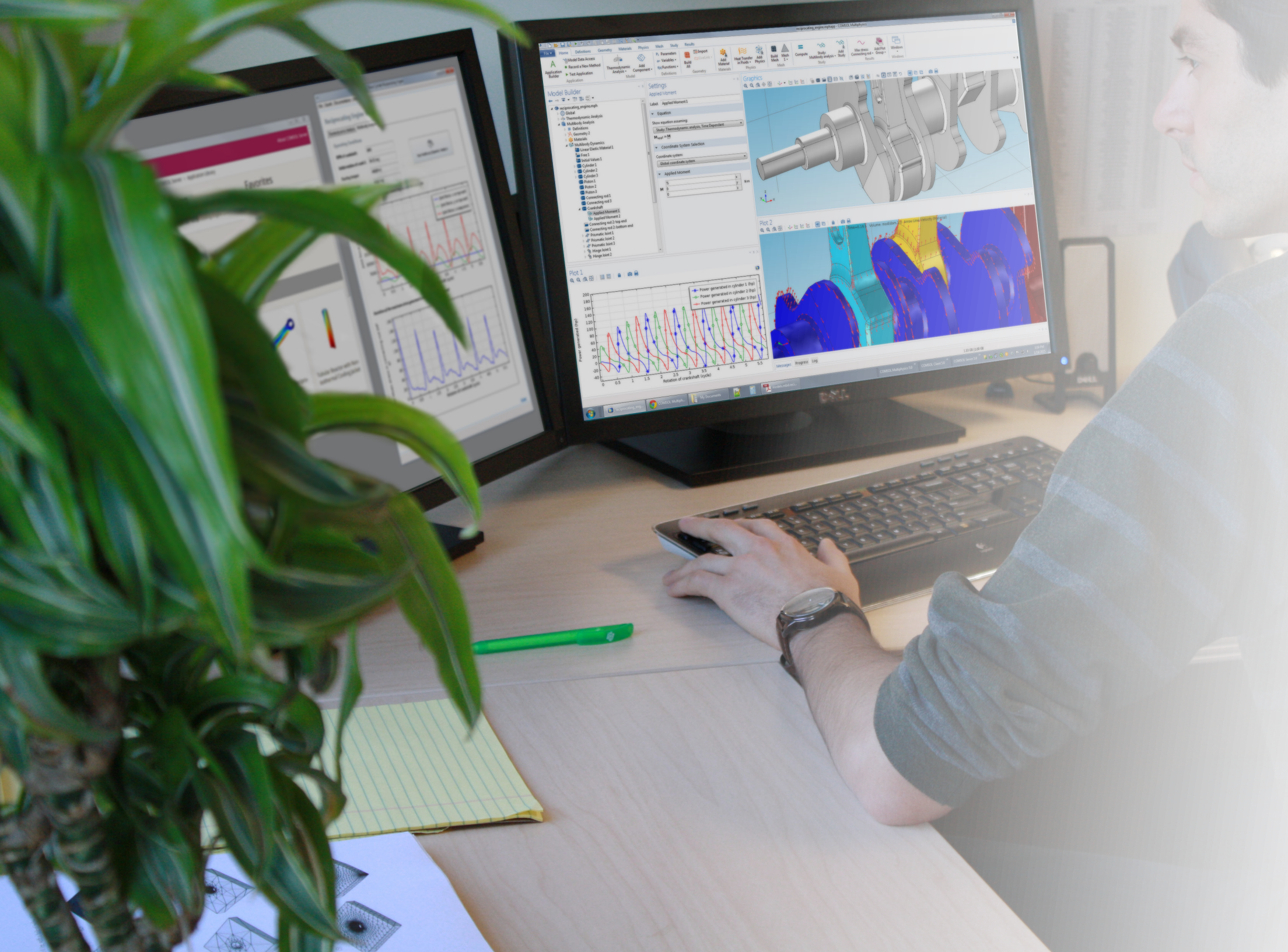
Are you ready to build your own simulation apps? COMSOL Multiphysics® and COMSOL Server™ make it easy.

CONTENTS

PRODUCT SUITE	10
COMSOL MULTIPHYSICS	12
COMSOL SERVER	22
APPLICATION SPECIFIC PRODUCTS	
ELECTRICAL	24
MECHANICAL	38
FLUID	52
CHEMICAL	64
MULTIPURPOSE	74
INTERFACING	80
USER SUPPORT	93
LICENSE OPTIONS	94



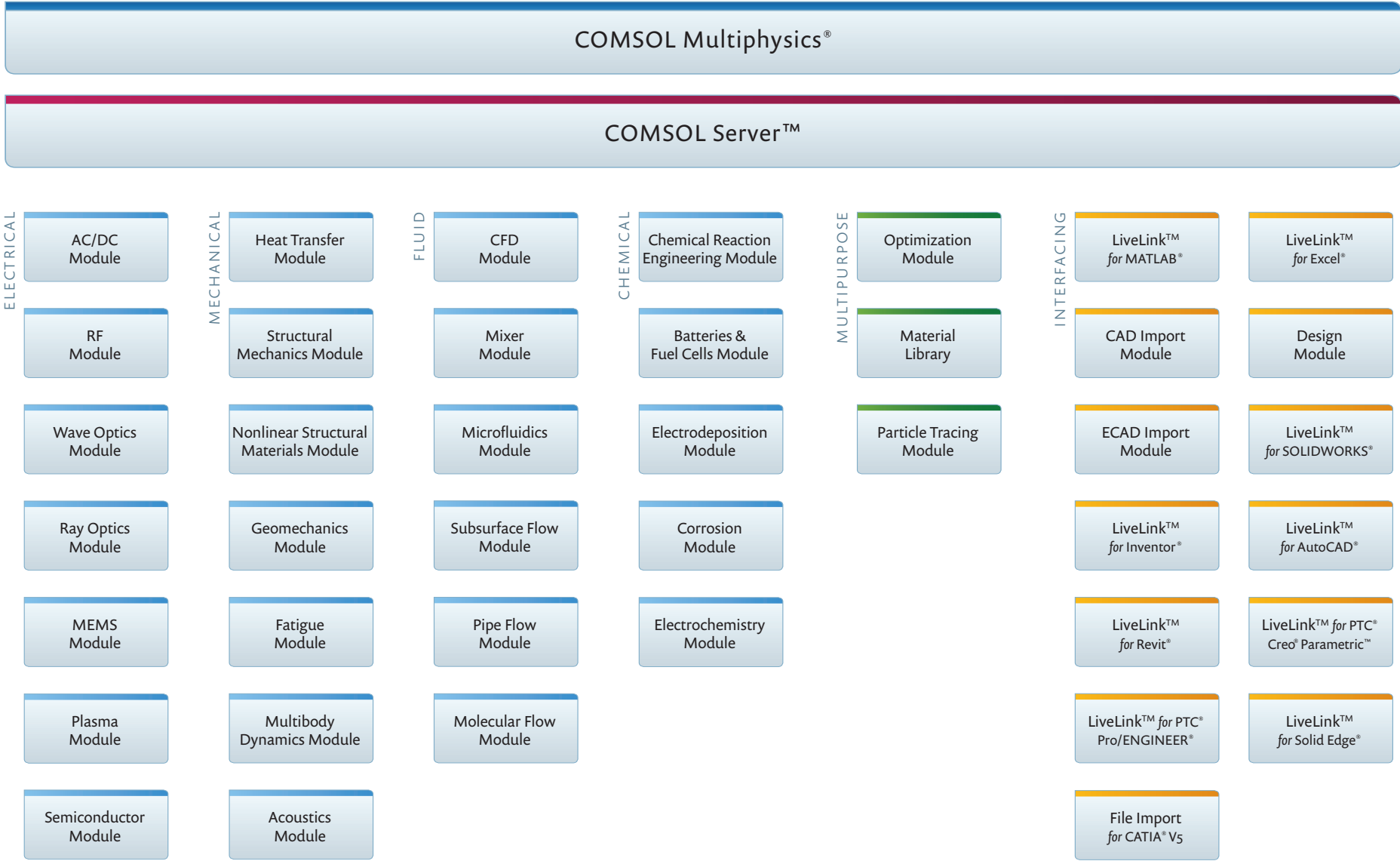
How do you create the best design and share your simulation expertise?



Through powerful computational tools. From Model to App.



With simulation apps that can be easily shared. Worldwide.



PRODUCT SUITE

COMSOL Multiphysics® _____ 12

COMSOL Server™ _____ 22

Electrical

AC/DC Module _____ 24

RF Module _____ 26

Wave Optics Module _____ 28

Ray Optics Module _____ 30

MEMS Module _____ 32

Plasma Module _____ 34

Semiconductor Module _____ 36

Mechanical

Heat Transfer Module _____ 38

Structural Mechanics Module _____ 40

Nonlinear Structural Materials Module _____ 42

Geomechanics Module _____ 44

Fatigue Module _____ 46

Multibody Dynamics Module _____ 48

Acoustics Module _____ 50

Fluid

CFD Module _____ 52

Mixer Module _____ 54

Microfluidics Module _____ 56

Subsurface Flow Module _____ 58

Pipe Flow Module _____ 60

Molecular Flow Module _____ 62

Chemical

Chemical Reaction Engineering Module _____ 64

Batteries & Fuel Cells Module _____ 66

Electrodeposition Module _____ 68

Corrosion Module _____ 70

Electrochemistry Module _____ 72

Multipurpose

Optimization Module _____ 74

Material Library _____ 76

Particle Tracing Module _____ 78

Interfacing

CAD Import Module _____ 80

File Import for CATIA® V5 _____ 80

Design Module _____ 82

LiveLink™ for SOLIDWORKS® _____ 83

LiveLink™ for Inventor® _____ 84

LiveLink™ for AutoCAD® _____ 85

LiveLink™ for Revit® _____ 86

LiveLink™ for PTC® Creo® Parametric™ _____ 87

LiveLink™ for PTC® Pro/ENGINEER® _____ 88

LiveLink™ for Solid Edge® _____ 89

ECAD Import Module _____ 90

LiveLink™ for MATLAB® _____ 91

LiveLink™ for Excel® _____ 92

COMSOL Multiphysics®

The COMSOL Multiphysics® software simulation environment facilitates all 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.

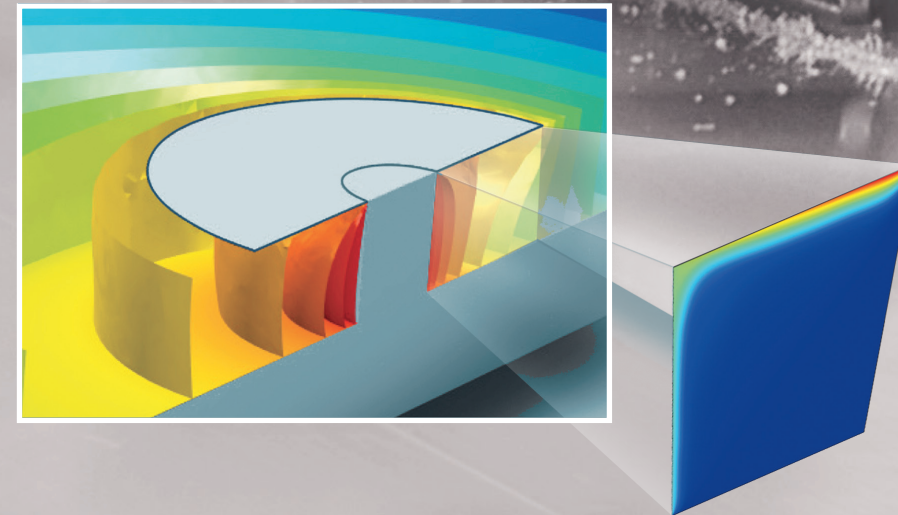
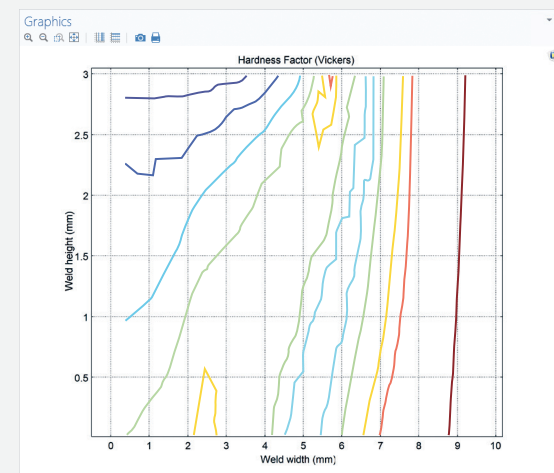
Model setup is quick, thanks to a number of predefined physics interfaces for applications ranging from fluid flow and heat transfer to structural mechanics and electrostatics. Material properties, source terms, and boundary conditions can all be spatially varying, time-dependent, or functions of the 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.

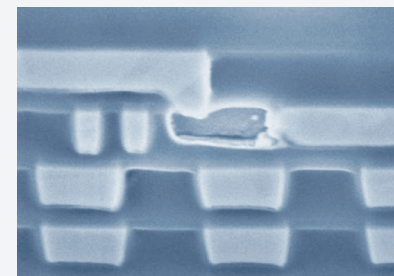
This model shows temperature distribution in a model that couples fluid flow with friction and heat transfer. The graph shows the hardness factors in the weld as calculated from the model.



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

HIGHLIGHTS

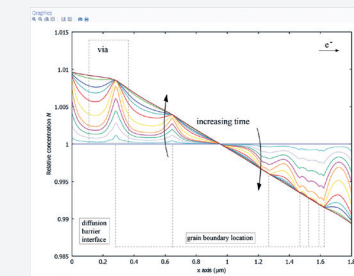
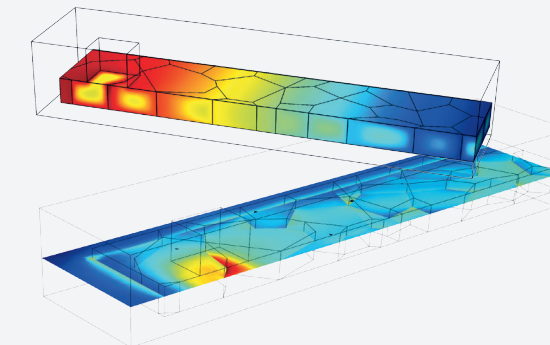
- Model Builder
- Application Builder
- Animation, image, and data export
- Automatic, swept, and boundary layer meshing
- Boundary and volume integration
- Component couplings for linking 1D, 2D, and 3D model components
- Core physics interfaces
- Fast Fourier Transform (FFT)
- Interpolation table import 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



ELECTRONICS

Model predicting a copper line interconnect failure. Simulation results show vacancy concentrations, and the resulting von Mises stresses these produce. The graph shows the evolution of the vacancy accumulation over time where peaks occur near the via and at the grain boundaries.

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've incorporated these qualities into the complete COMSOL Desktop® environment.

This sleek interface gives you full insight 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 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 your model is done in the Model Wizard by selecting one or more physics interfaces.

APPLICATION LIBRARIES

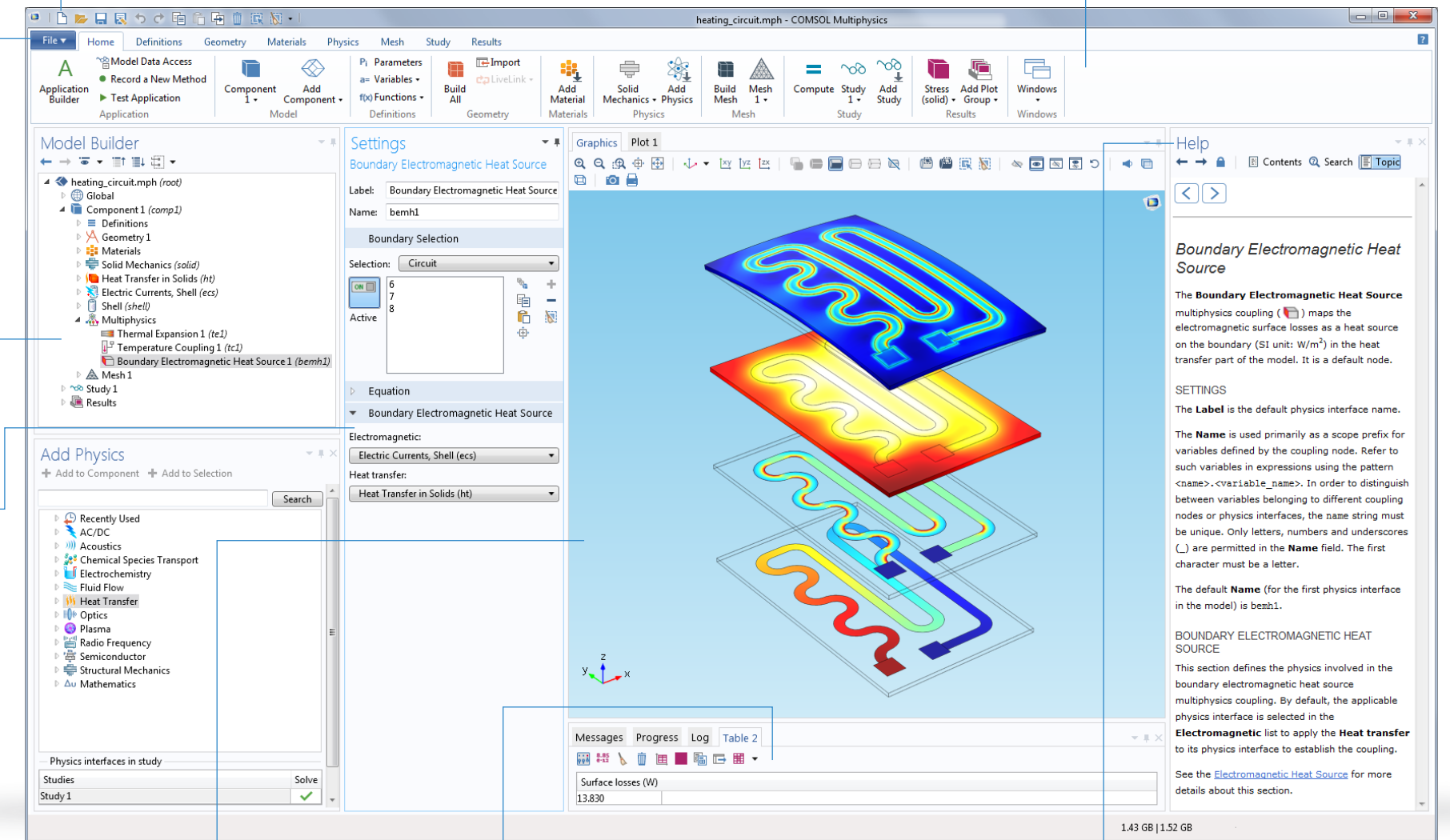
The Application Libraries consist of documented examples and simulation apps from all fields of engineering and science.

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 will 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 continually updated with online access to the Knowledge Base and Model 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

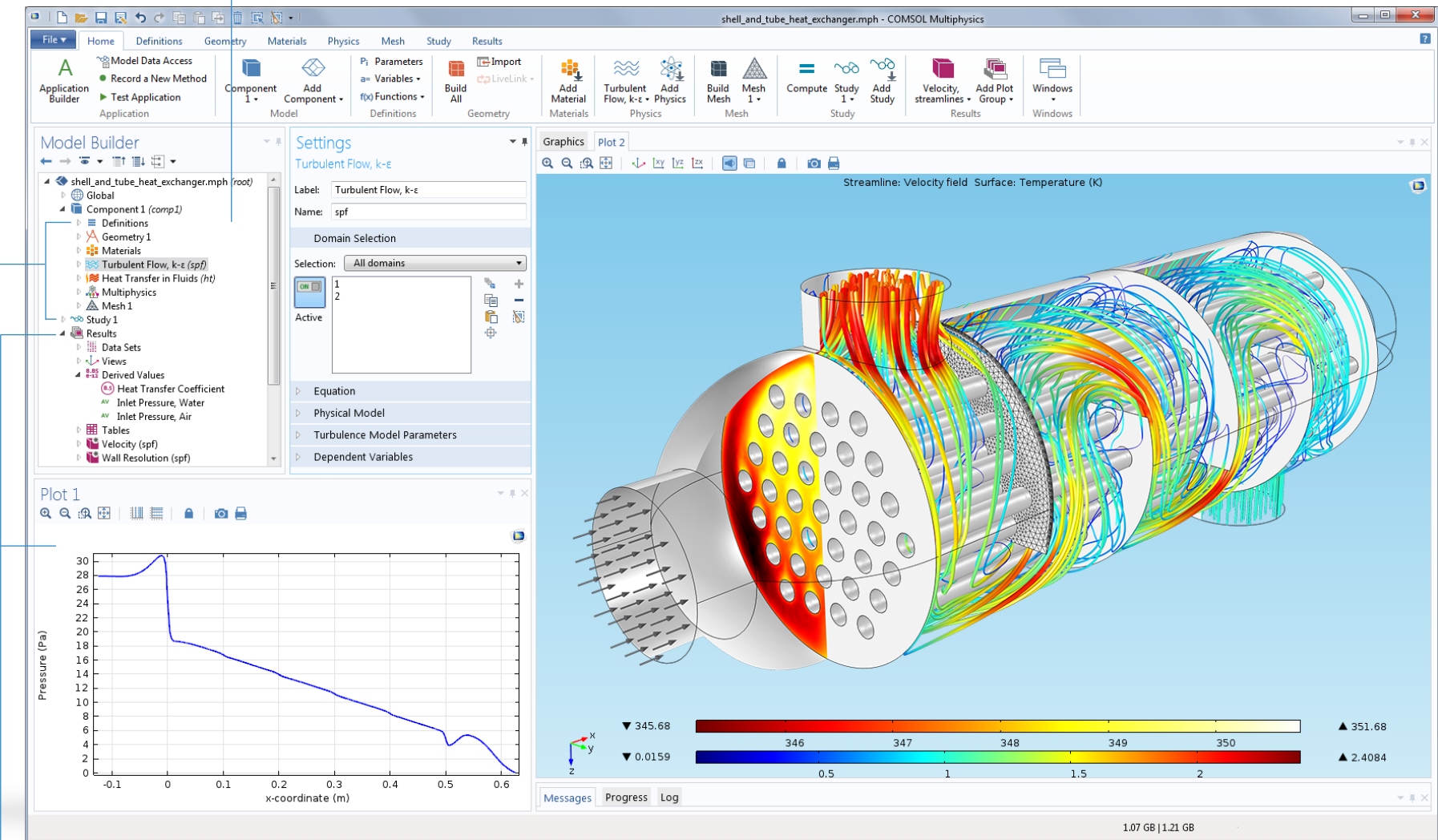
Automatically generate and customize your mesh for optimal resolution and solving.

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 processing 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 the tools needed to build and run simulation applications. Any COMSOL Multiphysics model can be turned into an app 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 app. It is a programming environment that allows you to write Java® code to modify the data structure representing a COMSOL Multiphysics® model.

"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.

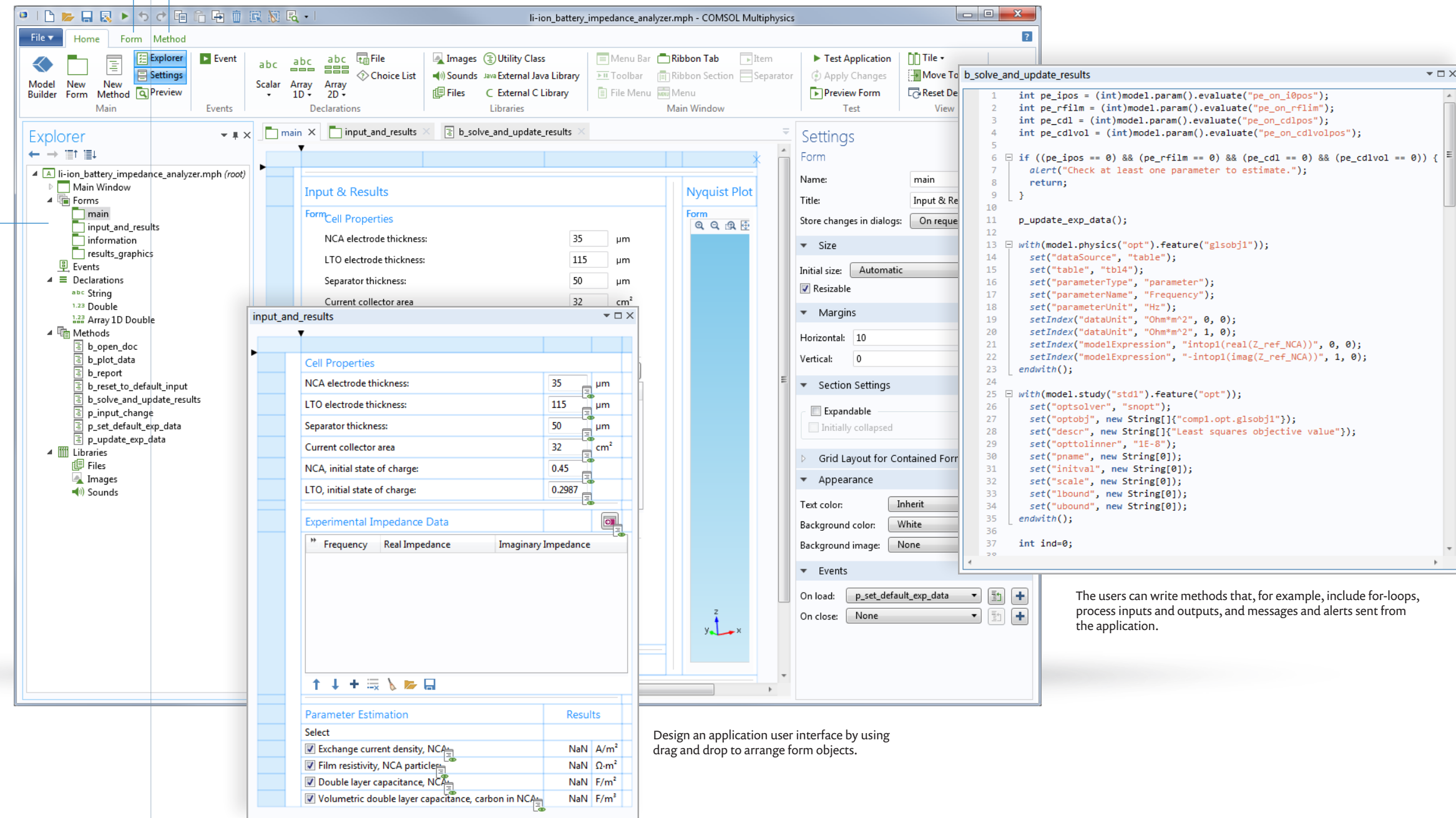
APPLICATION TREE
The application tree gives an overview of forms, events, declarations, methods, and libraries used in the application.

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.



Design an 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 application, 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. Apps can also be password protected.

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



APPS DISTRIBUTION

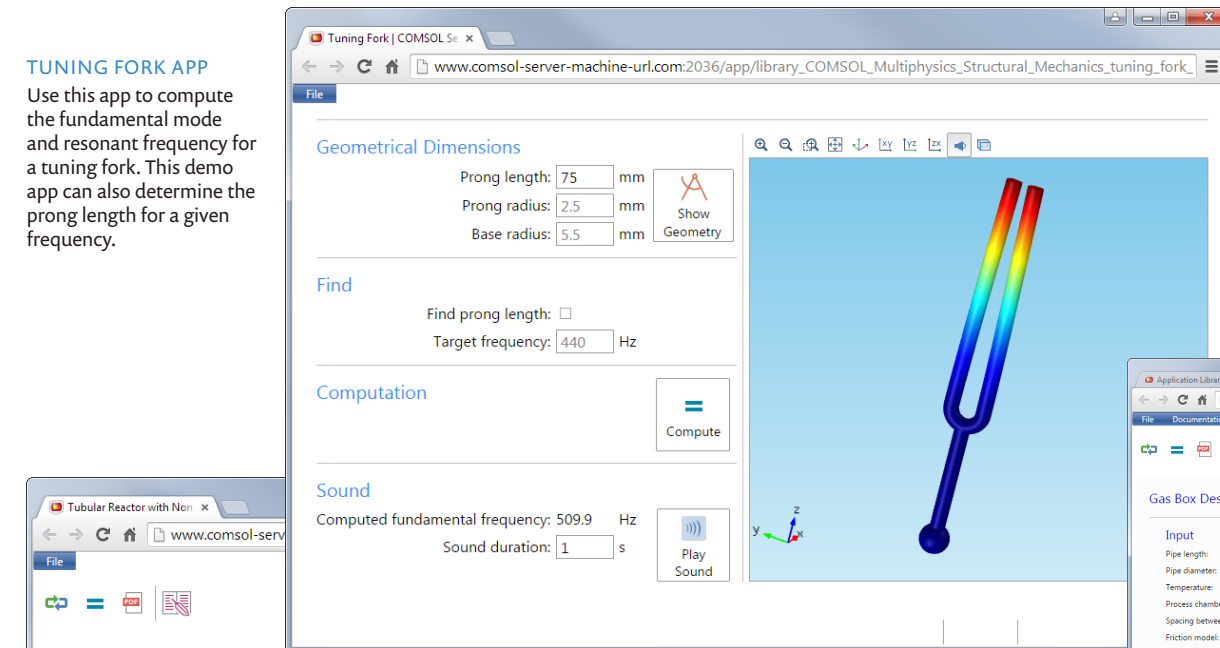
Apps created with the Application Builder can be accessed and used, both within an organization and worldwide, through COMSOL Server™.

“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 APEI employees.”

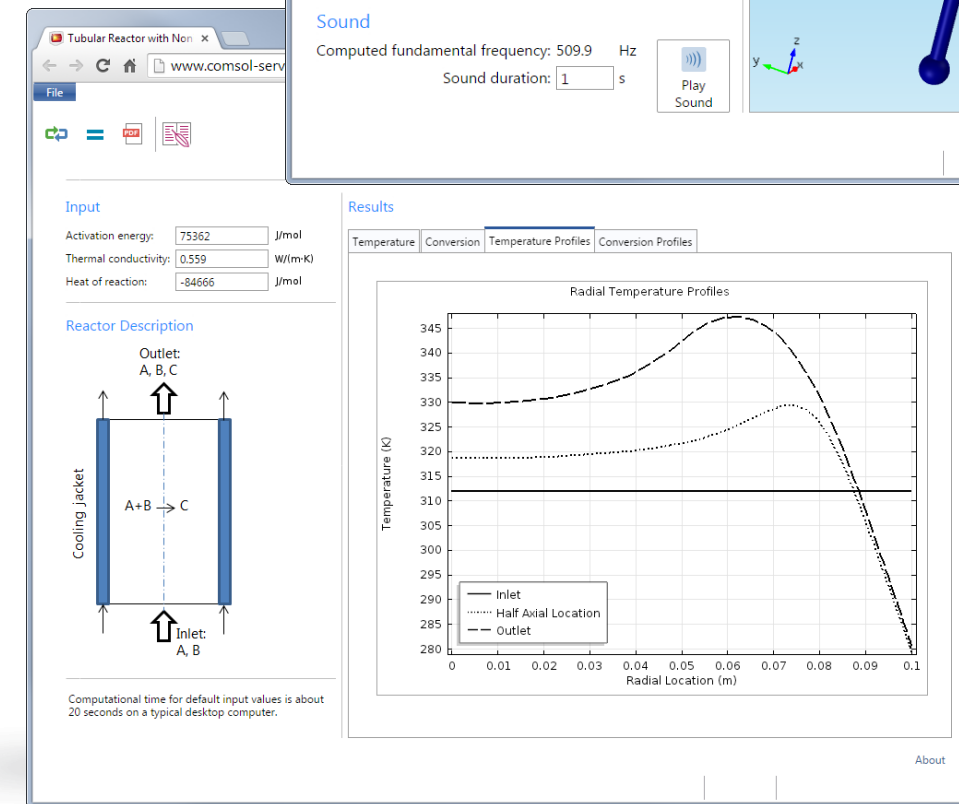
Brice McPherson, APEI, Fayetteville, Arkansas.

TUNING FORK APP

Use this app to compute the fundamental mode and resonant frequency for a tuning fork. This demo app can also determine the prong length for a given frequency.

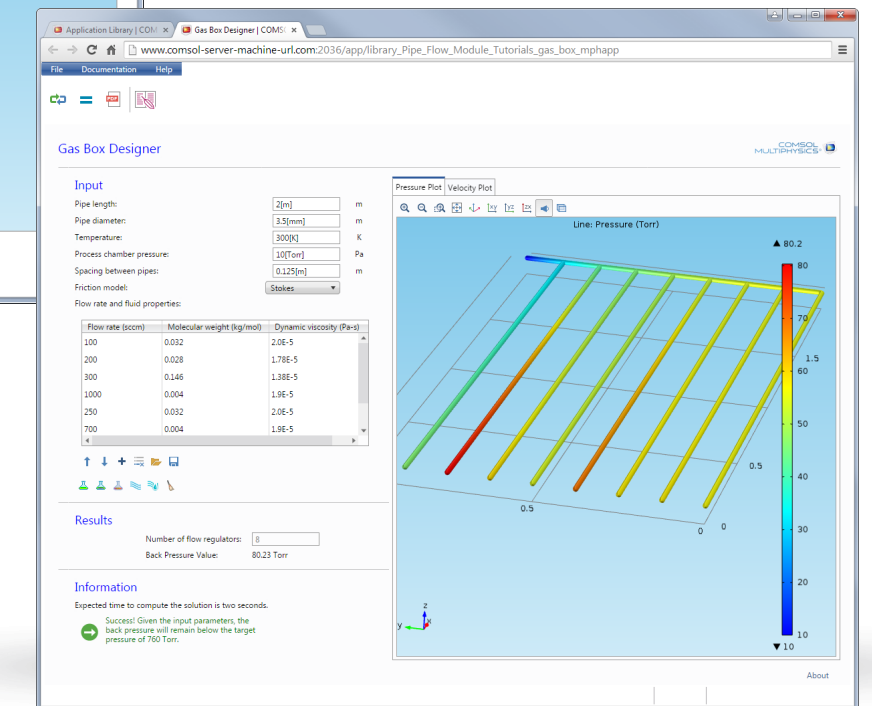


Demo Apps From The Application Library



TUBULAR REACTOR SIMULATOR APP

This app was created using a model of the reaction kinetics in a tubular reactor. A user may alter the reactor properties and boundary conditions in order to analyze the temperature and conversion profiles resulting from the exothermic reaction.



GAS BOX DESIGNER APP

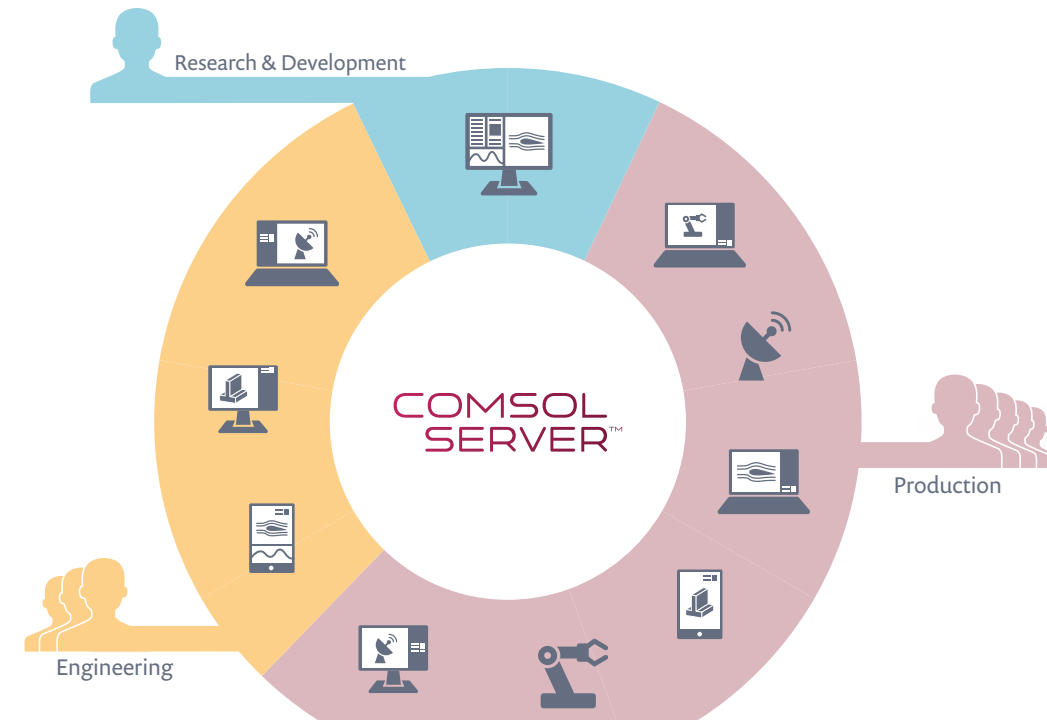
This app computes the back pressure in a network of pipes fed by a system of mass flow controllers (MFC). Each MFC supplies a given flow rate to one of the pipes, and can only function correctly if the back pressure remains below 760 Torr. Users can obtain results for chosen pipe dimensions, boundary conditions, and fluid models.

COMSOL Server™

COMSOL Server™ is the hub for running COMSOL applications and controlling their deployment, distribution, and use. After creating simulation apps in the Application Builder included in COMSOL Multiphysics®, the 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.

The server can be hosted in a corporate network or in the cloud enabling design teams, production departments, and others to share apps. Simulation applications can be run in a COMSOL Client for Windows® or in Google Chrome™, Firefox®, Internet Explorer®, Safari®, and other major web browsers.

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.



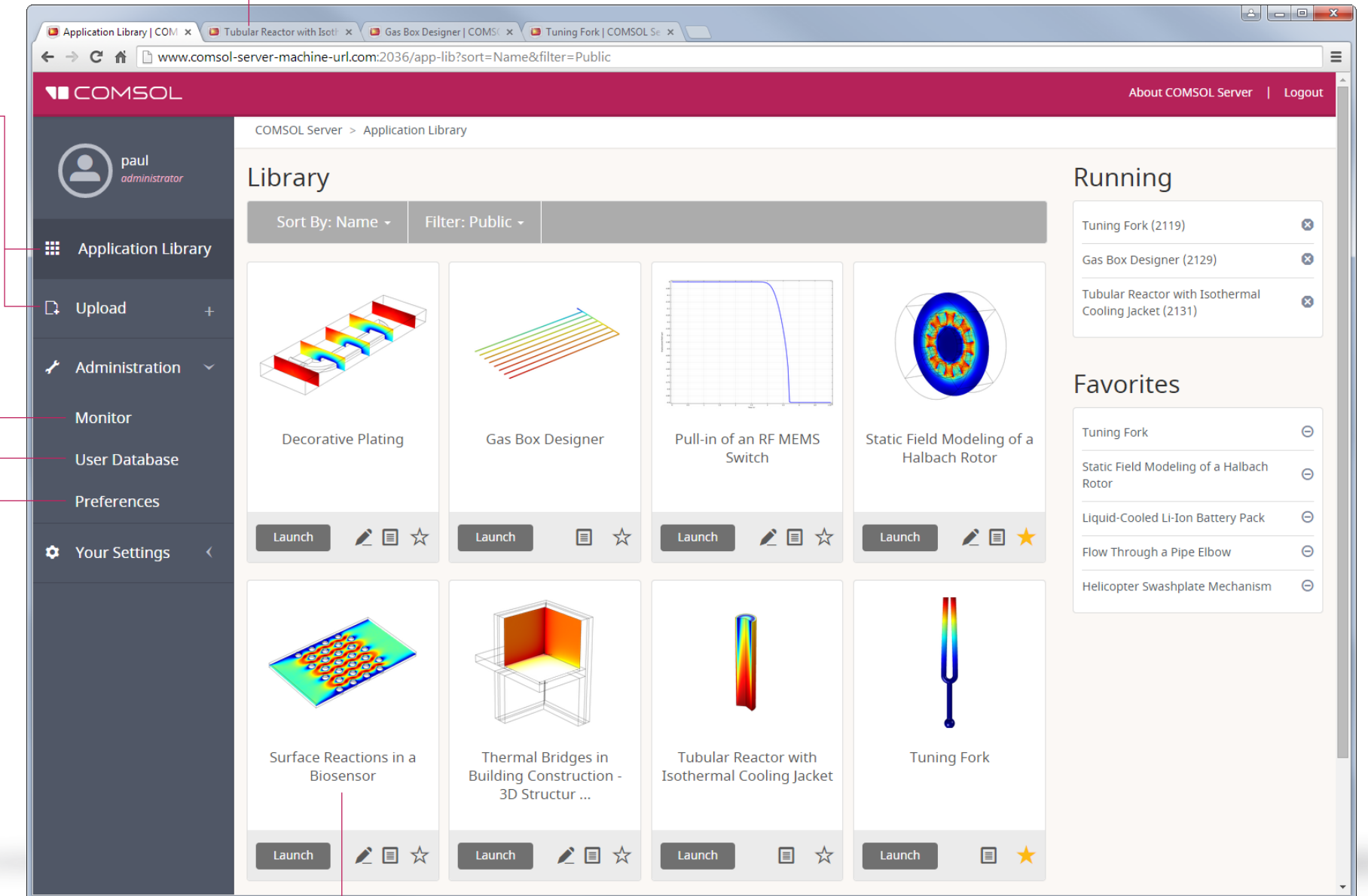
APPLICATION LIBRARY
Upload and display the list of applications that can be run.

MONITOR
Find information about start time and uptime, CPU and memory use, and an overview of running application 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 APPS
Applications can be run in separate tabs and windows.

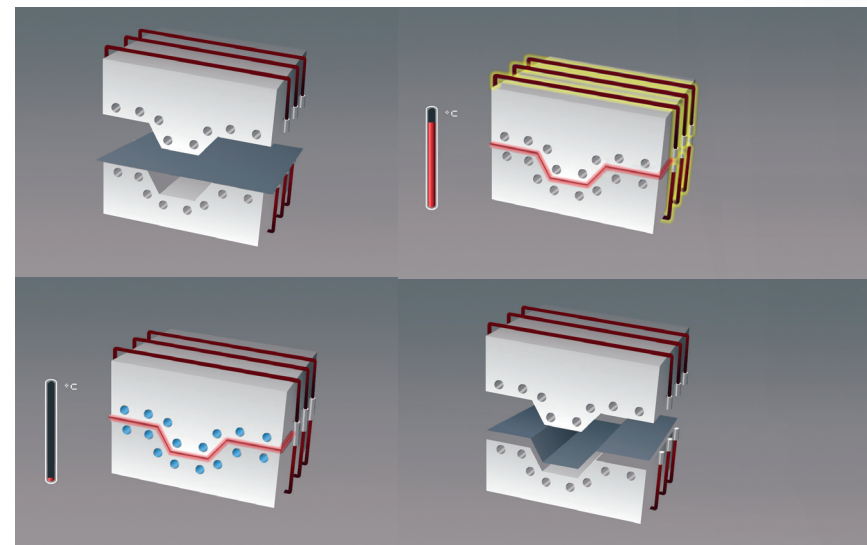


APP CARD
Edit, show general information, favorite, and launch an app.

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.

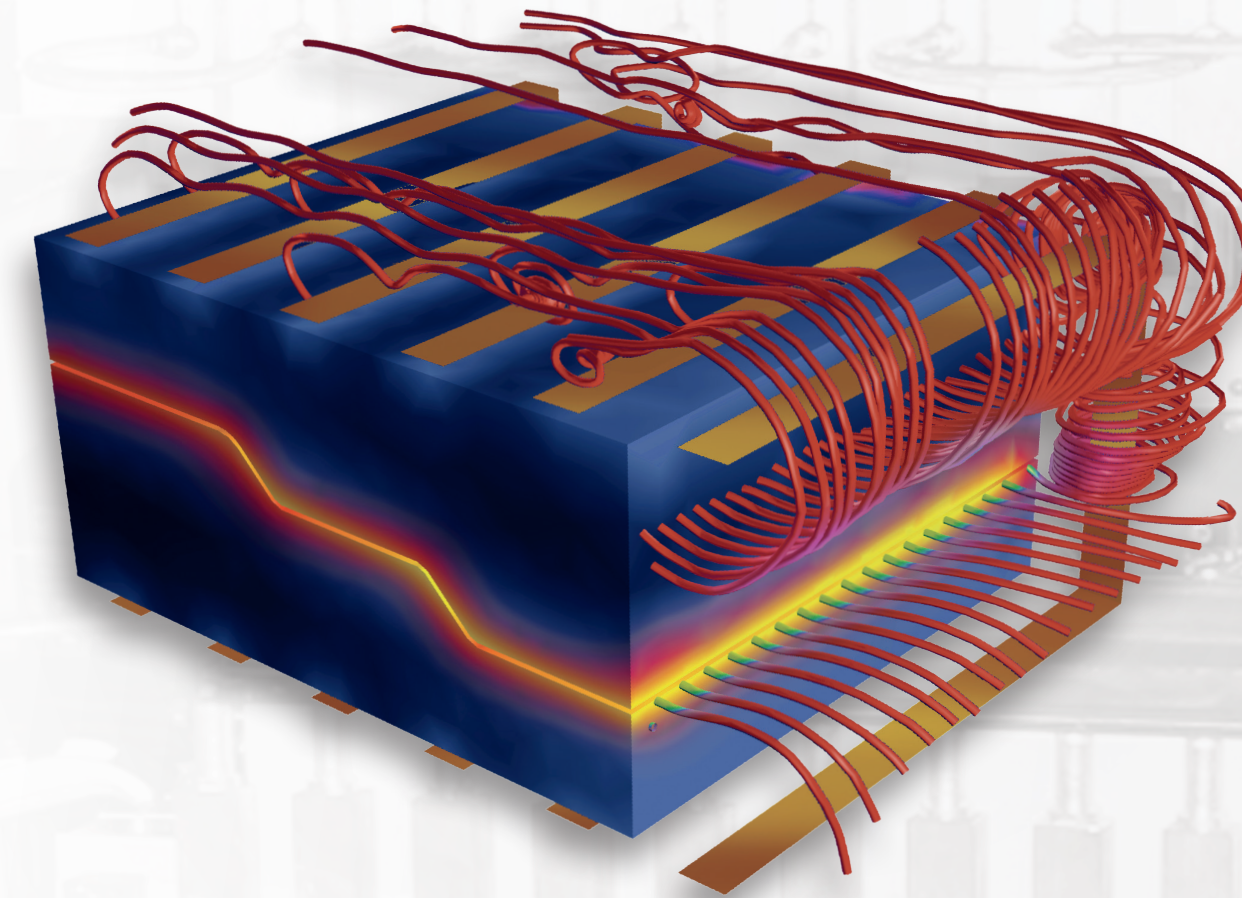
The capabilities of the AC/DC Module span electrostatics, magnetostatics, and electromagnetic quasi-statics with access to any derived field quantities and unlimited couplings to other physics. When considering your electrical components as part of a larger system, the AC/DC Module provides an interface 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.



COMPOSITES MANUFACTURING

A sheet of composite material is placed within a mold that presses down on the material while induction currents heat the two surfaces of the mold. When the final shape has been taken, water flows through pipes to cool down the material, which is then released.

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

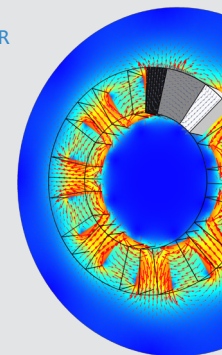


INDUCTION HEATING

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

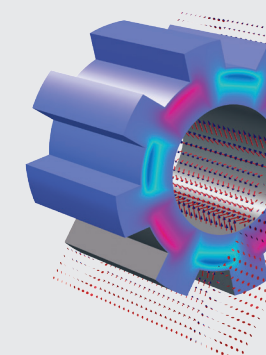
HALBACH ROTOR

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



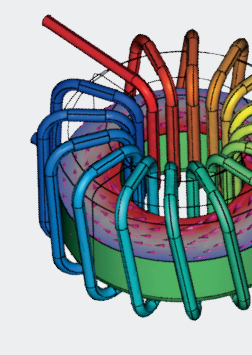
POWER ENGINEERING

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.



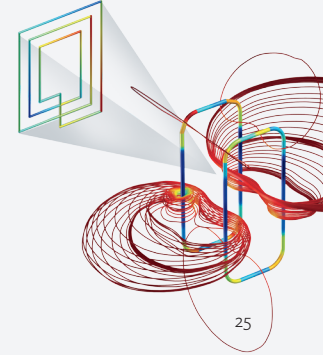
ELECTROMAGNETIC INDUCTOR

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.



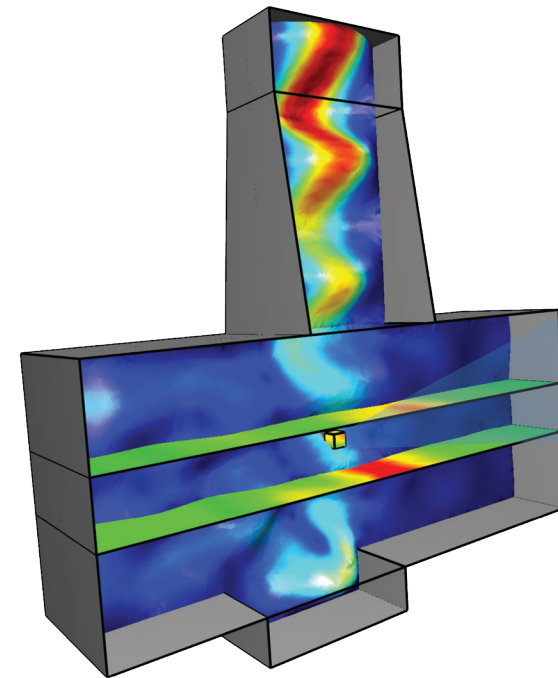
HIGHLIGHTS

- AC/DC current and field distribution
- Bioheating
- Coils and solenoids
- Combined SPICE circuit and field simulations
- Contact resistance
- Electromagnetic compatibility (EMC) and interference (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

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.



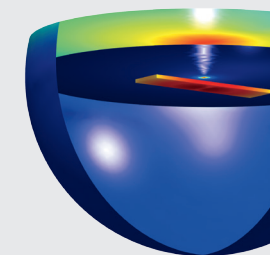
MINERAL PROCESSING

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.

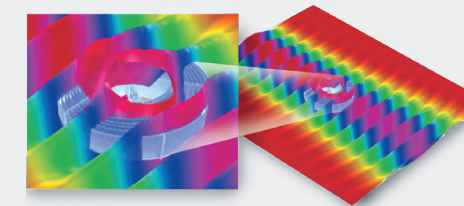
OIL PROSPECTING

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



MATERIALS SCIENCE

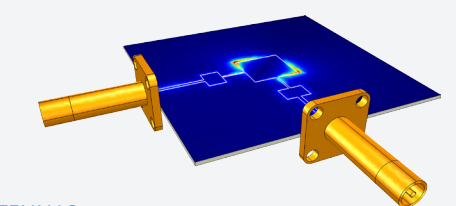
Electromagnetic metamaterials have a spatial resolution that is below that of an electromagnetic wavelength, so as to create a form of 'invisibility' over a narrow wave band. In this model, an object located in the center of this structure is invisible to microwave radiation.



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



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.

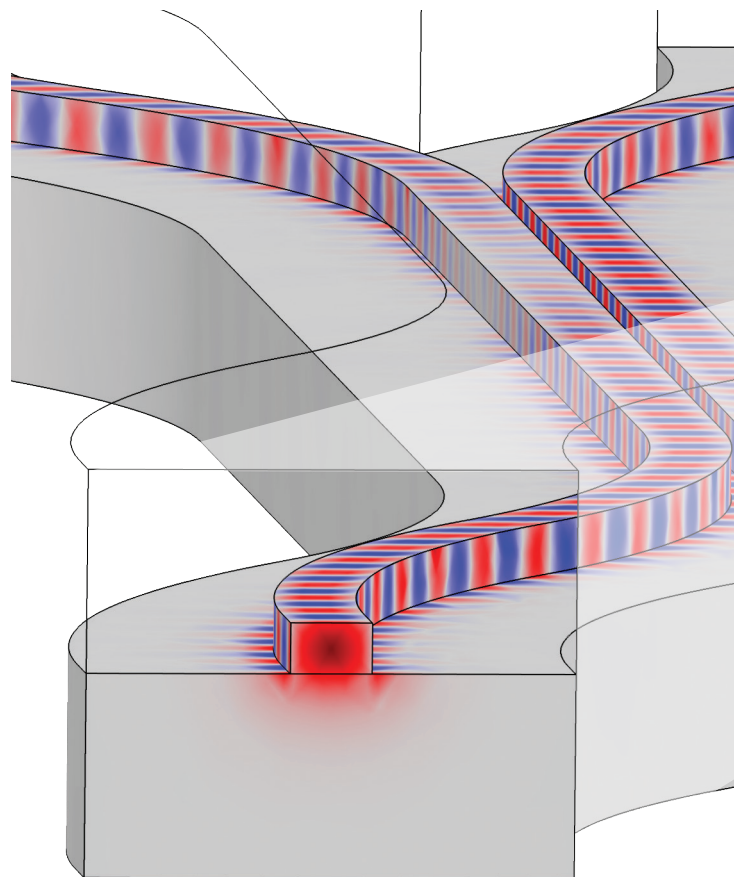
Wave Optics Module

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 innovative 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 in the Wave Optics Module for eigenfrequency mode analysis, frequency-domain, and time-domain electromagnetic simulation.

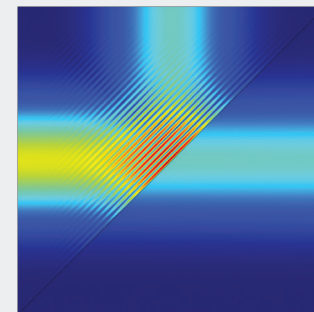
DIRECTIONAL COUPLER

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.



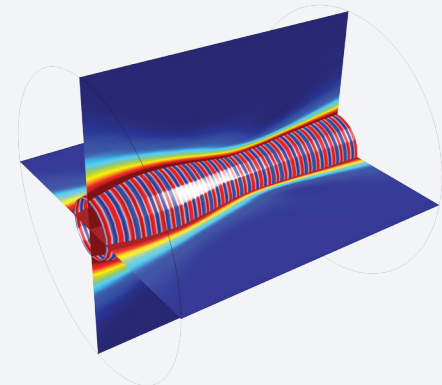
BEAM SPLITTER

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 a 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 design of high-power laser systems.



HIGHLIGHTS

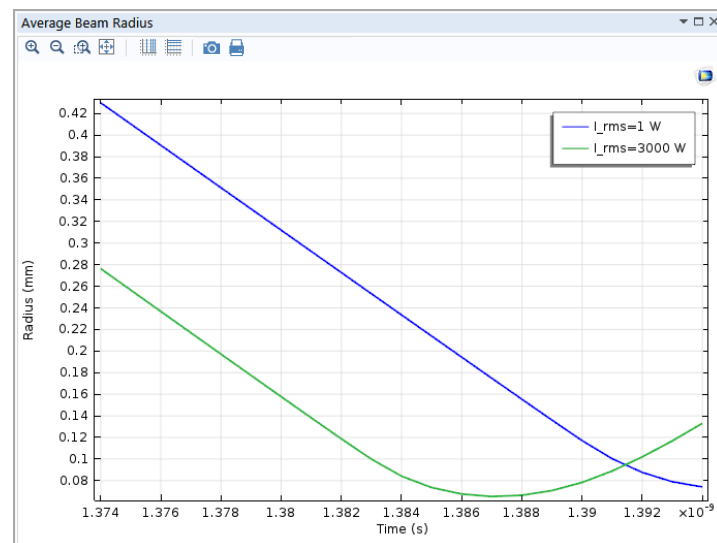
- Couplers
- Fiber Bragg gratings
- Fiber optics
- Harmonic generation and frequency mixing
- Integrated optics
- Lasers and amplifiers
- Laser heating
- Metamaterials
- Nonlinear optics
- Optical lithography
- Optical scattering
- Optical sensors
- Optoelectronics
- Photonic crystal fibers
- Photonic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

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.

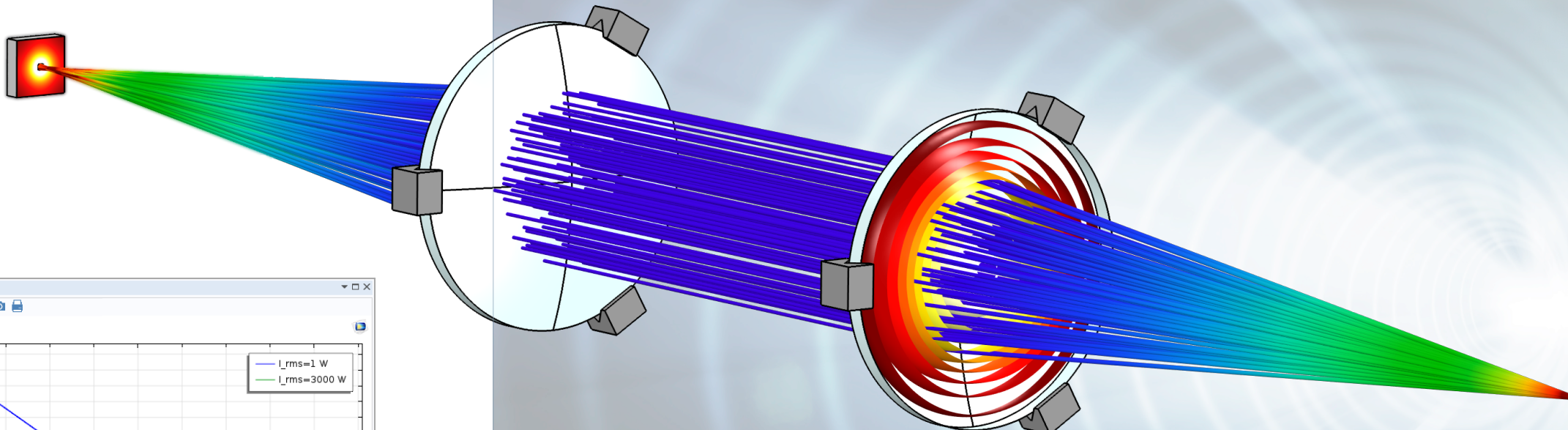
Rays can also undergo reflection and refraction at boundaries between different media. The Ray Optics Module contains a variety of boundary conditions, including combinations of specular and diffuse reflection. Rays can be released from within domains, from boundaries, or at a uniform grid of points. 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.



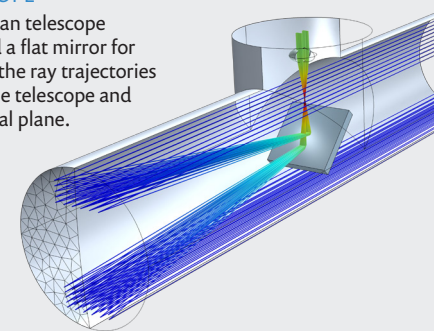
THERMALLY INDUCED FOCAL SHIFT

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.



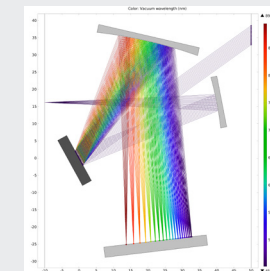
NEWTONIAN TELESCOPE

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.



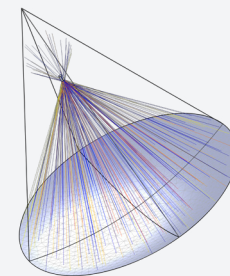
CZERNY-TURNER MONOCHROMATOR

A Czerny-Turner monochromator used for separating polychromatic light into monochromatic rays. Simulation results show the trajectories and wavelengths in the optical signal.



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.



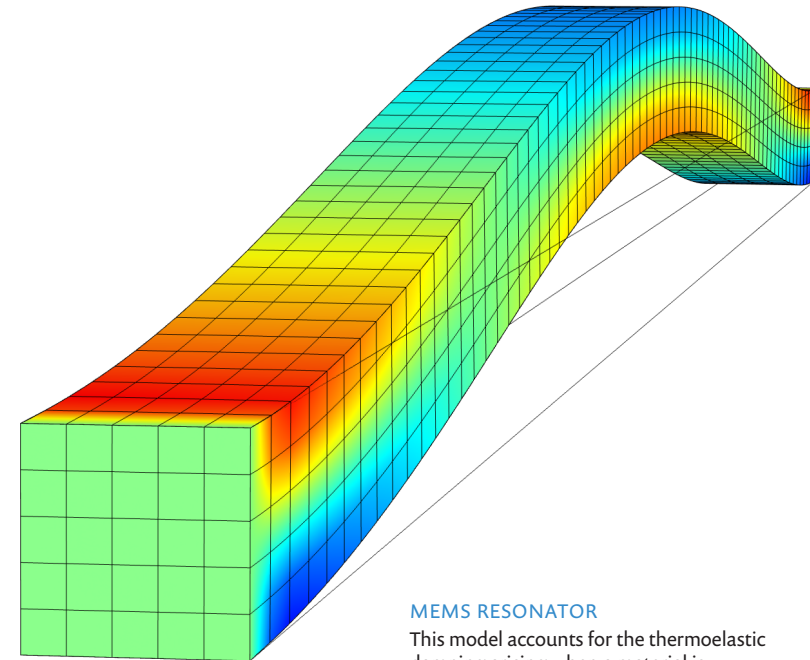
HIGHLIGHTS

- 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 coherent radiation

MEMS Module

The design and modeling of microscale electro-mechanical 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.



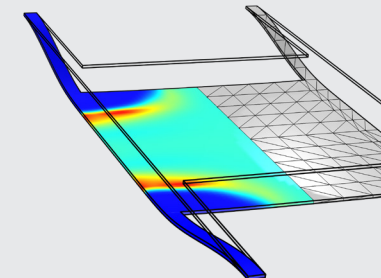
MEMS RESONATOR
This model accounts for the thermoelastic damping arising when a material is subjected to cyclic stress. Results show the mode shape and the corresponding temperature distribution within the beam when the resonator is operating at an eigenfrequency of 63.3 kHz. The computed quality factor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

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.

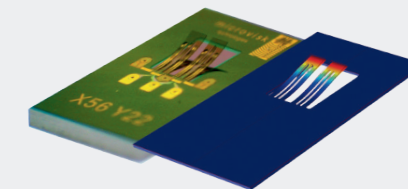
PULL-IN ANALYSIS OF AN RF MEMS SWITCH

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



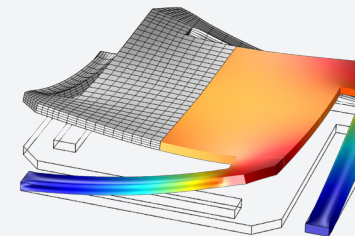
ACTUATORS & SENSORS

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 micro-cantilevers.



PRESTRESSED MICROMIRROR

The lift-off of this prestressed plated micromirror is controlled electrostatically. Results from a parametric study reveal how the prestress affect the displacements.



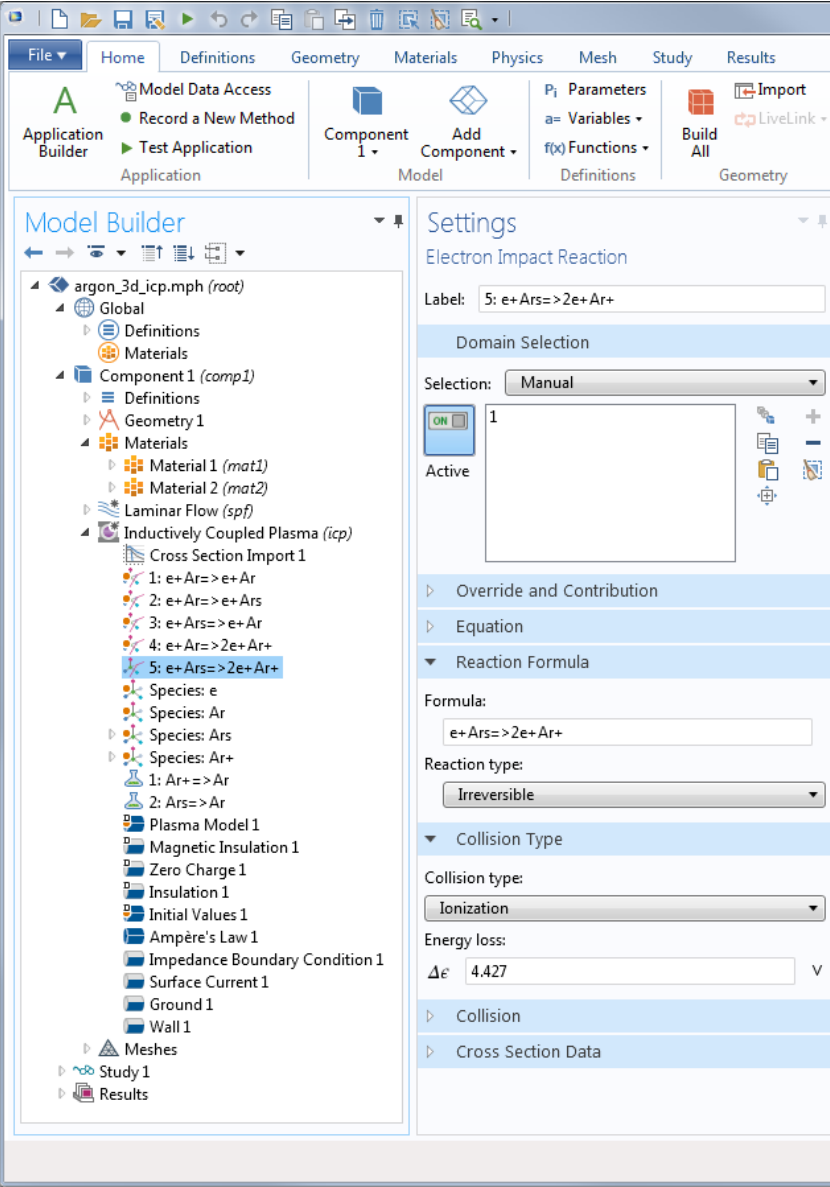
HIGHLIGHTS

- Accelerometers
- Actuators
- Anchor damping
- Bulk Acoustic Wave (BAW) devices
- Cantilever beams
- Fluid-structure interaction (FSI)
- Loss factors
- MEMS capacitors
- MEMS gyroscopes
- MEMS resonators
- MEMS 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

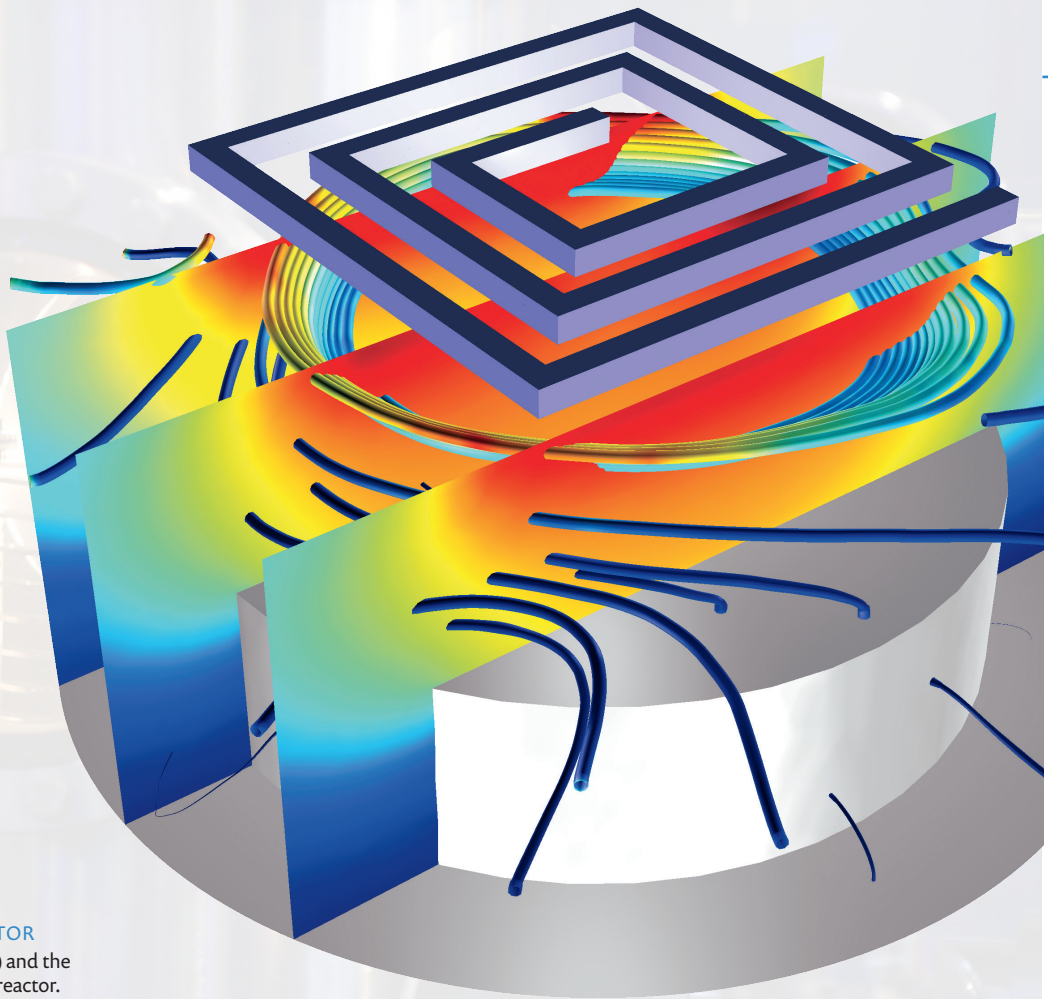
Plasma Module

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 non-equilibrium 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.



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.



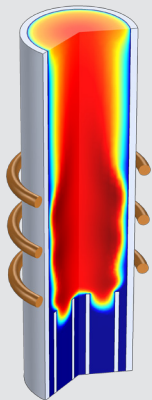
HIGHLIGHTS

- 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

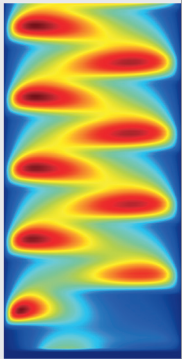
ICP TORCH

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



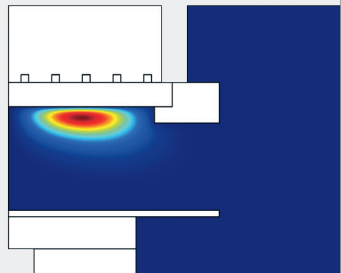
DIELECTRIC BARRIER DISCHARGE

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.



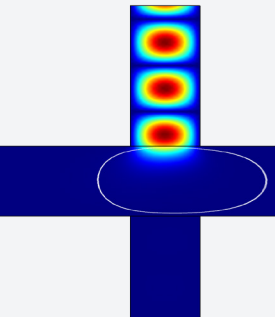
INDUCTIVELY COUPLED PLASMA

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

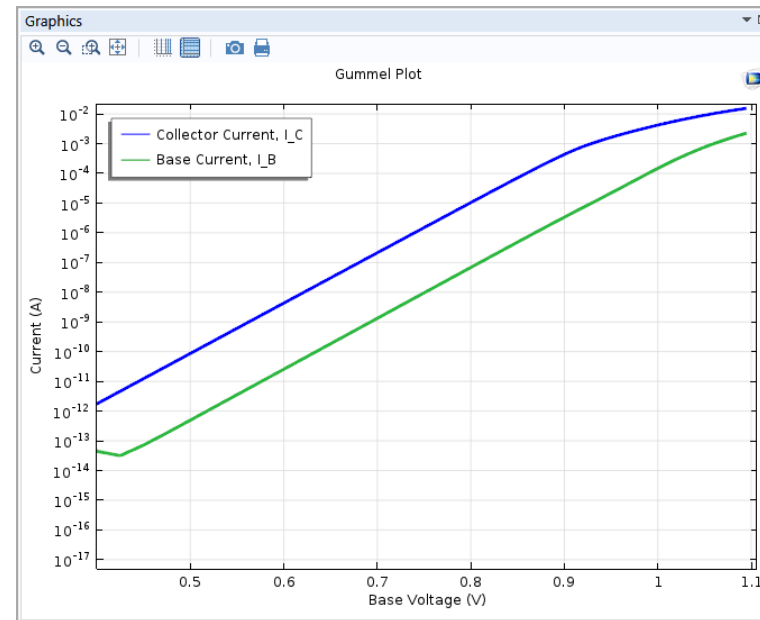
The surface color shows the magnitude of the high frequency electric field as it propagates down a waveguide and into a plasma. The contour indicates where the electron density is equal to the critical plasma density.



Semiconductor Module

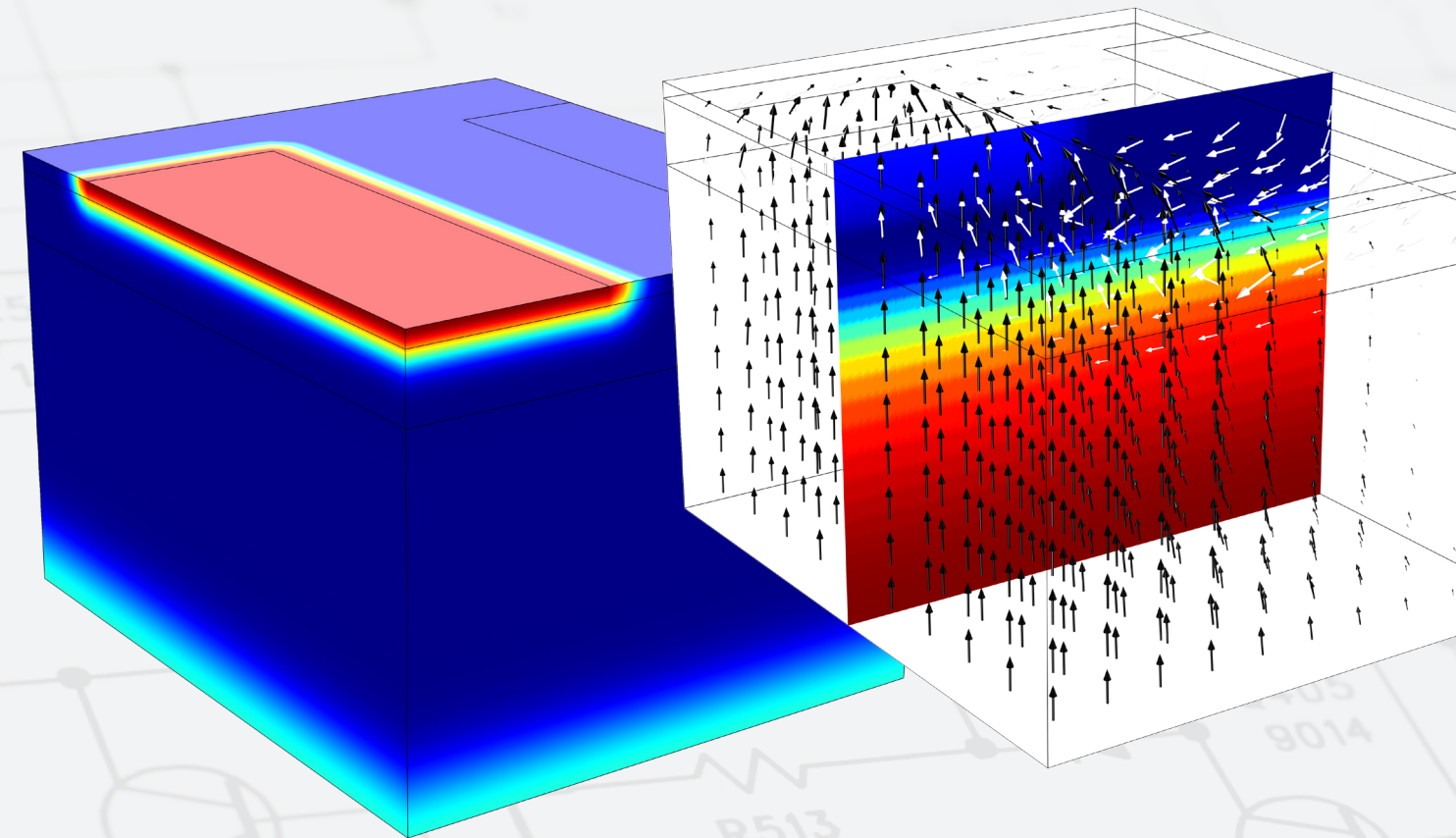
The Semiconductor Module allows for detailed analysis of semiconductor device operation at the fundamental physics level. The module is based on the drift-diffusion equations with both isothermal and non-isothermal transport models. A straightforward but powerful interface makes it easy to set up doping 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 pre-defined 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 that can be 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 metal contacts (both Schottky and ohmic contacts), gates, surface traps and Fowler-Nordheim tunneling to be modeled.



3D ANALYSIS OF A BIPOLAR TRANSISTOR

The device is simulated while operating in the common-emitter regime. Gummel plot, total net dopant concentration, and voltage and current density for a base current of 2 microamperes and collector-emitter voltage of 1.5 volts are shown.

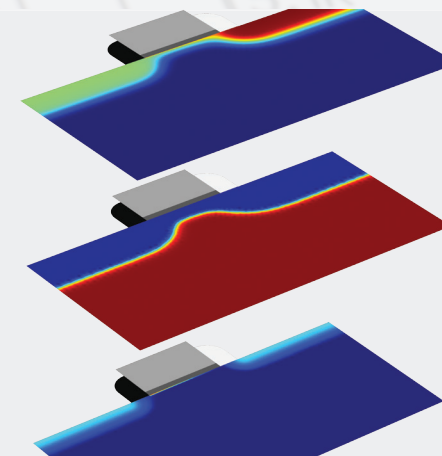


HIGHLIGHTS

- Bipolar transistors
- Bulk lasers
- EEPROM memory
- 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

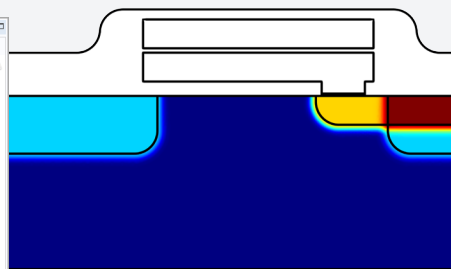
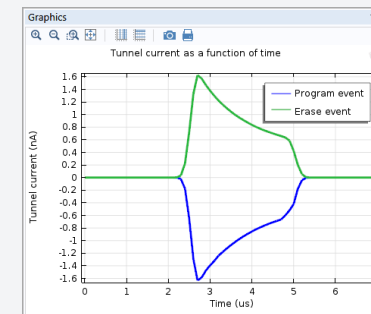
SEMICONDUCTOR DESIGN

2D model of a MOSFET transistor. Simulation results demonstrate the transistor operation where an applied gate voltage turns the device on and then the drain saturation current is 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 device. Tunnel current as a function of time throughout the program and erase events, and signed dopant concentration are shown.

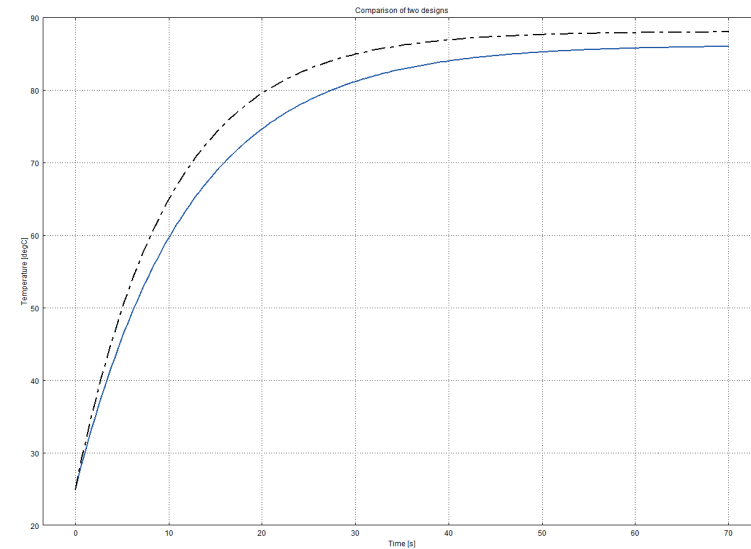


Heat Transfer Module

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, process design, phase change modeling, radiative heat transfer through both transparent and semi-transparent 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.



TEMPERATURE SENSOR

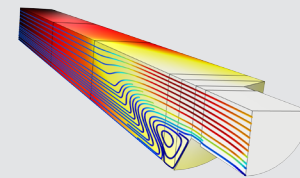
Top: Graph comparing temperature versus response time for two different sensor cap designs. Left: Photograph and model plot of the temperature distribution in a temperature sensor and on the sensing element inside the cap. An X-ray picture of the device is also shown.



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

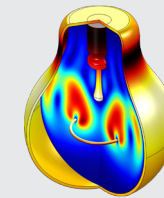
HIGHLIGHTS

- 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
- Friction stir welding
- Furnace and burner design
- Heat and moisture transport
- Heat exchangers and cooling flanges
- Heat transfer in porous media
- Laser welding and laser heating
- Material heat treatment
- Power electronics and electronic cooling
- Radiative heat transfer
- Resistive and induction heating
- Thermal contact and friction
- Thermal performance of constructions
- Thermoelectric effect



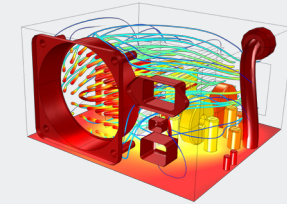
CONTINUOUS CASTING

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



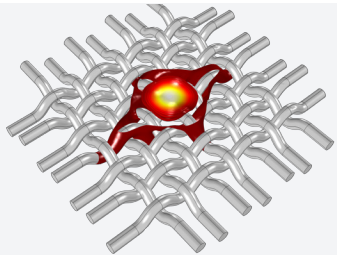
LIGHT BULB

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



POWER SUPPLY COOLING

Simulation of the thermal behavior of an enclosed computer power supply unit.



CARBON FIBERS

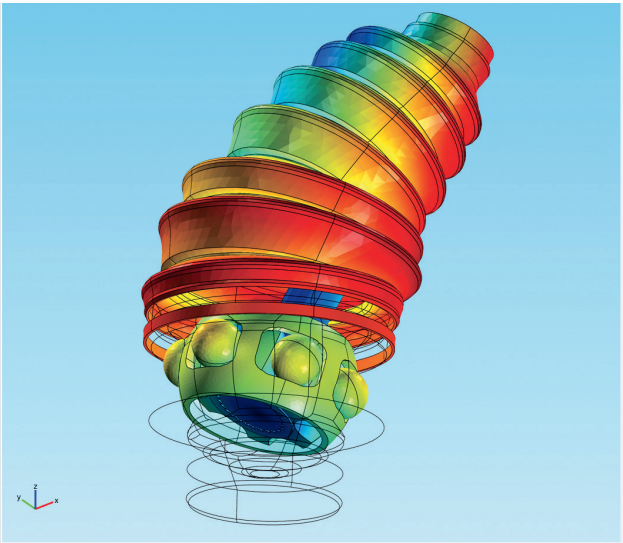
Heat transfer analysis of thermally anisotropic carbon fibers.

Structural Mechanics Module

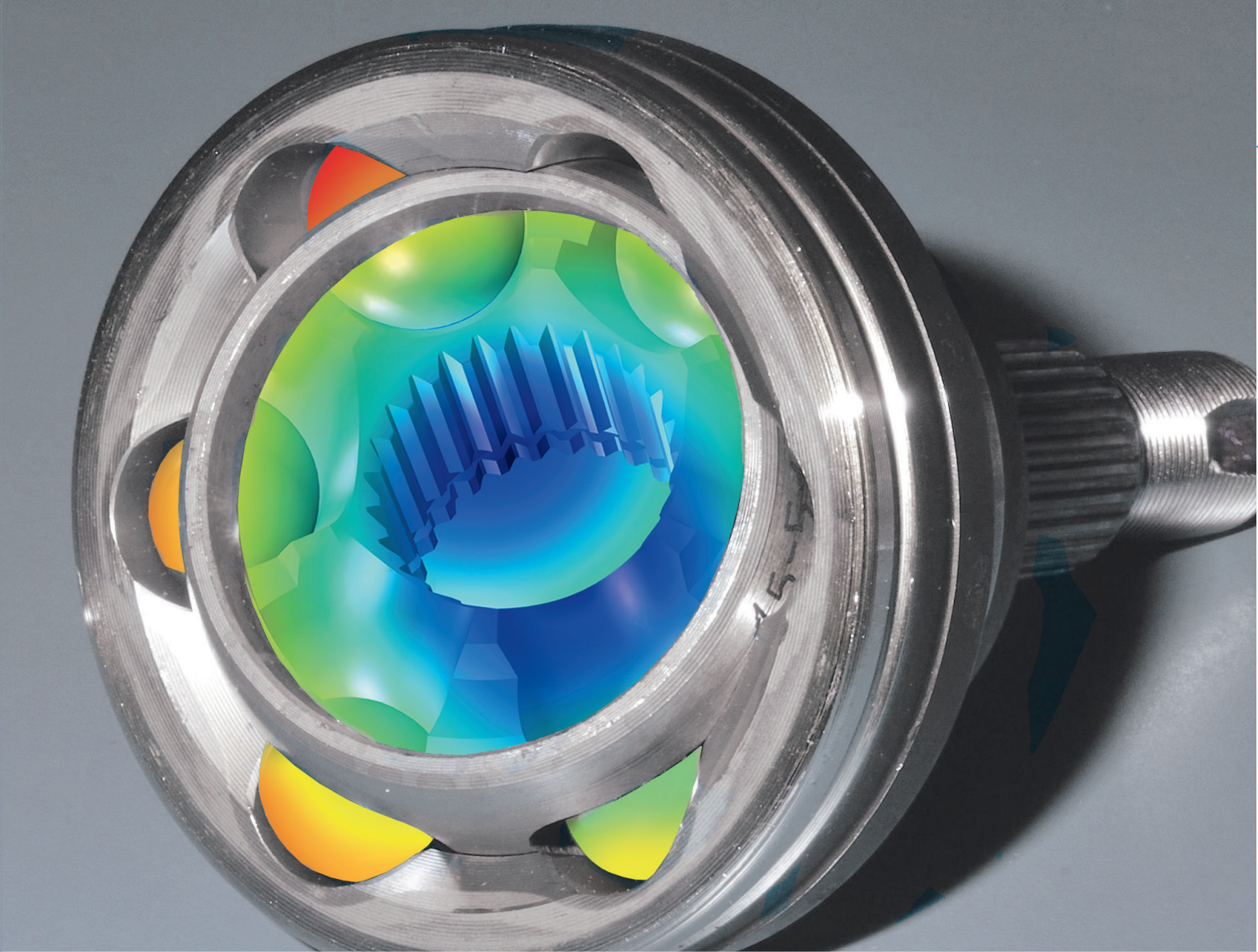
The Structural Mechanics Module is dedicated to the analysis of components and subsystems where it is necessary to evaluate deformations under loads. It also contains special user interfaces for the modeling of plates, shells, membranes, beams, and trusses.

The Structural Mechanics Module is used for a wide range of analysis types including: stationary, time-dependent, eigenmode, parametric, quasi-static, frequency-response, and buckling. Its user interfaces allow for large deformation analysis with geometrical nonlinearity and mechanical contact. Predefined multiphysics couplings are available for heat transfer, acoustics, fluid flow, and electrostatics. This allows for the easy set-up 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 into 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.

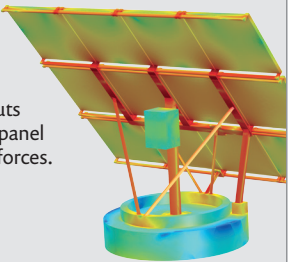


HIGHLIGHTS

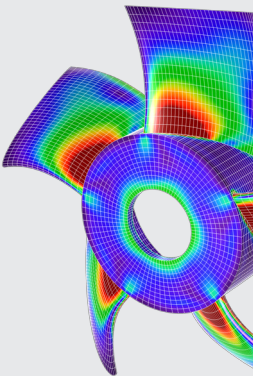
- Bolt pre-tension
- Buckling and postbuckling
- Built-in connections between solids, shells, and beams
- Centrifugal, Coriolis, and Euler forces
- Contact and friction
- Dynamic analysis of prestressed structures
- Elastic waves
- Euler and Timoshenko beams
- Fluid-structure interaction (FSI)
- Geometric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

* Together with the CFD Module

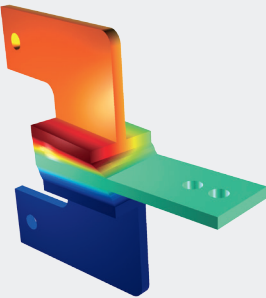
SOLAR PANEL
Stresses on the struts supporting a solar panel subjected to wind forces.



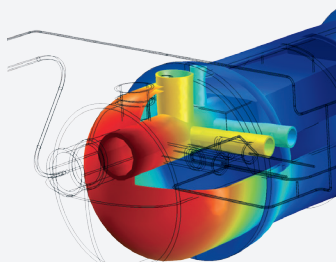
FAN ASSEMBLY
Vibrational analysis of a fan assembly using non-conforming meshes with hanging nodes. Swept meshes are used in all domains, resulting in a mix of hexahedral and prism elements.



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.



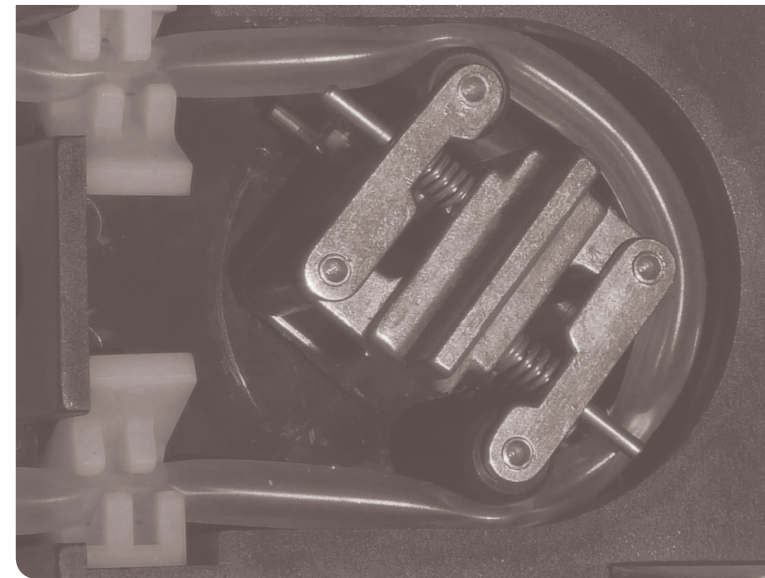
KNOCK-OUT DRUM
Deformations in a knock-out drum connected to a petroleum refinery.
Model courtesy of Stig Sandström, SSM Engineering, Sweden.



Nonlinear Structural Materials Module

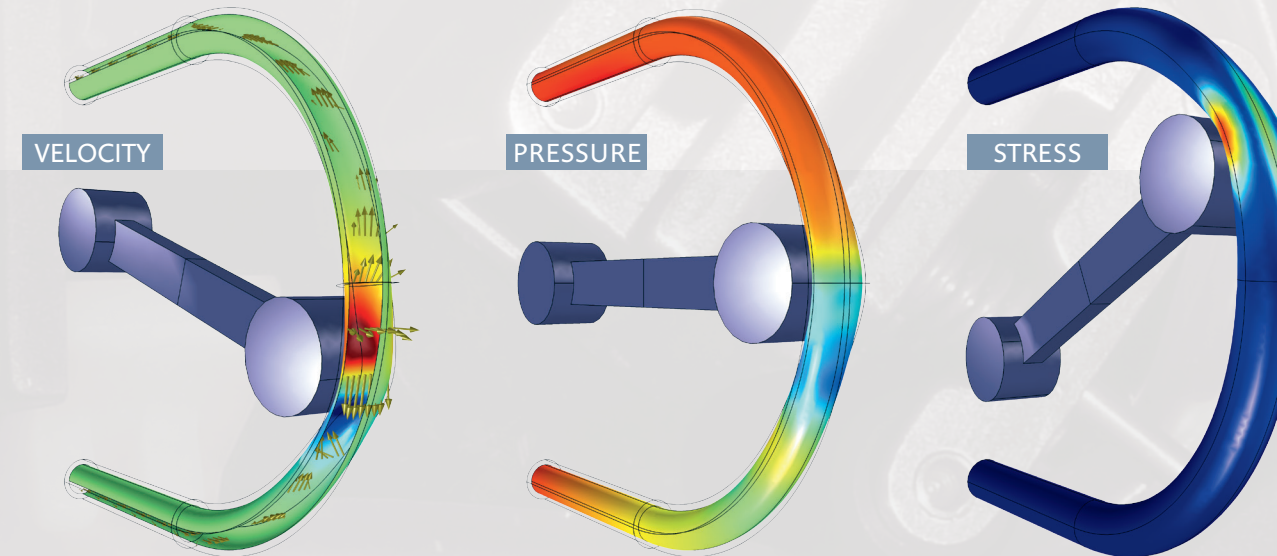
The Nonlinear Structural Materials Module augments the mechanical capabilities of the Structural Mechanics and MEMS Modules by adding nonlinear material models. When the mechanical stress in a structure becomes large, certain nonlinearities in the material properties force the user to abandon linear 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, large strain plastic deformation, and nonlinear elastic material models. 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.



PUMPING

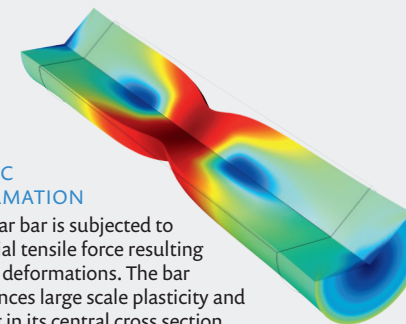
This model of a peristaltic pump accounts for the fluid structure interaction caused by the roller squeezing the tubing's wall and the pressure resulting from the fluid flow within the tubing. The model considers large deformations, contact, and the hyperelastic behavior of the tubing material, modeled 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.

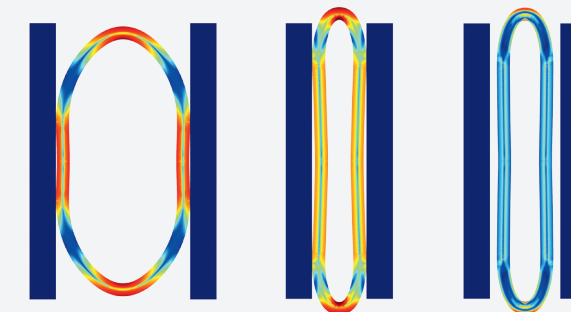
PLASTIC DEFORMATION

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 & large strain plasticity
- Nonlinear elastic models
 - Bilinear elastic
 - Uniaxial data
 - Ramberg-Osgood
 - Power law
- Polymers
- Rubber
- Temperature-dependant plasticity
- Tresca, von Mises, and Hill orthotropic plasticity
- User-defined creep, hyperelasticity, nonlinear elasticity, and plasticity

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 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: Cam-Clay, Drucker-Prager, Mohr-Coulomb, Matsuoka-Nakai, and Lade-Duncan. Other nonlinear elastic material models, such as Ramberg-Osgood; and hyperbolic law models, such as Duncan-Chang, Duncan-Selig, and Hardin-Drnevich soil models are also available.

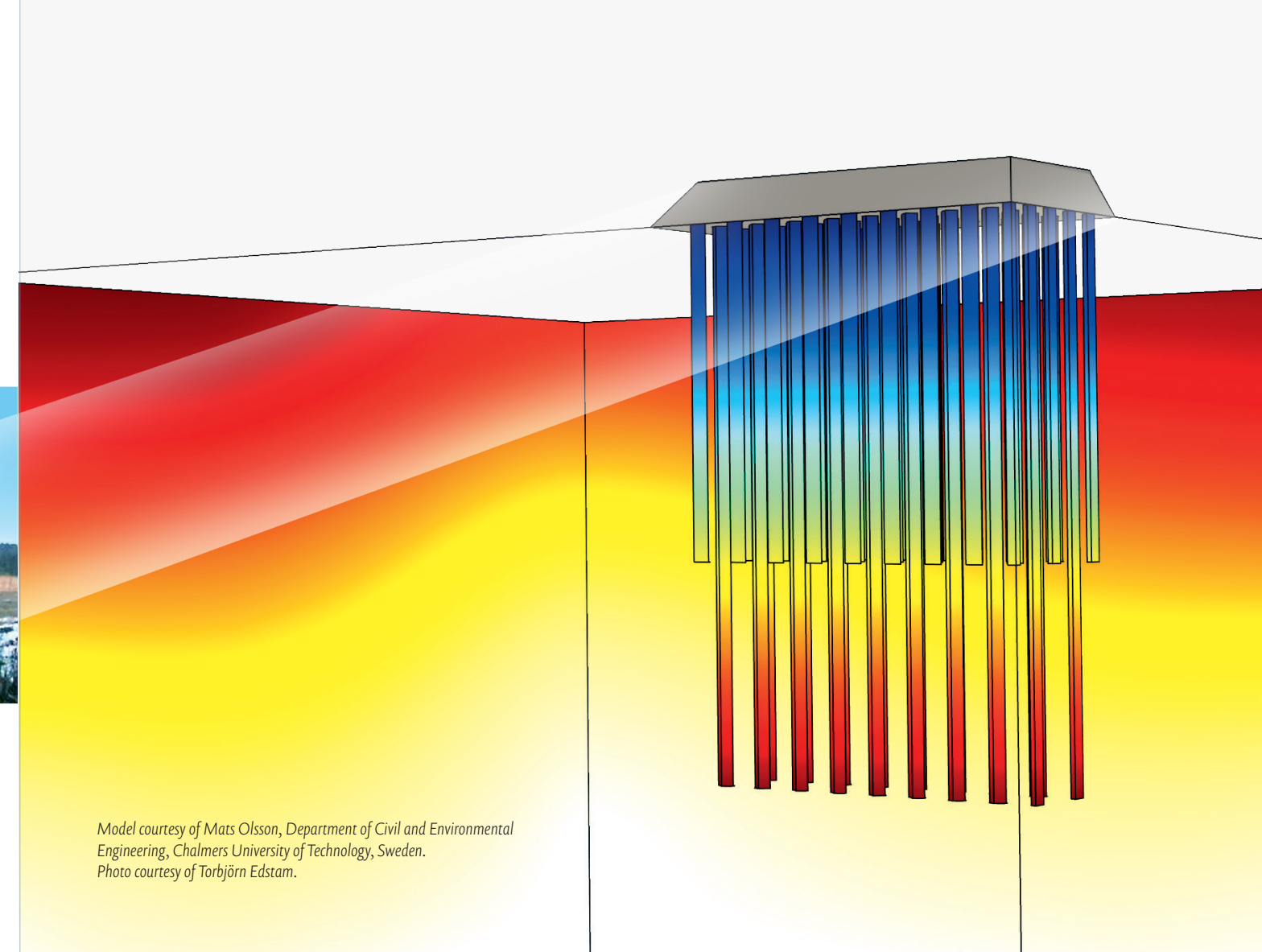
In addition to the built-in plasticity models, user-defined yield functions can be created by the versatile user interface provided by the COMSOL Multiphysics environment. Dependencies of a computed temperature field as well as other field quantities can be blended into these material definitions.

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

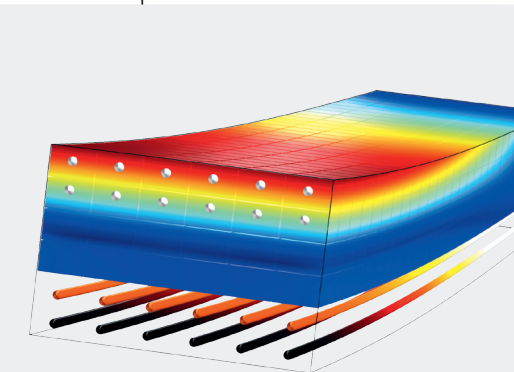
Long-term settlement in clay constitutes an engineering challenge in, for example, road and railway design and construction where supporting lime-cement columns are used. A test site was constructed to evaluate, among other things, the deformation properties of the supporting columns and surrounding 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).



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

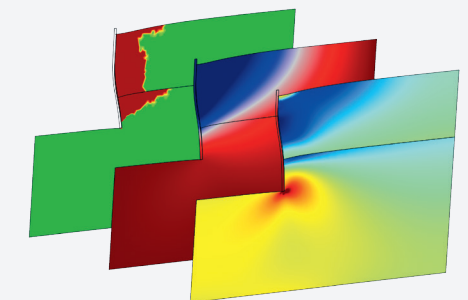
CIVIL ENGINEERING

The simulation shows how force is transferred from a concrete beam to its steel reinforcement bars during tension failure. Shown are the von Mises stresses in the concrete and the axial stresses in the bars.



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.



HIGHLIGHTS

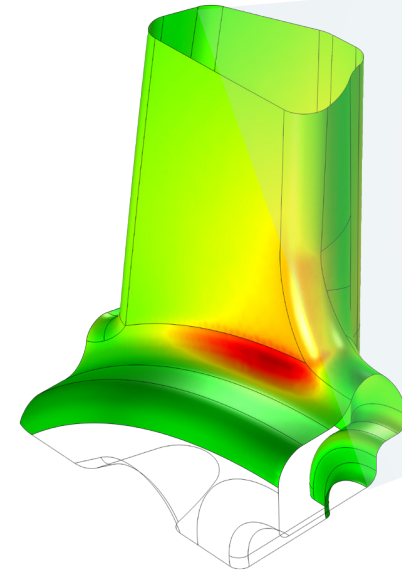
- Bresler-Pister and Ottosen concrete models
- Concrete and brittle materials
- Creep
- Drucker-Prager and Mohr-Coulomb soil models
- Ductile materials and saturated soils
- Embankments
- Excavations
- Foundations
- Hoek-Brown rock model
- Matsuoka-Nakai and Lade-Duncan soil models
- Modified Cam-Clay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

Fatigue Module

When structures are subjected to repeat 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.

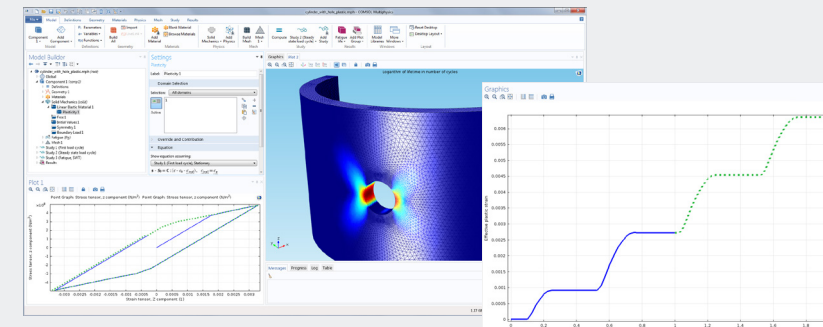
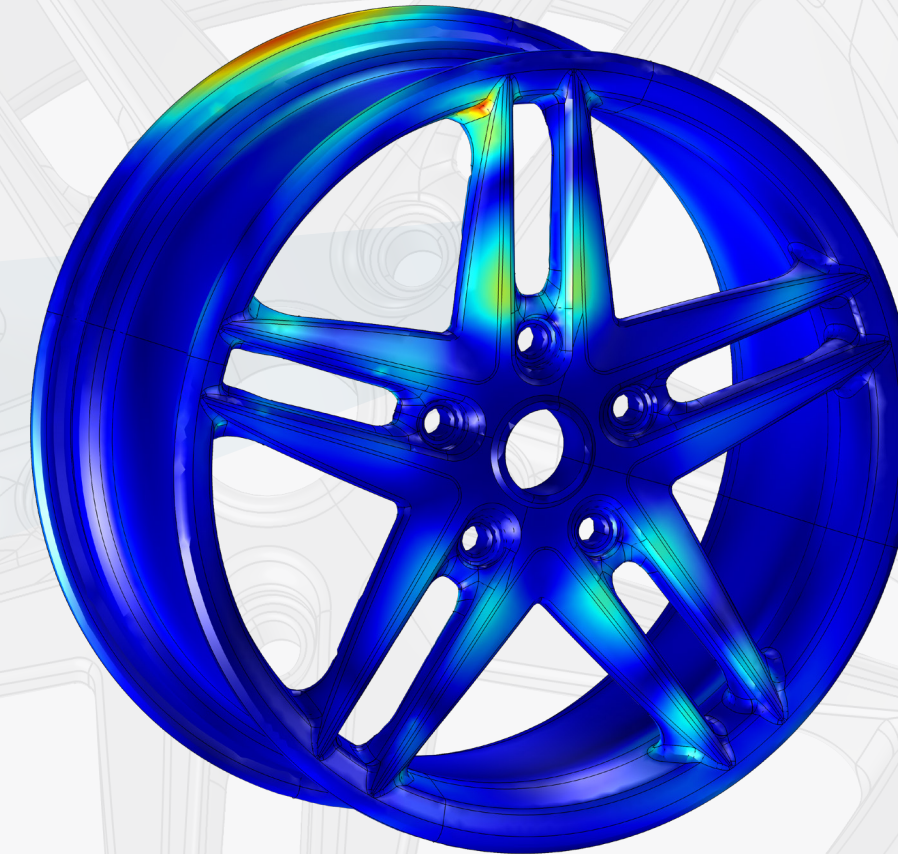
The stress-life and the strain-life models evaluate fatigue in structures subjected to proportional loading, while the stress-based and the strain-based models are suitable in the case of non-proportional loading. With these models you can evaluate fatigue in the high-cycle and low-cycle regime. In applications involving nonlinear materials, you can use energy-based and strain-based methods to simulate 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, multibodies, 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 stress 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).



ELASTOPLASTIC LOW-CYCLE FATIGUE ANALYSIS

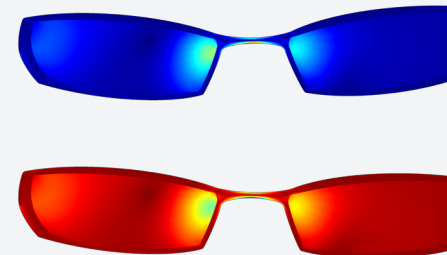
Low-cycle strain-based fatigue resulting from plastic deformation near the hole in a cylinder is simulated. The visualizations (from left to right) show a stress-strain curve for the last two load cycles, a surface plot of lifetime logarithm, and the accumulated effective plastic strain curve.

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
- 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
 - E-N curve criterion
 - Coffin-Manson criterion
 - Combined Basquin & Coffin-Manson criterion
- Stress-based fatigue
 - Findley criterion
 - Mataka criterion
 - Normal stress criterion
- Stress life models
 - S-N curve criterion
 - Basquin criterion
 - Approximate S-N curve criterion

EYEGLASS FRAME FATIGUE

Simulation of metal eyeglass frames with a thin section between the two halves. Results predict strain and fatigue when the eyeglasses are subjected to bending.

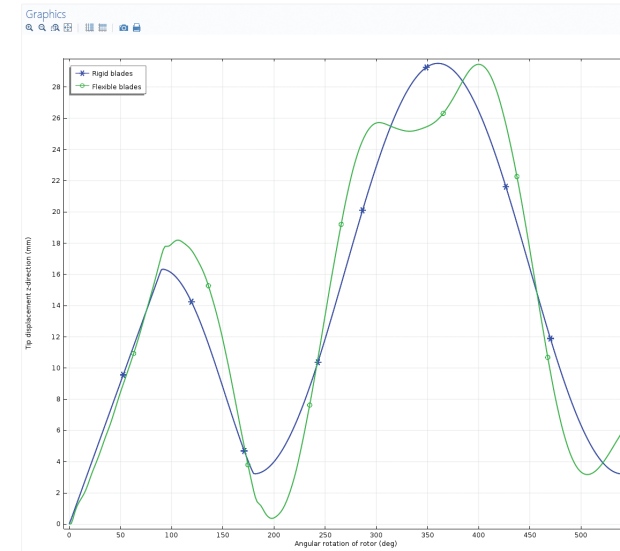


Multibody Dynamics Module

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.

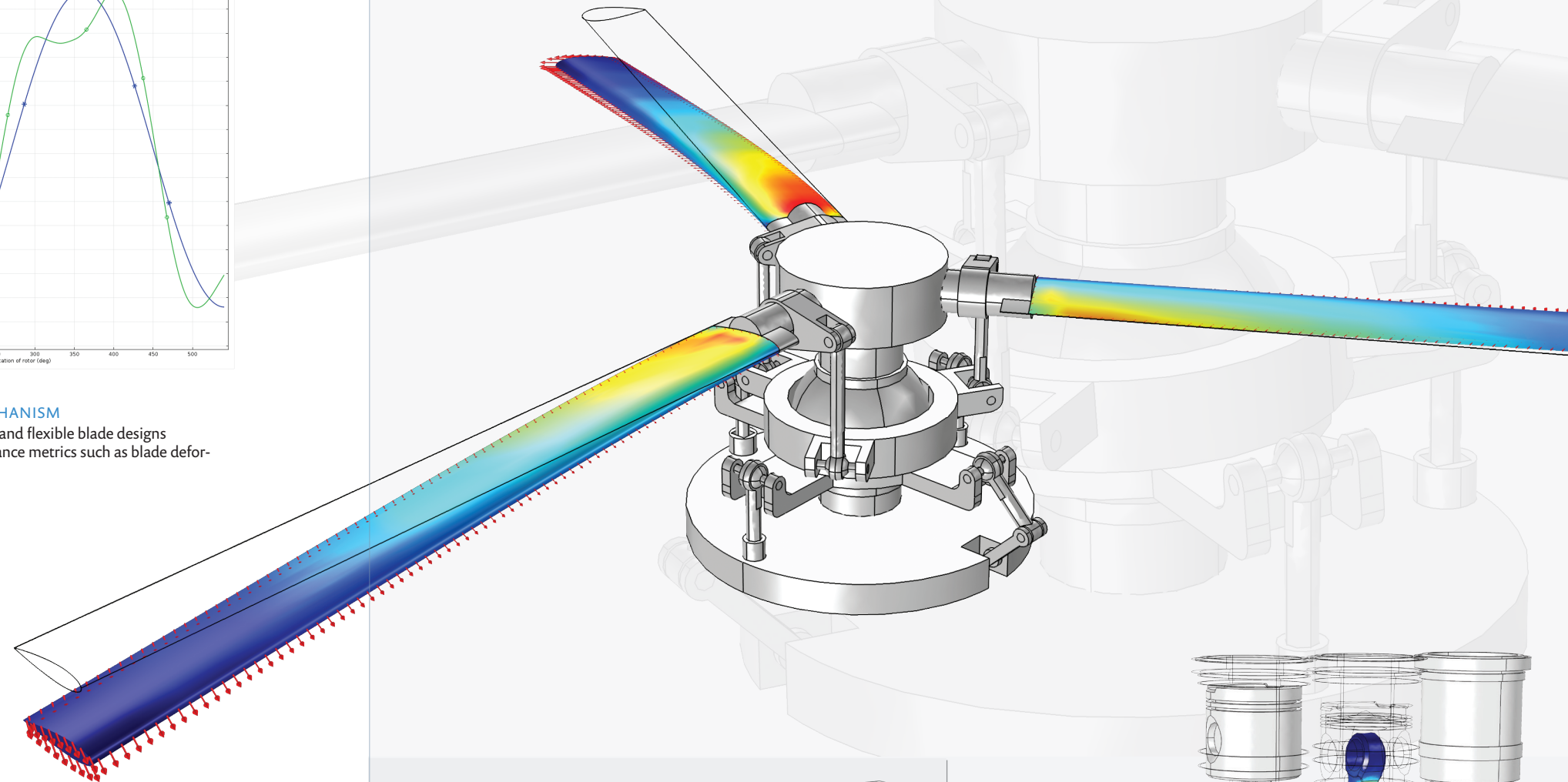
The relative motion of the bodies can be restricted by specifying rotational and translational constraints on joints.

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 these 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 analysis can be performed. Joints can be assigned linear 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.



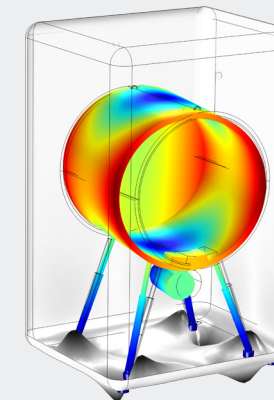
HELICOPTER SWASHPLATE MECHANISM

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



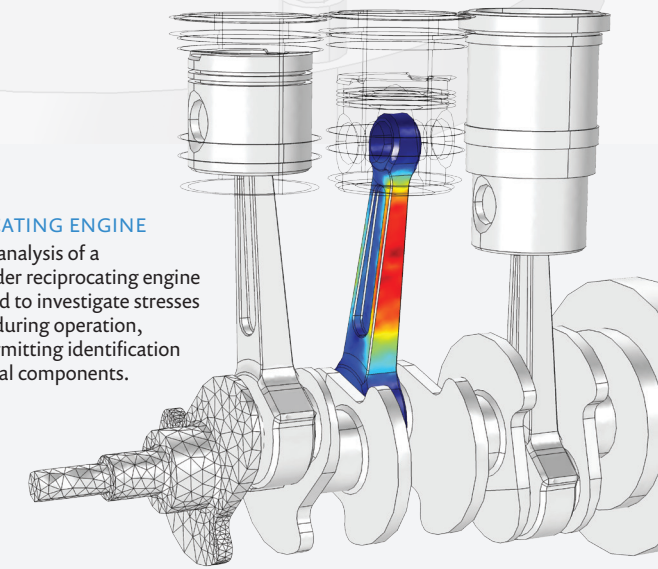
VIBRATION IN A WASHING MACHINE ASSEMBLY

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.



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

Acoustics Module

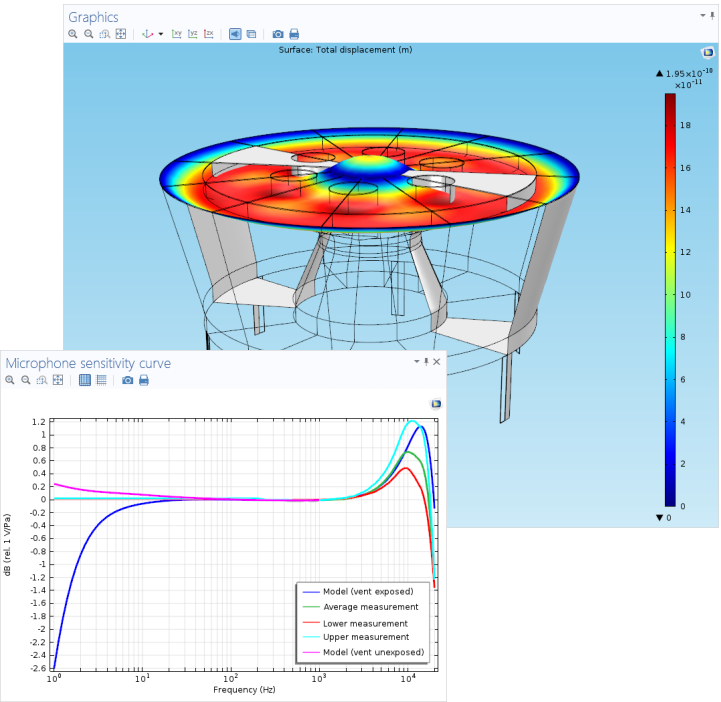
The Acoustics Module is a world-class solution to your acoustics modeling needs. This module is designed specifically for those who work with devices that produce, measure, and utilize acoustic waves. Application areas include speakers, microphones, hearing aids, and sonar devices, as well as noise control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

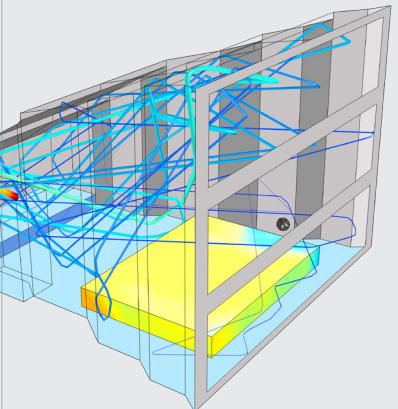
Electromechanical acoustic simulation of a pressure response type microphone for medium and high audio levels and for coupler measurements. Results show the sensitivity level and membrane deformation.

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

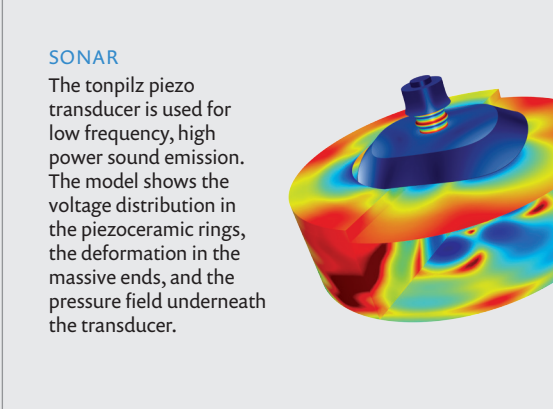


HIGHLIGHTS

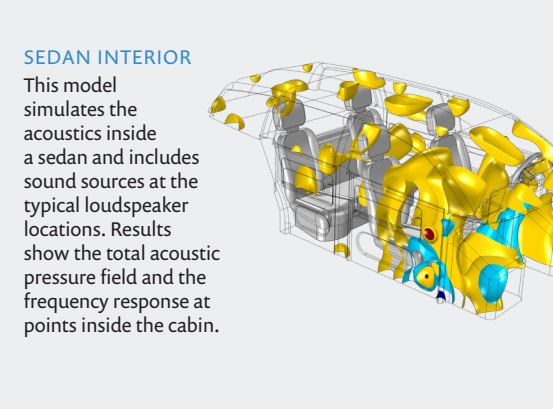
- Acoustic diffusion equation
- Acoustic-structure interaction
- Elastic waves
- Electroacoustic transducers and speakers
- Hearing aids
- Linearized Euler equations
- Linearized Navier-Stokes equations
- Linearized potential flow
- Loudspeakers and microphones
- MEMS acoustics sensors
- MEMS microphones
- Noise and vibration of machinery
- Noise reducing materials and insulation
- Piezoacoustics
- Poroelastic waves
- Ray tracing
- Reactive and absorptive mufflers
- Room acoustics
- SONAR devices
- Structural vibrations
- Thermoacoustics



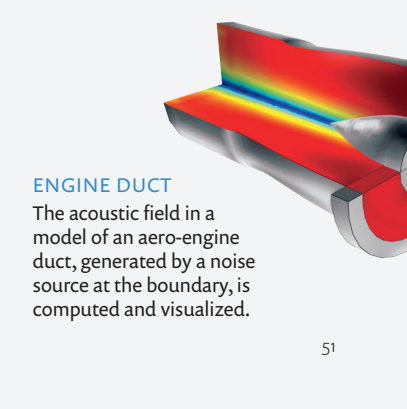
CONCERT HALL
This simulation of a small concert hall uses ray acoustics to determine sound pressure and impulse response.



SONAR
The tonpilz 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 aero-engine duct, generated by a noise source at the boundary, is computed and visualized.

CFD Module

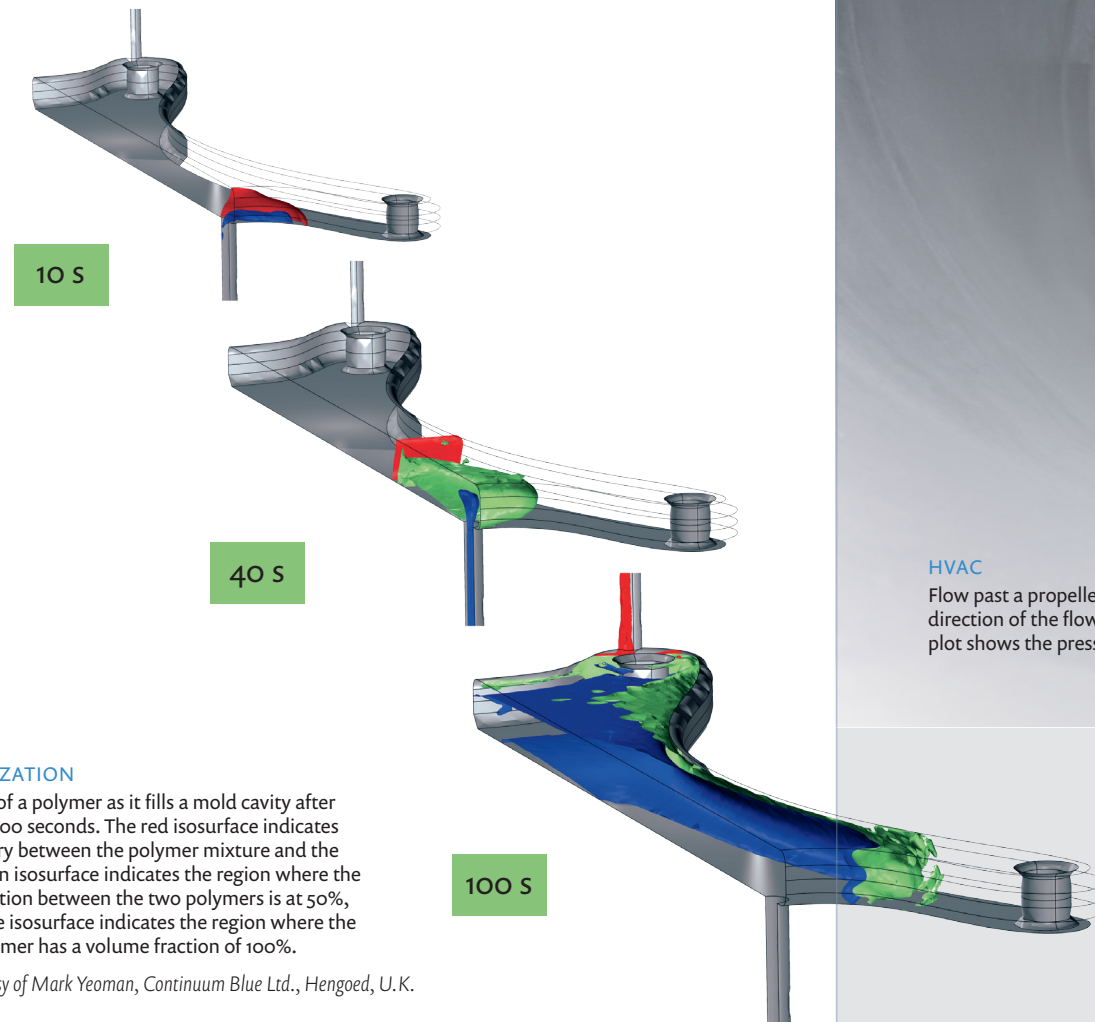
The CFD Module is the premier tool in the COMSOL product suite for sophisticated fluid flow simulations. Incompressible as well as compressible flows can be combined with advanced turbulence models, as well as heat and mass transfer. An important characteristic of the CFD Module is its ability to accurately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

A specific interface for modeling reacting flows allows for the simulation of transport and chemical reactions in dilute as well as in 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 the interface tracking of two-phase flows, formulations are provided using the level-set and phase-field methods.

POLYMERIZATION

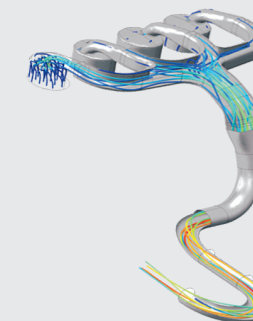
Simulation of a polymer as it fills a mold cavity after 10, 40, and 100 seconds. The red isosurface indicates the boundary between the polymer mixture and the air, the green isosurface indicates the region where the volume fraction between the two polymers is at 50%, and the blue isosurface indicates the region where the second polymer has a volume fraction of 100%.

Model courtesy of Mark Yeoman, Continuum Blue Ltd., Hengoed, U.K.



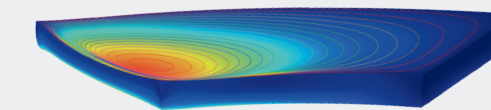
HVAC

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.



EXHAUST MANIFOLD

Flow from a car engine through a manifold to the exhaust system.

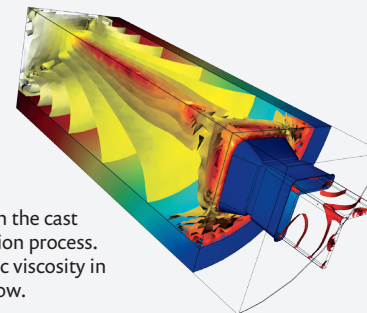


LUBRICATING OIL FILM

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.



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, nozzles
- Fluidized beds and sprays
- Fluid-structure interaction (FSI)*
- Gas bubble flow
- Heat exchangers and cooling flanges
- High Mach number flow
- Lubrication and elastohydrodynamics
- Medical/biophysical applications such as flow in blood vessels
- Mixers and stirred vessels
- Non-isothermal flow
- Non-Newtonian flow
- Polymer flow and viscoelastic flow
- Porous media flow
- Reacting flows**
- Sedimentation, emulsions, and suspensions
- Turbulent flow

* Together with the Structural Mechanics Module or MEMS Module

** Together with the Chemical Reaction Engineering Module

Mixer Module

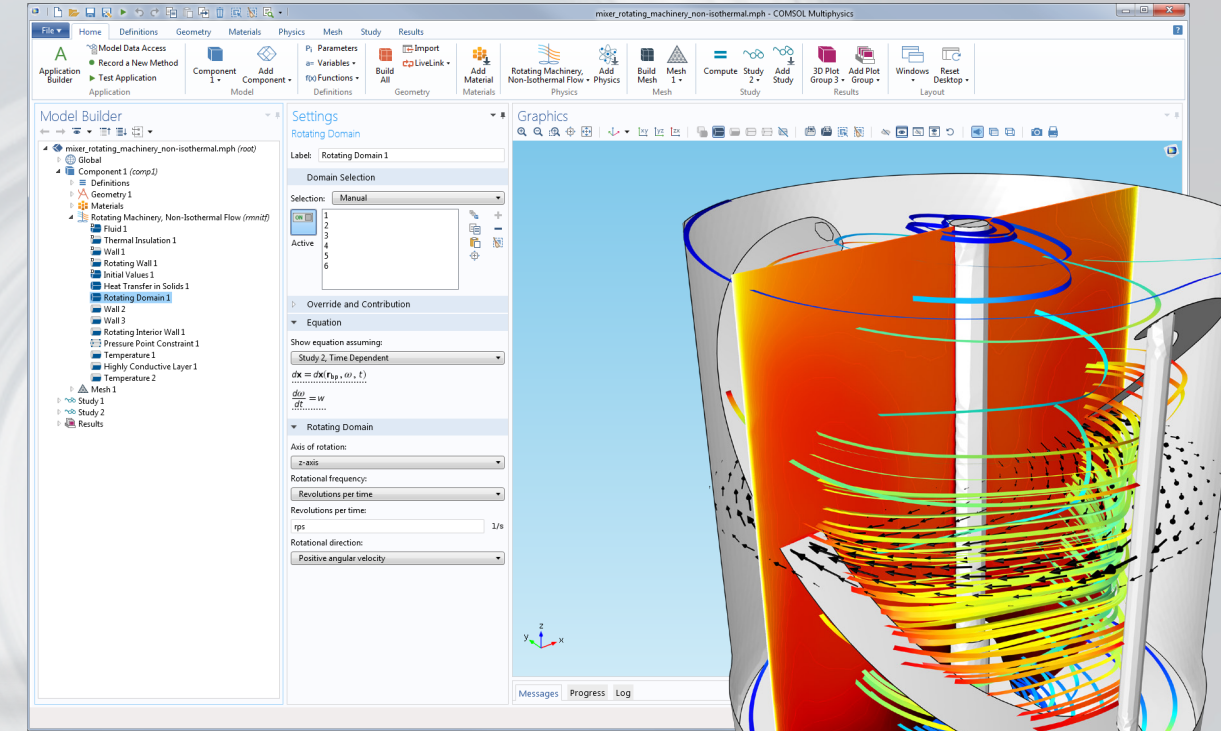
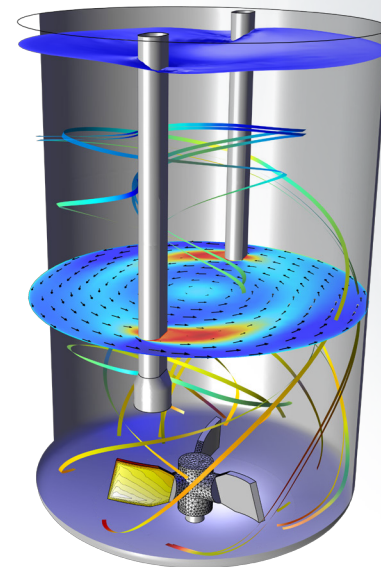
An add-on to the CFD Module, the Mixer Module allows you to analyze fluid mixers and stirred reactors. The Mixer Module includes a series of physics interfaces that are based on the conservation laws for momentum, mass, and energy, in addition to species material balances in fluids. They are applicable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

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

The Mixer Module also supplies a Frozen Rotor feature for simulating rotating flow. This feature significantly reduces the computational resources required to simulate pseudo-steady state conditions. The frozen rotor approximation is appropriate for mixers without baffles, pipes, or other geometric entities whose positions need to be modeled relative to the rotating machinery, or when the entire system rotates, such as in centrifugal separation.

TURBULENT MIXER WITH FREE SURFACE

Simulation of a turbulent mixer with two rods and a three-bladed impeller. The blue surface at the top shows the deformation of the free surface. Pressure contours are plotted on the front blade. The center slice and ribbons show the direction and magnitude of the flow field.



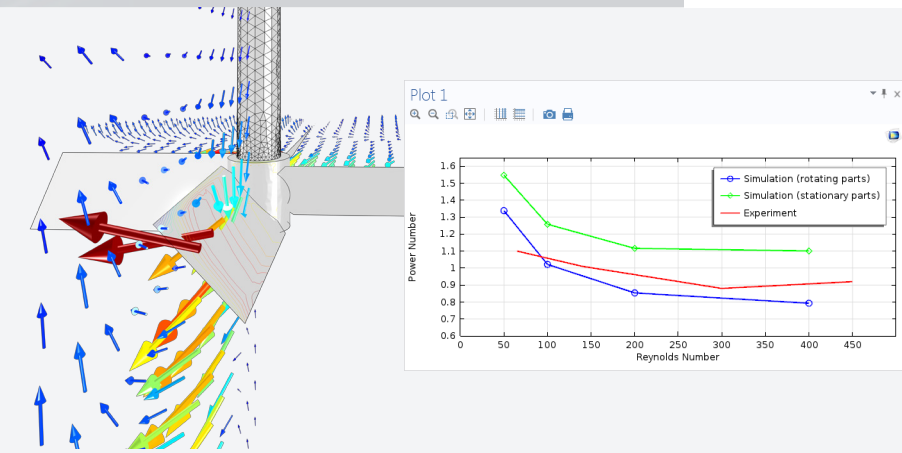
NON-ISOTHERMAL MIXER

Simulation of a batch reactor equipped with a helical ribbon impeller where temperature-dependent fluid properties and heat transfer in solids is included. The temperature distribution in the mixer is shown here in a 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).



HIGHLIGHTS

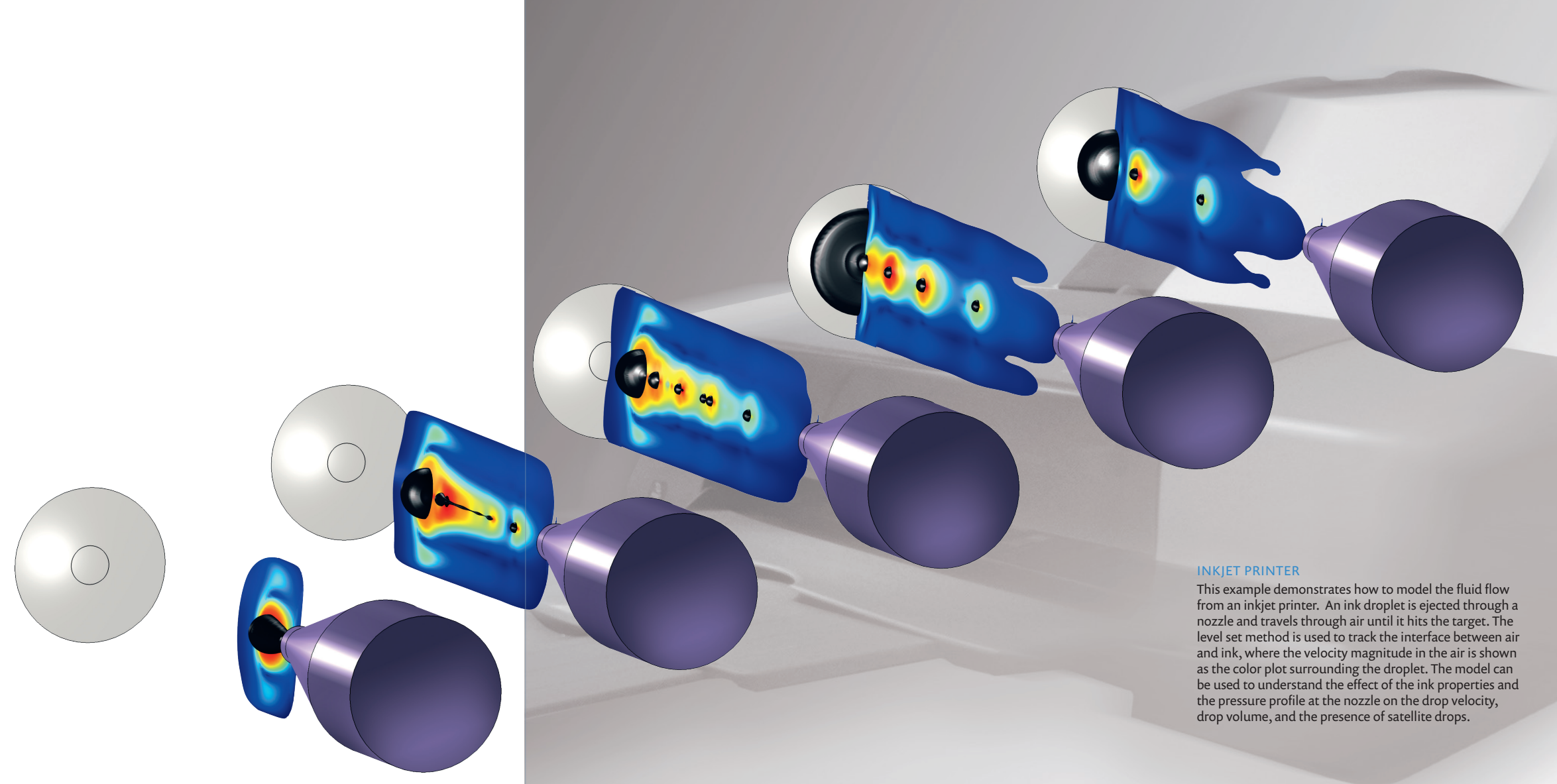
- 2D and 3D stationary and time-dependent studies
- Batch stirred reactors
- Continuous stirred reactors
- Fluid and chemical mixing
- Free fluid surfaces including the effects of surface tension forces and contact angles
- Laminar, turbulent, incompressible, and weakly compressible flows
- Mixing of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

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 will 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.

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.



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.

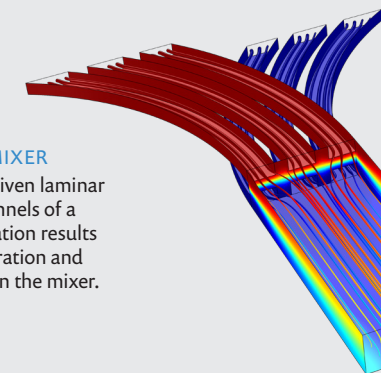
HIGHLIGHTS

- 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-phase flow

* Together with the Structural Mechanics Module or MEMS Module

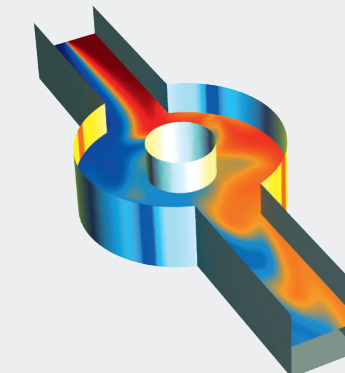
MICROFLUIDICS MIXER

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



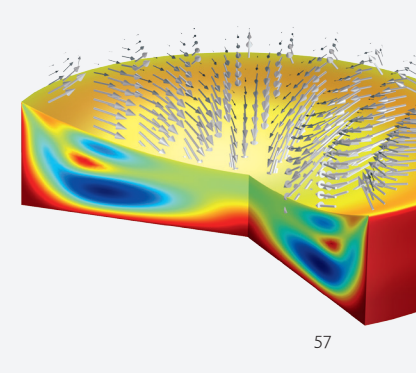
ELECTROOSMOSIS

Microlaboratories for biochemical applications often require rapid mixing of different fluid streams. This model takes advantage of electroosmosis to mix fluids. The system applies a time-dependent electric field, and the resulting electroosmosis perturbs the parallel streamlines and concentration distribution in an otherwise highly-ordered laminar flow.



ELECTROWETTING

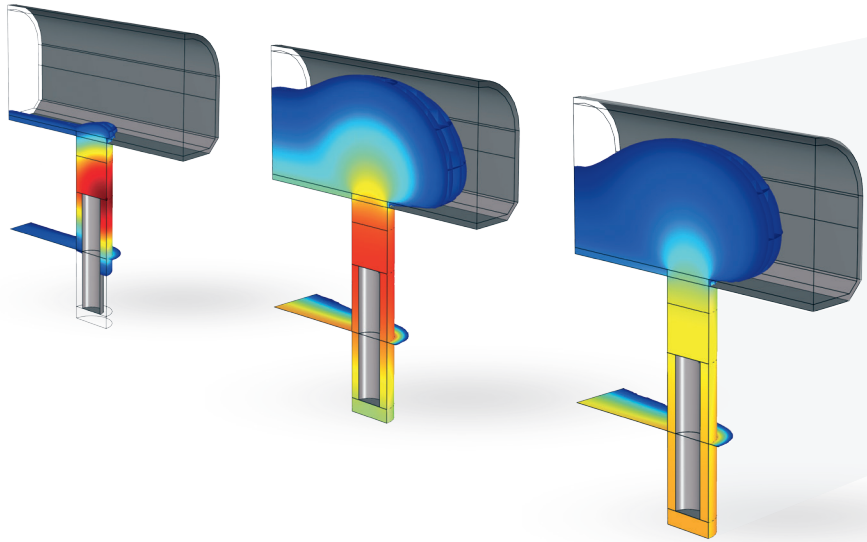
Simulation of a variable-focus liquid lens for a miniature camera during the transition between focal lengths. The lens is made from the interface between two fluids - the lower phase velocity field is colored, while the upper phase is shown by an arrow plot.



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 non-negligible. 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.

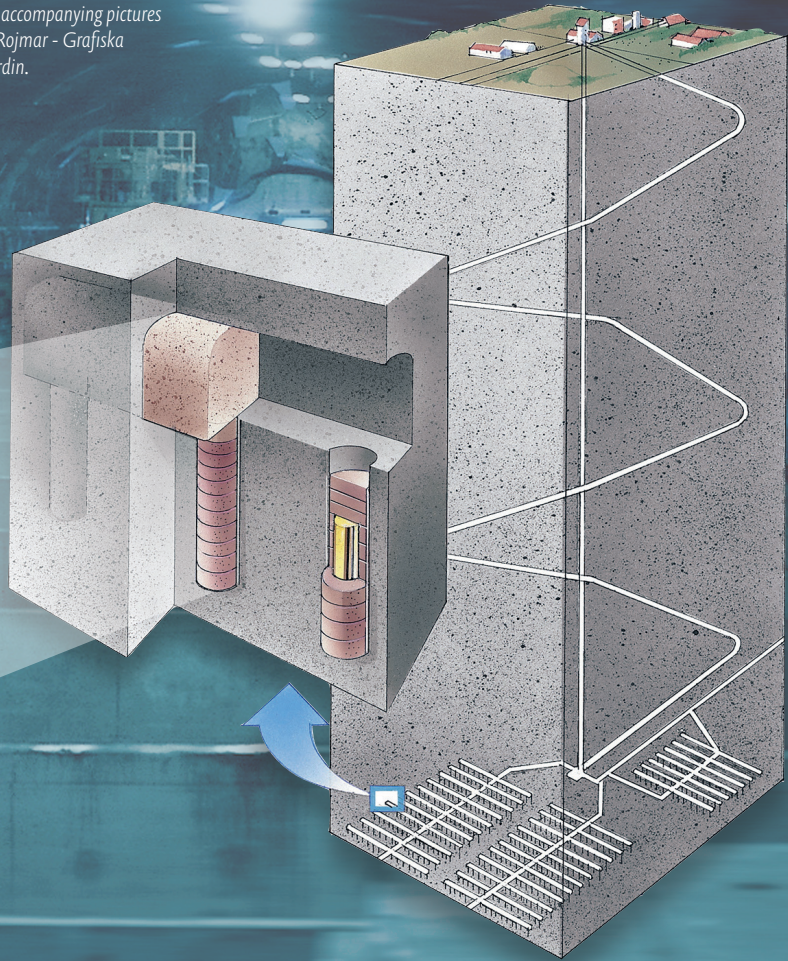
For heat transfer simulations, background geotherms are available as well as automated calculation of effective thermal properties for multicomponent systems, including heat transfer in solid, liquids, porous media, and fractures. Compaction and subsidence modeling is enabled by a very powerful user 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.

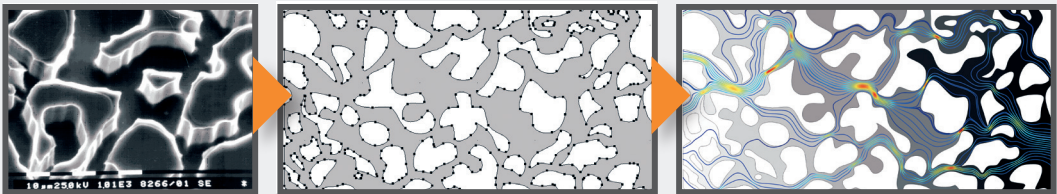
The model has been produced by Veli-Matti Pulkkanen, VTT Technical Research Centre of Finland. The accompanying pictures are Copyright © SKB. Illustrator: Jan M Rojmar - Grafiska Illustrationer. Photographer: Bengt O Nordin.



SUBSURFACE FLOW

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

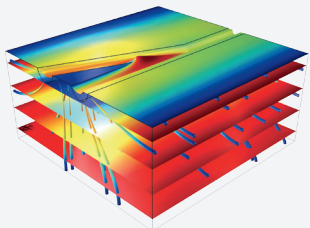
Model courtesy of Arturo Keller, University of California, Santa Barbara.



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.



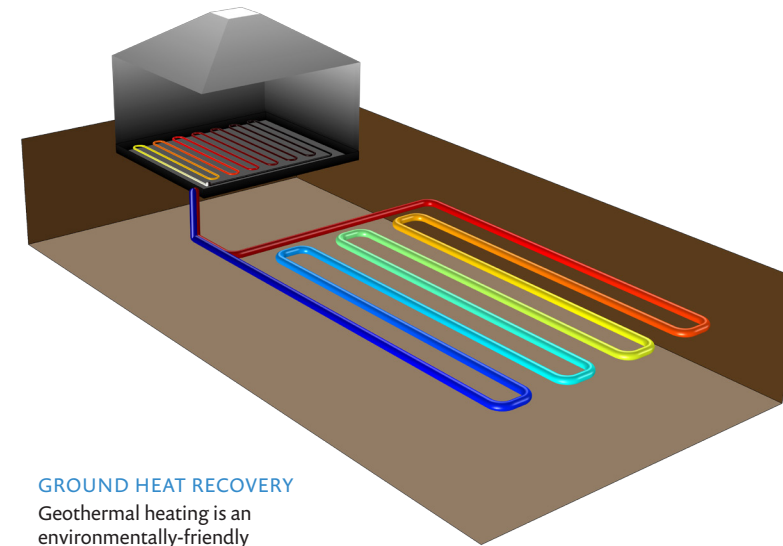
HIGHLIGHTS

- 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

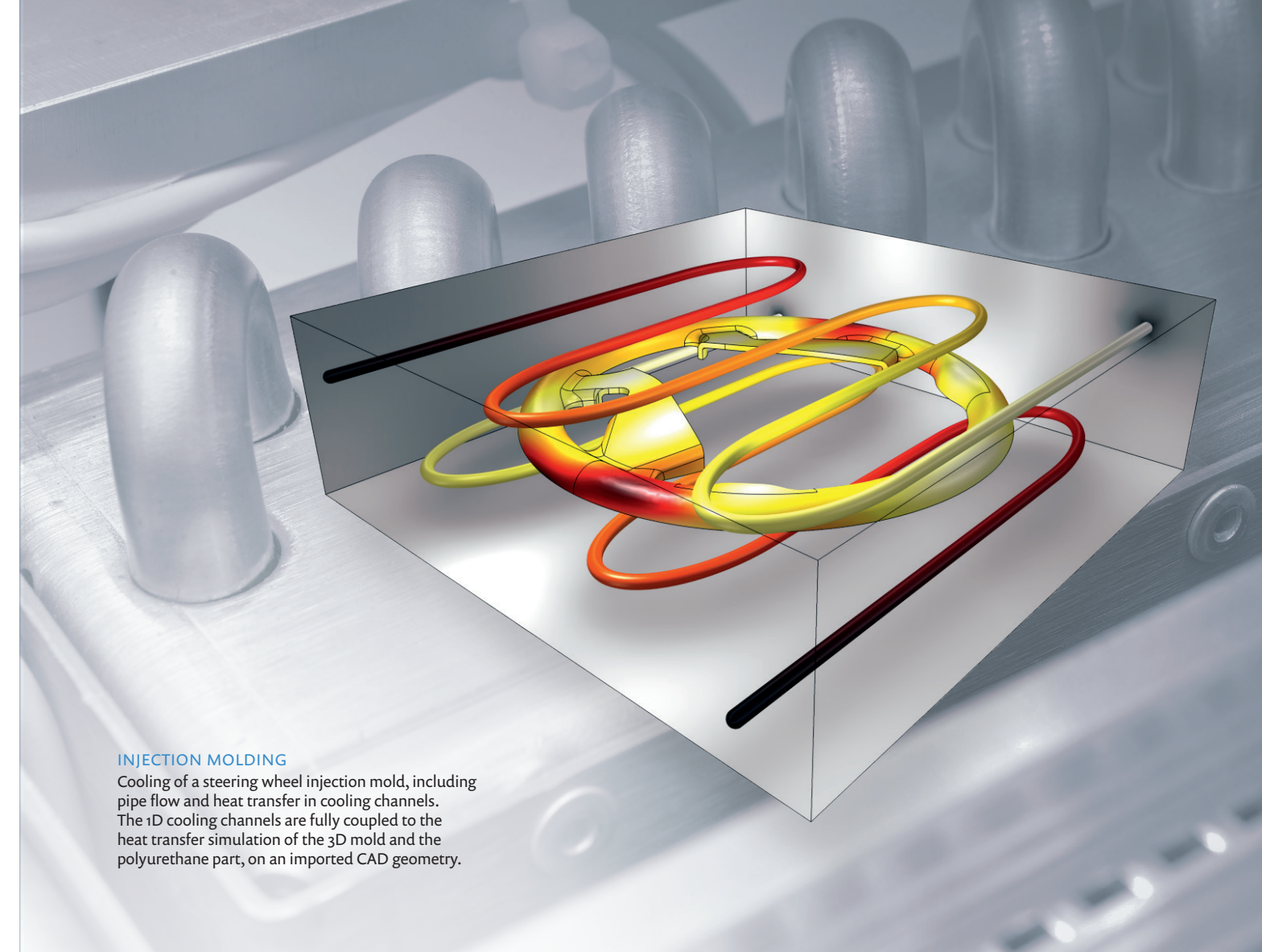
Pipe Flow Module

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 which have lengths large enough that flow in them 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.



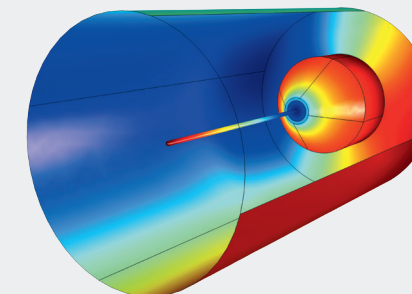
GROUND HEAT RECOVERY
Geothermal heating is an environmentally-friendly and energy-efficient way to heat homes. This simulation analyzes the thermal behavior of a pipe pattern embedded in subsurface soil.



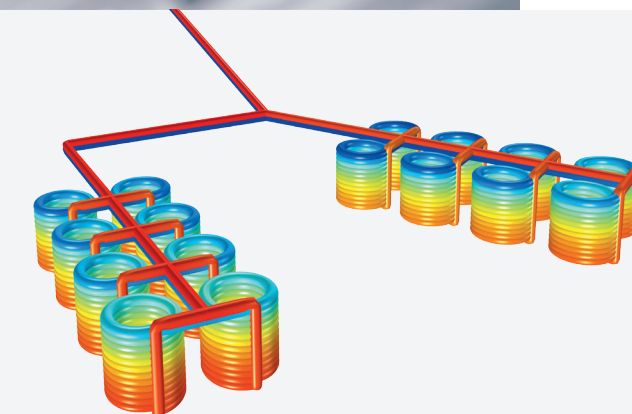
INJECTION MOLDING
Cooling of a steering wheel injection mold, including pipe flow and heat transfer in cooling channels. The 1D cooling channels are fully coupled to the heat transfer simulation of the 3D mold and the polyurethane part, on an imported CAD geometry.

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
- Non-isothermal pipe flow
- Oil refinery pipe systems
- Pressure drop calculation in piping and channels
- Radiant floor heating
- Ventilation and air conditioning systems



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.



GEOHERMAL 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.

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 non-isothermal 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.

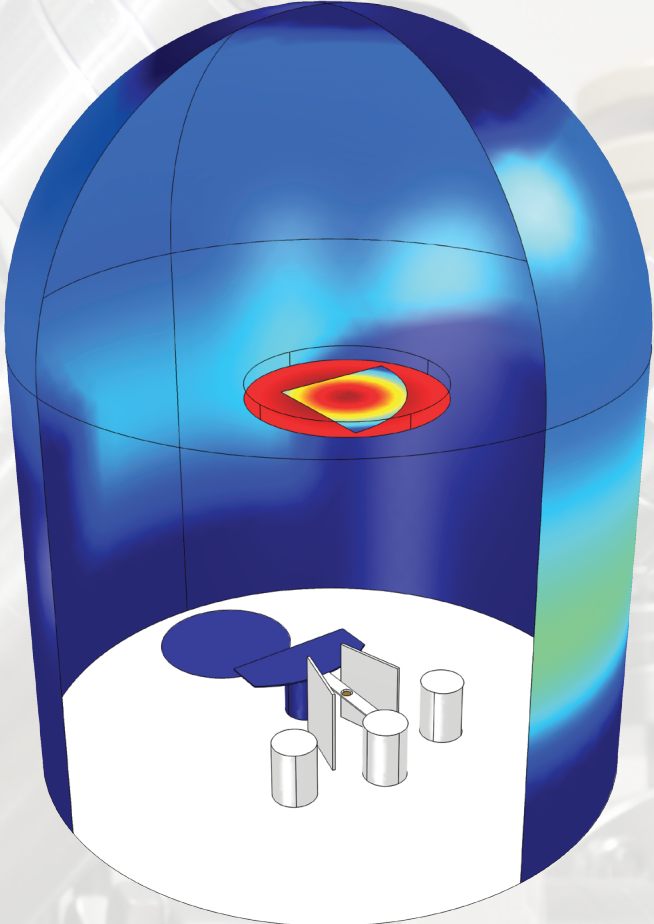
This module is ideal for the simulation of vacuum systems including those used in semiconductor processing, particle accelerators, and mass spectrometers. Small channel applications can also be addressed.

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 surfaces only. 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 or 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.

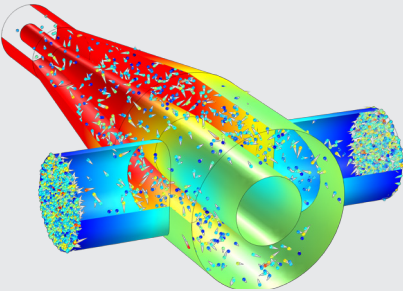


FILM DEPOSITION

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

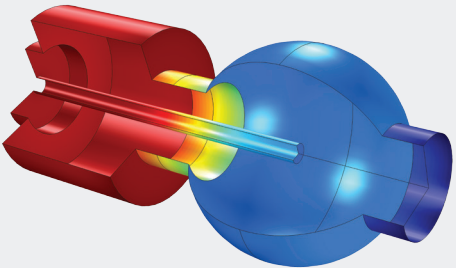
RF COUPLER

This model computes the transmission probability through an RF coupler.



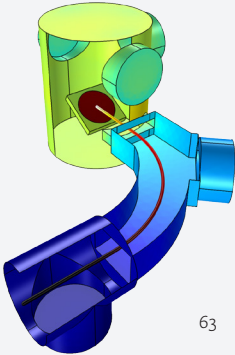
LOAD LOCK VACUUM SYSTEM

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 is 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.



HIGHLIGHTS

- Mass spectrometers
- Materials processing equipment
- Nanopore flow
- Particle accelerators
- Semiconductor processing equipment
- Vacuum systems

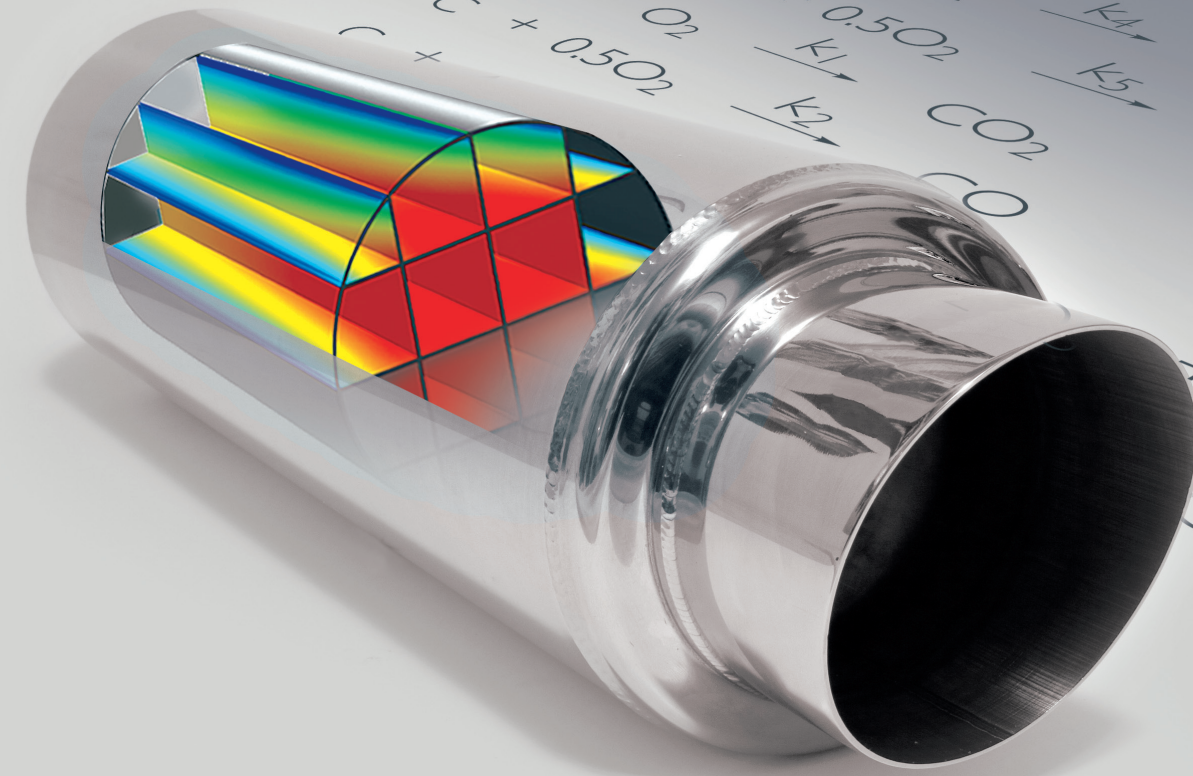
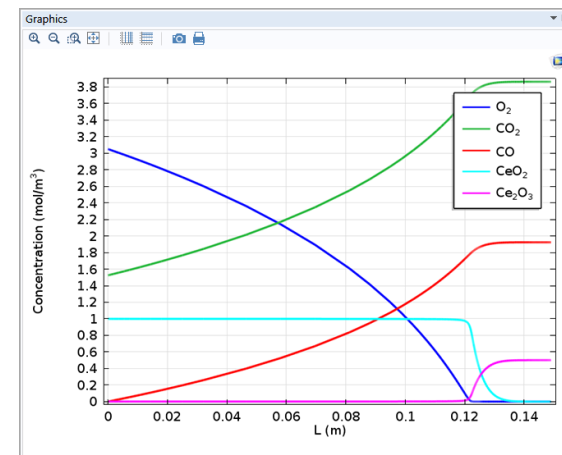
Chemical Reaction Engineering Module

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 as well as equation-based modeling. This allows for the inclusion of arbitrary expressions, functions, and source terms in the material property, transport, and reaction kinetic 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.

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 non-catalytic reactions in the reactor volume and on a moving surface. All reactions are temperature- and material transport dependent. The graph shows the concentrations of the carbon monoxide, carbon dioxide, and the cerium additive along the filter module length in a pre-study of the chemical kinetics.



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
- Pre-burners 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

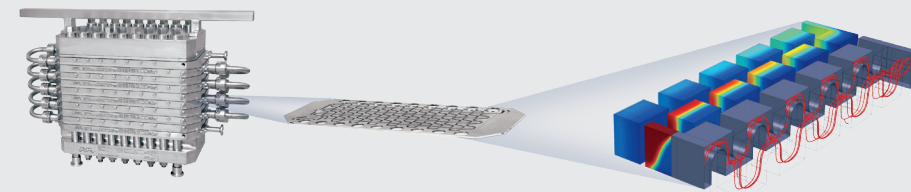


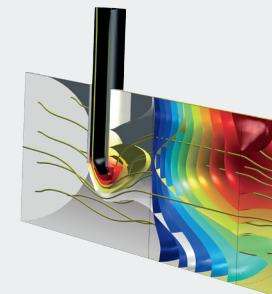
PLATE REACTOR

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.

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

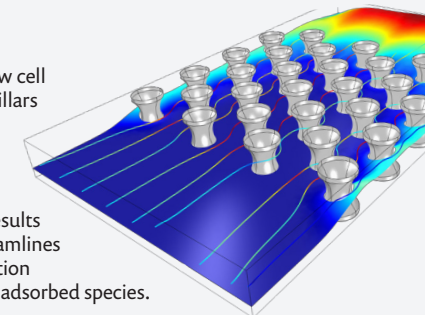
POROUS REACTOR

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 the concentration, for one of the reactants and the product species.



BIOSENSOR

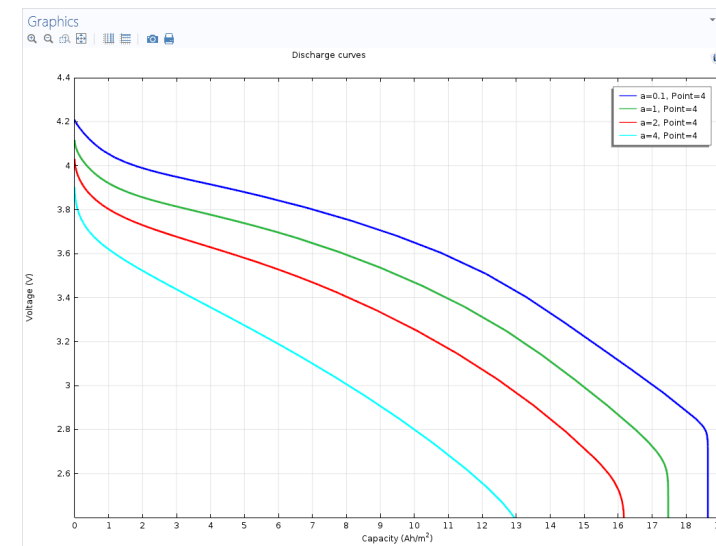
Simulation of a flow cell containing micropillars coated with an active material to support the adsorption of an analyte species. Results show velocity streamlines and the concentration distribution of the adsorbed species.



Batteries & Fuel Cells Module

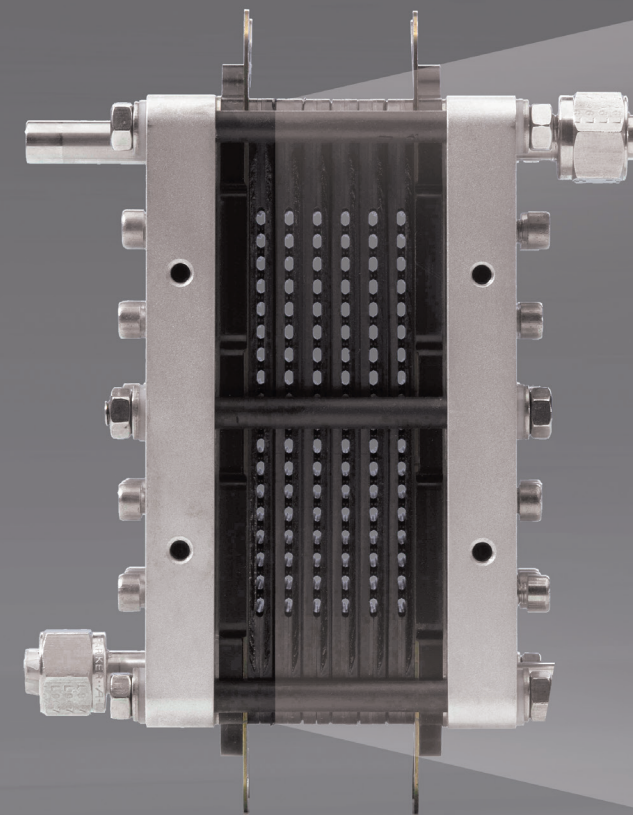
The Batteries & Fuel Cells Module provides a full set of easy-to-use tools for the simulation of fundamental processes in the electrodes and electrolyte of lithium-ion batteries, nickel metal-hydride batteries, solid oxide fuel cells, and proton exchange membrane fuel cells. With it you can quickly and 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 surface-to-surface radiation and turbulent flow, can be performed through the powerful capabilities of COMSOL Multiphysics.



LITHIUM-ION BATTERY

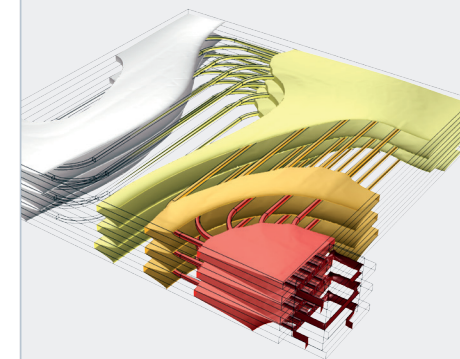
Discharge curves for various discharge rates in a lithium-ion battery. The different curves correspond to a full discharge in 10 h ($a=0.1$), 1 h ($a=1$), 30 min ($a=2$), and 15 min ($a=4$).



PEMFC

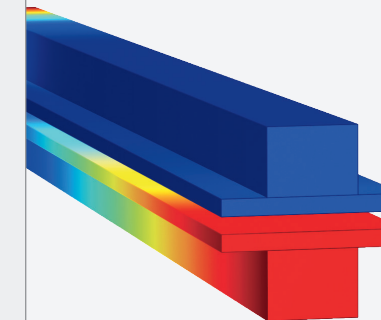
The hydrogen concentration through the channels of a bipolar plate considering the effects of the edges is shown.

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



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

Current density distribution in a solid oxide fuel cell. The model includes full coupling between 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.

HIGHLIGHTS

- Fuel Cells
 - Alkaline
 - Direct Methanol
 - Molten Carbonate
 - Proton Exchange Membrane
- Batteries
 - Lead Acid
 - Lithium Ion
 - Nickel Hydride
- Flow Batteries
 - Soluble Lead
 - Vanadium

CAPABILITIES

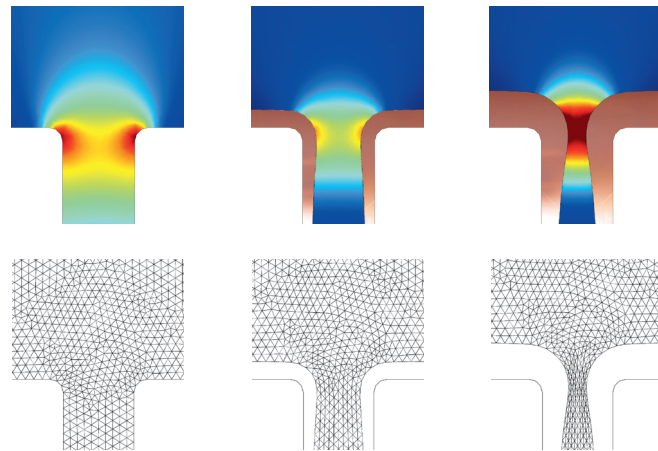
- Aging mechanisms due to structural, thermal, and chemical effects
- Detailed electrochemical analysis of cells
- Full transient studies including double layer capacitance
- Pre-defined impedance spectroscopy study for electrode kinetics and aging investigations
- Primary, secondary, and tertiary current distribution
- Short circuits in batteries
- Thermal management
- Thermal runaway in batteries

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 of the electrode can also be simulated with moving boundaries.

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.



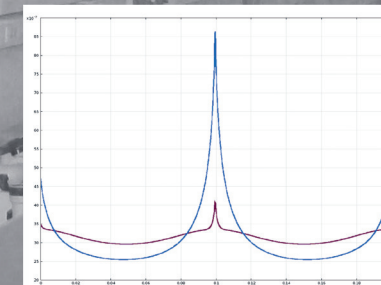
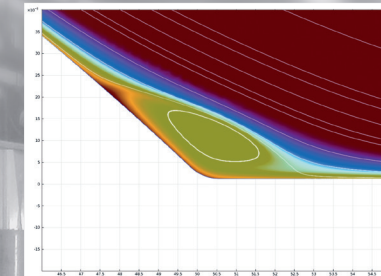
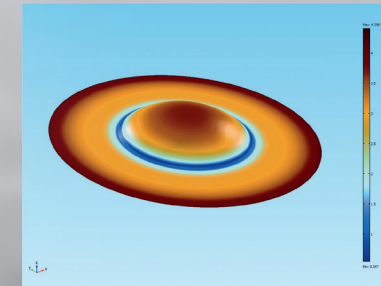
ELECTRODEPOSITION

This model demonstrates the effect of a moving boundary 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 non-uniform 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.

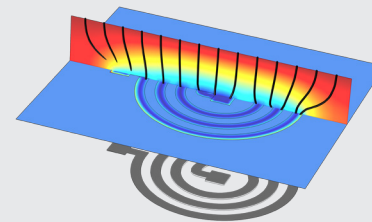


HIGHLIGHTS

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

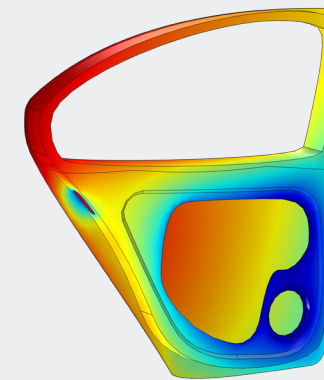
INDUCTOR COIL

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



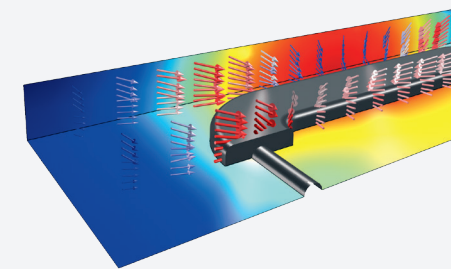
CAR DOOR

Simulation of deposited paint thickness on an electrocoated car door.



DECORATIVE ELECTROPLATING

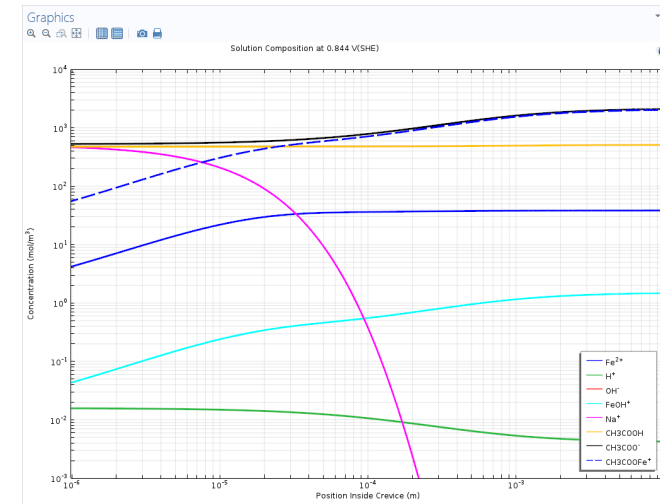
In this example, the anode is planar and dissolves, while the cathode is a furniture fitting that is to be decorated by the dissolved metal. Shown is the thickness of the deposited layer at the cathode as well as the pattern caused by dissolution of the anode surface. The model assumes secondary current 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. 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 pre-defined physics interfaces. These 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 these scales is necessary and also achievable with the Corrosion Module.

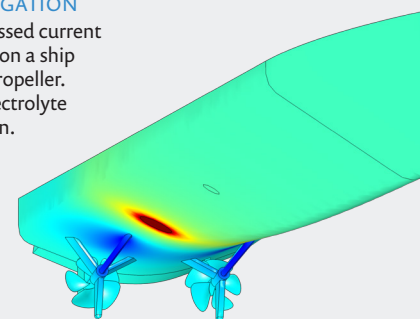


CREVICE CORROSION

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

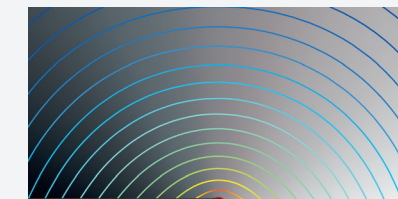
CORROSION MITIGATION

Simulation of impressed current cathodic protection on a ship hull with a coated propeller. Results show the electrolyte potential distribution.

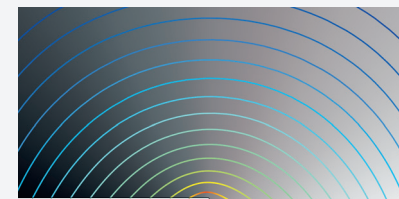


GALVANIC CORROSION

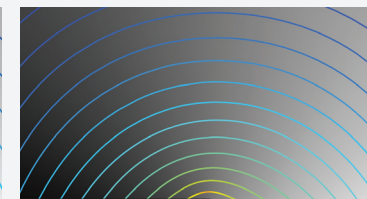
The corrosion of a magnesium alloy (AE44) connected to mild steel in a salt-water solution. The electrode material removal is also modeled, as it is an important variable to be considered when performing such simulations.



t=0 hours



t=28 hours



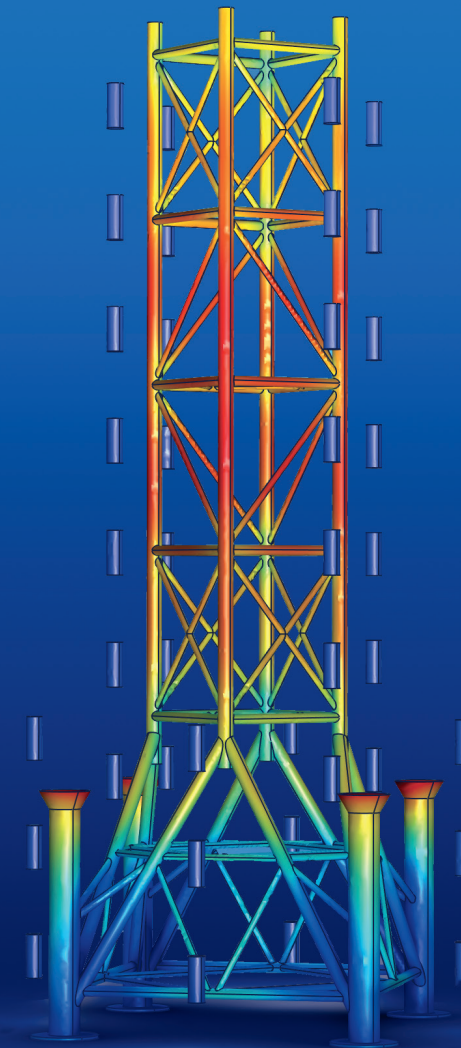
t=72 hours

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

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. Visualized is the electrolyte potential on the surface of the structure.

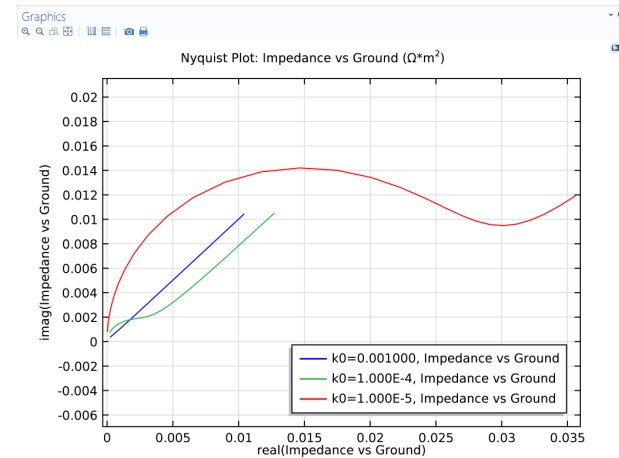


Electrochemistry Module

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 or to the industrial chemical engineer. 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.

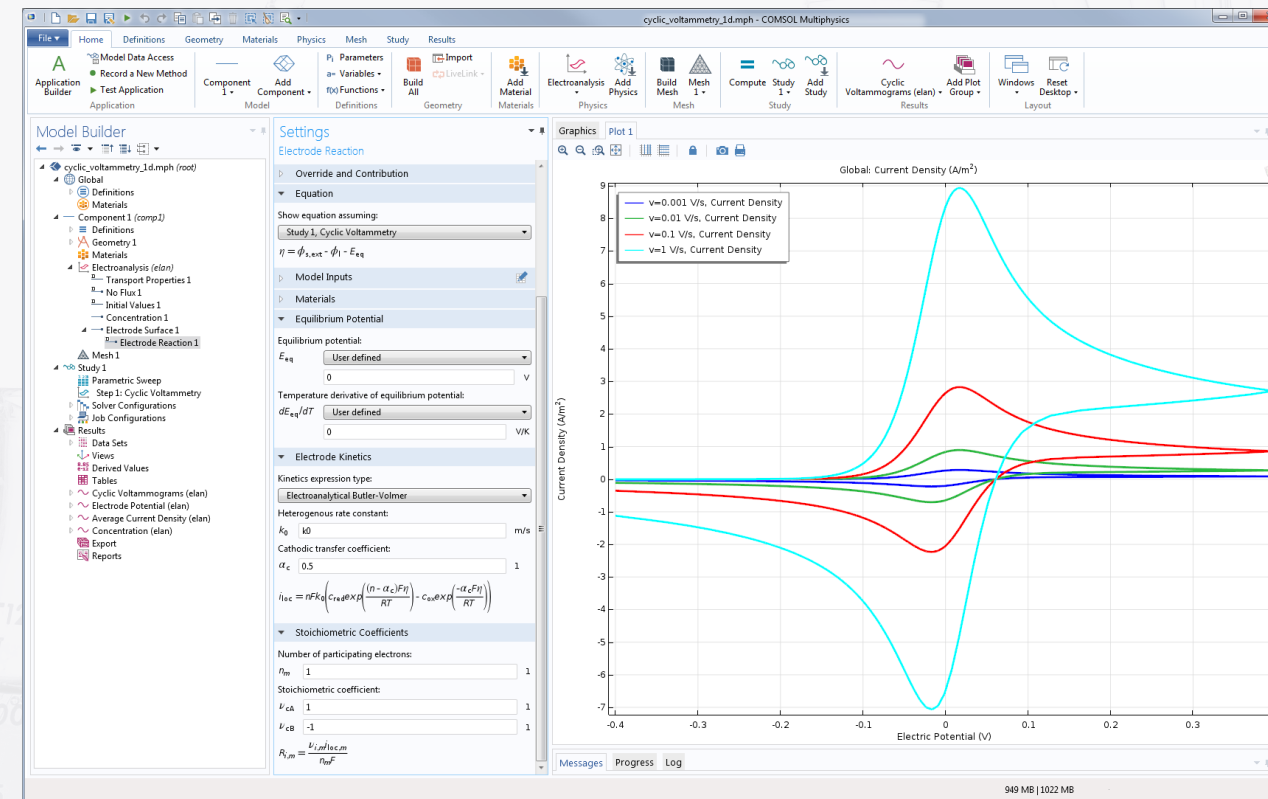
Dedicated interfaces in the Electrochemistry Module enable the definition of voltammetry, amperometry, potentiometry, electrochemical impedance, and coulometry studies. Exchange current densities and activation overpotentials can then be determined from the combined experiment and simulation results.

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.

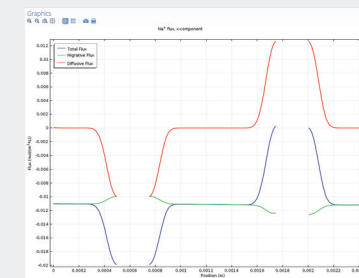


ELECTROCHEMICAL ANALYSIS

Cyclic voltammetry is a common technique for electrochemical analysis. During simulation, the potential at the working electrode is swept over a voltage range while the current is recorded. This waveform provides information about the reactivity and mass transport properties of an electrolyte.

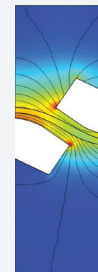
ELECTRODIALYSIS

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.



HIGHLIGHTS

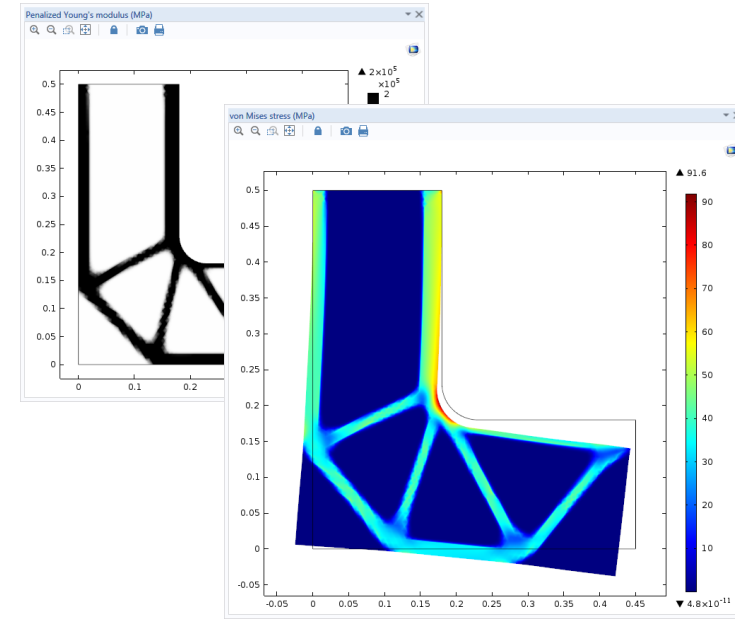
- Bioelectrochemistry
- Chlor-alkali electrolysis
- Control of electrochemical reactions in biomedical implants
- Desalination of seawater
- Electroanalysis
- Electrochemical sensors
- Electrodialysis
- Electrolysis
- Electrolyte waste treatment
- Gas sensors
- Glucose sensors
- Hydrogen and oxygen production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

Optimization Module

No matter the engineering discipline, once you have a model of a product or a 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. The following derivative-based methods are available: SNOPT, MMA, and Levenberg-Marquardt. The following derivative-free methods are available: 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, as well as 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.

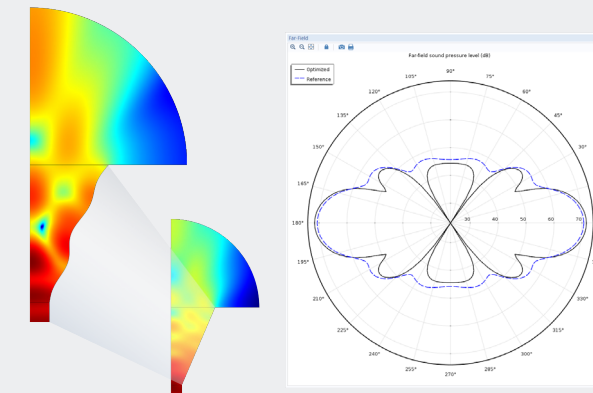


TOPOLOGICAL OPTIMIZATION

The SIMP model (Solid Isotropic Material with Penalization) for structural topology optimization is used to minimize weight given a minimum stiffness of 40%, the value computed from the fully solid design. Such a design reduces both weight and costs while maintaining the specified load carrying capacity, therefore making efficient use of the material.

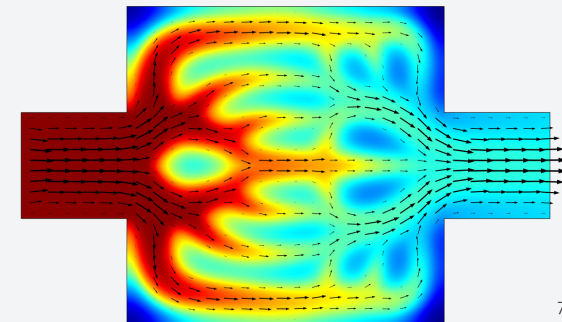
SHAPE OPTIMIZATION OF A HORN

A horn has the initial shape of an axisymmetric cone with a straight boundary. This 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.



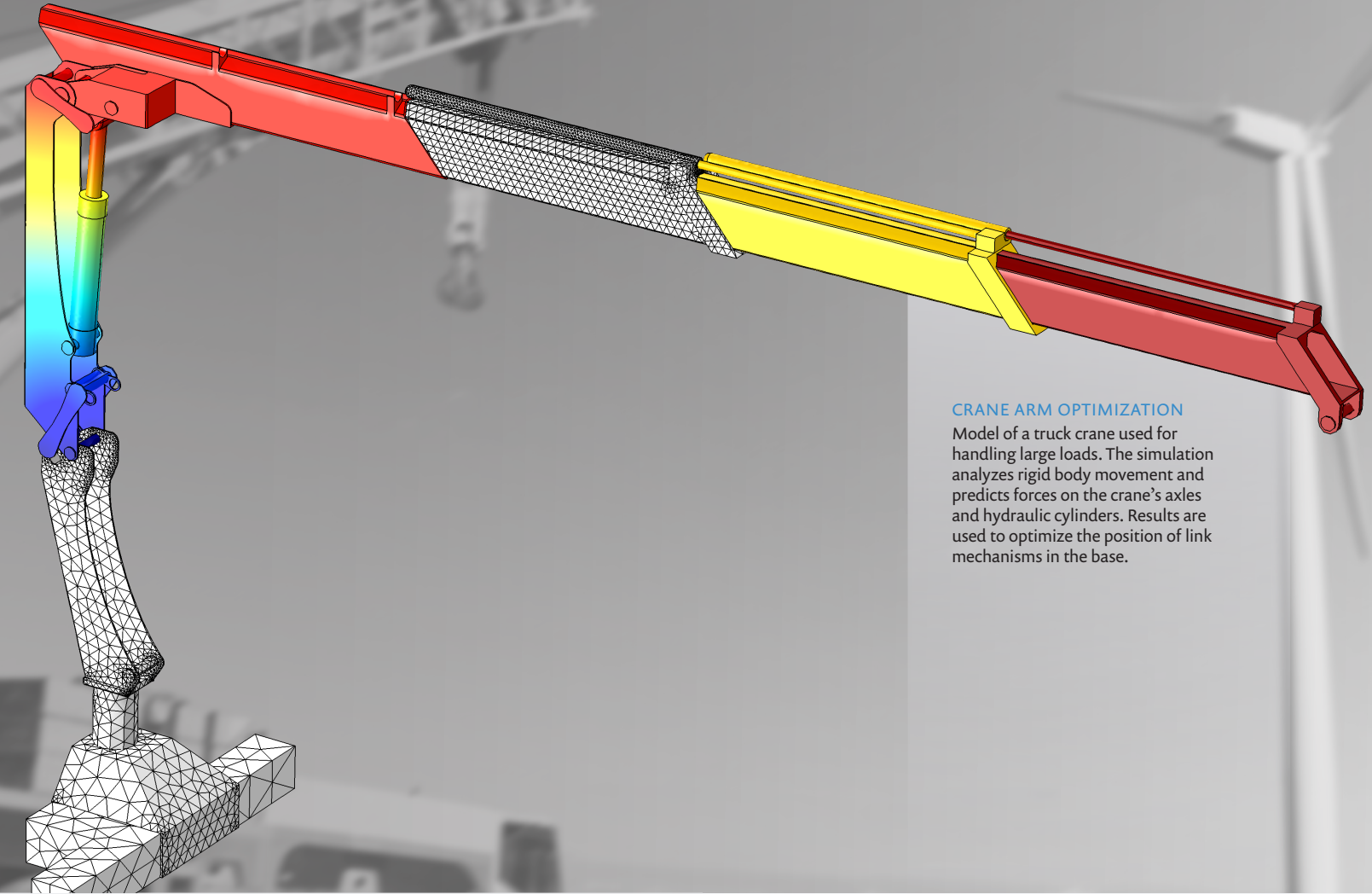
REACTOR OPTIMIZATION

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



CRANE ARM OPTIMIZATION

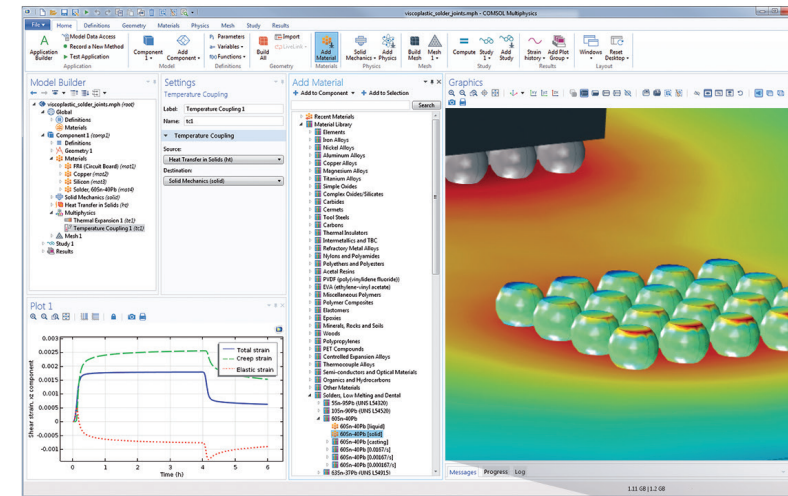
Model of a truck crane used for handling large loads. The simulation analyzes rigid body movement and predicts forces on the crane's axles and hydraulic cylinders. Results are used to optimize the position of link mechanisms in the base.



Material Library

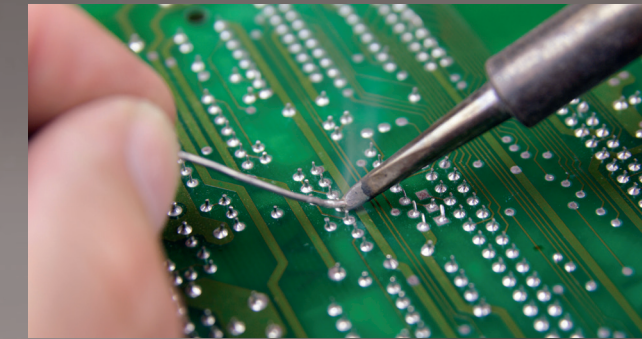
COMSOL Multiphysics gives you complete control over definitions and use of your material properties through the Model Builder and Material Browser. The Material Browser allows you to manage all your model's materials in one place and can be complemented by the Material Library.

The Material Library contains data for more than 2,700 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 plastic flow clearly appears after about 40 s of the loading. The model results show a closeup 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
- Polyamids and polyesters
- Polypropylenes and PET
- Resistance and magnetic alloys
- Salts, fuel cell, battery, and electro-ceramics
- Semiconductors, optical, and related materials
- Silicides and Borides
- Solders, dental, and Co alloys

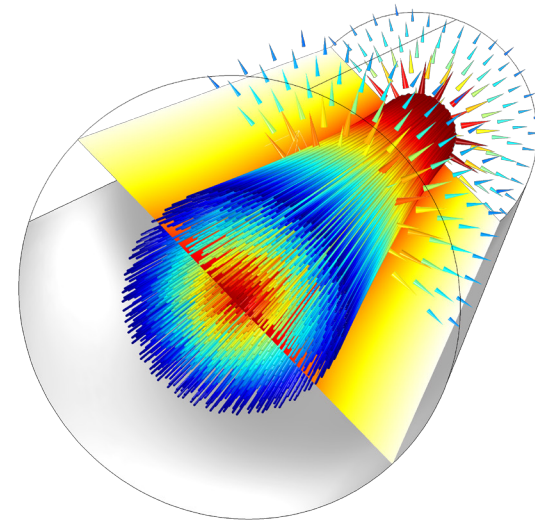
Particle Tracing Module

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.

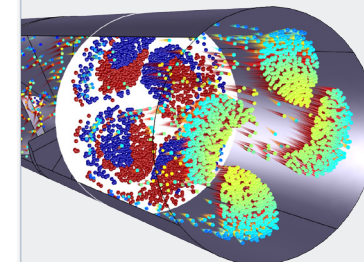
CVD CHAMBER

The Particle Tracing Module offers many ways to release particles. Particles may be released from an inlet surface or within a domain, and their initial positions may either be based on the finite element mesh or prescribed via a user-defined density function. It is also possible to release particles from a regular grid of points. In addition, the particle velocity may be initialized using a user-defined expression; a prescribed normal velocity; a Maxwellian distribution; or by releasing particles in a cone, sphere, or hemisphere. 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.



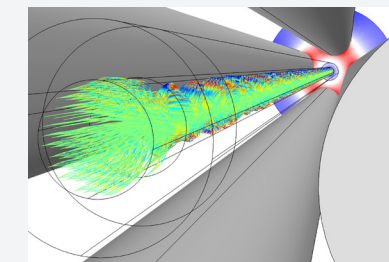
PARTICLE-FIELD INTERACTIONS

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.



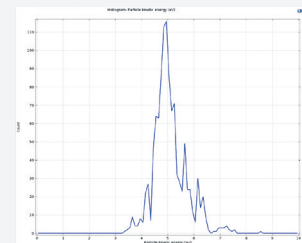
LAMINAR MIXER

In static mixers, a fluid is pumped through a pipe containing stationary blades to promote mixing with small pressure loss. In this example, the particle trajectories with comet tails indicate the particle velocity at several distinct times. Poincaré sections, shown in red and blue, show the distribution of particles at several cross-sections within the mixer. Because the motion of the dispersed phase is dominated by convection rather than diffusion, the performance of the mixer is best evaluated using a particle tracing approach.



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 and 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.



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

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

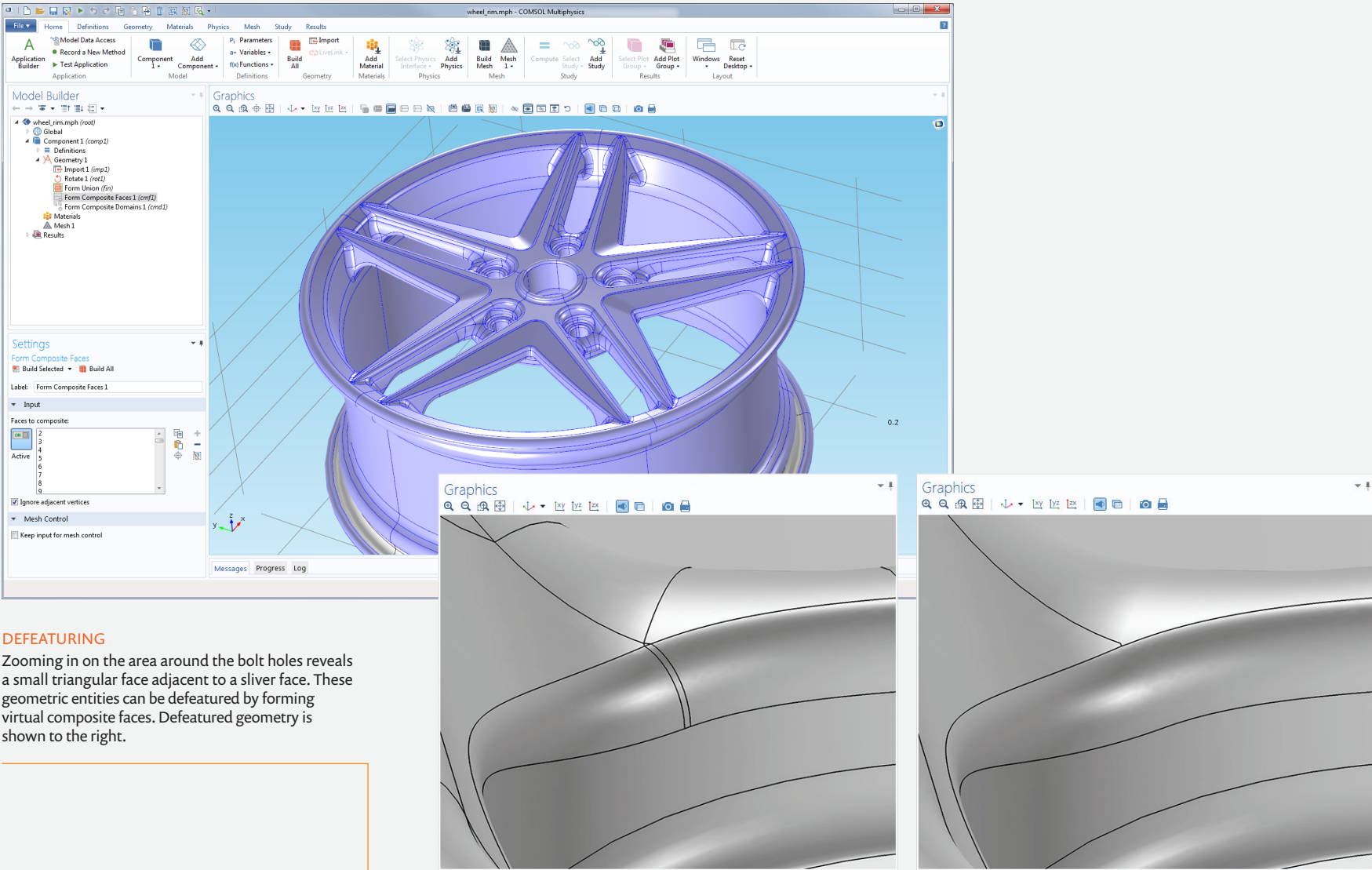
All the capabilities of the CAD Import Module are also available in the LiveLink products for SOLIDWORKS®, Autodesk® Inventor®, Autodesk® AutoCAD®, Autodesk® Revit®, PTC® Creo® Parametric™, PTC® Pro/ENGINEER® and Solid Edge® CAD systems, all of which deliver further seamless integration between your 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.



SUPPORTED CAD FILE FORMATS

- ACIS® (.sat, .sab, .asat, .asab)
- AutoCAD® (.dwg, .dxf)
- CATIA® V5 (.CATPart, .CATProduct) *
- PTC® Creo® Parametric™ (.prt, .asm)
- IGES (.iges, .igs)
- Inventor® (.ipt, .iam)
- NX® (.prt)
- Parasolid® (.x_t, .xmt_txt, .x_b, .xmt_bin)
- PTC® Creo® Parametric™ (.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.



DEFEATURING

Zooming in on the area around the bolt holes reveals a small triangular face adjacent to a sliver face. These geometric entities can be defeatured by forming virtual composite faces. Defeatured 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.

Besides adding support for the import of popular CAD file formats, the Design Module also enables automatic and interactive repair and defeaturing of the imported designs. Creating fluid volumes inside imported solid objects also becomes a lot easier by using the cap faces tool.

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, it enables modifying or constructing designs from the ground up and sharing them with other tools.

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

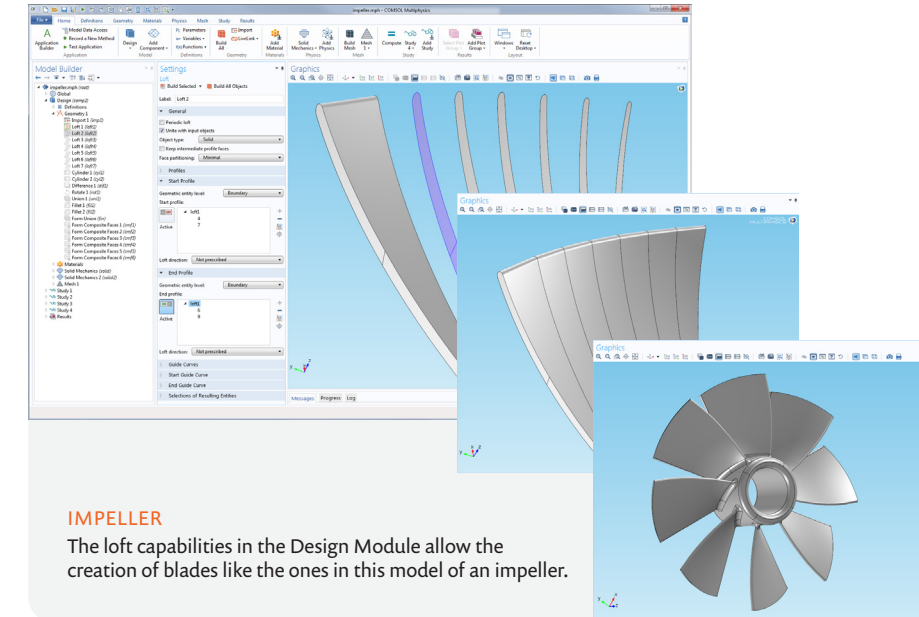


SUPPORTED CAD FILE FORMATS

See CAD Import Module

SUPPORTED OPERATIONS

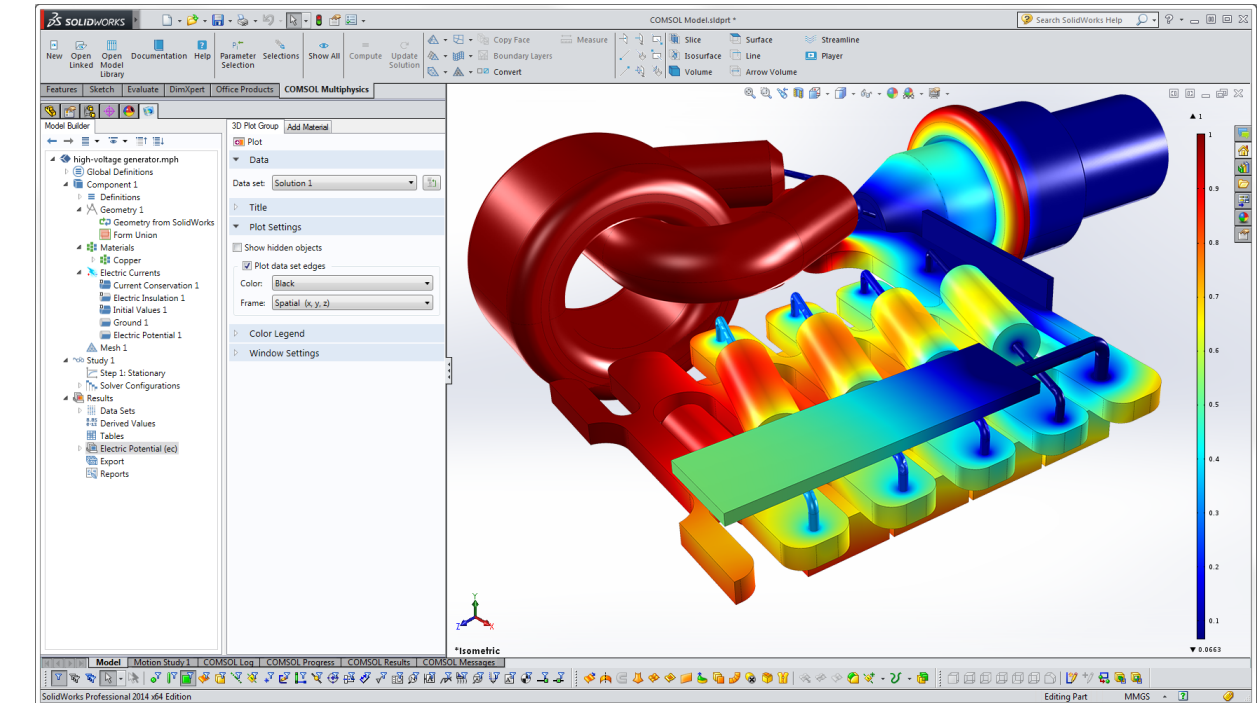
- | | |
|----------------------|------------|
| Repair and Defeating | 3D chamfer |
| Cap Faces | Thicken |
| Loft | Midsurface |
| 3D fillet | |



LiveLink™ for SOLIDWORKS®

ONE WINDOW INTERFACE

The LiveLink™ interface for the SOLIDWORKS® software CAD system has been extended with a One Window Interface where 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

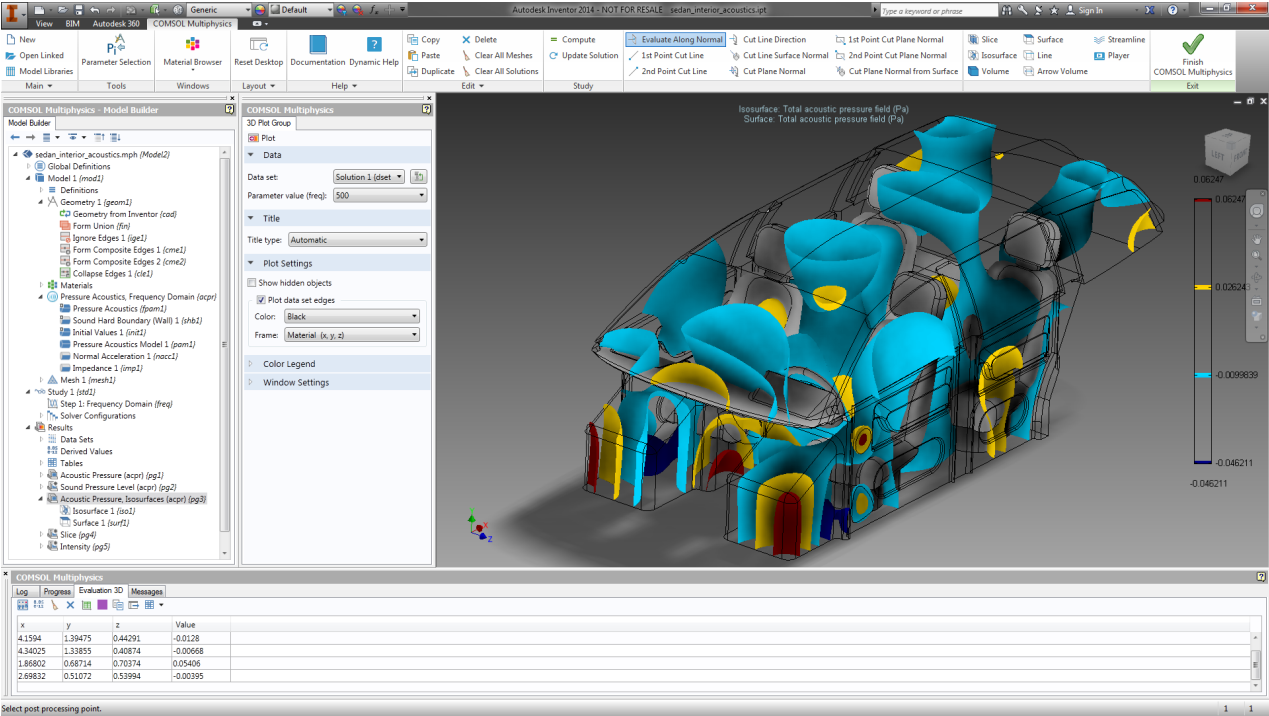
Simulation of the electromagnetic field in a high-voltage generator in an X-ray device. The design is created in the SOLIDWORKS® software CAD system and brought into COMSOL using the LiveLink™ interface. From here, different parts can be selected, physical properties defined, and the model solved.

Model courtesy of Comet AG, Flamatt, Switzerland.

LiveLink™ for Inventor®

ONE WINDOW INTERFACE

The LiveLink™ interface for the Autodesk® Inventor® software CAD system has been extended with a One Window Interface where 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.



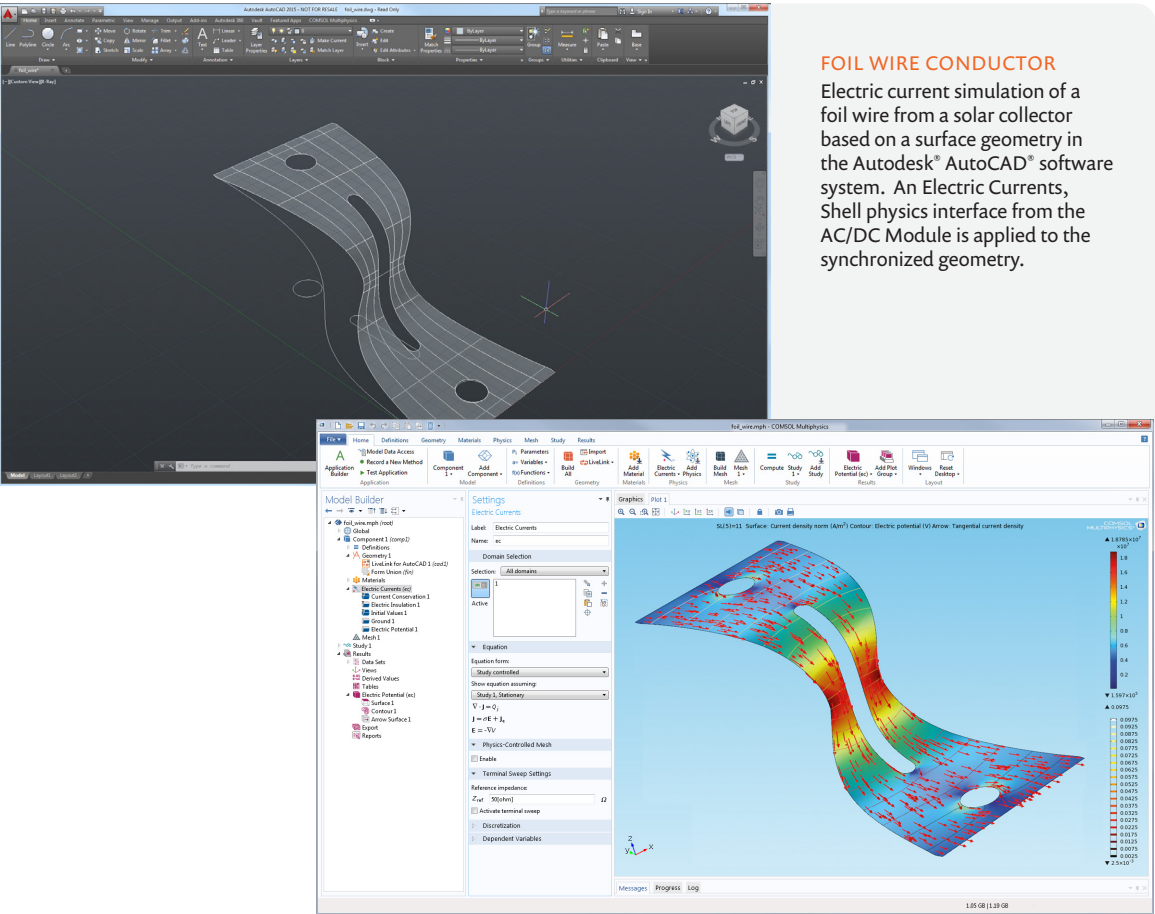
ACOUSTICS ANALYSIS

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®.



Autodesk and Inventor are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries.

LiveLink™ for AutoCAD®



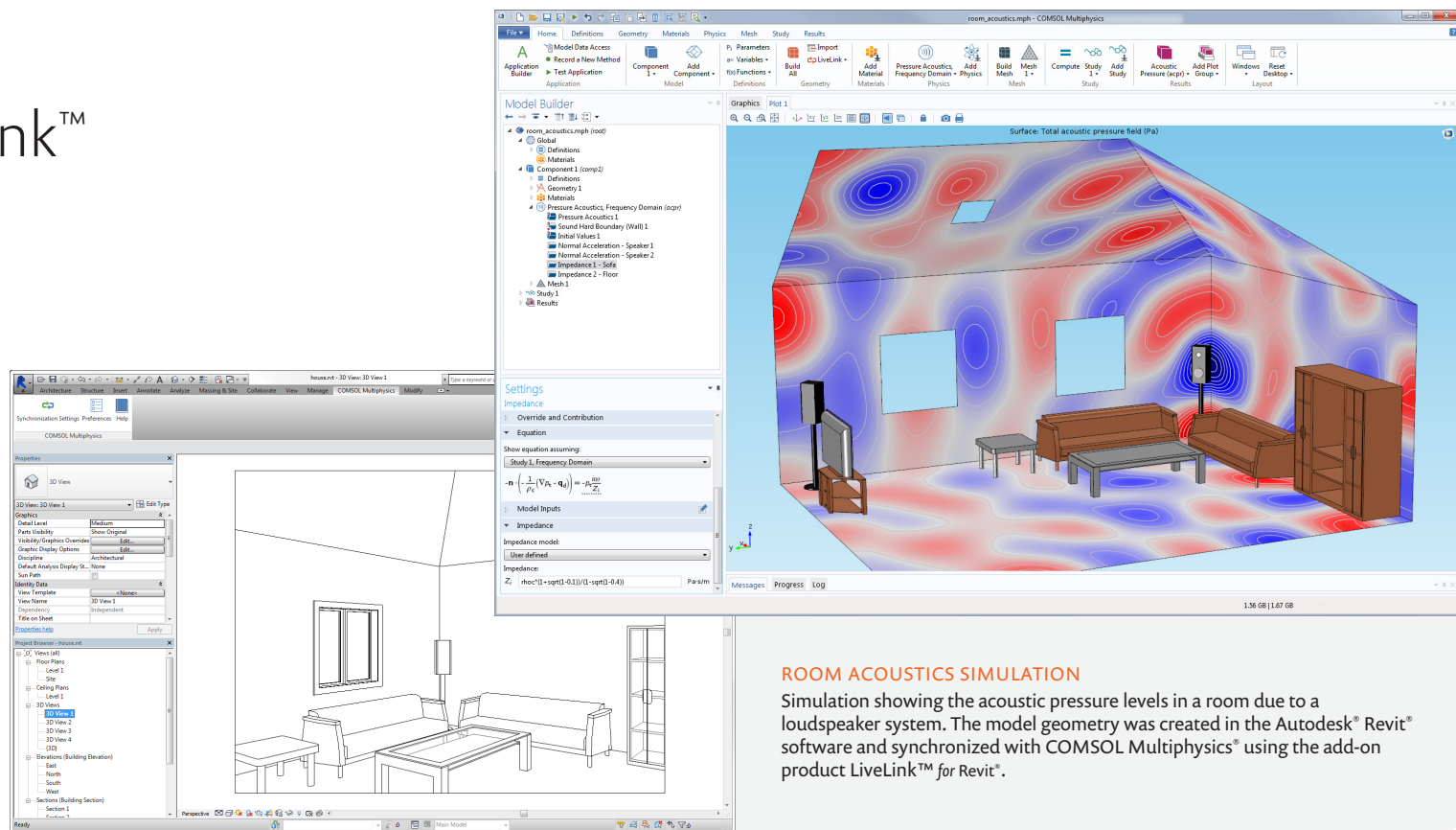
FOIL WIRE CONDUCTOR

Electric current simulation of a foil wire from a solar collector based on a surface geometry in the Autodesk® AutoCAD® software system. An Electric Currents, Shell physics interface from the AC/DC Module is applied to the synchronized geometry.

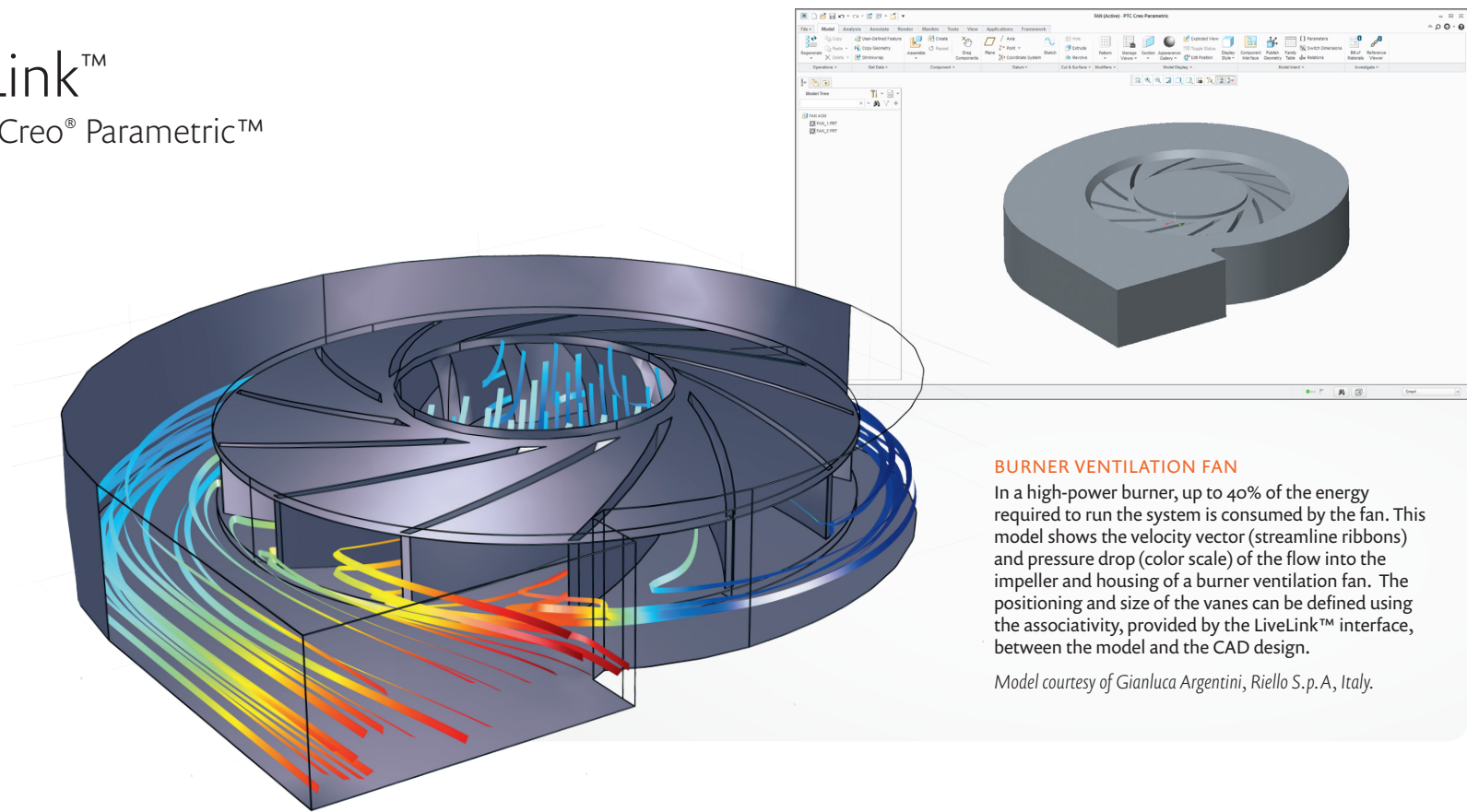


Autodesk and AutoCAD are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries.

LiveLink™ for Revit®



LiveLink™ for PTC® Creo® Parametric™



Powered by
PARASOLID

SPATIAL

AUTODESK
Authorized Developer

Autodesk and Revit are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries.

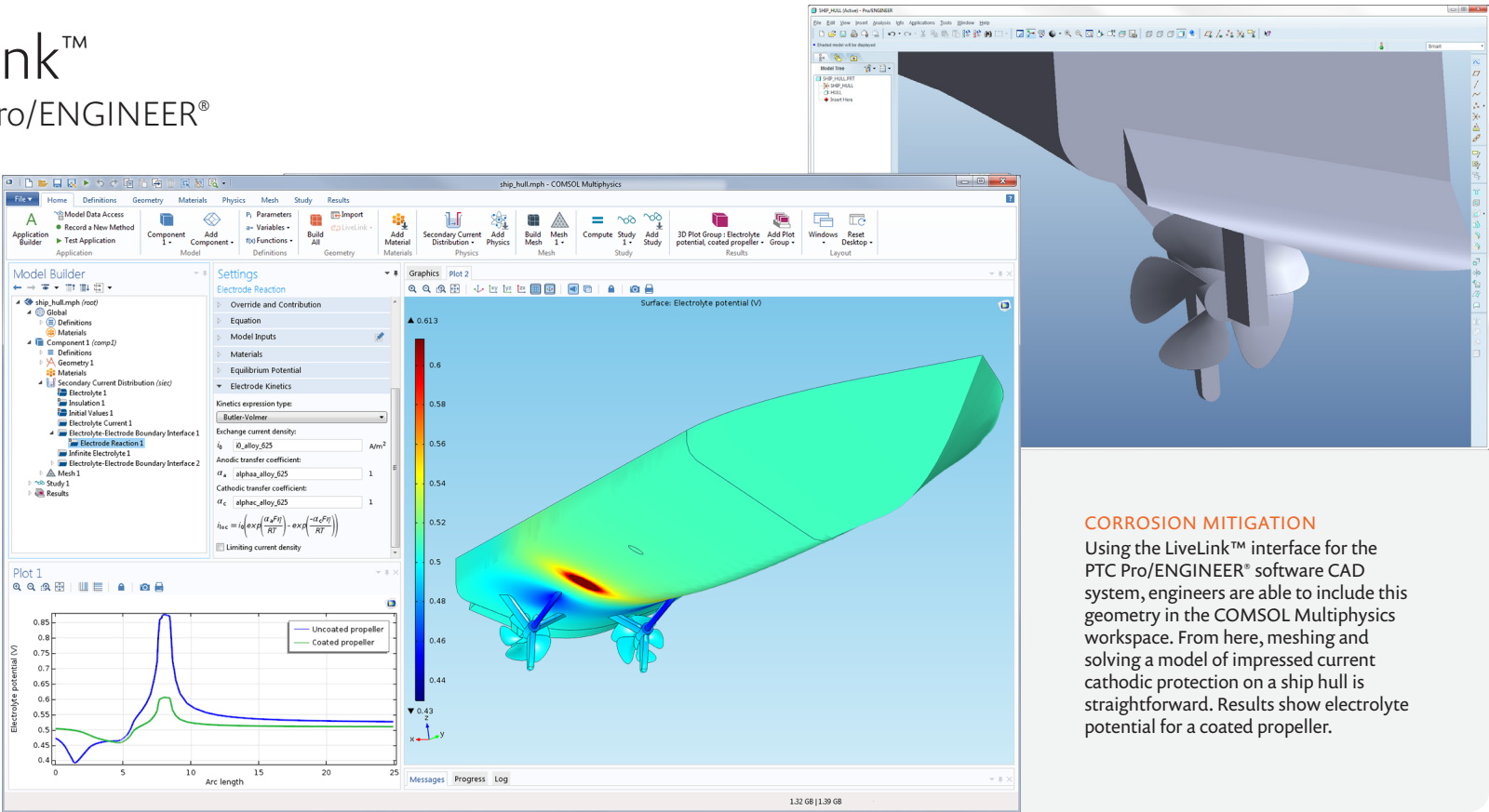
Powered by
PARASOLID

SPATIAL

PTC® PartnerAdvantage

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®



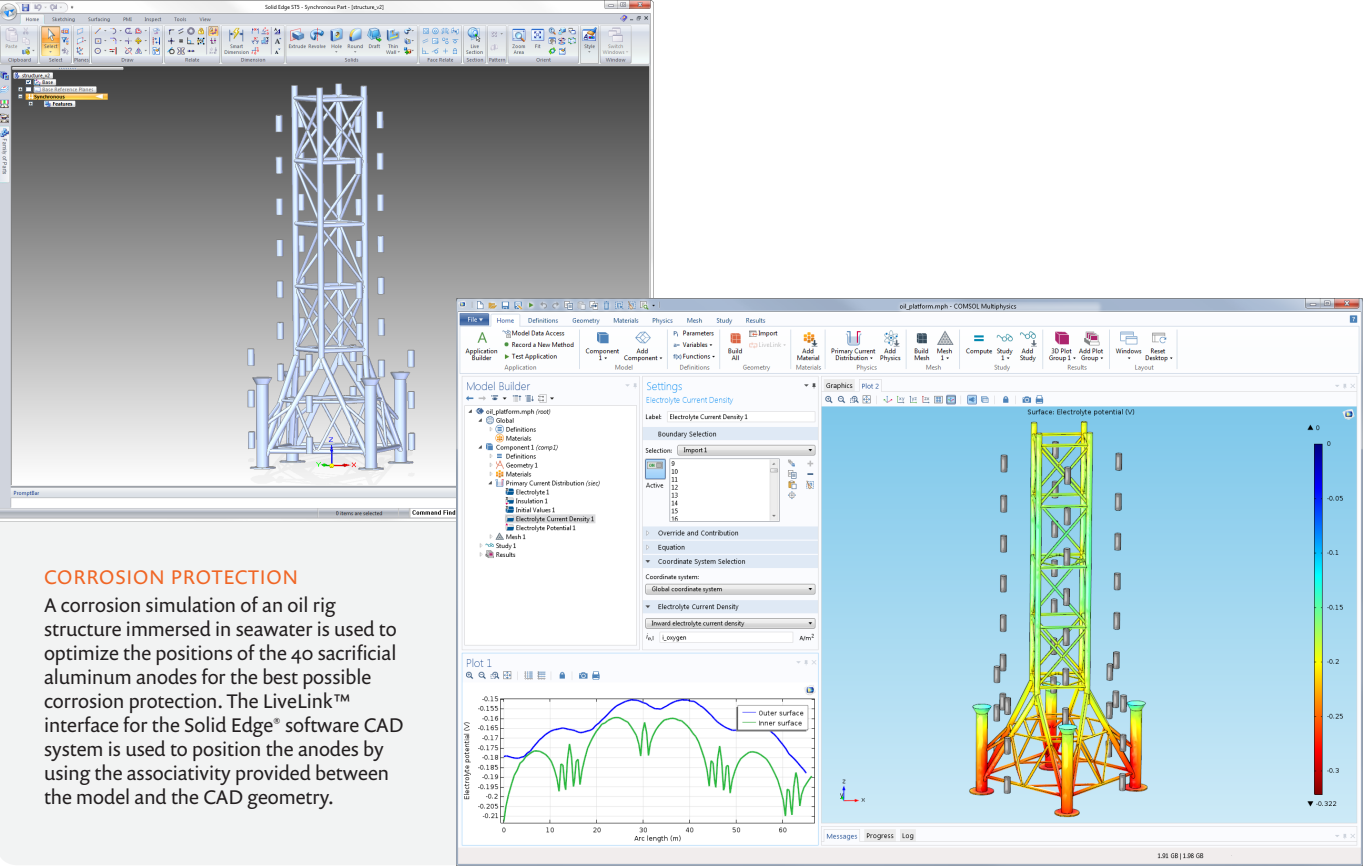
Powered by
PARASOLID

DS
SPATIAL

PTC® PartnerAdvantage

PTC and Pro/ENGINEER are trademarks or registered trademarks of PTC Inc. or its subsidiaries in the USA and in other countries.

LiveLink™
for Solid Edge®



Powered by
PARASOLID

DS
SPATIAL

Solution
Partner
PLM

SIEMENS

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.

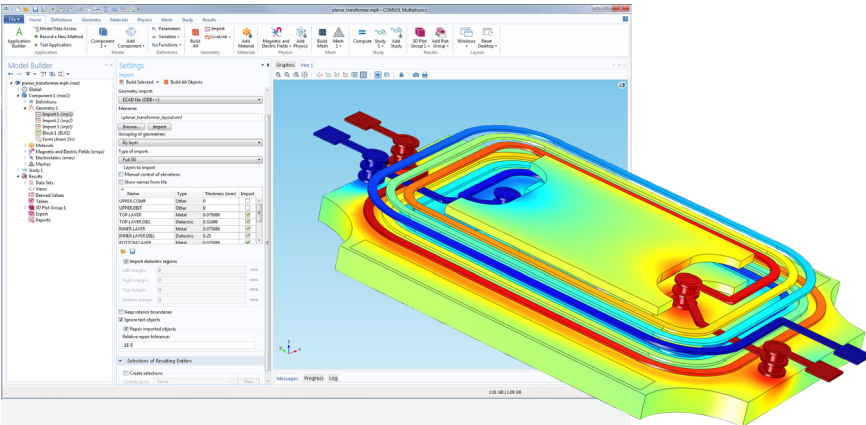
The ECAD Import Module supports the popular GDSII, ODB++, and NETEX-G file formats. As part of the import process, you can select which subset of cells, nets and layers to import. You can also edit layer thicknesses, control the geometric representation of bond wires, and include selected dielectric 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 is converted to a 3D geometry model.

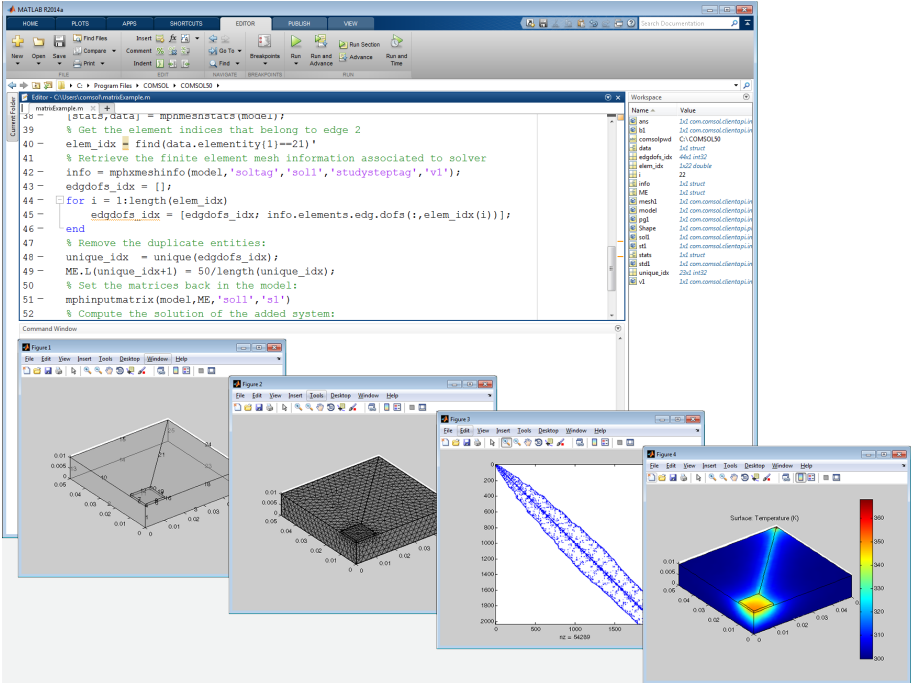
LiveLink™ for MATLAB®

LiveLink™ for MATLAB® allows the user to control a model both from MATLAB® software and from the COMSOL Desktop® — at the same time. This streamlines the workflow and you can use MATLAB® software as a scripting language for making updates or extracting results from the model and, at the same time, have the comfort of being able to view the model settings and results in the COMSOL Desktop®.

- Enhance your in-house MATLAB® software code with powerful multiphysics simulations
- Base your geometry modeling on probabilistic or image data
- Perform arbitrary statistical analysis on simulation results
- Use multiphysics models together with genetic or simulated annealing algorithms
- Export COMSOL models in state-space matrix format for incorporating in control systems
- Call MATLAB® software functions from the COMSOL Desktop®



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 vs. time, the finite element mesh, and the sparsity pattern of the stiffness matrix.

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). ODB++ is a trademark of Mentor Graphics Corporation.

LiveLink™ for Excel®

Extend your modeling capacity by running COMSOL Multiphysics® simulations from a spreadsheet with LiveLink™ for Excel®. Parameters and variables defined and modeled in COMSOL Multiphysics® are instantly available in the Microsoft® Excel® program and automatically synchronized to your COMSOL® model.

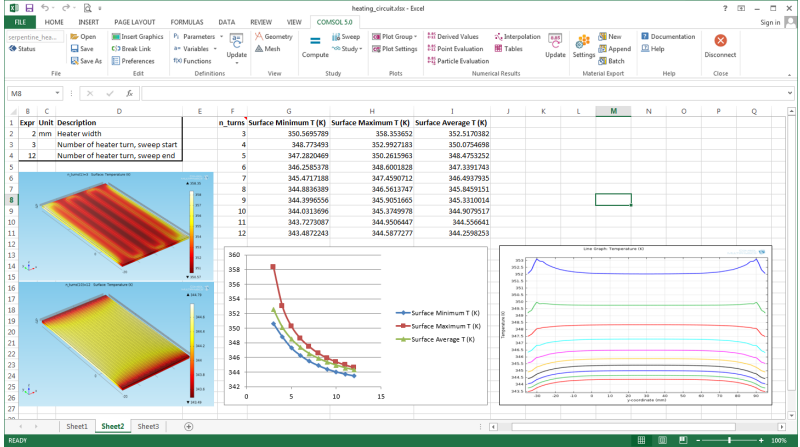
LiveLink™ for Excel® adds a COMSOL Multiphysics® tab to the Excel® ribbon for controlling the parameters, variables and mesh or for running a simulation. This allows for a simplified workflow where you only display and edit the most important simulation parameters, which can be changed on the fly or as part of an extensive parameter study controlled from an Excel® spreadsheet.

COMSOL Multiphysics images and graphs can be embedded into the spreadsheet environment. Numerical data can be imported from a COMSOL® model into Excel® for reporting and future analysis. In addition, LiveLink™ for Excel® adds the capability to import/export Excel® program files for parameter and variable lists in the COMSOL Desktop® environment. LiveLink™ for Excel® requires Excel® 2007, 2010 or 2013 for Windows® operating system.

SUPPORTED FILE FORMATS

- Microsoft Excel® 2013 workbook (.xlsx)
- Microsoft Excel® 2010 workbook (.xlsx)
- Microsoft Excel® 2007 workbook (.xls)

Microsoft, Excel and Windows are either registered trademarks or trademarks of Microsoft Corporation in the USA and/or other countries.



PARAMETERIZED STUDY

LiveLink™ for Excel® is here used for a parameterized multiphysics simulation of an 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 Excel® program for easy access to parameters, variables, functions, geometry, mesh, solvers, and results.

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 assure 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. And we're 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 (www.comsol.com/products/specifications), or integrate with CAD software (www.comsol.com/products/specifications/cad).

✓ Continuous Training

Training courses are offered frequently at locations worldwide. Available training covers everything from an introduction to COMSOL Multiphysics 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 Multiphysics to specific physics or applications. Videos and archived webinars are available at www.comsol.com/video.

✓ 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 (www.comsol.com/community/forums). Check out the COMSOL Blog (www.comsol.com/blogs) for posts on simulation topics including the electrical, mechanical, fluid, and chemical disciplines.

✓ Certified Consultants

COMSOL Certified Consultants are there to help when you do not have the resources to tackle modeling problems on your own. COMSOL Certified Consultants are renowned for their domain expertise and for their extensive experience using COMSOL Multiphysics.

✓ COMSOL Conference

Industry analysts have described the COMSOL Conference as the premier multiphysics event. Held in leading centers of science and technology world wide, 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.

License Options - COMSOL Multiphysics®

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

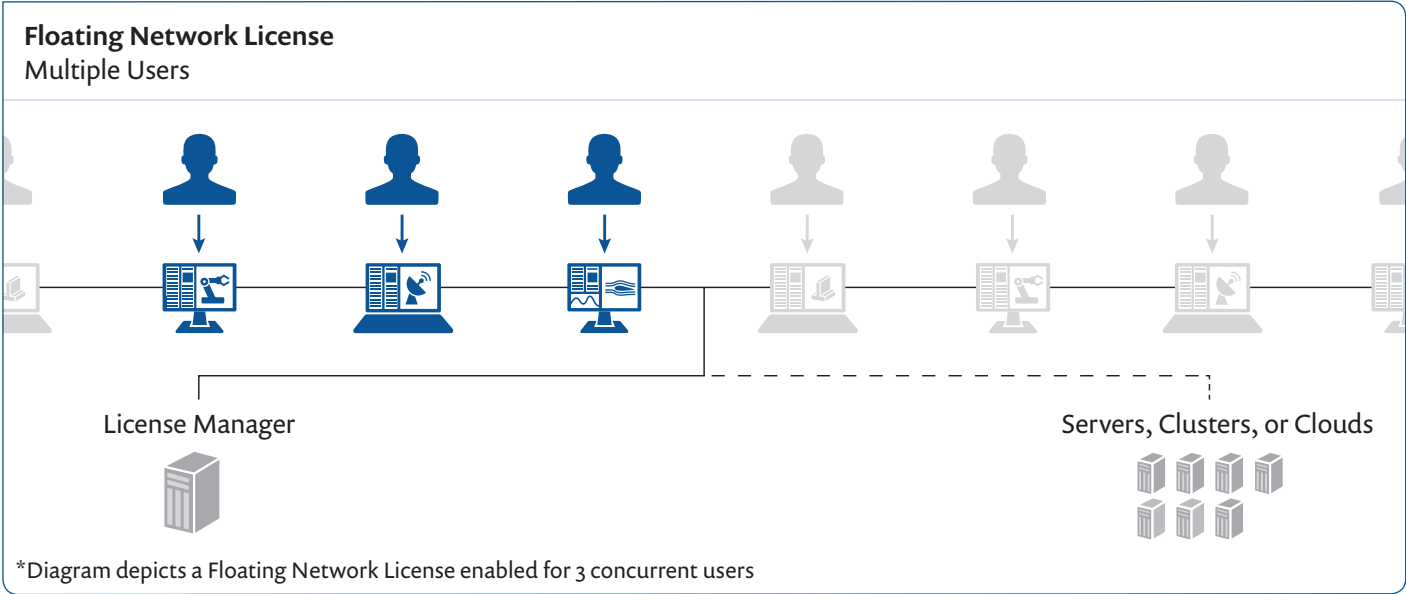
CPU-Locked Single User License (CPU): You can install the software on one computer and different users can take turns using COMSOL on that computer, one session at a time. Not for use over a network.

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, or 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 school network. Two teaching assistants can also use it to prepare class materials.

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

SUPPORTED PLATFORMS: Windows®, Linux®, OS X



High-Performance Computing: All license options include support for multicore computing with no limit on the number of cores. The Floating Network License includes additional support for cluster and cloud computing with no limit on the number of compute nodes.

License Options - COMSOL Server™

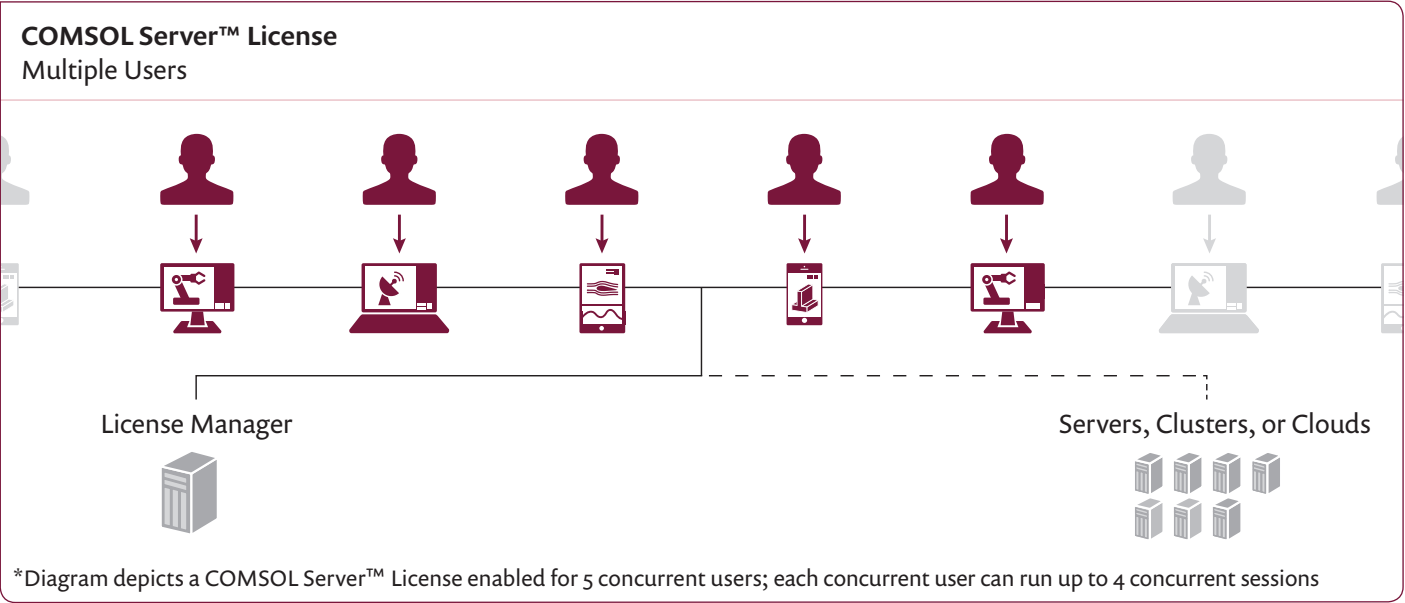
COMSOL Server License (CSL): Licensed on a concurrent user basis. Each concurrent user may run up to four simulation applications. The CSL version of the software enables you to run simulation apps built with COMSOL Multiphysics. The CSL allows you to host and run those applications within your organization 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 simulation applications. The ASL version of the software enables you to run simulation apps built with COMSOL Multiphysics. The ASL allows you to host and run those applications 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.

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

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



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

