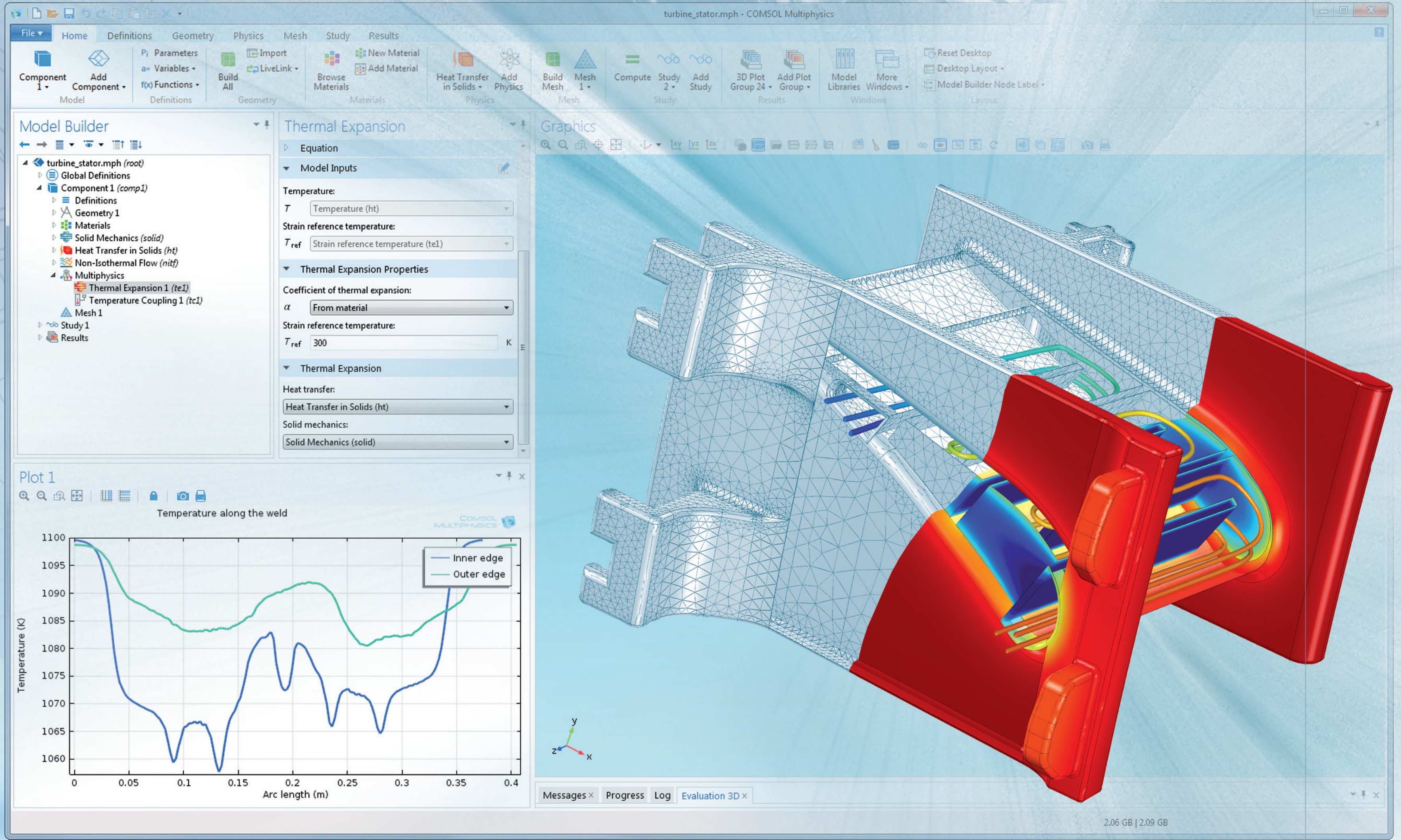
COMSOL MULTIPHYSICS®

PRODUCT BOOKLET

VERSION 4.4

COMSOL



Thermal Expansion

Equation

Model Inputs

Temperature:

T Temperature (ht)

Strain reference temperature:

T_{ref} Strain reference temperature (te1)

Thermal Expansion Properties

Coefficient of thermal expansion:

α From material

Strain reference temperature:

T_{ref} 300 K

Thermal Expansion

Heat transfer:

Heat Transfer in Solids (ht)

Solid mechanics:

Solid Mechanics (solid)

TURBINE STATOR BLADE

A stator blade in the turbine stage of a jet engine is heated by the combustion gases. To prevent the stator from melting, air is passed through a cooling duct in the blade. The resulting temperature gradients introduce significant stresses.

Multiphysics Simulation.

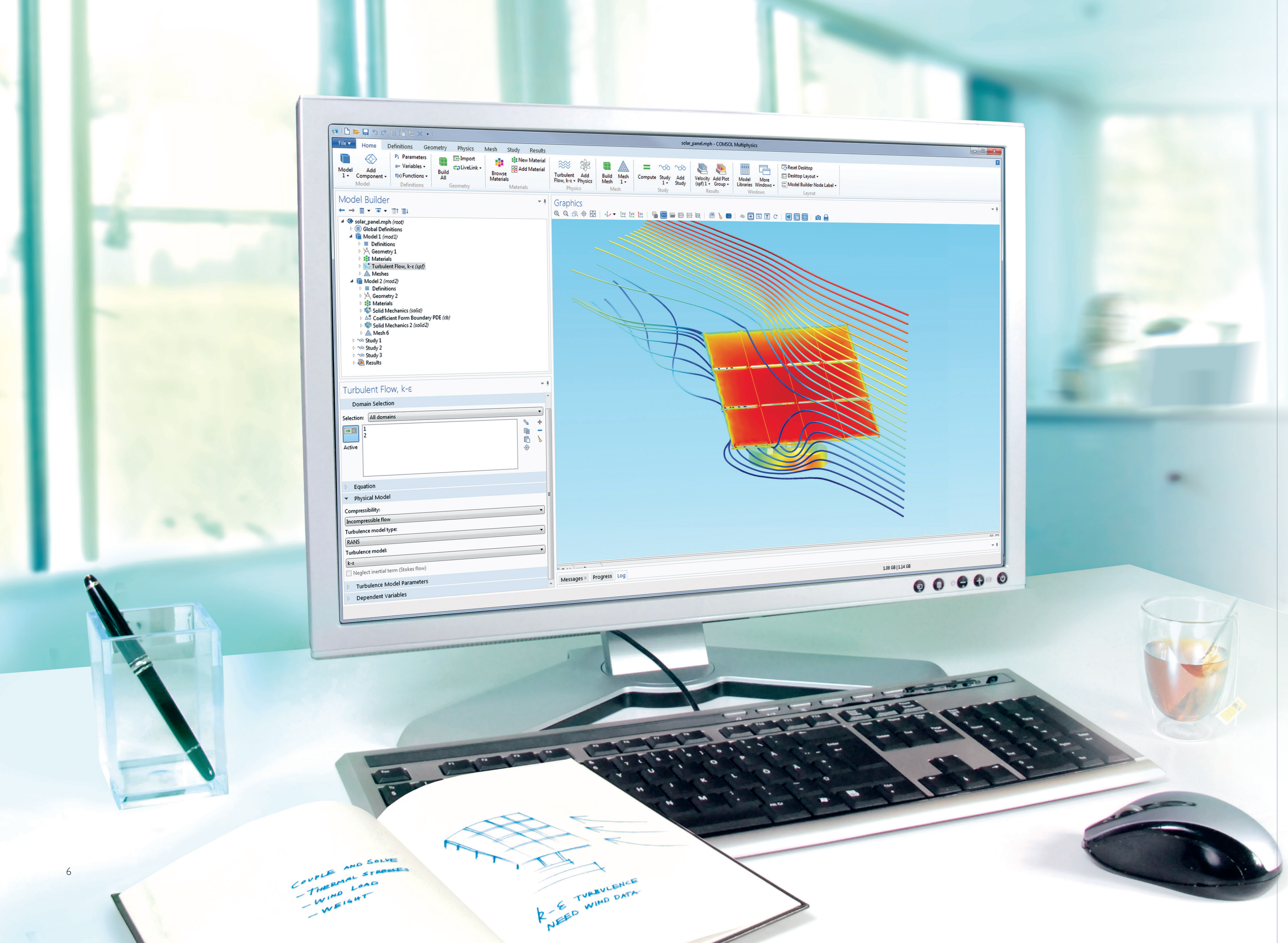
In today's fast-paced research and development culture, simulation power gives you the competitive edge. COMSOL Multiphysics® is the ideal tool to build simulations that accurately replicate the important characteristics of your designs. Its unparalleled ability to include all relevant physical effects that exist in the real world is known as multiphysics.

This approach delivers results—tangible results that save precious development time and spark innovation. COMSOL Multiphysics brings you this remarkable technology in an easy-to-use, intuitive interface, making it accessible to all engineers including designers, analysts, and researchers.

TABLE OF CONTENTS

© 2006-2014 COMSOL. The software is protected by U.S. Patents 7,519,518; 7,596,474; 7,623,991; 8,219,373; and 8,457,932. Patents pending. COMSOL, the COMSOL logo, COMSOL Multiphysics, COMSOL Desktop, and LiveLink are registered trademarks or trademarks of COMSOL AB. ACIS and SAT are registered trademarks of Spatial Corporation. AutoDesk, AutoCAD and Inventor are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. CATIA is a registered trademark of Dassault Systèmes S.A. or its subsidiaries in the US and/or other countries. Microsoft, Excel and Windows are are either registered trademarks or trademarks of Microsoft Corporation in the United States and/or other countries. Parasolid and Solid Edge are trademarks or registered trademarks of Siemens Product Lifecycle Management Software Inc. or its subsidiaries in the United States and in other countries. SolidWorks is a registered trademark of Dassault Systèmes SolidWorks Corp., PTC, Creo, Creo Parametric and Pro/ENGINEER are trademarks or registered trademarks of PTC Inc. or its subsidiaries in the U.S. and in other countries. MATLAB is a registered trademark of The MathWorks, Inc. Amazon Web Services, the "Powered by Amazon Web Services" logo, Amazon EC2 and Amazon Elastic Compute Cloud are trademarks of Amazon.com, Inc. or its affiliates in the United States and/or other countries. Mac and Macintosh are trademarks of Apple Inc., registered in the U.S. and other countries. Linux is a registered trademark of Linus Torvalds. Red Hat is a registered trademark of Red Hat, Inc. in the U.S. and other countries. CHEMKIN is a registered trademark of Reaction Design Corp. in the United States and other countries. Nastran is a registered trademark of NASA. Neither COMSOL nor any of the COMSOL products listed herein are affiliated with, endorsed by, sponsored by, or supported by any of these other trademark owners. Other product names, brand names or logos are trademarks or registered trademarks of their respective holders.

How do you verify and optimize your design?



> *"COMSOL Multiphysics provides an excellent platform that allows us to examine all physics within one easy-to-use environment and optimize system operation before we start building prototypes."*
>
> **Karl-Anders Weiss**, Fraunhofer Institute for Solar Energy Systems, Freiburg, Germany.

Through powerful simulation tools.

Companies are gravitating toward using simulation to drive the design. What is being realized are the advantages of simulation not simply to verify design decisions, but to explore and develop these new, innovative ideas; come up with viable concepts; and investigate all of their "what-ifs" in terms of use of materials or manufacturing methods.

A critical feature of these processes is making sure that the necessary tools are provided that let you quickly create and test new ideas. This is what COMSOL Multiphysics is offering: a single, integrated solution that can address the widest range of applications.

HEARING IMPLANT

Over the years, Cochlear Ltd. has helped more than 250,000 people in over 100 countries connect to a world of hearing. A new type of hearing implant, Codacs™, provides mechanical stimulations directly to the cochlea. Shown is a simulation of the electromagnetic fields within the balanced armature.

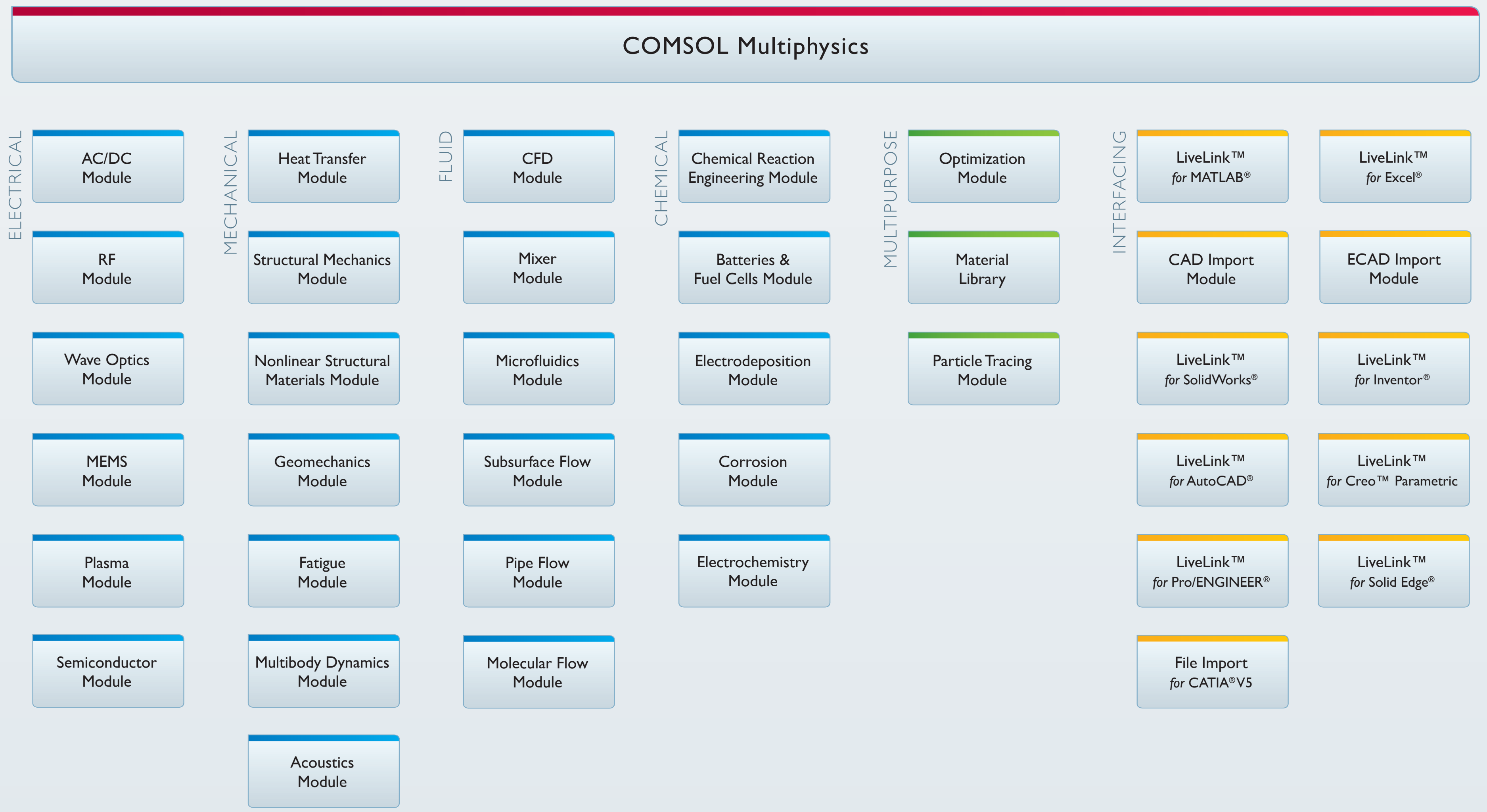
Model and pictures courtesy of Patrik Kennes, Cochlear Technology Centre, Belgium.

> *"The Direct Acoustic Cochlear Implant was developed from the ground up using COMSOL Multiphysics. Through COMSOL, we were able to avoid a time-consuming and costly trial-and-error design approach whereby we would have to build many prototypes to determine the appropriate part dimensions."*
>
> **Patrik Kennes,** Cochlear Technology Centre, Belgium.

With real world precision.

From its inception, the COMSOL® platform was designed to address multiphysics problems to help you predict what will happen to a desired level of accuracy. The software starts with first principles like transport phenomena, electromagnetic field theory, and solid mechanics as basic fibers. Then, in a practical and flexible user interface, you can weave these fibers together in a self-consistent way to solve your particular simulation needs. The end result is a model you can trust, because you control every aspect of the underlying physics.

Our suite of best-in-class solvers comes standard and includes automatic detection of your model's characteristics. Mesh generation is also an automated process. Powerful tools like parametric sweeps, interactive meshing, and custom solver sequences together with materials and other model properties help you quickly adapt to the ebbs and flows of your requirements.



PRODUCT SUITE

Electrical

Mechanical

Fluid

Chemical

Multipurpose

Interfacing

COMSOL Multiphysics

The COMSOL Multiphysics 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 a platform for the application specific modules.

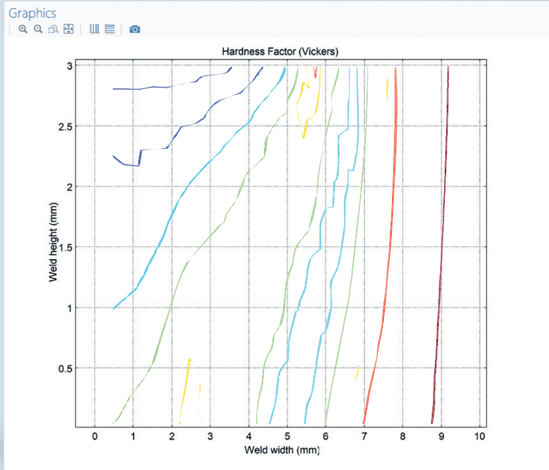
Model set-up is quick, thanks to a number of predefined physics interfaces for applications ranging from fluid flow and heat transfer to structural mechanics and electrostatics. Material properties, source terms, and boundary conditions can all be spatially varying, time-dependent, or functions of the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

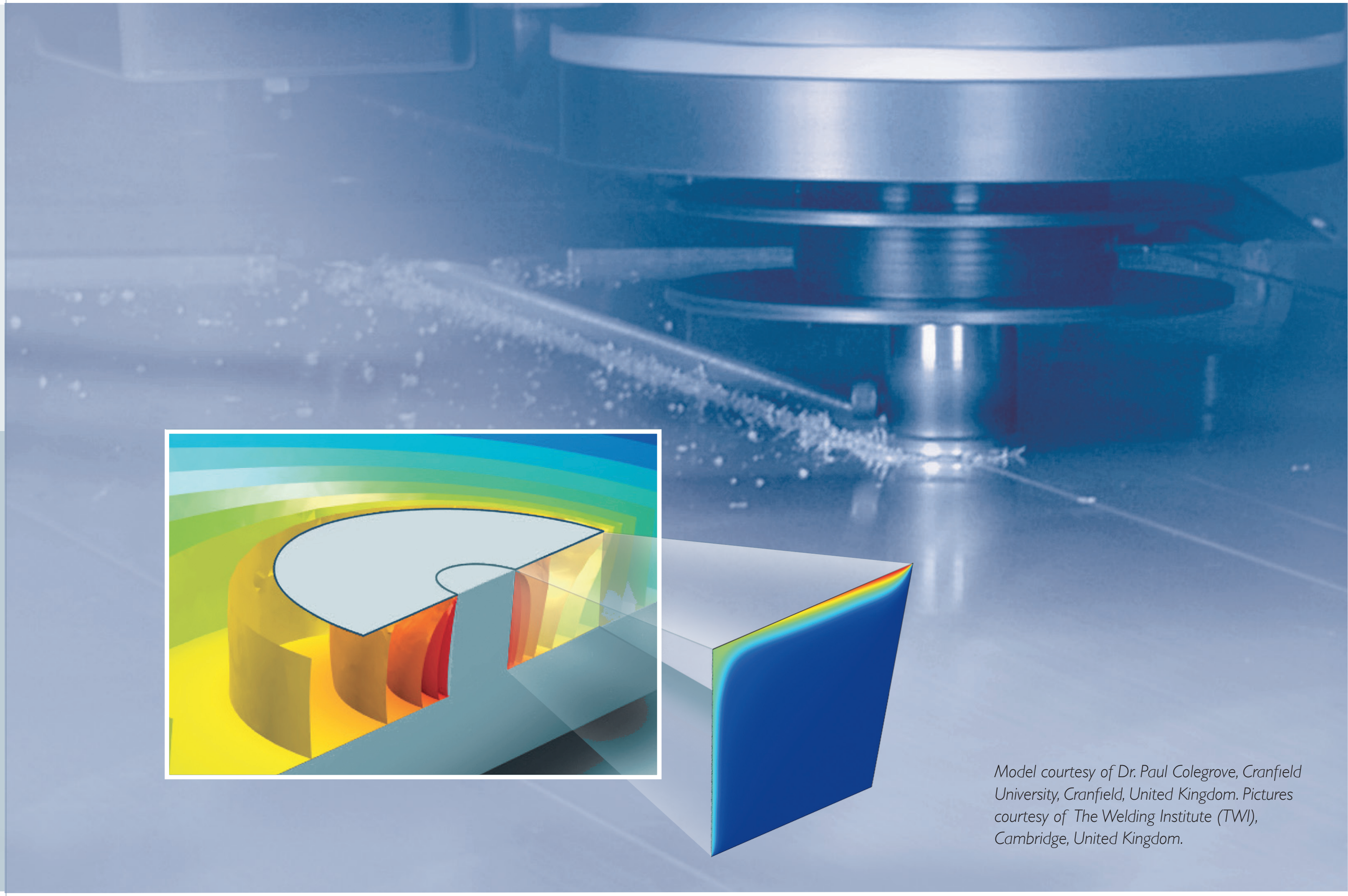
As an alternative to writing your own simulation code, the COMSOL Multiphysics user interface gives you the option to specify your own partial or ordinary differential equations (PDEs or ODEs) and link them with other physics interfaces. When combined with the CAD Import Module or one of the LiveLink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

WELDING

Friction stir welding is a solid phase welding process which 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 with friction and heat transport. The graph shows the hardness factors in the weld as calculated from the model.

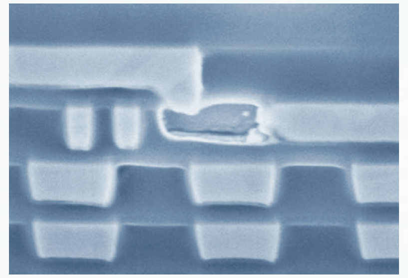




Model courtesy of Dr. Paul Colegrove, Cranfield University, Cranfield, United Kingdom. Pictures courtesy of The Welding Institute (TWI), Cambridge, United Kingdom.

HIGHLIGHTS

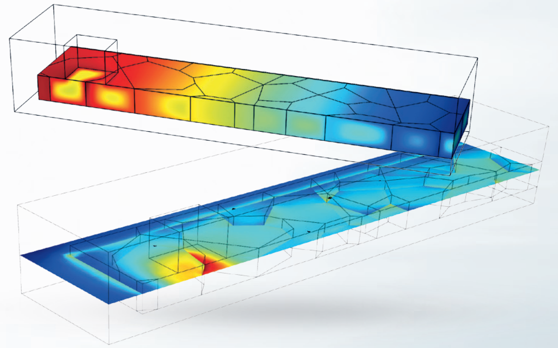
- Animation, image, and data export
- Automatic, swept, and boundary-layer meshing
- Boundary and volume integration
- Component couplings for linking 1D, 2D, and 3D model components
- Core physics interfaces
- Fast-Fourier transform (FFT)
- Interpolation table import for 1D, 2D, and 3D data sets
- Linear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

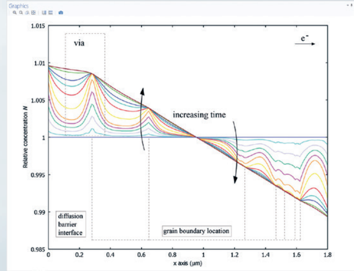


ELECTRONICS

Model predicting a copper line interconnect failure. Model 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.





ALL INCLUSIVE.

Practical simulation software—the kind that gets results—must be straightforward to use regardless of your modeling experience, while being constructive and powerful enough to achieve your objectives. We've incorporated these qualities into the all-inclusive COMSOL Desktop® environment.

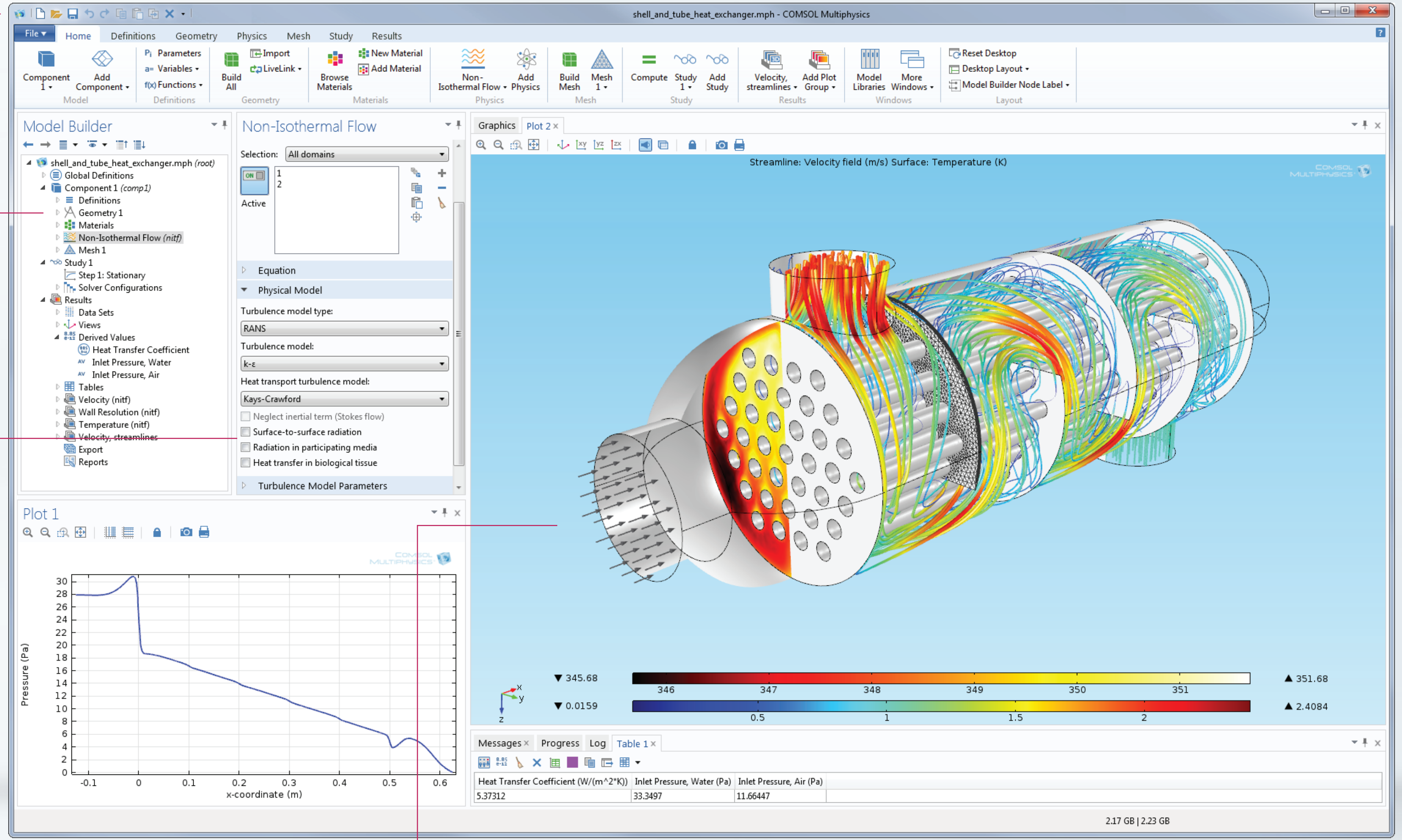
This sleek interface gives you full insight and control over the modeling process as it concisely reflects the COMSOL Multiphysics architecture. The modeling process is integrated and intuitive, guiding you quickly through the building of your simulations. The COMSOL Desktop structure is uniform throughout, no matter which physics or application is being simulated. This encourages cross-discipline collaboration so engineering teams can develop better models faster.

> *"COMSOL Multiphysics version 4 is a significant advance of simulation power with its fresh, new interface and insightful approach to the overall management of [simulation] data,"*
>
> **Bill Vetterling,** ZINK Imaging, Bedford, Massachusetts.

COMSOL DESKTOP
Customize the appearance by positioning the docked windows to your liking.

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

SETTINGS
Through its integration with the Graphics window, you can control and update model properties in the Settings window.



Model Builder

- shell_and_tube_heat_exchanger.mph *(root)*
  - Global Definitions
  - Component 1 *(comp1)*
    - Definitions
    - Geometry 1
    - Materials
    - Non-Isothermal Flow *(nitf)*
    - Mesh 1
  - Study 1
    - Step 1: Stationary
    - Solver Configurations
  - Results
    - Data Sets
    - Views
    - Derived Values
      - Heat Transfer Coefficient
      - Inlet Pressure, Water
      - Inlet Pressure, Air
    - Tables
    - Velocity (nitf)
    - Wall Resolution (nitf)
    - Temperature (nitf)
    - Velocity, streamlines
    - Export
    - Reports

Non-Isothermal Flow

Selection: All domains
Active
1
2
Equation
Physical Model
Turbulence model type: RANS
Turbulence model: k-ε
Heat transport turbulence model: Kays-Crawford
☐ Neglect inertial term (Stokes flow)
☐ Surface-to-surface radiation
☐ Radiation in participating media
☐ Heat transfer in biological tissue
Turbulence Model Parameters

Heat Transfer Coefficient (W/(m^2*K))	Inlet Pressure, Water (Pa)	Inlet Pressure, Air (Pa)
5.37312	33.3497	11.66447

GRAPHICS
Ultrafast graphic presentation, stunning visualization, and multiple plots.

STRUCTURED.

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

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

> *"The COMSOL Multiphysics environment is so easy to understand and use, it didn't take long before several of us could model with it and collaborate."*
>
> **Jan Hoffman**, Amsterdam Water Supply, Amsterdam, The Netherlands.

MATERIALS

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

GEOMETRY

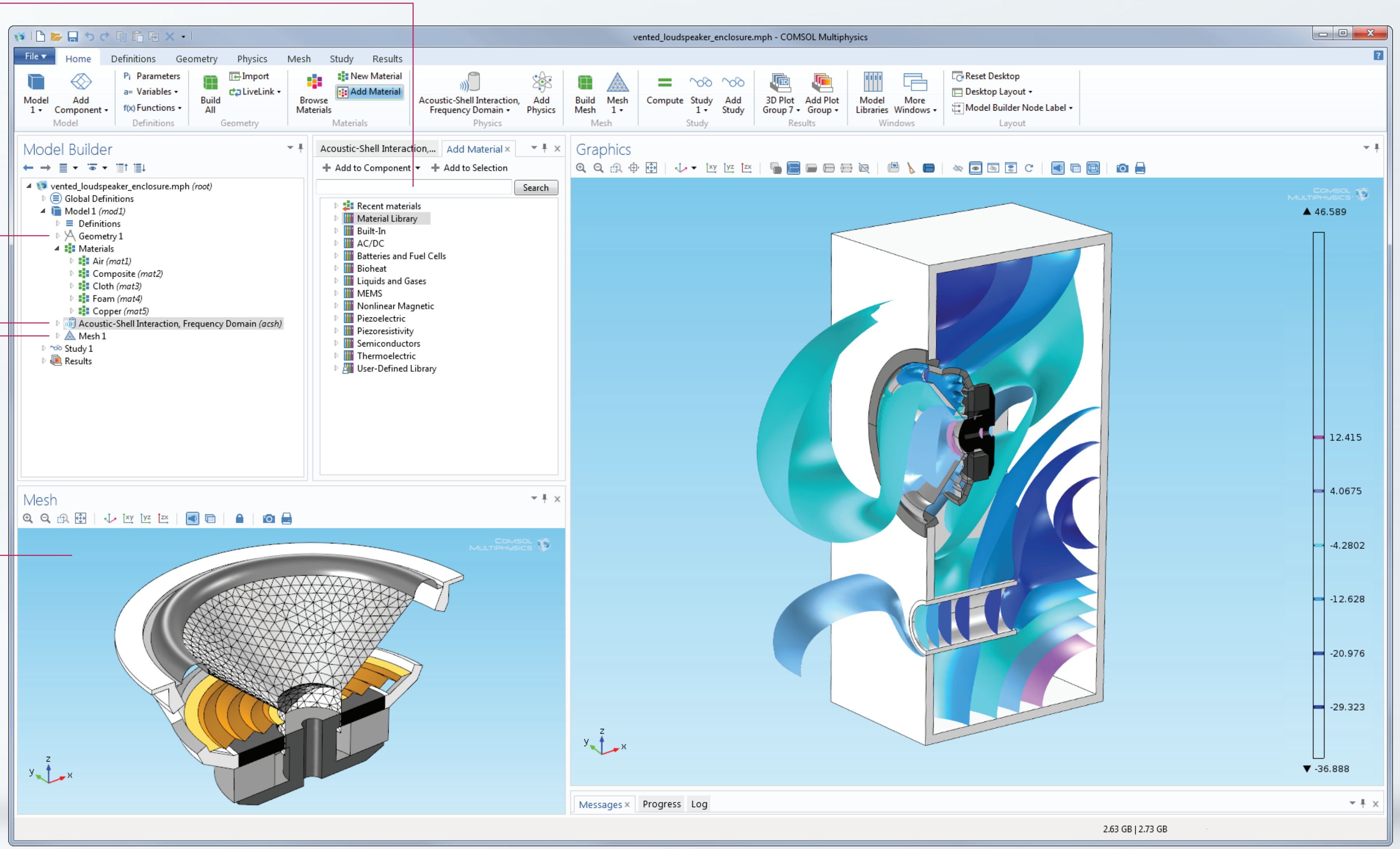
Use the built-in drawing tools or import files from a CAD package to create your geometries in the Geometry branch.

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.



IN CONTROL.

The Model Builder allows instant access to any part of the model settings. You can even record the steps of your set-up as a sequence of nodes, pausing the process at any stage to investigate, refine and optimize a modeling feature.

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.

MODEL WIZARD

Quick setup of your model is done in the Model Wizard by selecting one or more physics interfaces.

MODEL LIBRARY

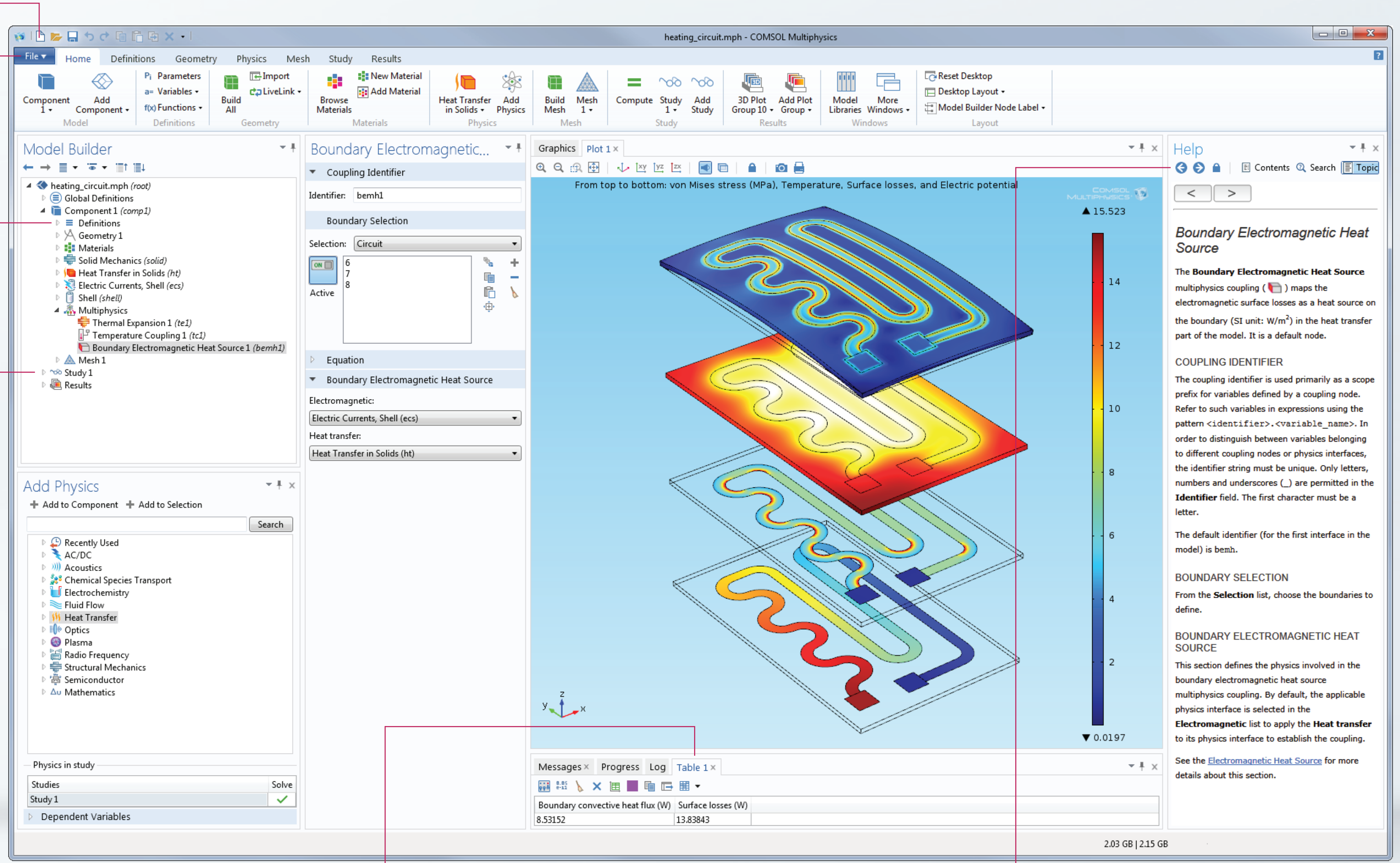
The Model Library consists of documented examples from all fields of engineering and science.

DEFINITIONS

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

STUDY

Configure parametric studies and other sequences of Solvers.



Identifier: bemh1

Selection: Circuit

6
7
8

Electromagnetic: Electric Currents, Shell (ecs)

Heat transfer: Heat Transfer in Solids (ht)

Boundary convective heat flux (W)	Surface losses (W)
8.53152	13.83843

Boundary Electromagnetic Heat Source

The **Boundary Electromagnetic Heat Source** multiphysics coupling ( ) maps the electromagnetic surface losses as a heat source on the boundary (SI unit: W/m²) in the heat transfer part of the model. It is a default node.

COUPLING IDENTIFIER

The coupling identifier is used primarily as a scope prefix for variables defined by a coupling node. Refer to such variables in expressions using the pattern `<identifier>.<variable_name>`. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the identifier string must be unique. Only letters, numbers and underscores (_) are permitted in the **Identifier** field. The first character must be a letter.

The default identifier (for the first interface in the model) is `bemh`.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define.

BOUNDARY ELECTROMAGNETIC HEAT SOURCE

This section defines the physics involved in the boundary electromagnetic heat source multiphysics coupling. By default, the applicable physics interface is selected in the **Electromagnetic** list to apply the **Heat transfer** to its physics interface to establish the coupling.

See the Electromagnetic Heat Source for more details about this section.

TABLE

Displays tables with sampled field values selected by clicking in the Graphics window. The tables also display integral and variable evaluations defined in the Results branch.

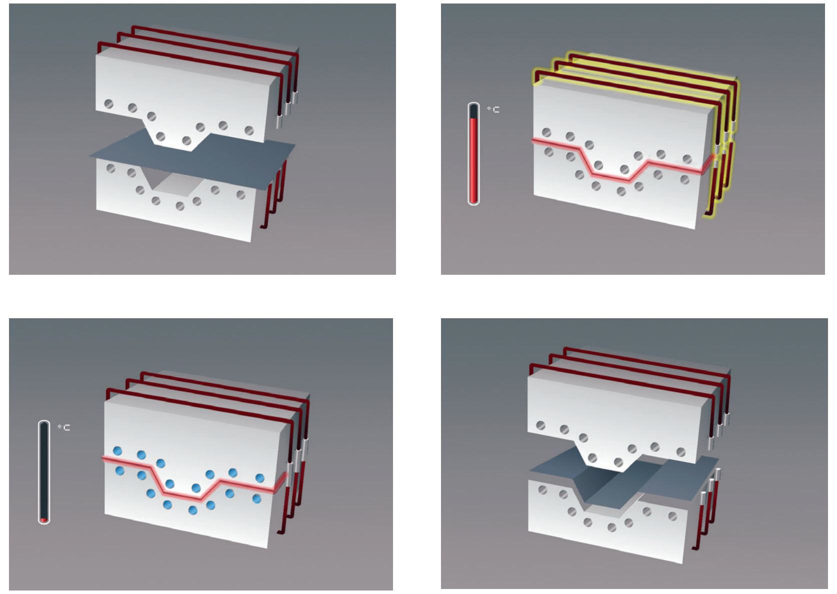
DYNAMIC HELP

Continually updated with online access to the Knowledgebase and Model Gallery. The Help feature enables easy browsing with extended search functionality.

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.



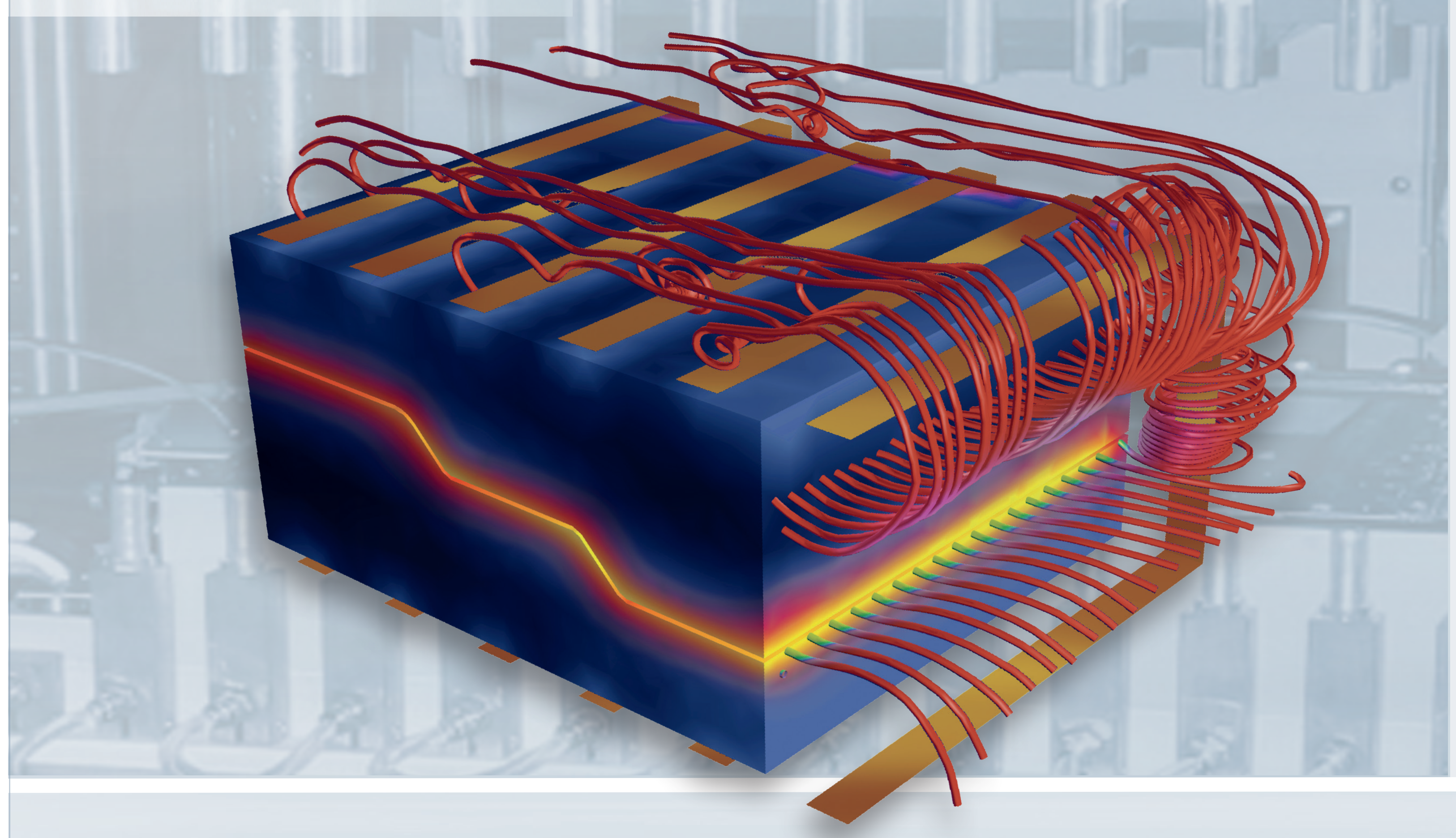
COMPOSITES MANUFACTURING

A sheet of composite material is placed within a mold that then 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.

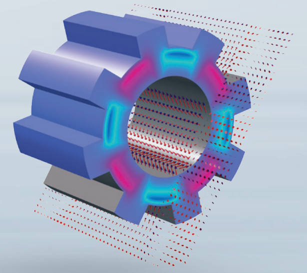


HIGHLIGHTS

- 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

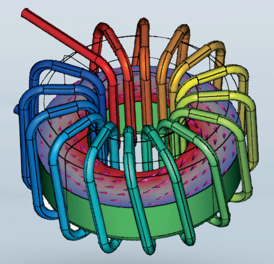
POWER ENGINEERING

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 is the magnetic fields inside and around the generator.



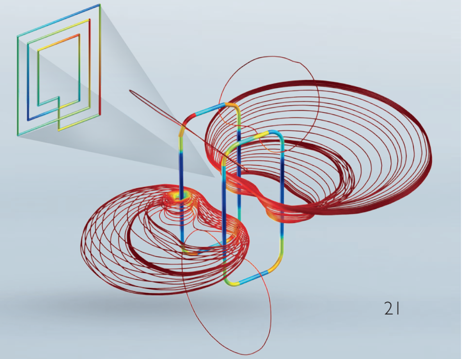
ELECTROMAGNETIC INDUCTOR

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



RFID TAGS

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



RF Module

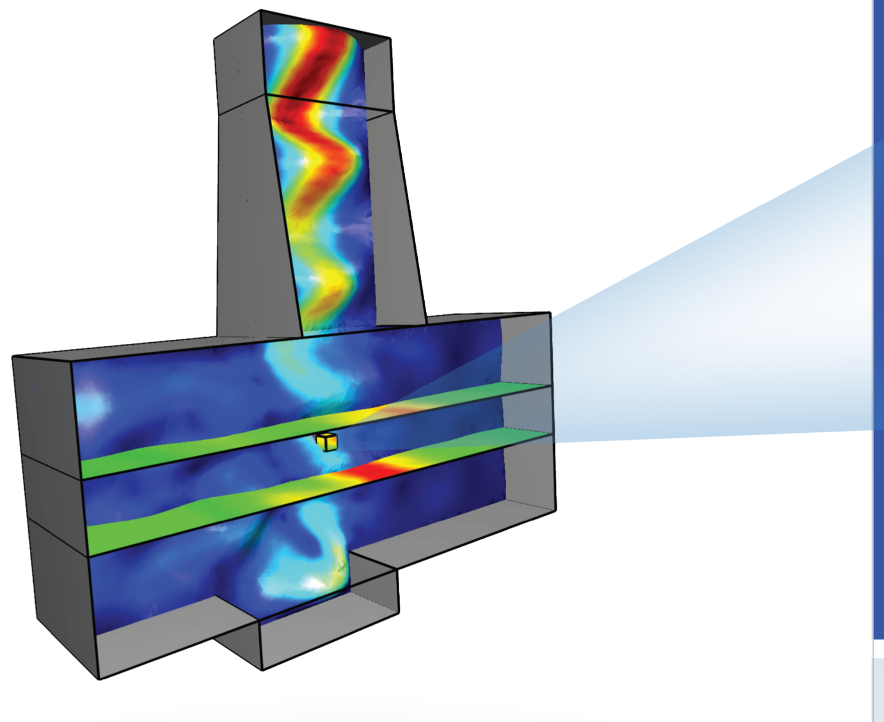
Modeling in the RF and microwave regimes requires modeling the propagation of electromagnetic waves in and around structures that can be metallic, dielectric, gyromagnetic, or even metamaterials with engineered properties. The RF Module offers you the tools to meet this challenge by including port and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

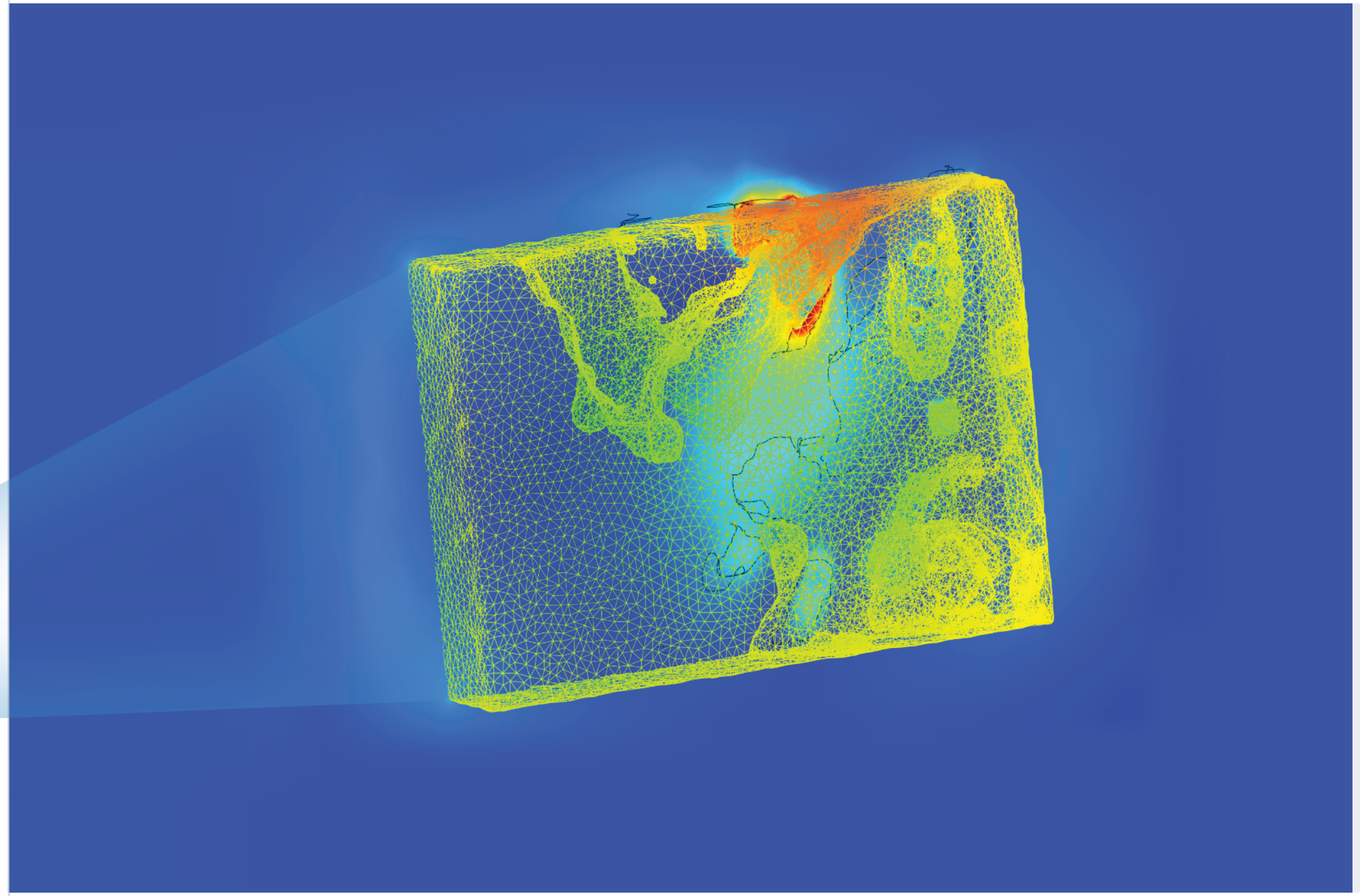
The RF Module completes the modeling experience by providing S-parameter computation and far-field analysis. Taken together with the unsurpassed ability of COMSOL Multiphysics to couple to other physics, such as electromagnetic RF heating, stress, and deformation effects, you have the industry's leading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

MINERAL PROCESSING

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

Model courtesy of Jan Przybyla, e2v, Chelmsford, UK.



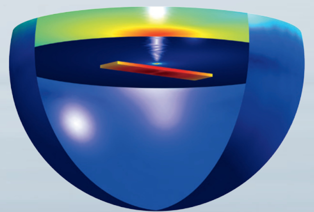


HIGHLIGHTS

- 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

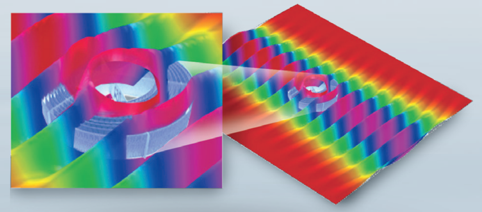
OIL PROSPECTING

The CSEM method has become popular within 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.

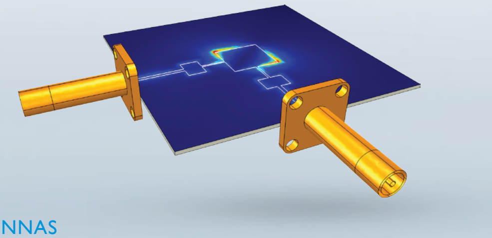


MATERIALS SCIENCE

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.

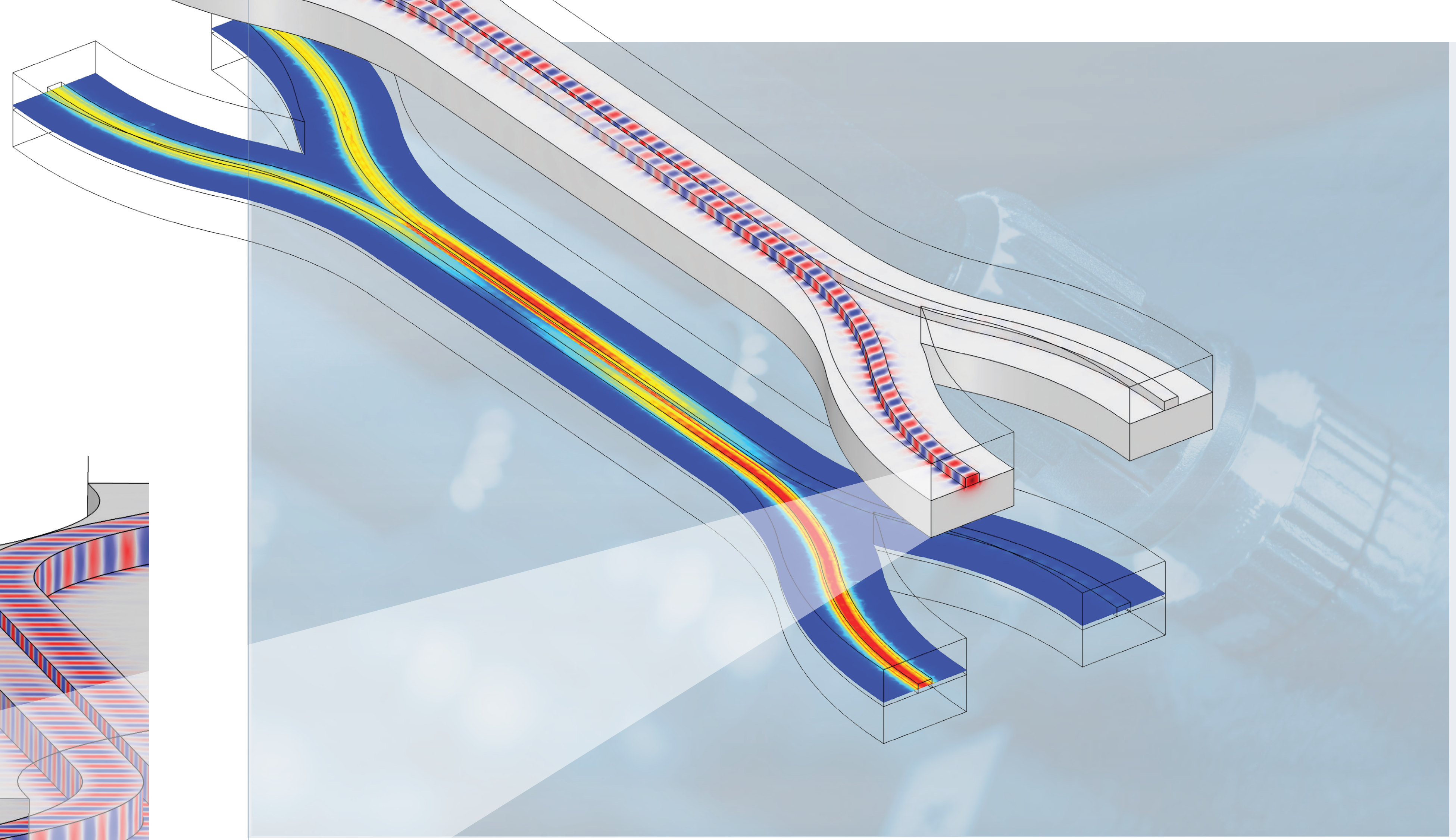


Model courtesy of Cummer and Schurig, Duke University, Durham, North Carolina.



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.

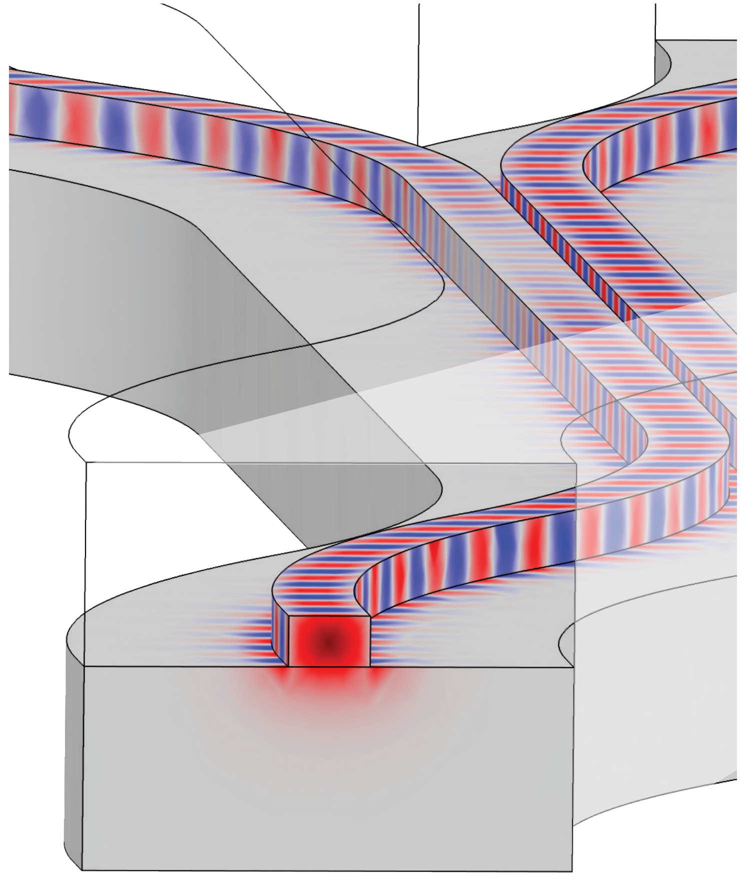


DIRECTIONAL COUPLER
Model of a directional coupler formed from two interacting waveguides. The left waveguide is excited and the simulation results reveal the optical coupling between the waveguides through visualization of the electric field.

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.

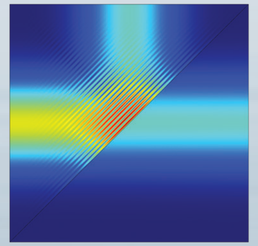
Support for generic anisotropic refractive index, permittivity, or permeability tensors allows for optical media that includes gyromagnetic materials or metamaterials with engineered properties. Several 2D and 3D formulations are available in the Wave Optics Module for eigenfrequency mode analysis, frequency-domain,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



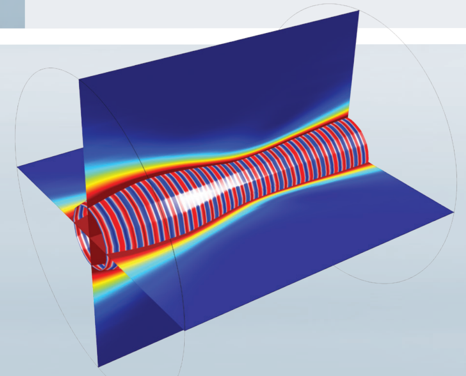
HIGHLIGHTS

- 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

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.



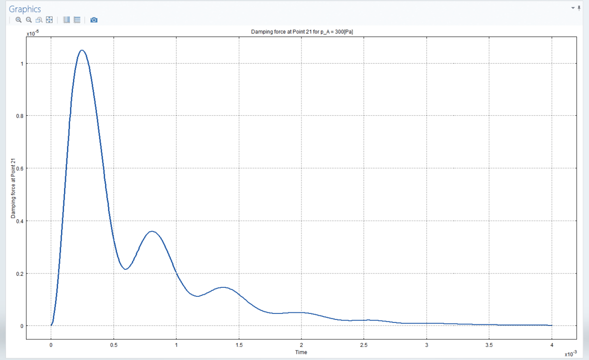
SELF-FOCUSING
A Gaussian beam is launched into a BK-7 optical glass. The material has an intensity-dependent refractive index. For the center of the beam, the refractive index is the largest. This induced refractive index profile counteracts the diffractive effects and actually focuses the beam. Self-focusing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

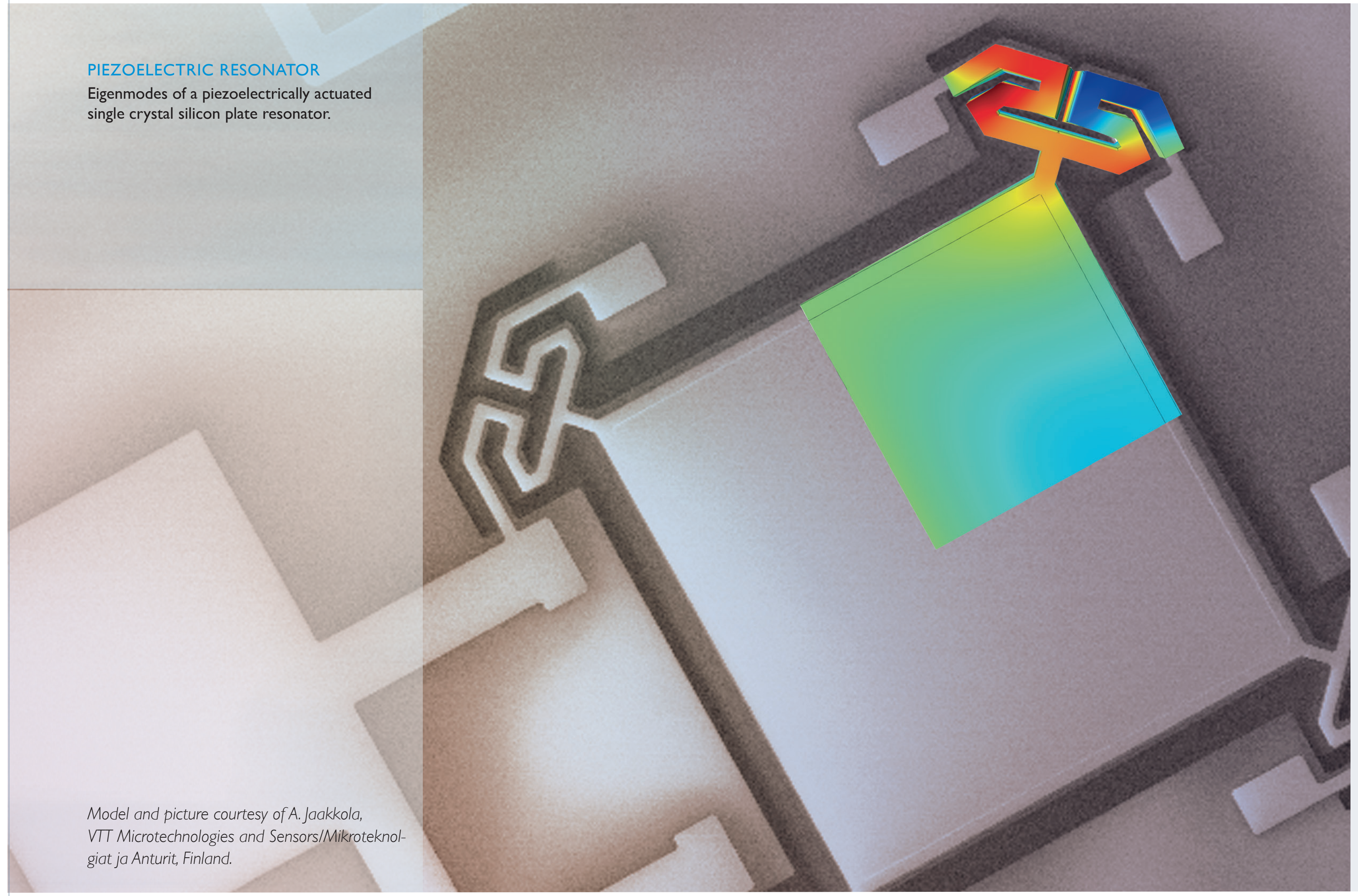


MEMS Module

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. MEMS devices and sensors may even utilize multiphysics phenomena for its very function or for increased sensitivity. To this end, the MEMS Module provides user 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 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 available in COMSOL 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.





PIEZOELECTRIC RESONATOR

Eigenmodes of a piezoelectrically actuated single crystal silicon plate resonator.

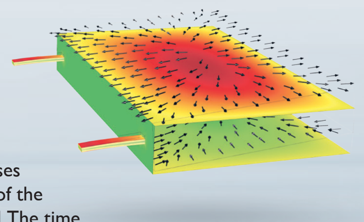
Model and picture courtesy of A. Jaakkola, VTT Microtechnologies and Sensors/Mikroteknologiat ja Anturit, Finland.

HIGHLIGHTS

- Accelerometers
- Actuators
- Bulk Acoustic Wave (BAW) devices
- Cantilever beams
- Fluid-structure interaction (FSI)
- 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

ACCELEROMETERS

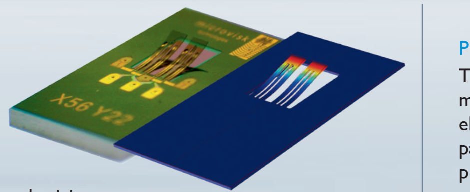
This example depicts the coupling of squeezed film gas damping to the mechanical displacements in a MEMS accelerometer. The direction of the gas flow and the von Mises stresses on the face and arms of the sensor are shown in the model. The time series plot to the left shows the gradual reduction of the damping force.



ACTUATORS & SENSORS

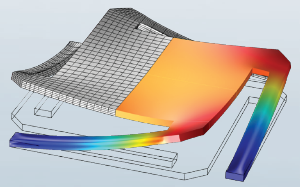
A two arm thermal actuator is activated through thermal expansion, where the temperature profile is plotted on the resulting deformed shape. Relative visco-elasticity changes during the clotting of blood are then measured through deflections of the micro-cantilevers.

Model courtesy of Slava Djokov / Neal McLoughlin, Microvisk Technologies, North Wales, UK.



PRESTRESSED MICROMIRROR

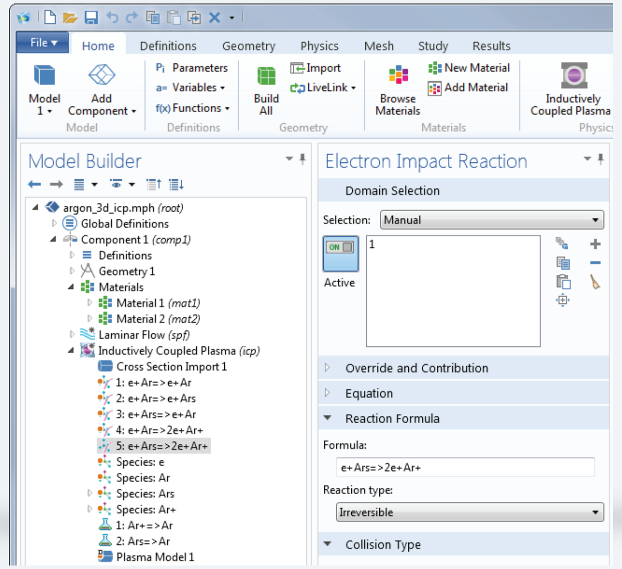
The lift-off of this prestressed plated micromirror is controlled electrostatically. Results from a parametric study reveal how the prestress affect the displacements.



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 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 to a 1D, 2D, or 3D model, 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. The module is accompanied by a suite of tutorial and industrially relevant models, which serve as both instructional examples and a foundation for future work.



Model Builder

- argon_3d_icp.mph *(root)*
  - Global Definitions
  - Component 1 *(comp1)*
    - Definitions
    - Geometry 1
    - Materials
      - Material 1 *(mat1)*
      - Material 2 *(mat2)*
    - Laminar Flow *(spf)*
    - Inductively Coupled Plasma *(icp)*
      - Cross Section Import 1
      - 1: e+Ar=>e+Ar
      - 2: e+Ar=>e+Ars
      - 3: e+Ars=>e+Ar
      - 4: e+Ar=>2e+Ar+
      - 5: e+Ars=>2e+Ar+
      - Species: e
      - Species: Ar
      - Species: Ars
      - Species: Ar+
      - 1: Ar+=>Ar
      - 2: Ars=>Ar
      - Plasma Model 1

Electron Impact Reaction

Domain Selection

Selection: Manual

1

Override and Contribution

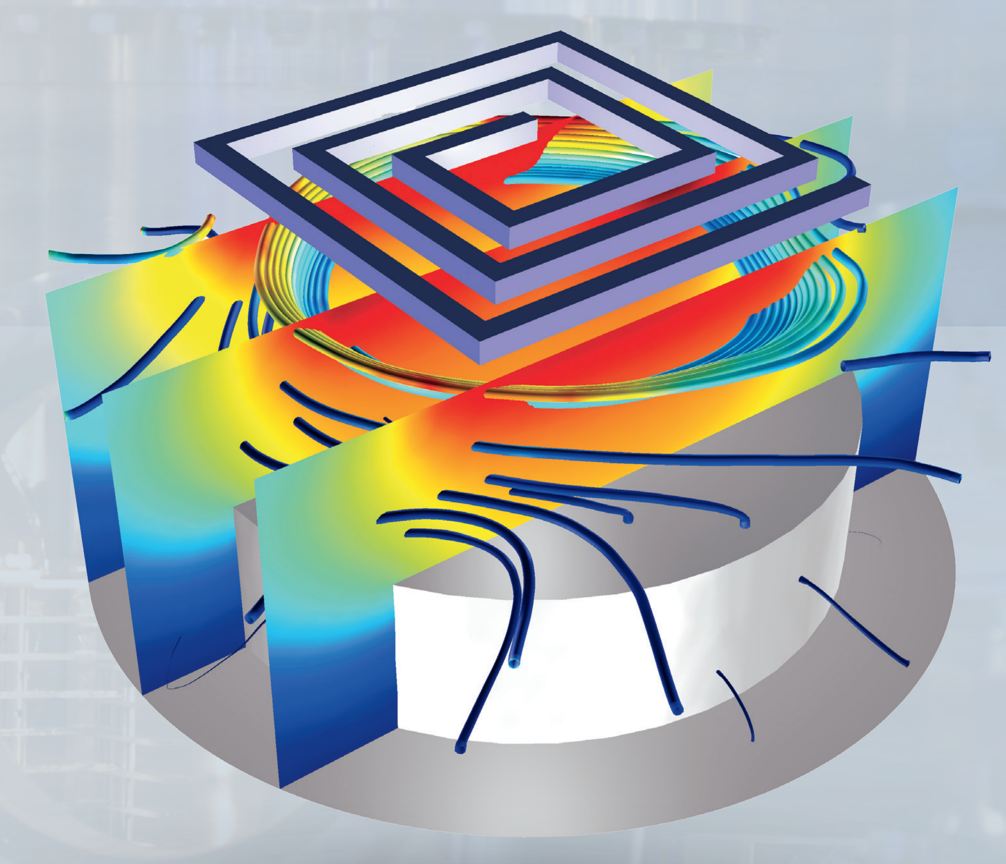
Equation

Reaction Formula

Formula: e+Ars=>2e+Ar+

Reaction type: Irreversible

Collision Type



3D MODEL OF AN ASYMMETRIC ICP REACTOR
The plot shows the mean electron energy (slice) and the electron current density (streamlines).

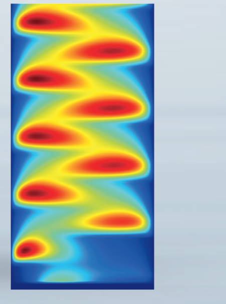
HIGHLIGHTS

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

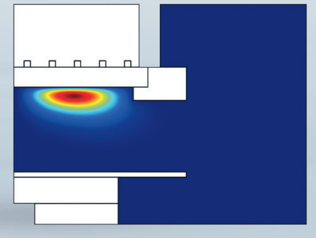
* *Together with the AC/DC Module*

** *Together with the RF Module*

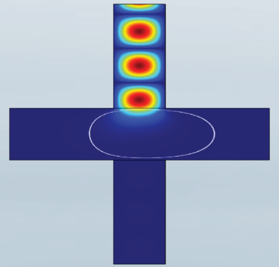
DIELECTRIC BARRIER DISCHARGE
Excited argon atoms can spontaneously decay back to the ground state releasing a photon in the ultraviolet spectrum. The plot to the right shows the concentration of excited argon atoms in a dielectric barrier discharge.



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.

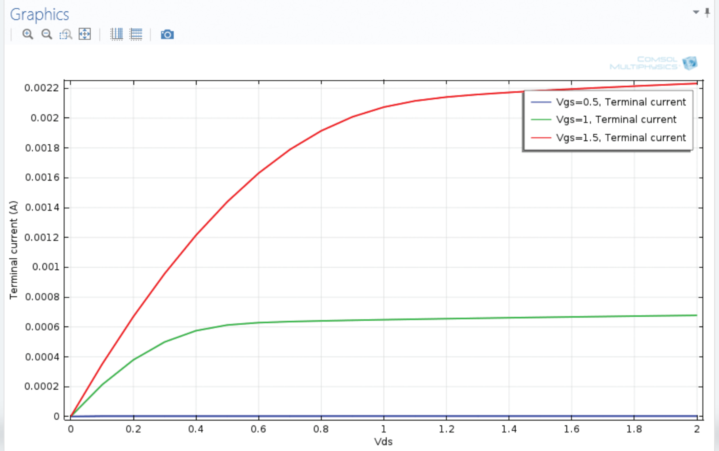


Semiconductor Module

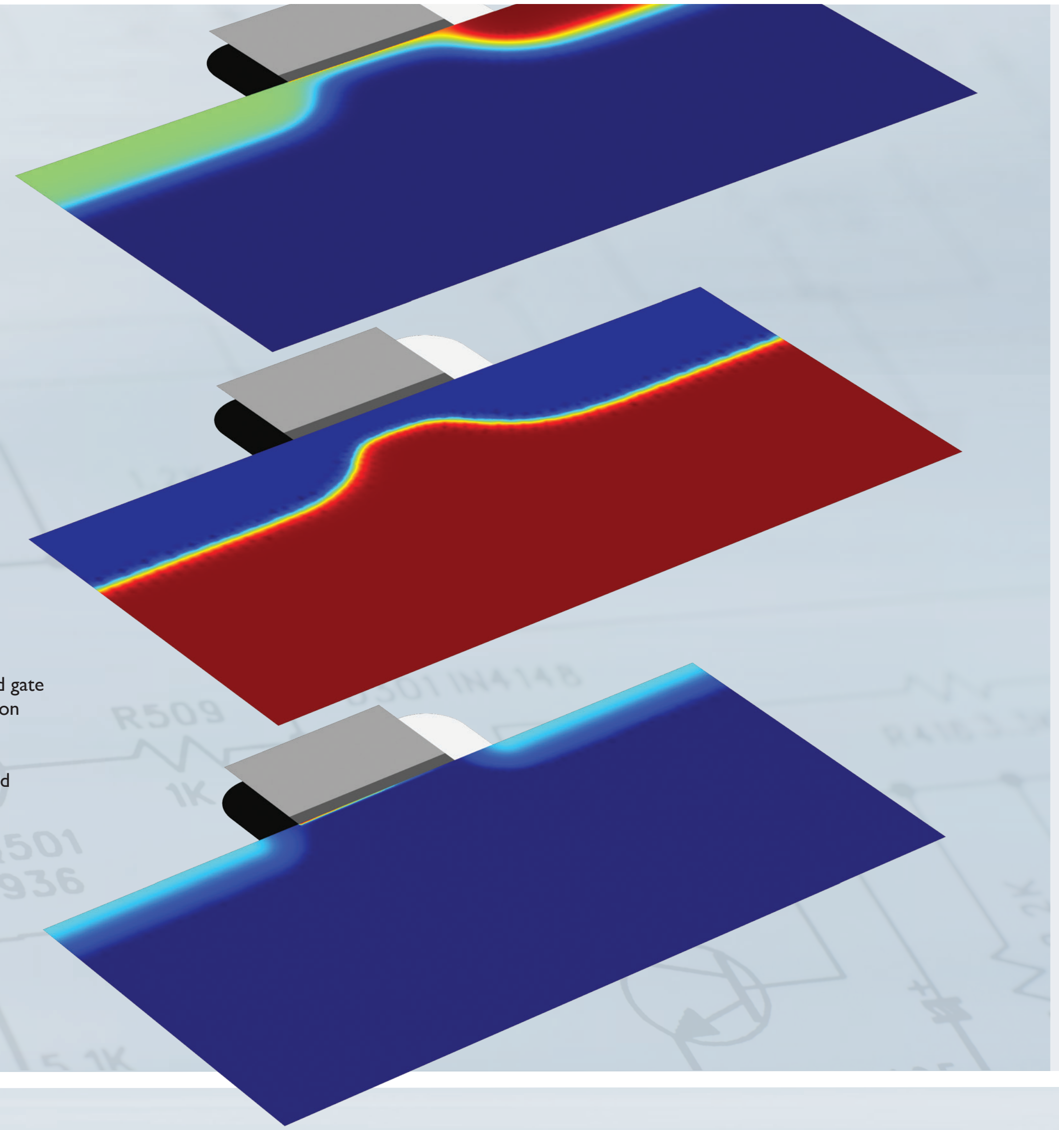
The Semiconductor Module allows for detailed analysis of semiconductor device operation at the fundamental physics level. The module is based on the drift-diffusion equations with isothermal and non-isothermal transport models. Two numerical methods are provided: the finite volume method with Scharfetter-Gummel upwinding and a Galerkin least-squares stabilized finite element method. The module provides an easy-to-use interface, greatly simplifying the task of semiconductor device simulation on the COMSOL platform.

Models for semiconducting and insulating materials in addition to boundary conditions for ohmic contacts, Schottky contacts, and gates are provided as dedicated features within the Semiconductor Module. Enhanced functionality is available for modeling electrostatics. System level and mixed device simulations are enabled through an interface for electrical circuits with SPICE import capability.

The Semiconductor Module is particularly relevant for simulating transistors including bipolar, metal semiconductor field-effect transistors (MESFETs), metal-oxide-semiconductor field-effect transistors (MOSFETs), Schottky diodes, thyristors, and P-N junctions.



DC characteristics of the simulated MOSFET transistor.



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.

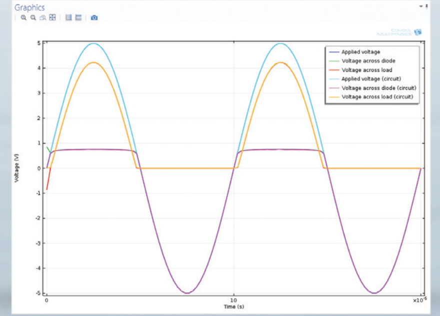
From top to bottom: electric potential, hole density, and electron density are shown.

HIGHLIGHTS

- Bipolar transistors
- Metal-oxide-semiconductor field-effect transistors (MOSFETs)
- Metal semiconductor field-effect transistors (MESFETs)
- P-N junctions
- Schottky diodes
- Thyristors

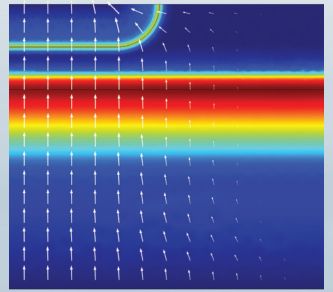
HALF-WAVE RECTIFIER

A device level simulation of a rectifier based on a p-n junction coupled to an electrical circuit. The results are compared with a lumped model of the diode and circuit. The figure shows input and output waveforms for both the full and lumped model simulations.



BIPOLAR TRANSISTOR

Simulation of an n-p-n bipolar transistor in the common-emitter configuration. The figure shows the current (arrows) and the electric field (color). The emitter contact is in the top-left of the figure, the base is in the top-right and the collector is at the base of the figure. Current flows from the collector into the emitter. The electric field is highest in the vicinity of the two junctions in the device.

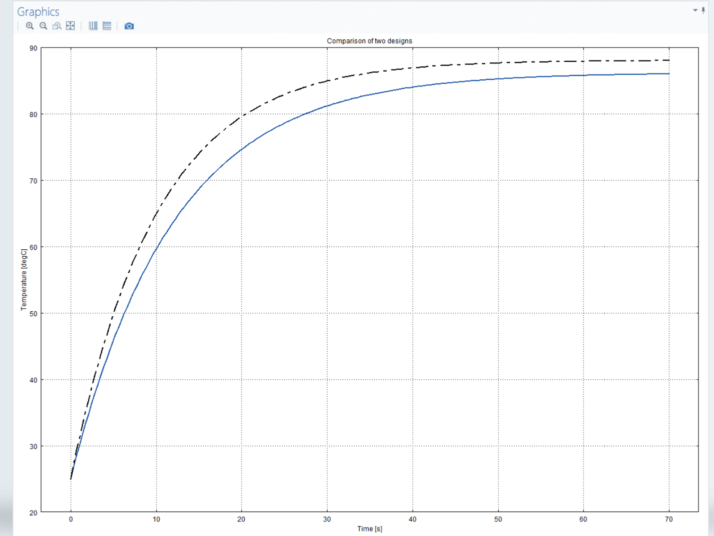


Heat Transfer Module

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 boundary conditions for describing convective and radiative effects, contact resistance, and thin highly-conductive shells, 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.

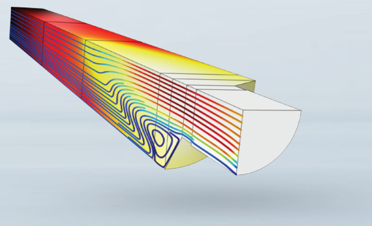


TEMPERATURE SENSOR
A photograph, X-ray picture, and model plot of the temperature distribution in a temperature sensor and on the sensing element inside the cap. The graph compares the temperature versus response time for two different designs of the cap.

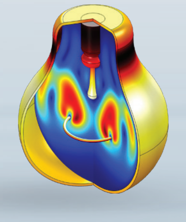
Model courtesy of Martin Šťas, Continental Corporation, Frenstat, Czech Republic.

HIGHLIGHTS

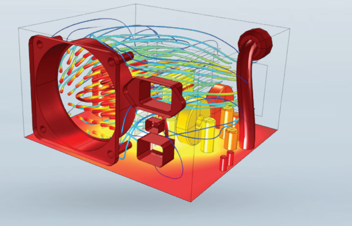
- Bioheat treatment and tissue necrosis
- Casting and thermal processing
- Conjugate heat transfer
- Disc brakes
- Drying and freeze drying
- Power electronics and electronic cooling
- Food processing, cooking, and sterilization
- Friction stir welding
- Furnace and burner design
- Heat and moisture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



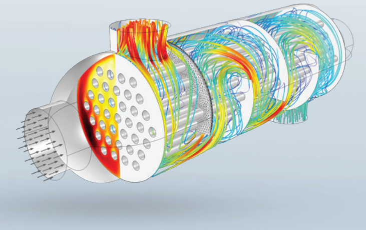
CONTINUOUS CASTING
Simulation of heat transport and the fluid-to-solid flow, including the phase transfer from melt to solid.



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.



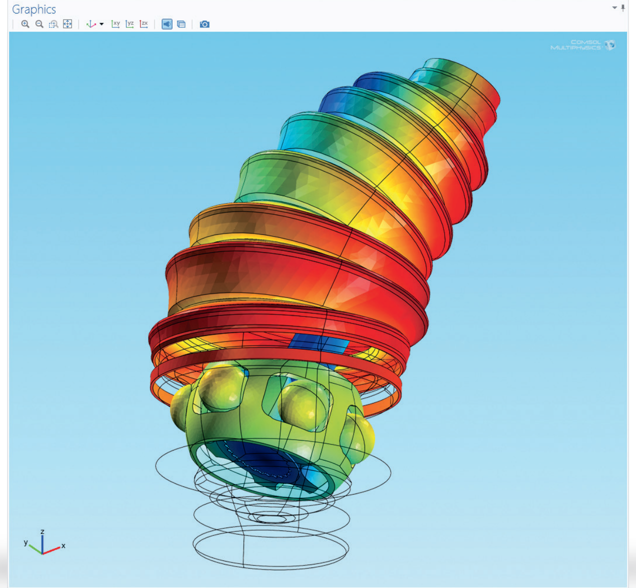
HEAT EXCHANGER
Simulation results reveal flow velocity, temperature distribution, and pressure within a shell and tube heat exchanger.

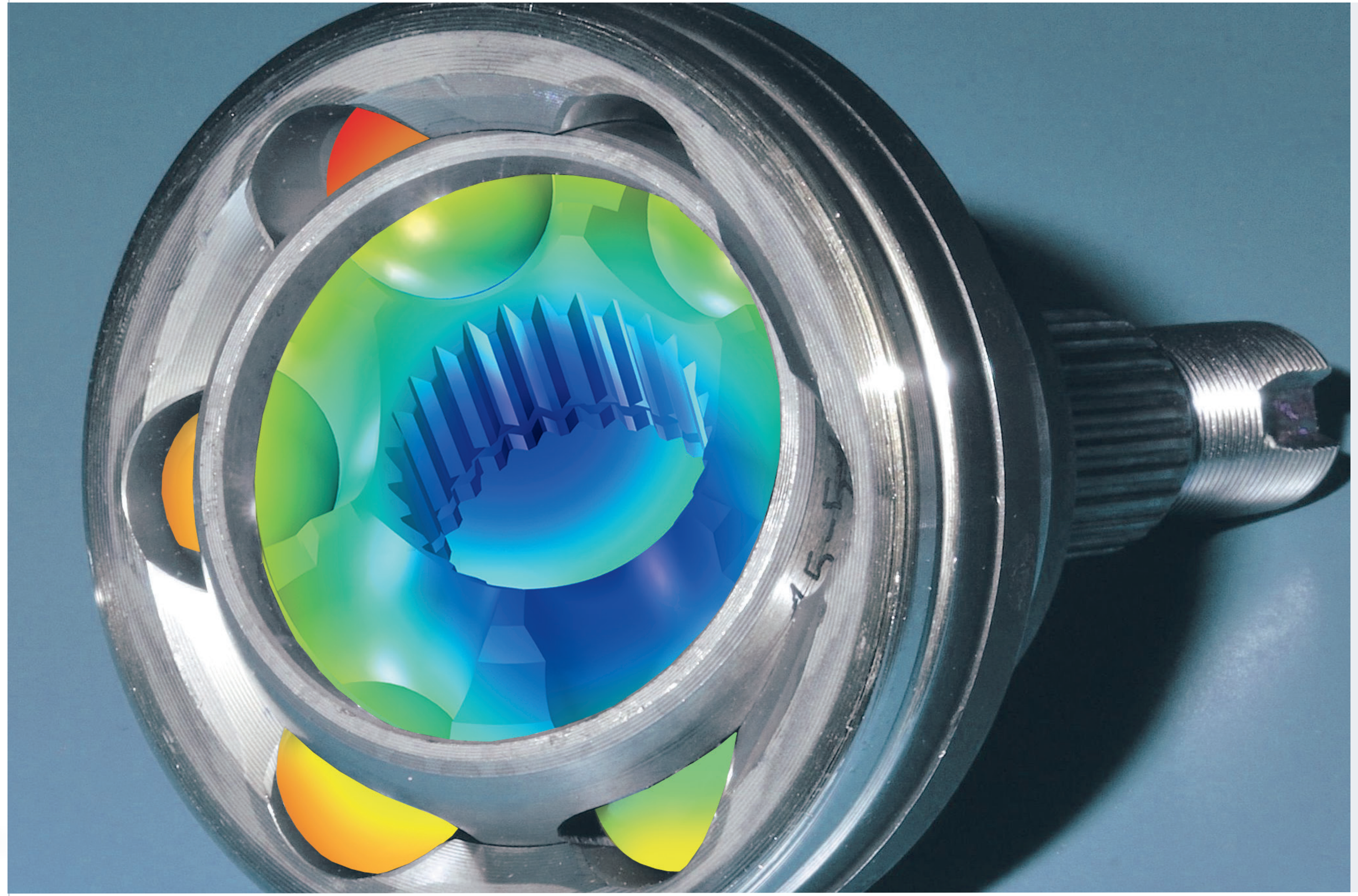
Structural Mechanics Module

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 shells, membranes, beams, and trusses.

The Structural Mechanics Module is used for a wide range of analysis types including: stationary, eigenmode, parametric, quasi-static, frequency-response, and prestressed. Its user interfaces allow for large deformation analysis with geometrical nonlinearity, mechanical contact, piezoelectric materials, thermal strain, fluid-structure interaction (FSI), and elastic waves.

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.

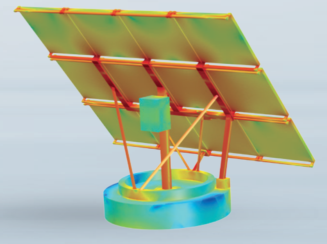
HIGHLIGHTS

- Buckling and postbuckling
- Centrifugal, Coriolis, and Euler forces
- Contact and friction
- Elastic waves
- Euler and Timoshenko beams
- Fluid-structure interaction (FSI)
- Geometric nonlinearity
- Infinite elements, PMLs, and low-reflecting boundaries
- Large deformation
- Lubrication and elastohydrodynamics*
- Membranes
- Modal analysis
- Periodic boundary conditions
- Piezoelectric devices
- Prestressed structures
- Rigid domains
- Shells and plates
- Springs and dampers
- Stress-optical effects
- Structural contact and friction
- Structural vibrations
- Thermal stress
- Trusses and cables
- Viscoelasticity

* *Together with the CFD Module*

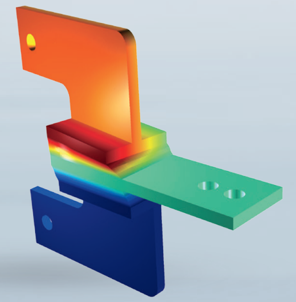
SOLAR PANEL

The von Mises stresses on the struts supporting a solar panel having being subjected to the forces from wind.



DAMPING ELEMENT

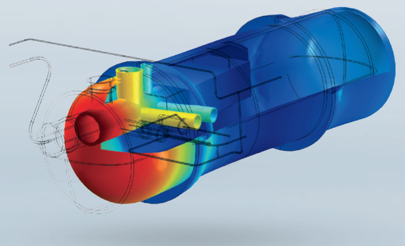
Structural analysis of a viscoelastic damping element used to stabilize tall buildings. Shown is the displacement as both the color and deformed shape plot.



KNOCK-OUT DRUM

Deformations in a knock-out drum connected to a petroleum refinery.

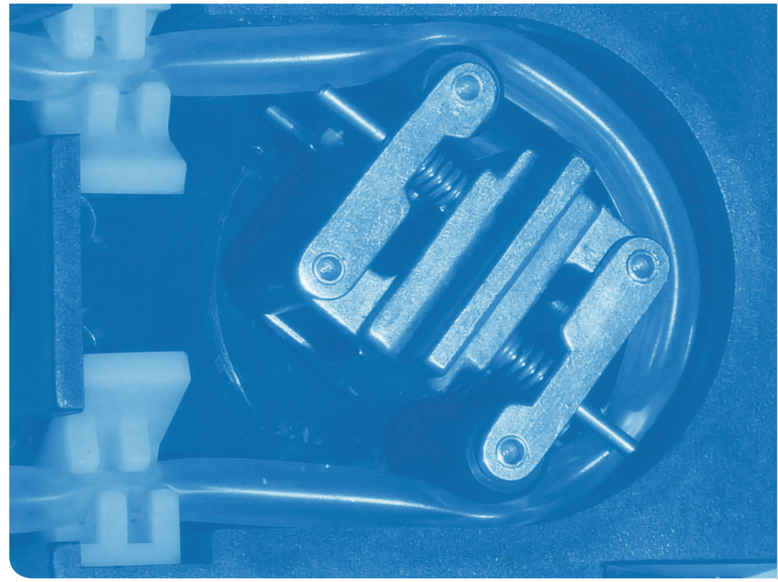
Model courtesy of Stig Sandström, SSM Engineering, Sweden.



Nonlinear Structural Materials Module

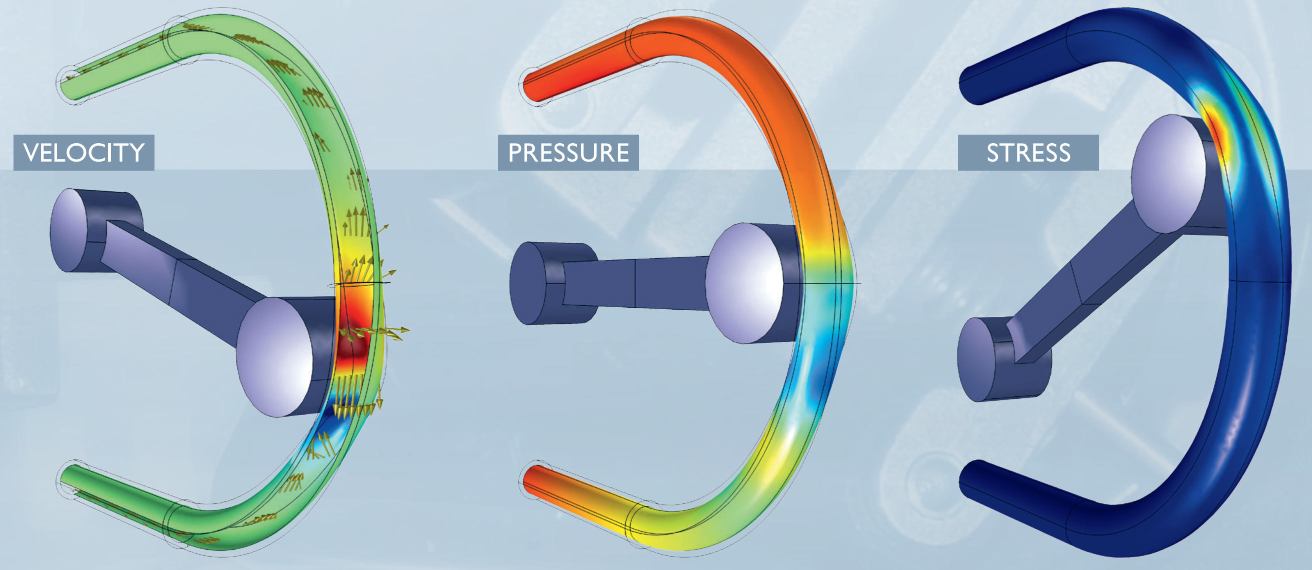
The Nonlinear Structural Materials Module augments the mechanical capabilities of the Structural Mechanics and MEMS Modules by adding nonlinear material models, including large strain plastic deformation. When the mechanical stress in a structure becomes large, certain nonlinearities in the material 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, and hyperelastic 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.

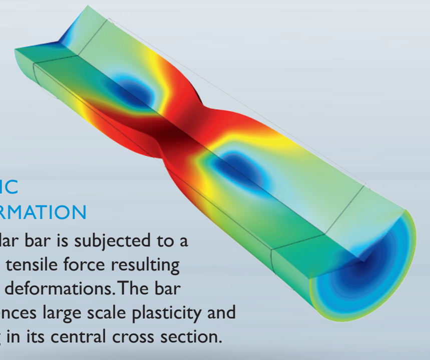


PUMPING

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.

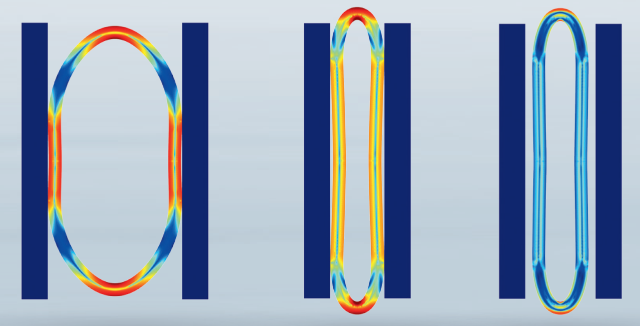


Model images are provided courtesy of Nagi Elabbasi of Veryst Engineering, Needham, MA. To read more, visit: comsol.com/papers/11574. Photograph of peristaltic pump first produced by Andy Dingley.



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.

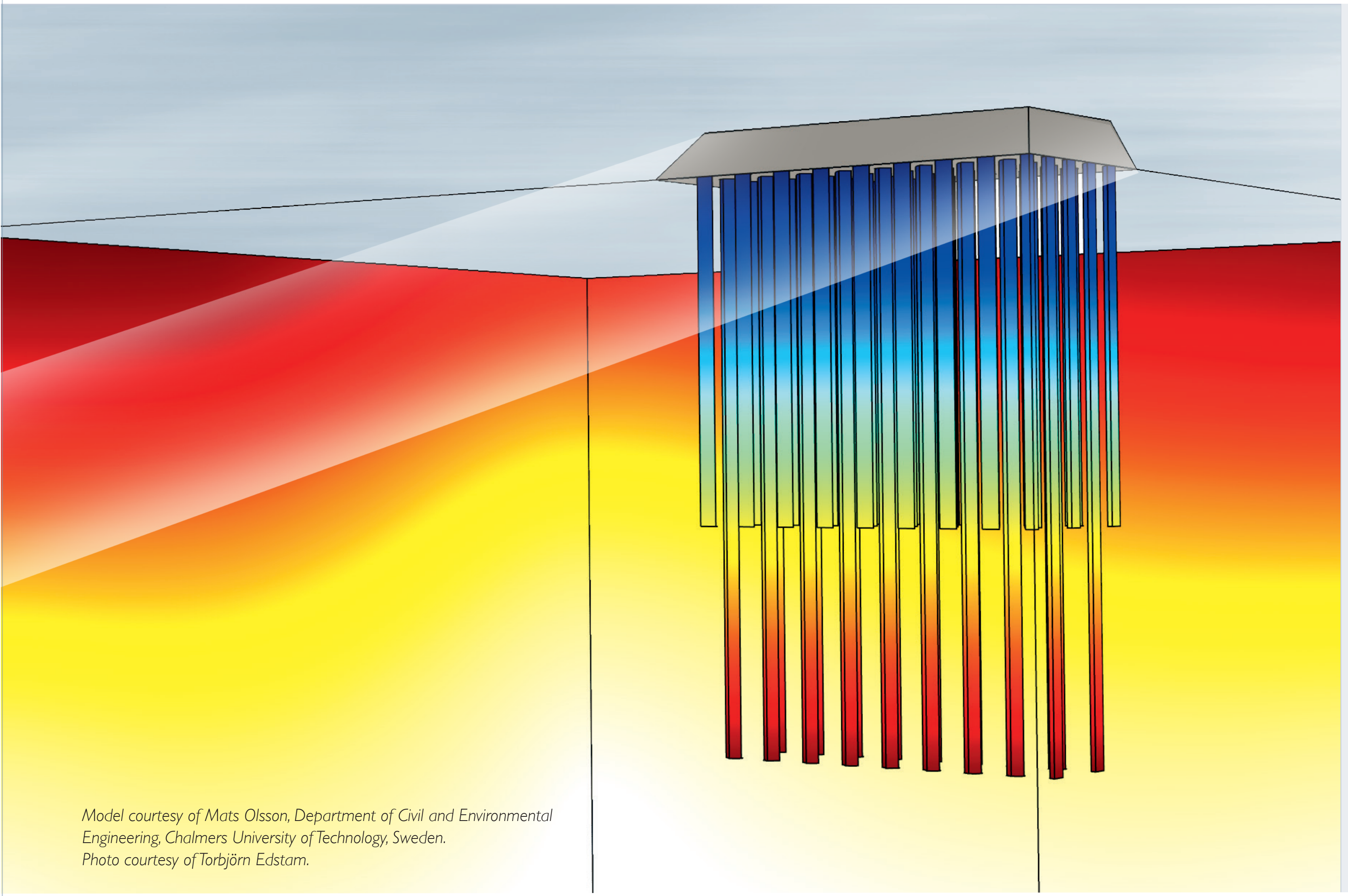
HIGHLIGHTS

- Anand viscoplasticity
- Arruda-Boyce hyperelasticity
- Biomechanics
- Blatz-Ko hyperelasticity
- Coble creep
- Creep
- Deviatoric creep
- Elastomers
- Elastoplasticity
- Gao hyperelasticity
- Garofalo creep
- Gent hyperelasticity
- Isotropic, kinematic, and perfectly plastic hardening
- Hill plasticity
- Hyperelastic materials
- Large deformation
- Large strain plasticity
- Mooney-Rivlin hyperelasticity
- Murnaghan hyperelasticity
- Nabarro-Herring creep
- Neo-Hookean hyperelasticity
- Nonlinear materials
- Norton and Norton-Bailey creep
- Ogden hyperelasticity
- Orthotropic plasticity
- Polymers
- Rubber
- Saint Venant–Kirchhoff hyperelasticity
- Storakers hyperelasticity
- Temperature-dependant plasticity
- User-defined creep, hyperelasticity, and plasticity
- Varga hyperelasticity
- Viscoplasticity
- Yeoh hyperelasticity

Geomechanics Module

The Geomechanics Module is a specialized add-on to the Structural Mechanics Module for simulation of geotechnical applications such as tunnels, excavations, slope stability, and retaining structures. The module features tailored user interfaces 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. In addition to the built-in plasticity models, user-defined yield functions can be created by the versatile equation-based 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.



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

SETTLEMENT IN EMBANKMENTS

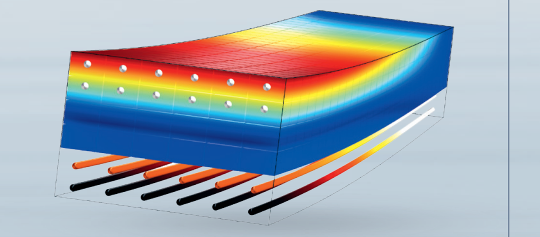
Long-term settlement in clay constitutes an engineering challenge in, for example, road and railway design and construction where supporting lime-cement columns are used. A test site was constructed to evaluate, among other things, the deformation properties of the supporting columns and surrounding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

HIGHLIGHTS

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

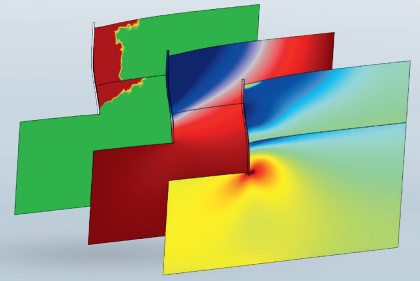
CIVIL ENGINEERING

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.

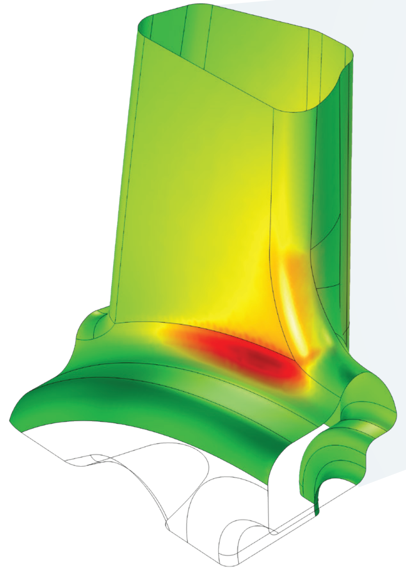


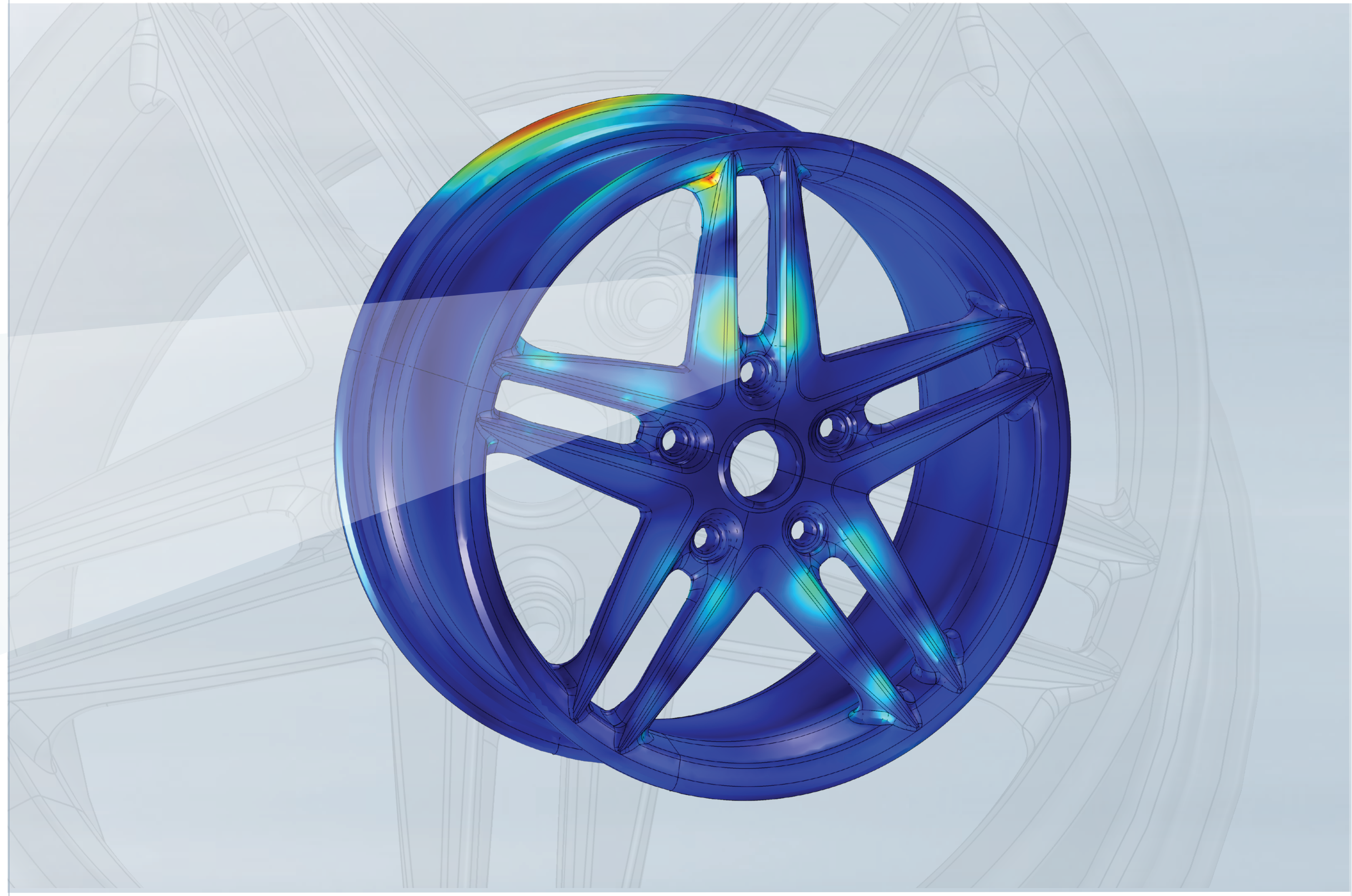
Fatigue Module

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.

With the stress-based and the strain-based critical plane methods, you can evaluate the high-cycle and low-cycle fatigue 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.



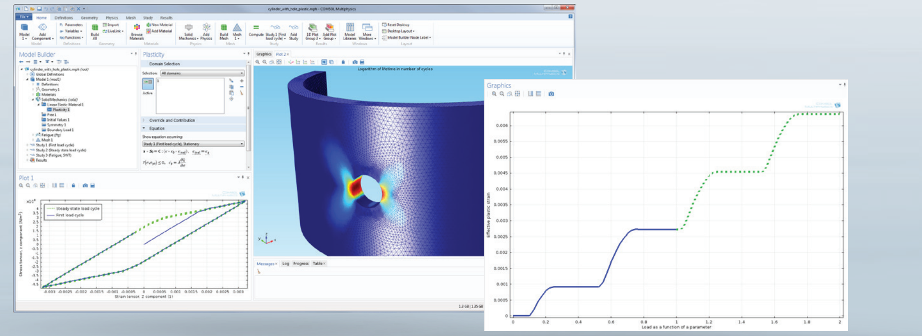


HIGH-CYCLE FATIGUE ANALYSIS OF A CAR WHEEL

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).

HIGHLIGHTS

- Cumulative damage analysis
    - *- Rainflow counting*
    - *- Palmgren-Miner linear damage rule*
- Energy-based fatigue
    - *- Darveaux criterion*
    - *- Morrow criterion*
- Fatigue usage factor computation
- High-cycle fatigue
- Low-cycle fatigue
- Stress-based fatigue
    - *- Findley criterion*
    - *- Matake criterion*
    - *- Normal stress criterion*
- Strain-based fatigue
    - *- Coffin-Manson criterion*
    - *- Fatemi-Socie criterion*
    - *- Mean stress correction*
    - *- Notch approximation*
    - *- Smith-Watson-Topper (SWT) criterion*
    - *- Wang-Brown criterion*



ELASTOPLASTIC LOW-CYCLE FATIGUE ANALYSIS

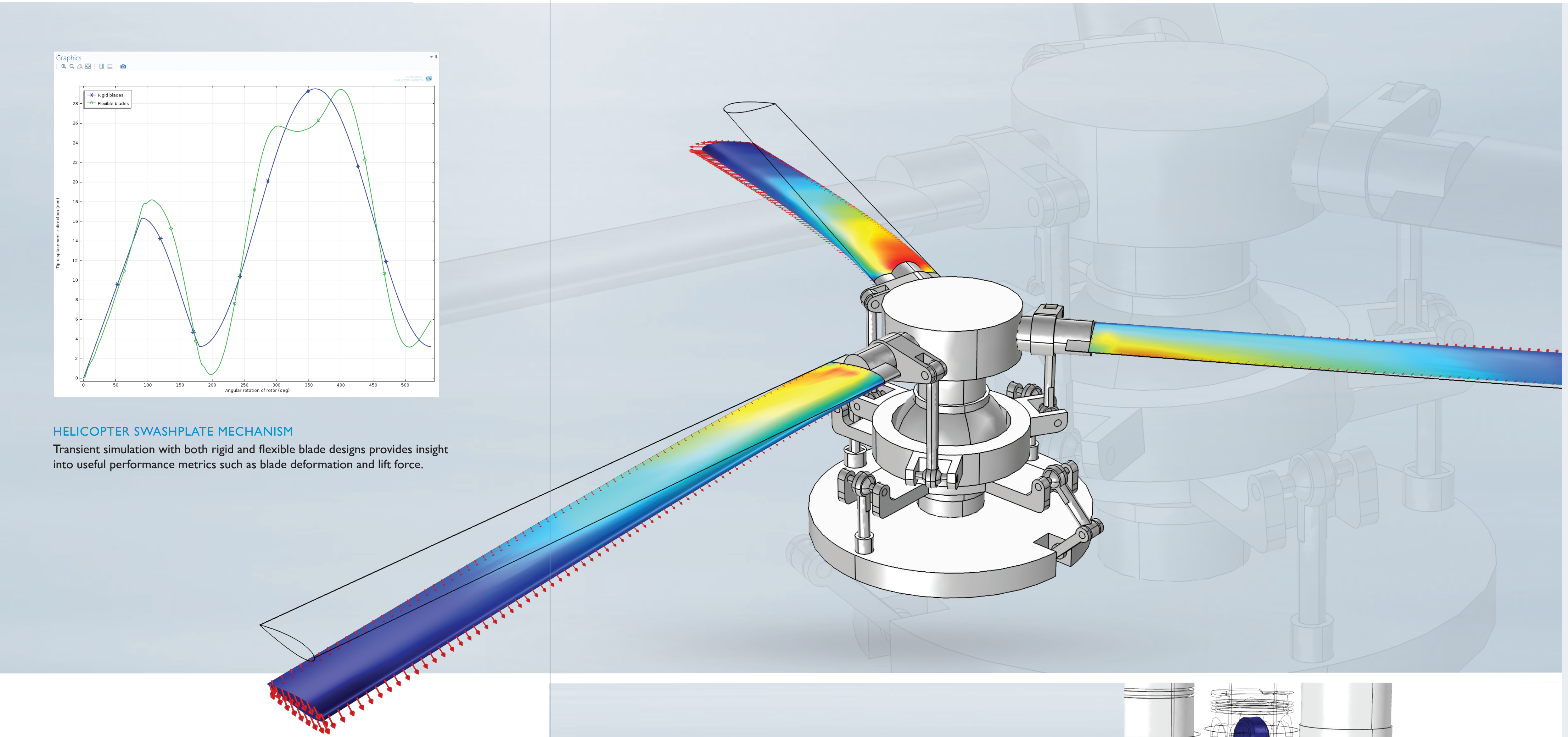
Low-cycle strain-based fatigue resulting from plastic deformation near the hole in a cylinder is simulated. The visualizations (from left to right) show a stress-strain curve for the last two load cycles, a surface plot of lifetime logarithm, and the accumulated effective plastic strain curve.

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 in addition to locking.

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 with the Nonlinear Structural Materials Module and the Geomechanics 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.

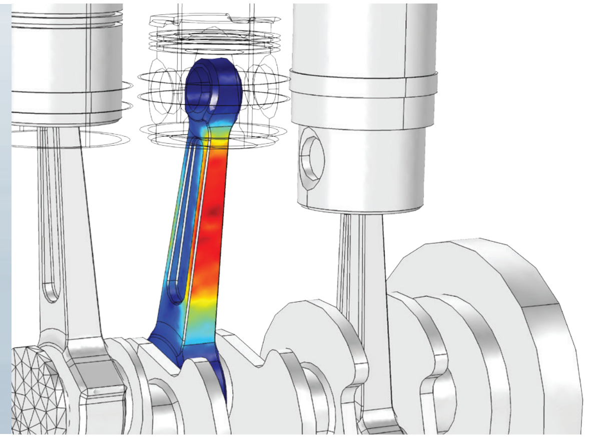


HELICOPTER SWASHPLATE MECHANISM

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

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

- Automotive and Aerospace
- Biomechanics
- Biomedical instruments
- Engine dynamics
- General dynamic simulations of mechanical assemblies
- Mechatronics and Robotics
- Vehicle dynamics

JOINT TYPES

- Ball (3D)
- Cylindrical (3D)
- Distance (2D, 3D)
- Fixed (2D, 3D)
- Hinge (2D, 3D)
- Planar (3D)
- Prismatic (2D, 3D)
- Reduced Slot (2D, 3D)
- Screw (3D)
- Slot (3D)
- Universal (3D)

CAPABILITIES

- Coupling multibody systems with other physical phenomena
- Eigenmodes of flexible multibody systems
- Fatigue analysis of critical flexible bodies by combining with the Fatigue Module
- Reaction forces and moments at a joint
- Relative displacement/rotation between two components and their velocities
- Stresses and deformations in flexible bodies

Acoustics Module

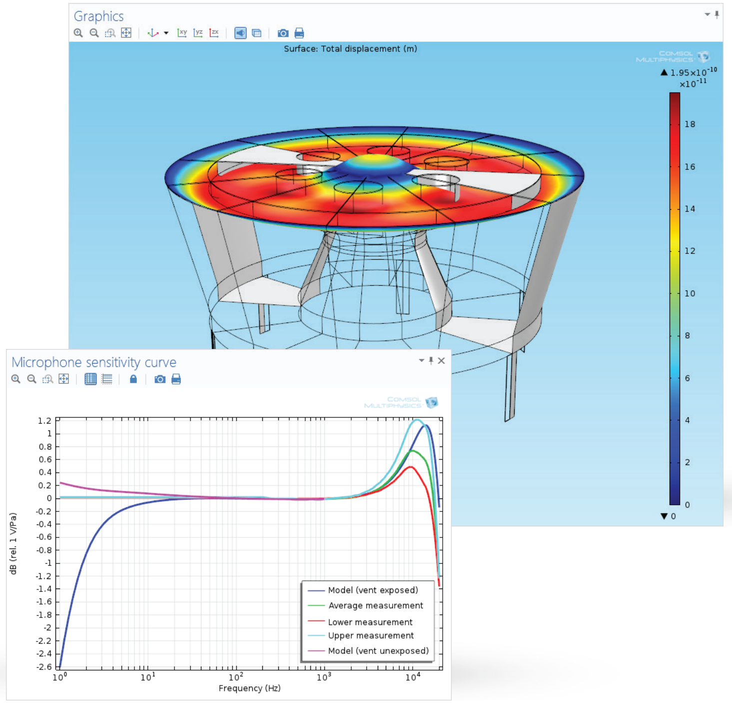
The Acoustics Module is a world-class solution to your acoustics modeling needs. This module is designed specifically for those who work with devices that produce, measure, and utilize acoustic waves. Application areas include speakers, microphones, hearing aids, and sonar devices, as well as noise control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

Easy-to-use physics interfaces provide the tools to model acoustic pressure wave propagation in air, water, and other fluids. Dedicated 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. You can also model vibrations and elastic waves in solids, piezoelectric materials, and poroelastic structures. Multiphysics user interfaces for acoustic-solid, acoustic-shell, and piezo acoustics 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.

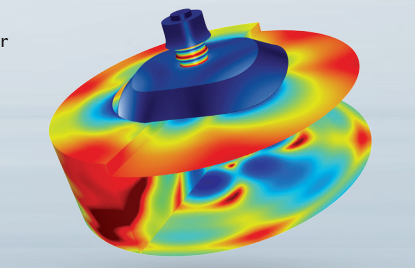


HIGHLIGHTS

- Acoustic-structure interaction
- Elastic waves
- Electroacoustic transducers and speakers
- Hearing aids
- Linearized Euler 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
- Reactive and absorptive mufflers
- SONARs
- Structural vibrations
- Thermoacoustics

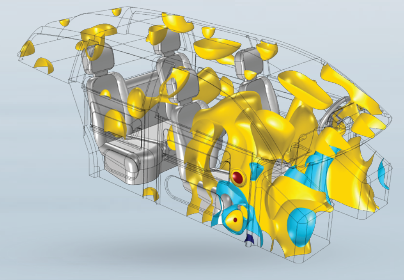
SONAR

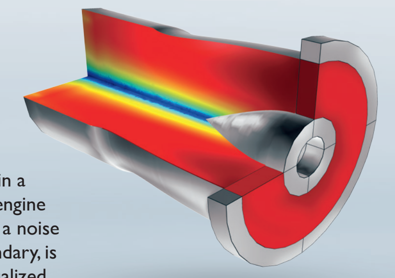
The tonpilz piezo transducer is used for low frequency, high power sound emission. The model shows the voltage distribution in the piezoceramic rings, the deformation in the massive ends, and the pressure field underneath the transducer.



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

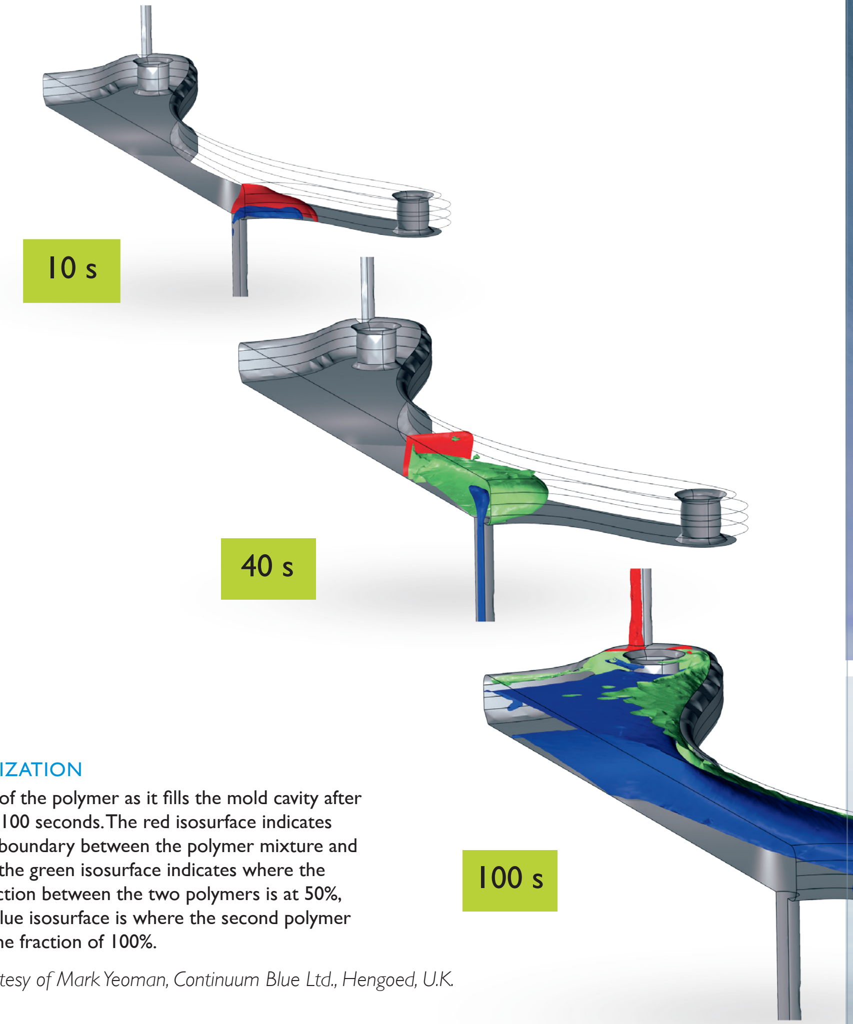
The acoustic field in a model of an aero-engine duct, generated by a noise source at the boundary, is computed and visualized.

CFD Module

The CFD Module is the premier tool in the COMSOL product suite for sophisticated fluid flow simulations. Compressible as well as incompressible flows can be combined with advanced turbulence models and forced and natural convection. An important characteristic of the CFD Module is its capability of precise multiphysics-flow simulations such as conjugate heat transfer with non-isothermal flow, fluid-structure interactions, non-Newtonian flow with viscous heating, and fluids with concentration-dependent viscosity. Porous-media flow user interfaces allow for isotropic or anisotropic media, as well as automatically combined free flow and porous domains. Tools for modeling of stirred vessels with rotating parts are available for both 2D and 3D flows.

The module's interfaces for homogeneous two-phase flow include a mixture model for fine particle suspensions and a bubbly flow model for macroscopic gas bubble flow. For interface tracking two-phase flow, formulations are provided using the level-set and phase-field methods.

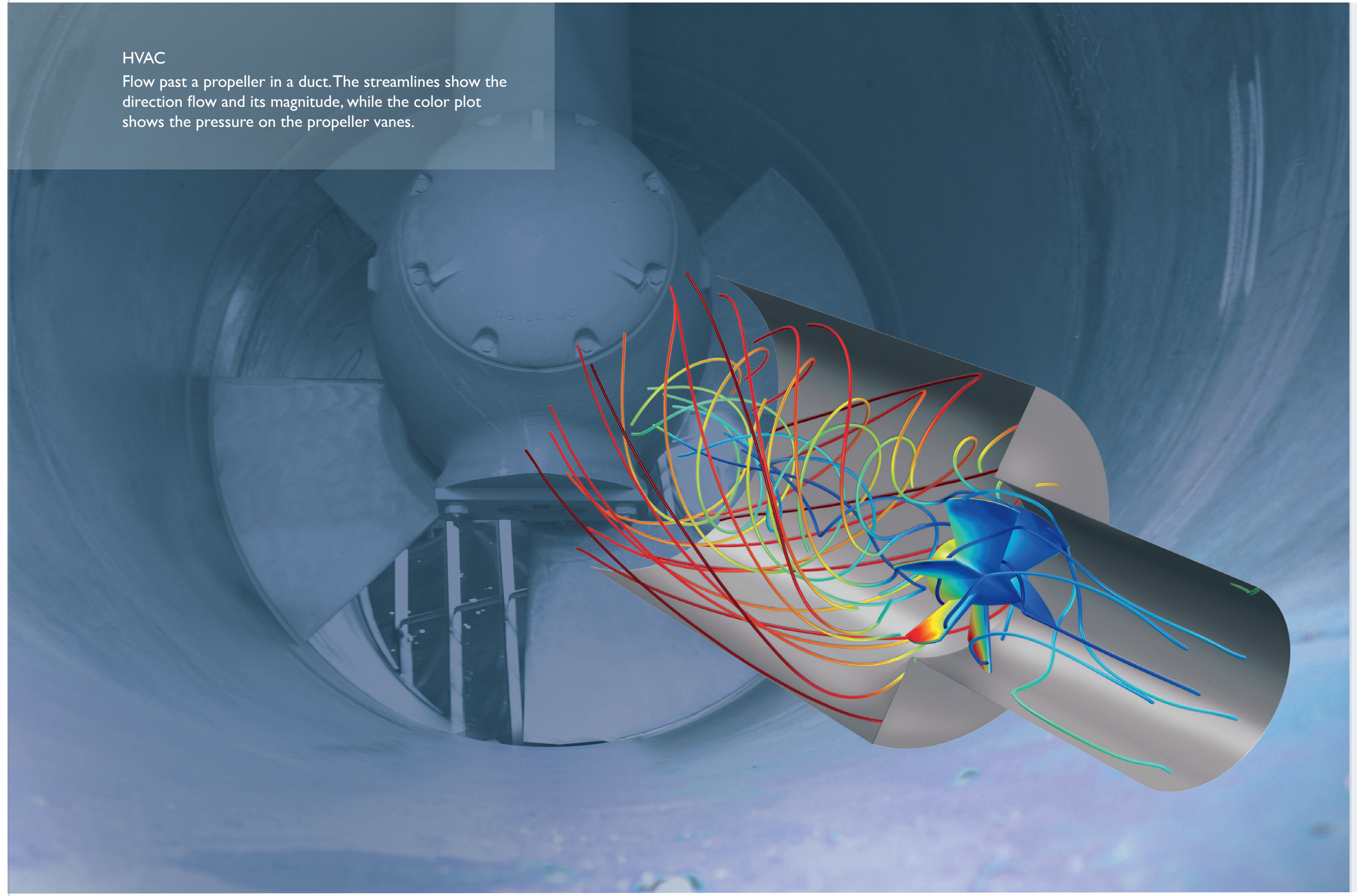
The tools available in the CFD Module for advanced transport and reacting flow simulations are automatically extended when combined with the Chemical Reaction Engineering Module. For fluid-structure interactions, the Structural Mechanics Module is compatible with the flow models of the CFD Module and makes available elastic solid-fluid couplings as well as lubricating flow and elastohydrodynamics.



POLYMERIZATION

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

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



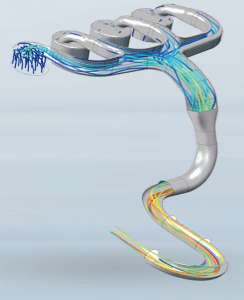
HVAC

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

HIGHLIGHTS

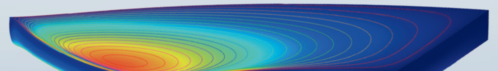
- Conjugate heat transfer
- Cyclones, filters and separation units
- Electronic cooling
- Fans, grills, and pumps
- Flow around vehicles and structures
- Flows in pipes, valves, joints, nozzles
- Fluidized beds and sprays
- Fluid-structure interaction (FSI)*
- Gas bubble flow
- Heat exchangers and cooling 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
- Sedimentation, emulsions, and suspensions
- Turbulent flow

* *Together with the Structural Mechanics Module or MEMS Module*



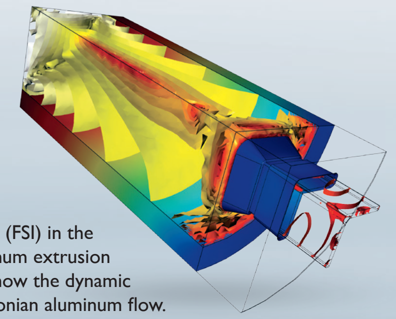
EXHAUST MANIFOLD

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

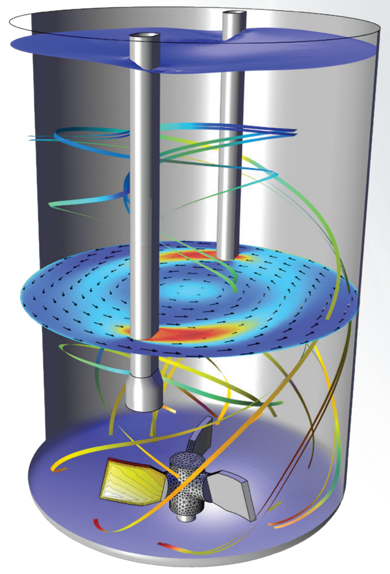
Fluid-structure Interaction (FSI) in the cast and mold of an aluminum extrusion process. The isosurfaces show the dynamic viscosity in the non-Newtonian aluminum flow.

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. These include 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 on the fluid flow equations.

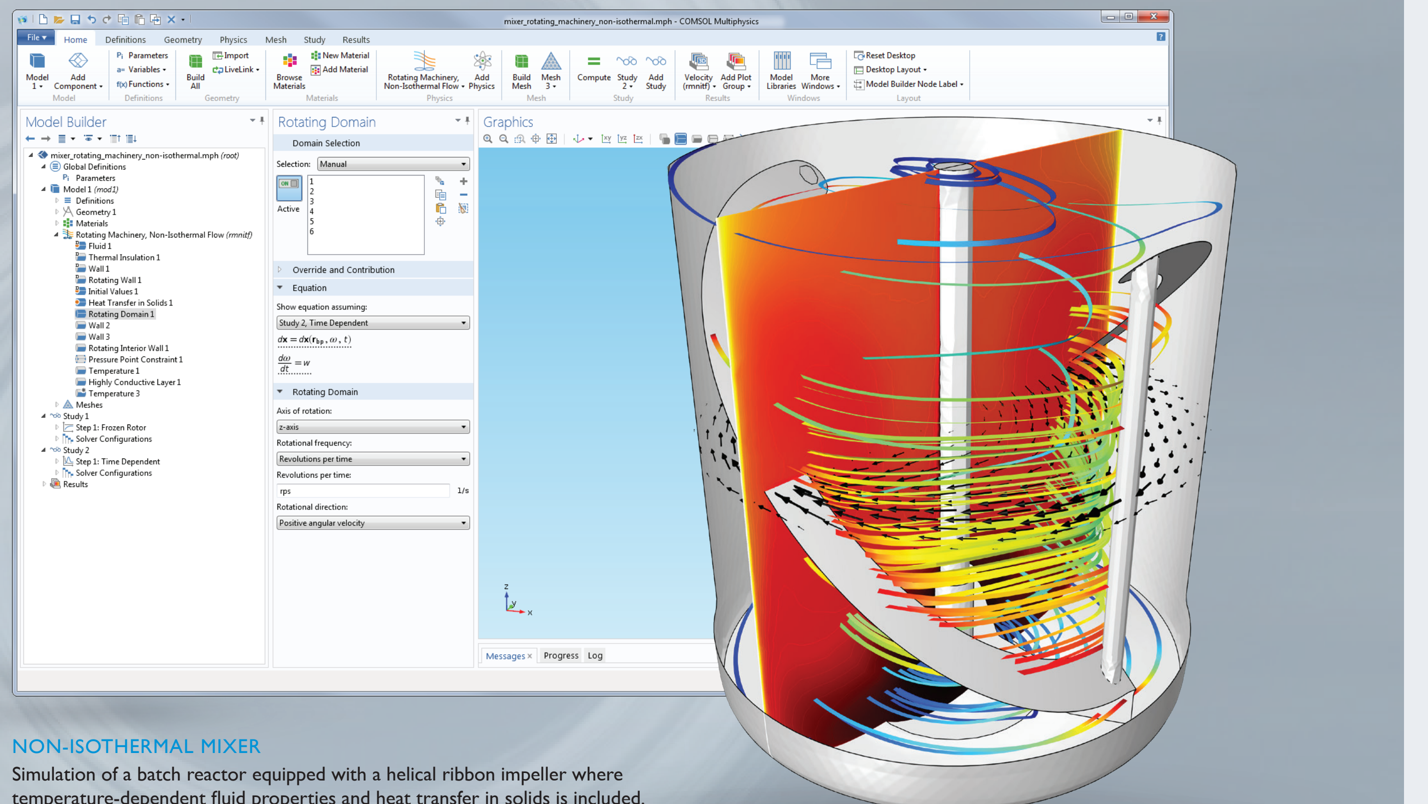
A fully time-dependent study for fluid flow affected by rotating machinery accounts for the movement of geometric parts relative to each other, and is the most accurate way to simulate mixing processes. COMSOL defines a modeling domain that encompasses the impeller or rotor, and another outside of this, 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 by assuming that the topology of the system relative to the rotating reference frame is fixed, or frozen. 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 represents the deformation of the free surface, and a pressure contour is shown on the front blade. The center slice and ribbons show the direction and magnitude of the flow field.



Selection: Manual

1
2
3
4
5
6

Show equation assuming: Study 2, Time Dependent

$d\mathbf{x} = d\mathbf{x}(\mathbf{r}_{bp}, \omega, t)$

$\frac{d\omega}{dt} = w$

Rotating Domain

Axis of rotation: z-axis

Rotational frequency: Revolutions per time

Revolutions per time: rps 1/s

Rotational direction: Positive angular velocity

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.

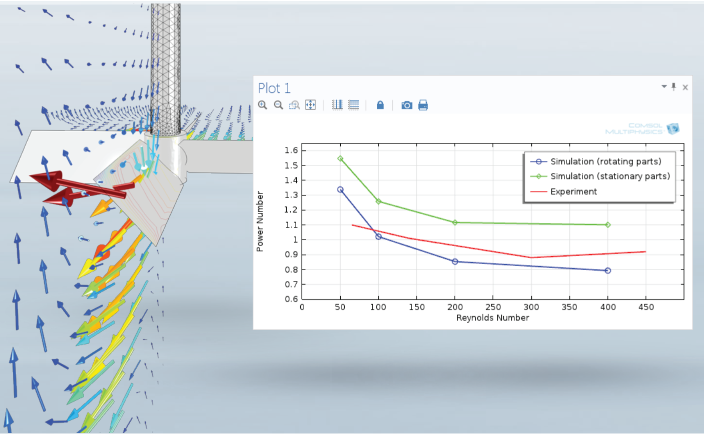
HIGHLIGHTS

- 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, or low Reynolds number k-epsilon model

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).

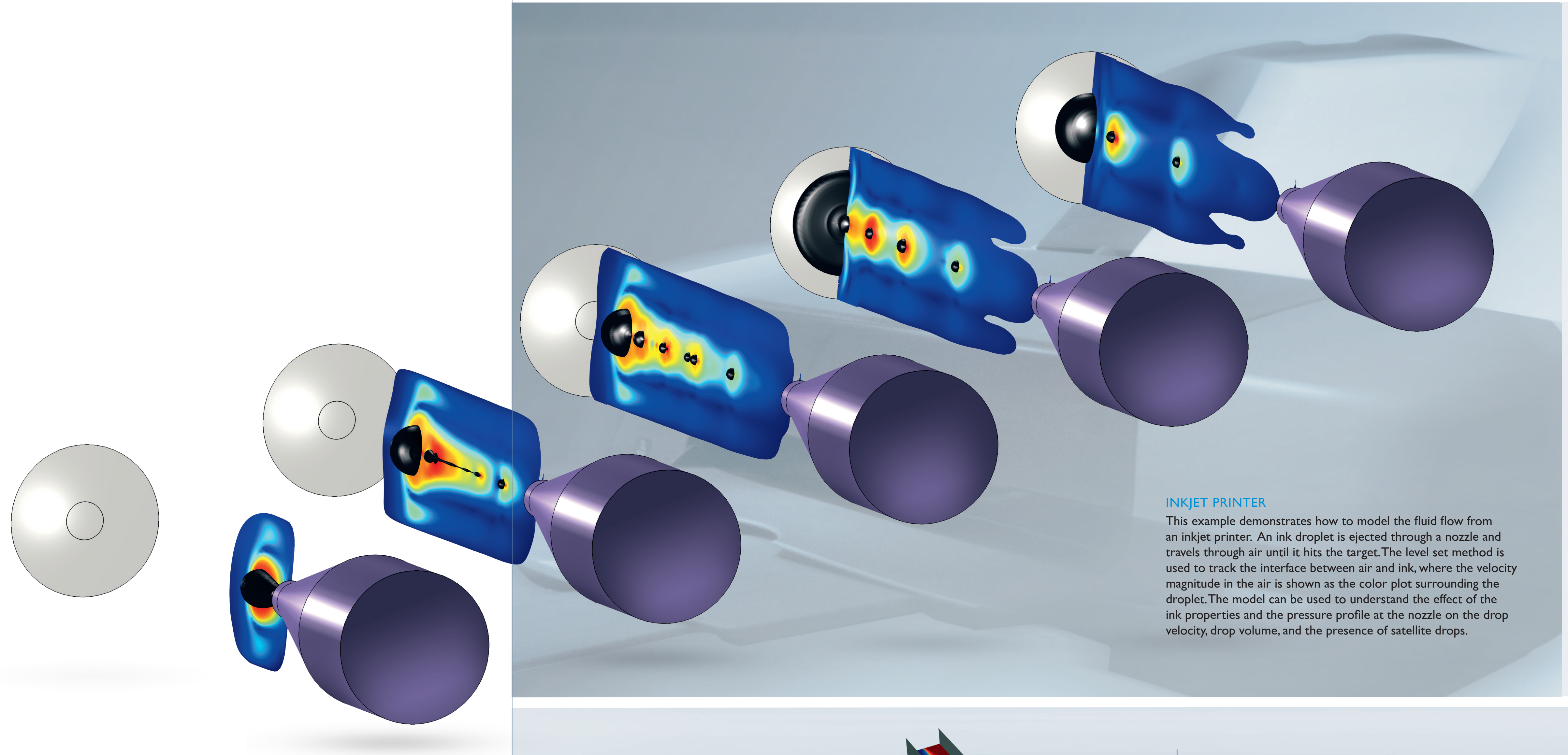


Microfluidics Module

The Microfluidics Module brings easy-to-use tools for the study of microfluidic devices. Important applications include simulations of lab-on-a-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 Maragoni 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.

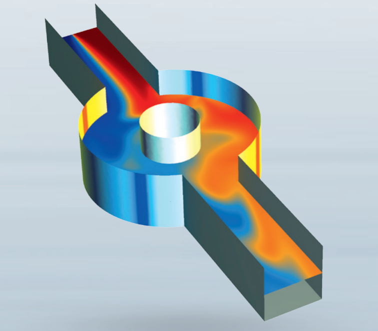
HIGHLIGHTS

- 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*

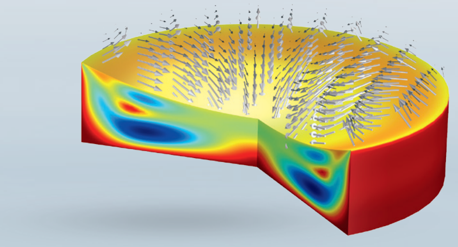
ELECTROOSMOSIS

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 (surface plot), and the resulting electroosmosis perturbs the parallel streamlines and concentration distribution (slice plot) 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.



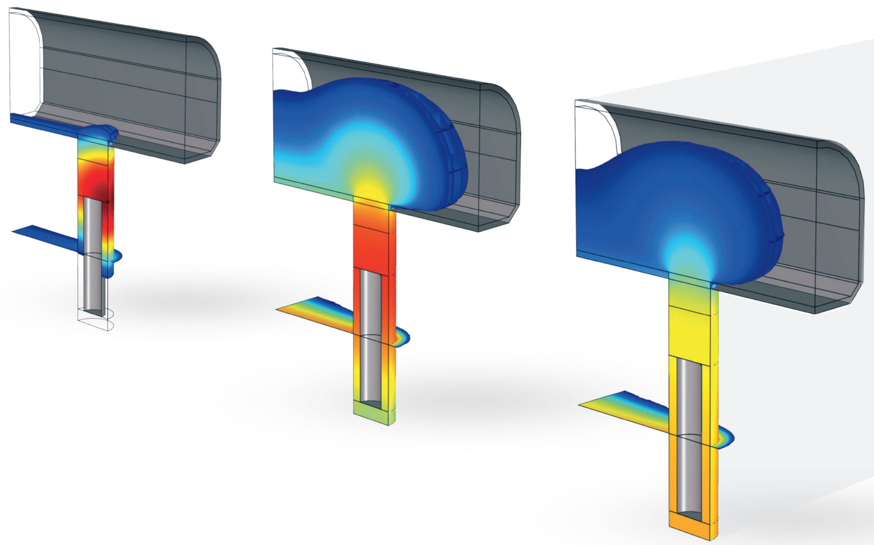
Subsurface Flow Module

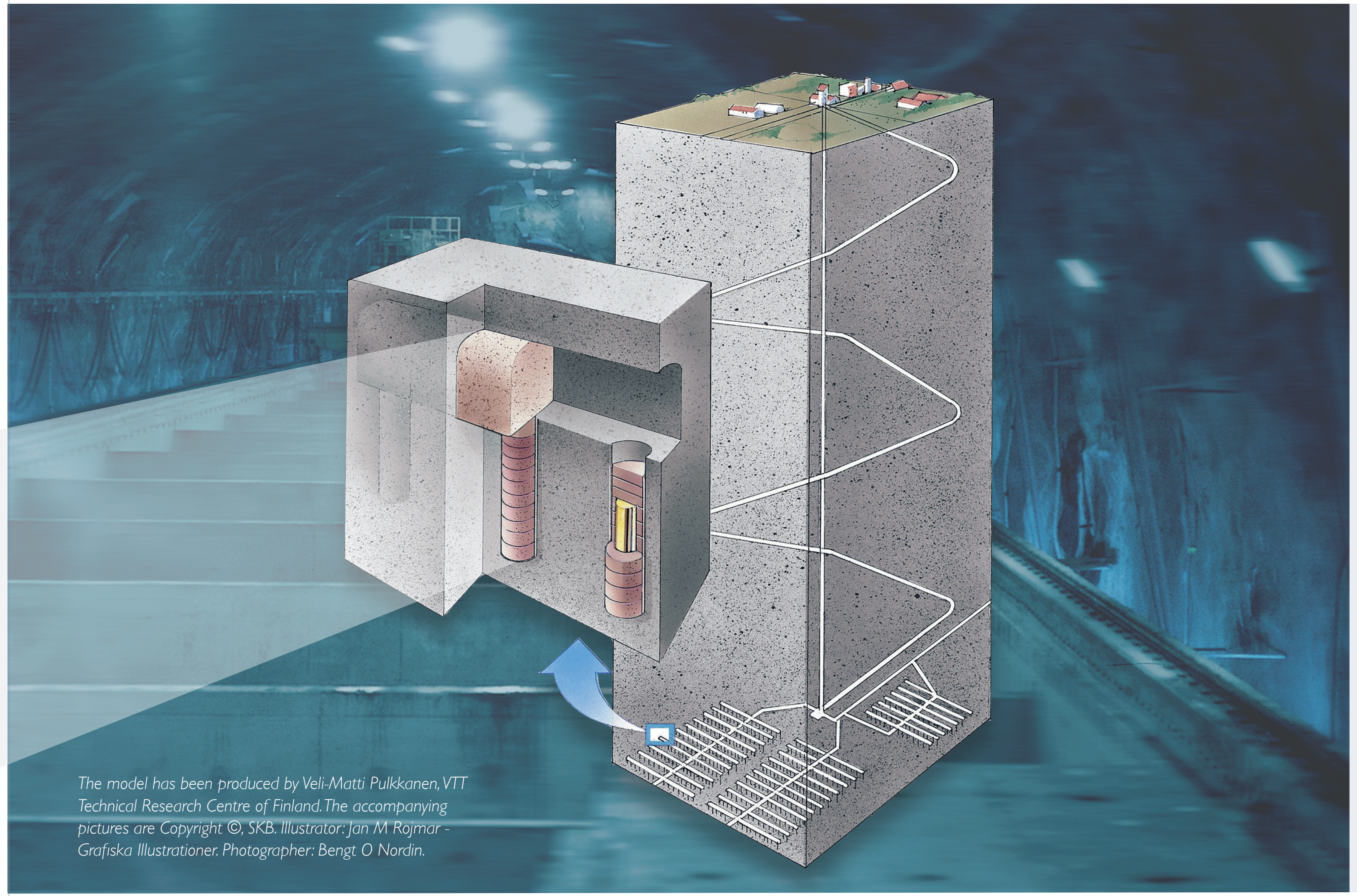
The Subsurface Flow Module is designed for geophysical and environmental phenomena studies such as the modeling of groundwater flow, the spread of pollution through soil, and oil and gas flow in porous media. The Richards' Equation interface describes nonlinear flow in variably saturated porous media, 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 solid, liquid, and gas phases for free, saturated, and variably saturated fluid flows, including solute transport in fractures.

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

NUCLEAR WASTE DISPOSAL

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





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

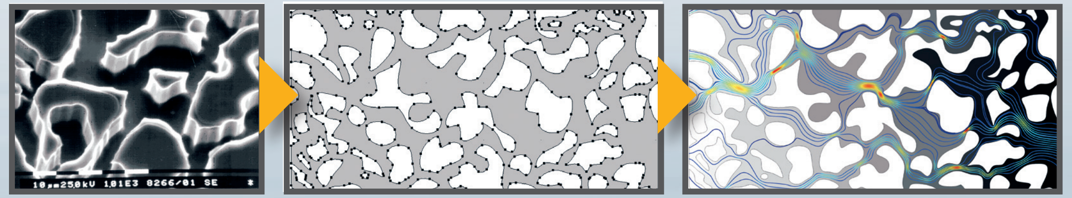
HIGHLIGHTS

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

SUBSURFACE FLOW

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

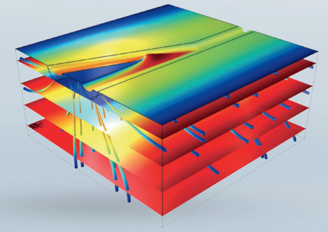
Model courtesy of Arturo Keller, University of California, Santa Barbara.



RESERVOIR MECHANICS

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

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



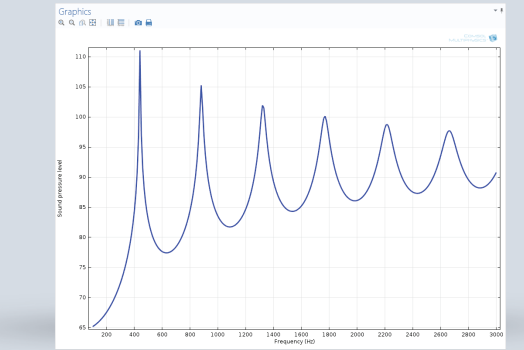
Pipe Flow Module

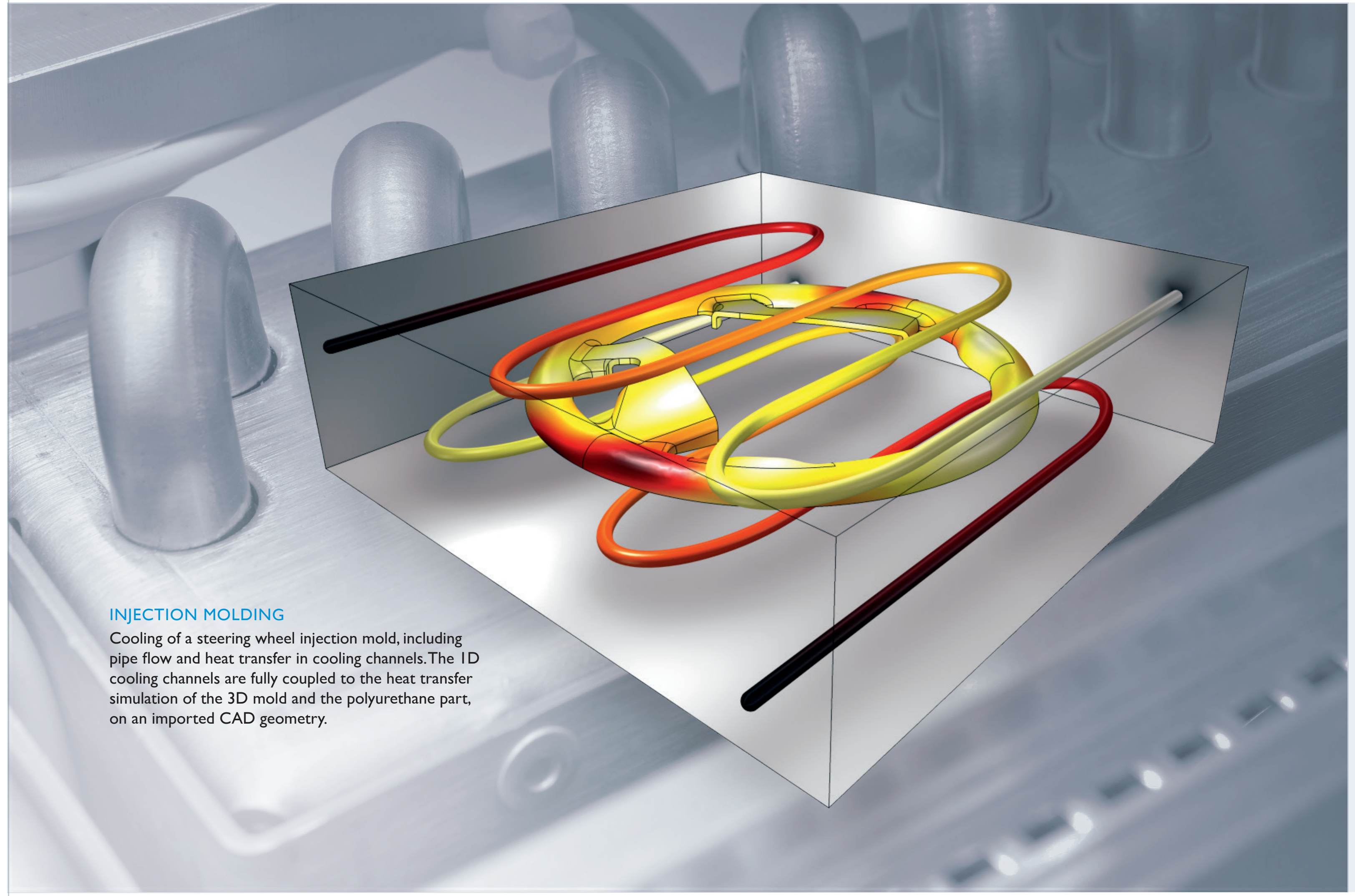
The Pipe Flow Module is used for simulations of fluid flow, heat and mass transfer, hydraulic transients, and acoustics in pipe and channel networks. Pipe flow simulations yield the velocity, pressure variation, concentrations and chemical reactions, and temperature along pipes and channels. The module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

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 industry. 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.

ORGAN PIPE DESIGN

Frequency response of an organ pipe showing the resonance peak of the fundamental frequency and five of the harmonics from 100 to 3,000 Hz. The model uses the Pipe Acoustics, Frequency Domain physics interface and includes the effects of the elastic pipe walls as well as the impedance at the open pipe end.



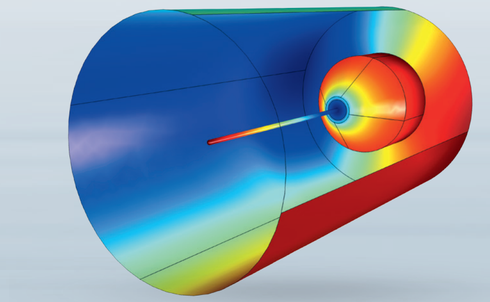


INJECTION MOLDING

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.

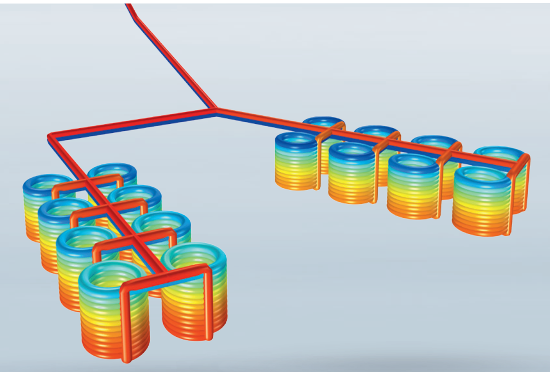
HIGHLIGHTS

- Chemical plant distribution systems
- Chemical reactions in pipes
- Cooling systems
- Geothermal systems
- Heat exchangers and cooling flanges
- Heat transfer in pipes
- Hydraulics
- Mass transfer in pipes
- Nonisothermal pipe flow
- Oil refinery pipe systems
- Pipe acoustics
- Pipe flow
- Water hammer
- Lubrication



MICROPHONES

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



GEOTHERMAL HEATING

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

Molecular Flow Module

The Molecular Flow Module is a tool that 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.

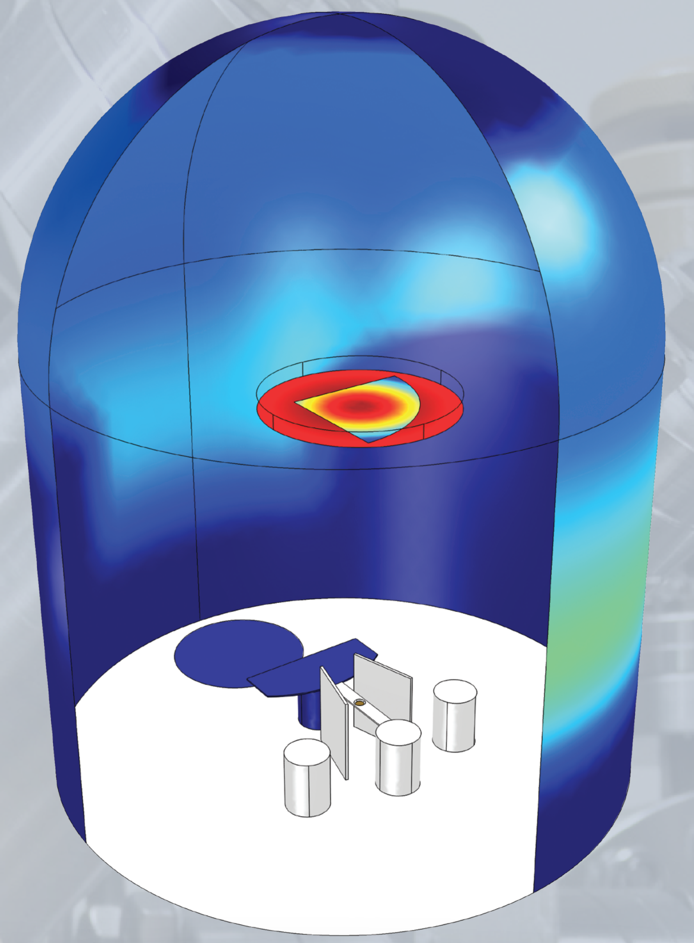
The Molecular Flow Module is ideal for the simulation of vacuum systems including those used in semiconductor processing, particle accelerators and mass spectrometers. Small channel applications can also be addressed.

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 to mesh 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 is 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

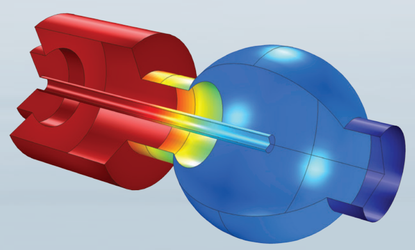
Thermally evaporated gold film thickness on both a sample and the interior surfaces of the evaporator chamber are shown. The thickness is computed as a function of time with a transient simulation.

HIGHLIGHTS

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

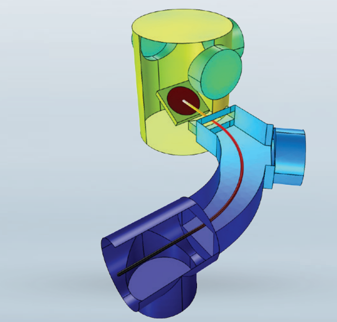
LOAD LOCK VACUUM SYSTEM

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.



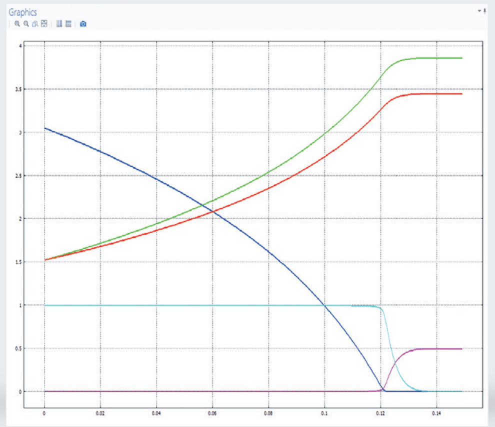
Chemical Reaction Engineering Module

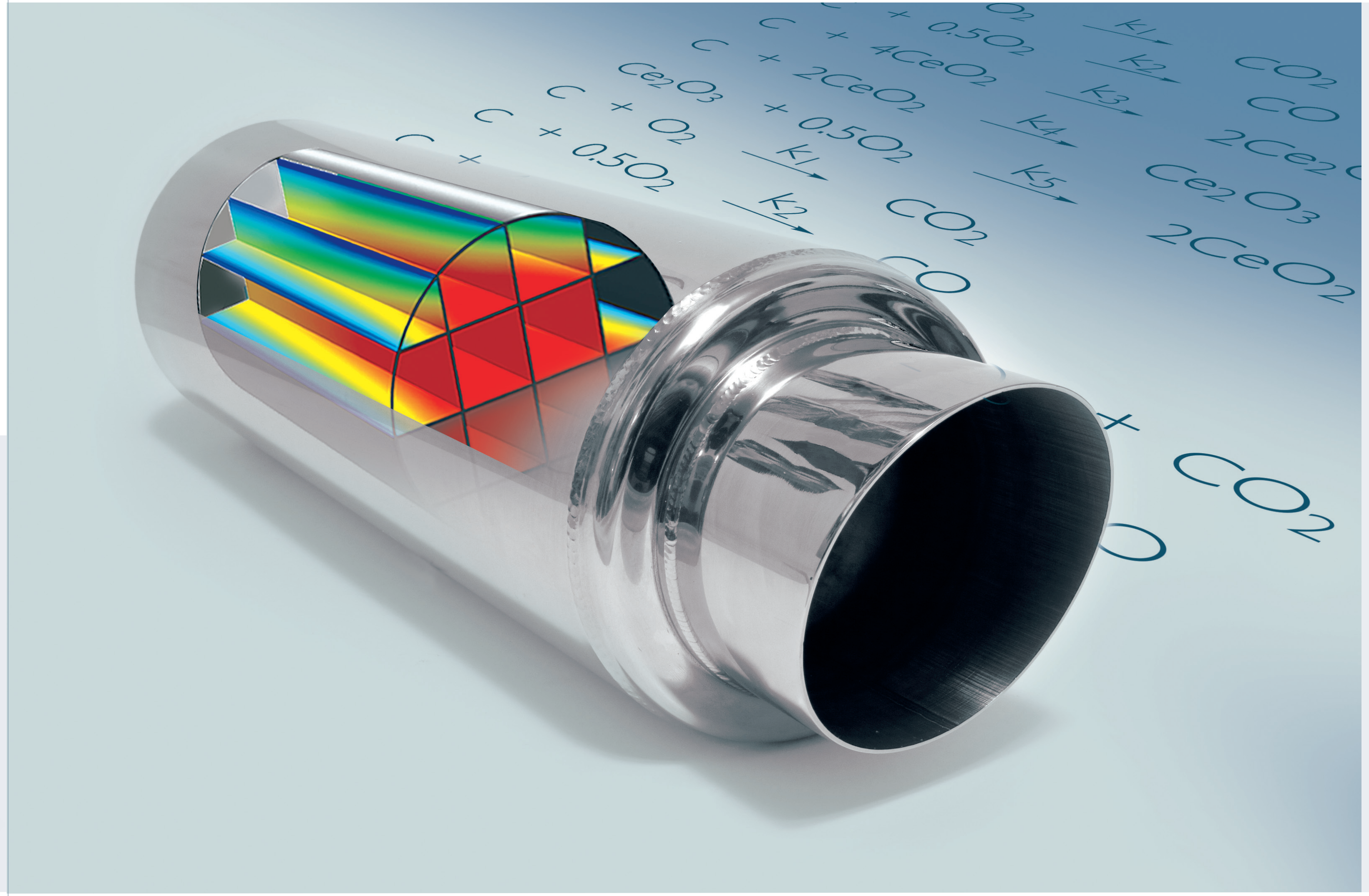
The Chemical Reaction Engineering Module is optimized for the modeling of reactors, filtration and separation units, and other equipment common in the chemical and similar industries. It is specifically designed to easily couple fluid flow and mass and energy transport to chemical reaction kinetics. 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 in a pre-study of the chemical kinetics. The model shows the carbon dioxide concentration in the filter.





HIGHLIGHTS

- Batch reactors, fermenters, and crystallizers
- Biochemistry and food science
- Chemical reactor sizing and optimization
- Chromatography and electrophoresis
- Cyclones, separators, scrubbers, and leaching units
- Exhaust after-treatment and emission control
- Filtration and sedimentation
- Kinetics modeling in chemical reactors
- Microfluidics and lab-on-chip devices
- Multicomponent and membrane transport
- Packed bed reactors
- Pharmaceutical synthesis
- Plug-flow and tubular reactors
- Polymerization kinetics and manufacture
- Pre-burners and internal combustion engines
- Reformers and catalytic converters
- Semiconductor manufacture and CVD
- Surface chemistry kinetics and adsorption

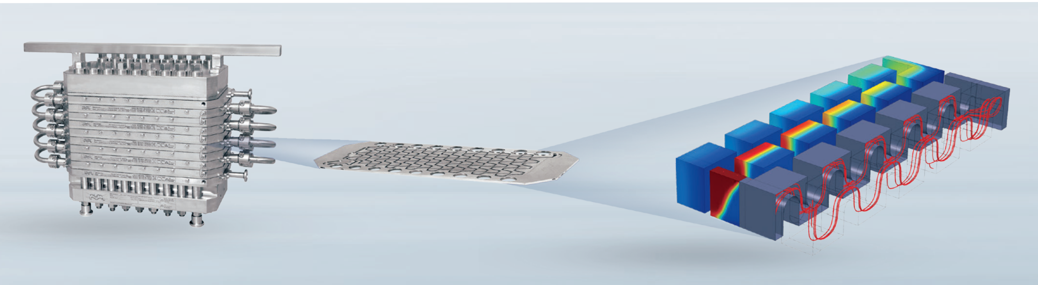
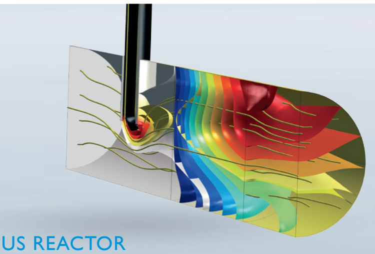


PLATE REACTOR

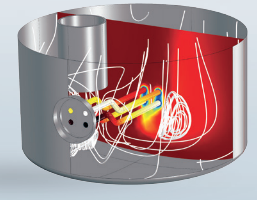
Flow streamlines and concentration in a plate reactor. One species is included at the beginning of the reactor while another is introduced half-way through it.

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 flow streamlines and concentration isosurfaces for one of the reactants and the product species are shown.



SCALE REMOVAL

The cleaning of scale from the heating element in the boiler is very much dependent on the flow of the species. This plot shows the concentration distribution of the scale (surface plot) and cleaning chemical (slice plot), and flow (streamlines).

CHEMKIN is a registered trademark of Reaction Design Corp. In the United States and other countries.

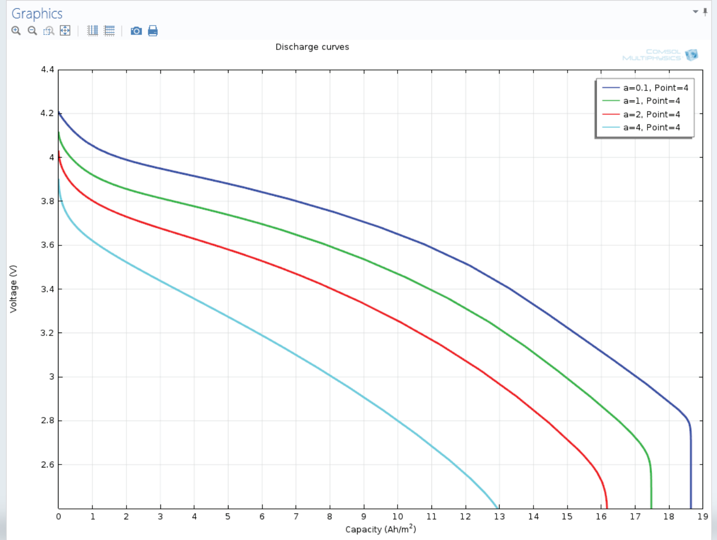
Batteries & Fuel Cells Module

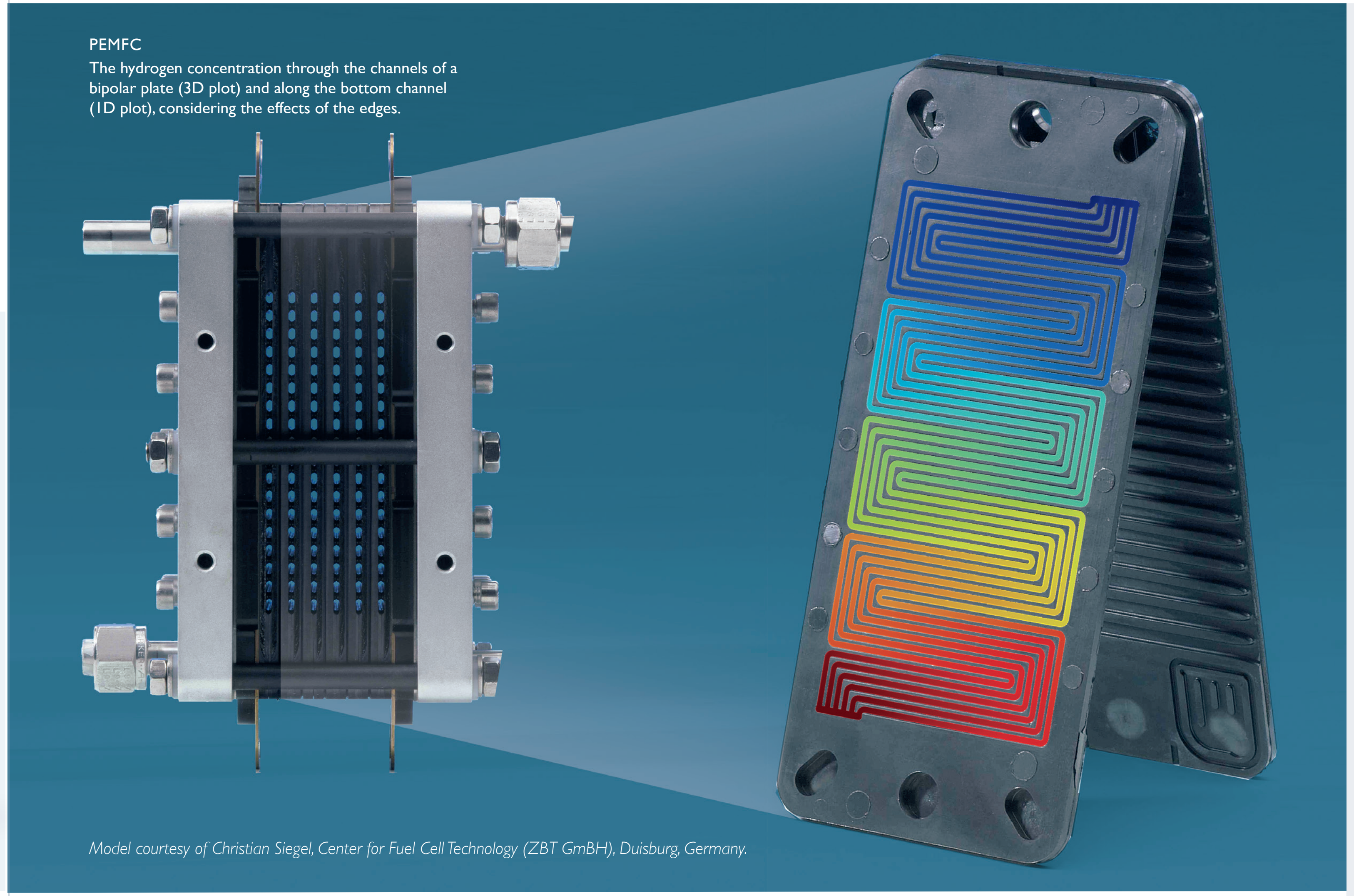
The Batteries & Fuel Cells Module provides a full set of easy-to-use tools for simulation of fundamental processes in the electrodes and electrolyte of lithium-ion batteries, nickel metal-hydride batteries, solid oxide fuel cells, and proton exchange membrane fuel cells. With it you can quickly and accurately 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. 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).

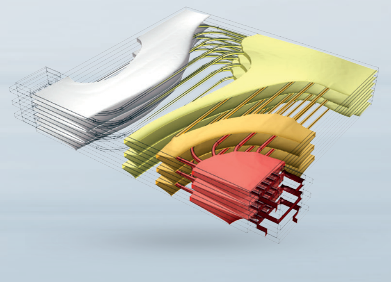




PEMFC

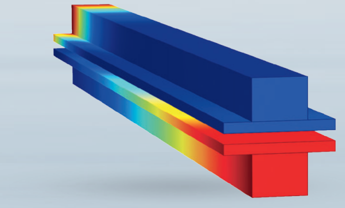
The hydrogen concentration through the channels of a bipolar plate (3D plot) and along the bottom channel (1D plot), considering the effects of the edges.

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 a detailed electrochemical model of the batteries coupled to a thermal analysis for the batteries and the components in the battery pack, and the fluid flow in the cooling channels.



SOFC

Current density distribution in a solid oxide fuel cell, including 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
- Solid Oxide

BATTERIES
- Lead Acid
- Lithium Ion
- Nickel Hydride

FLOW BATTERIES
- Vanadium
- Soluble lead

CAPABILITIES

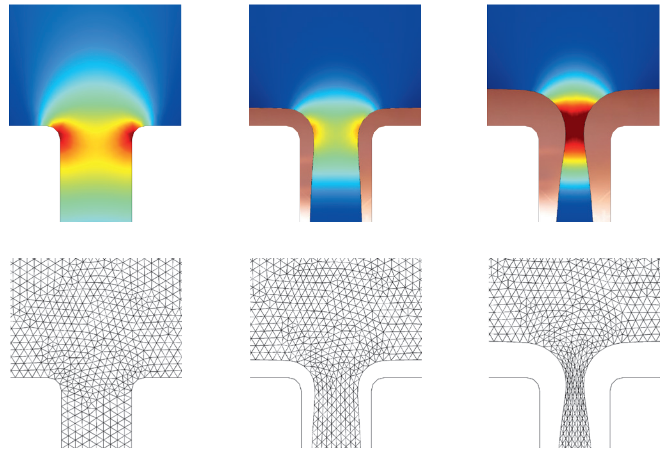
- Aging mechanisms due to structural, thermal, and chemical effects
- Detailed electrochemical analysis of cells
- Full transient studies including double layer capacitance
- Impedance spectroscopy pre-defined study for electrode kinetics and aging investigations
- Primary, secondary, and tertiary current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

Electrodeposition Module

Modeling and simulation are cost effective ways 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 simulate 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.

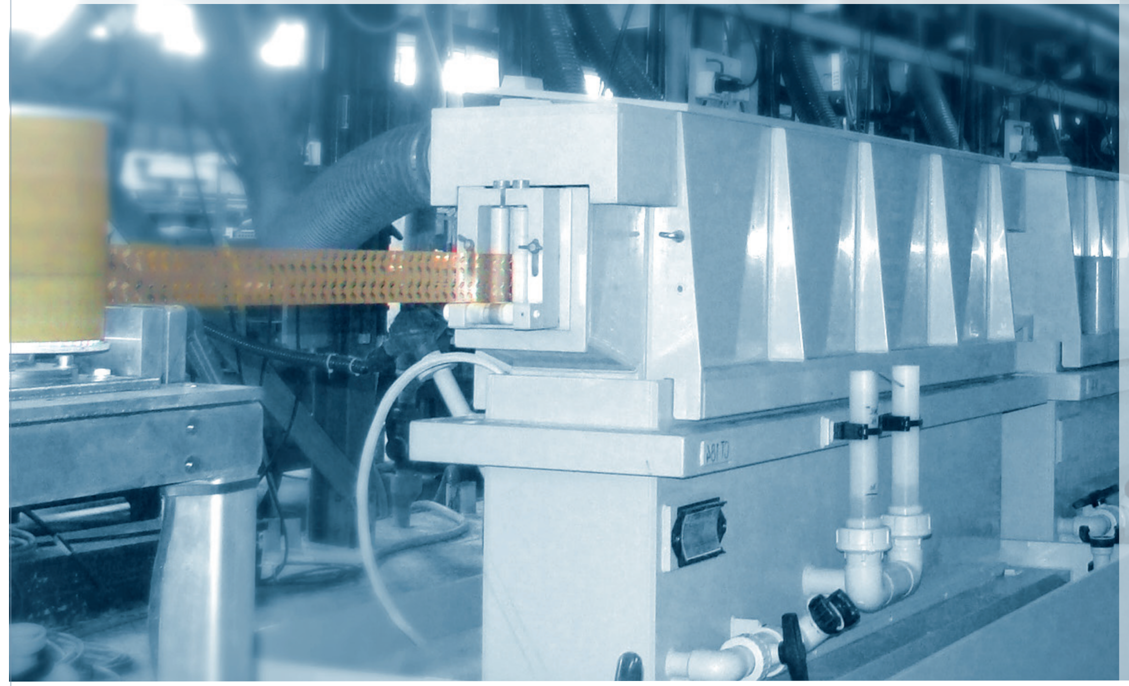
The Electrodeposition Module is applicable to a variety of diverse applications including metal deposition for electronics and electrical parts, corrosion and wear protection, decorative electroplating, electroforming of parts with thin and complex structure and metal electrowinning.

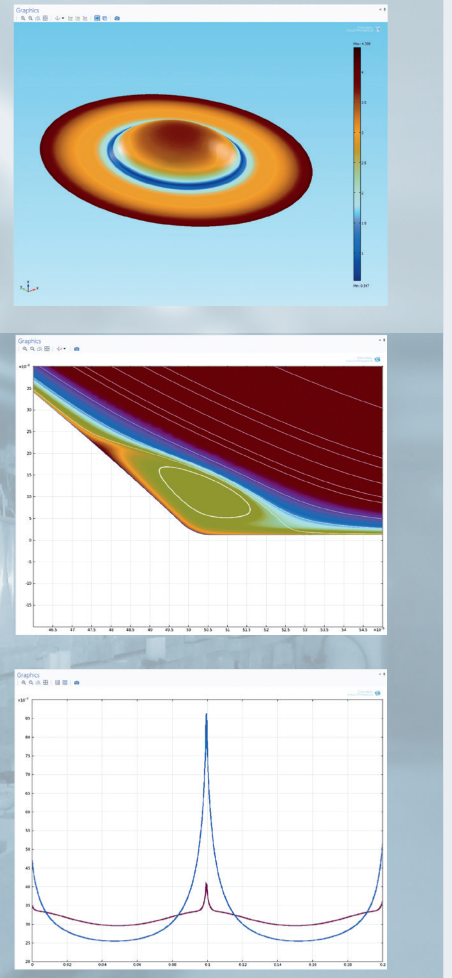


ELECTRODEPOSITION
This model demonstrates the effect of a moving boundary in the application of copper electrodeposition on circuit boards. The model is time-dependent and results clearly show that the mouth of the trench narrows, due to the non-uniform deposition of the copper.

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 does not (blue line).

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

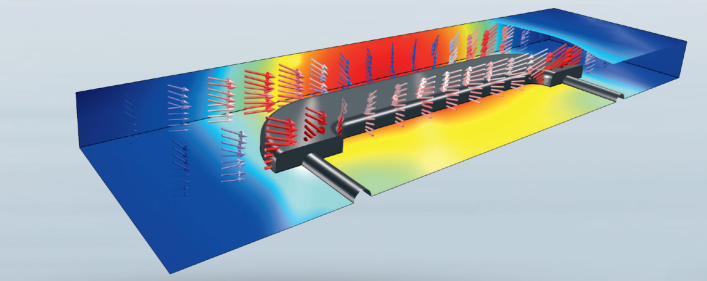


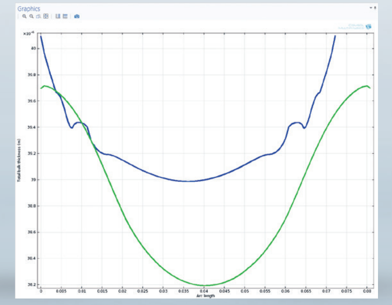


HIGHLIGHTS

- 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

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.

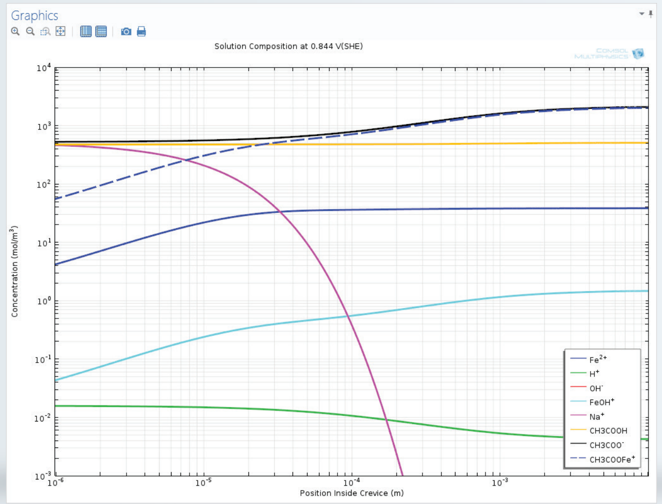




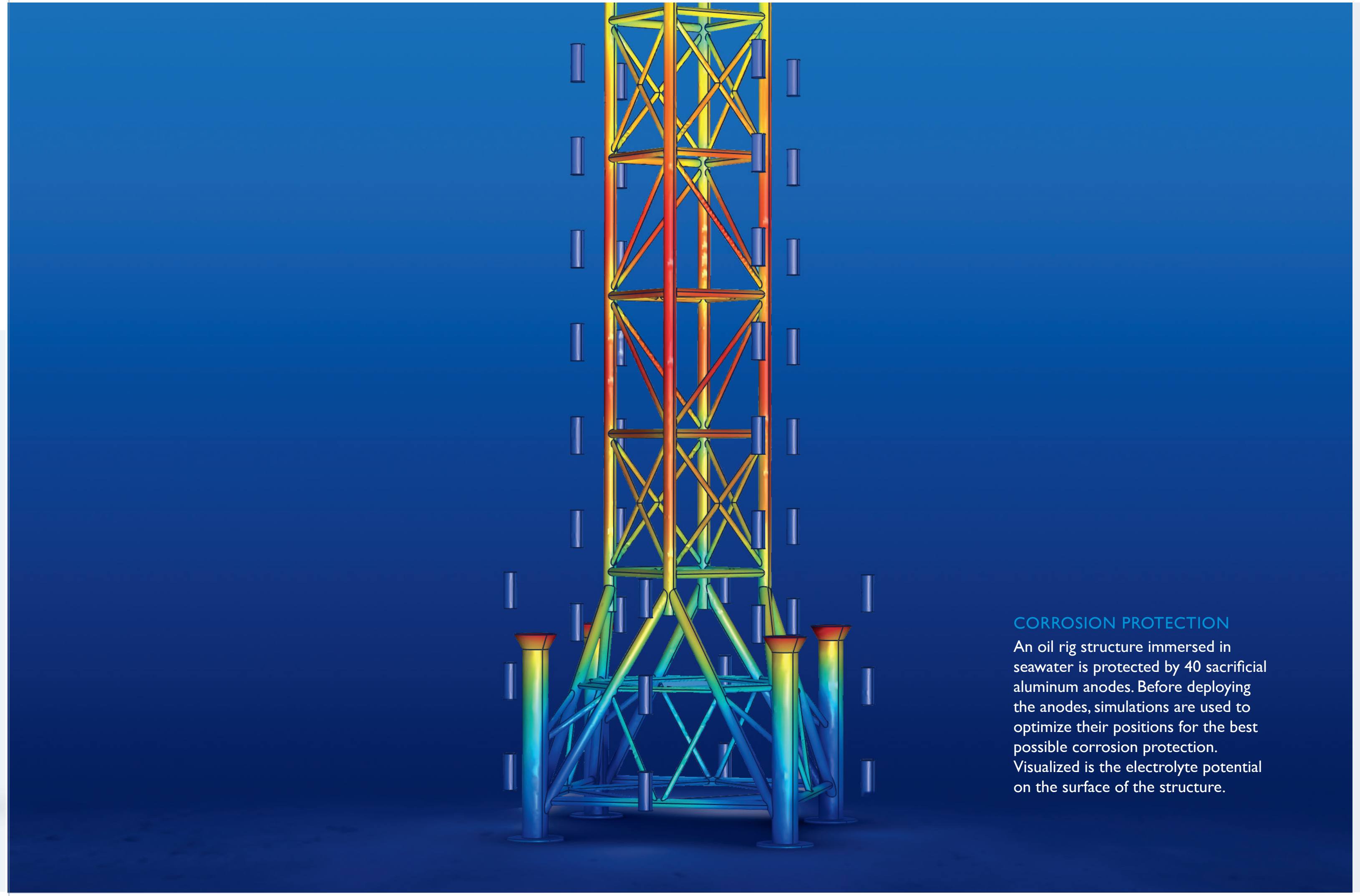
Corrosion Module

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 user 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 of the simulation. 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).

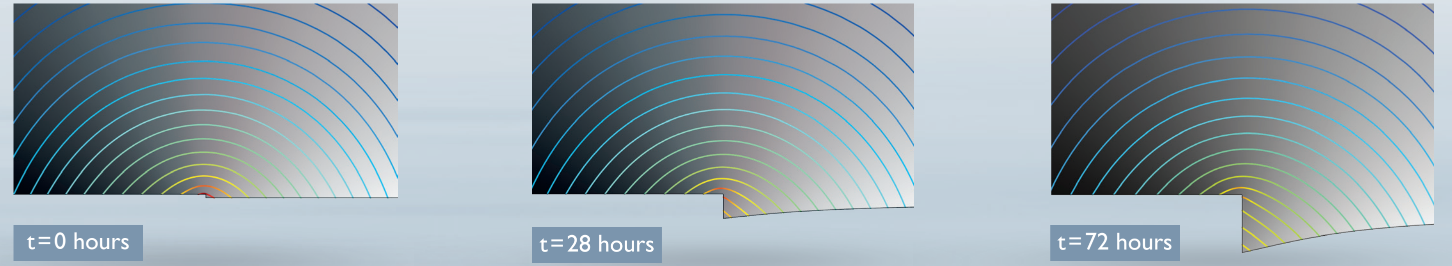


CORROSION PROTECTION
An oil rig structure immersed in seawater is protected by 40 sacrificial aluminum anodes. Before deploying the anodes, simulations are used to optimize their positions for the best possible corrosion protection. Visualized is the electrolyte potential on the surface of the structure.

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

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.

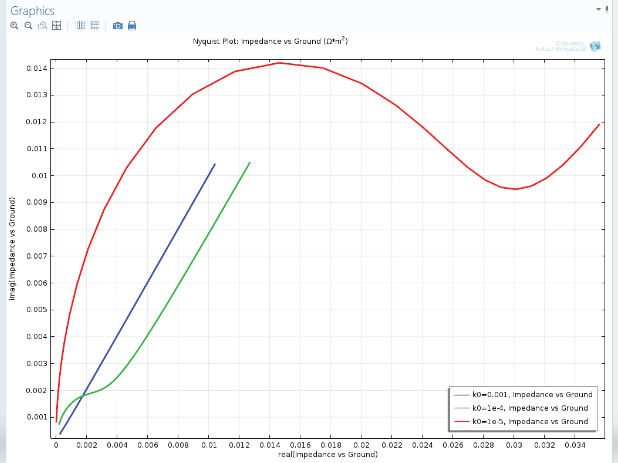


Electrochemistry Module

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.

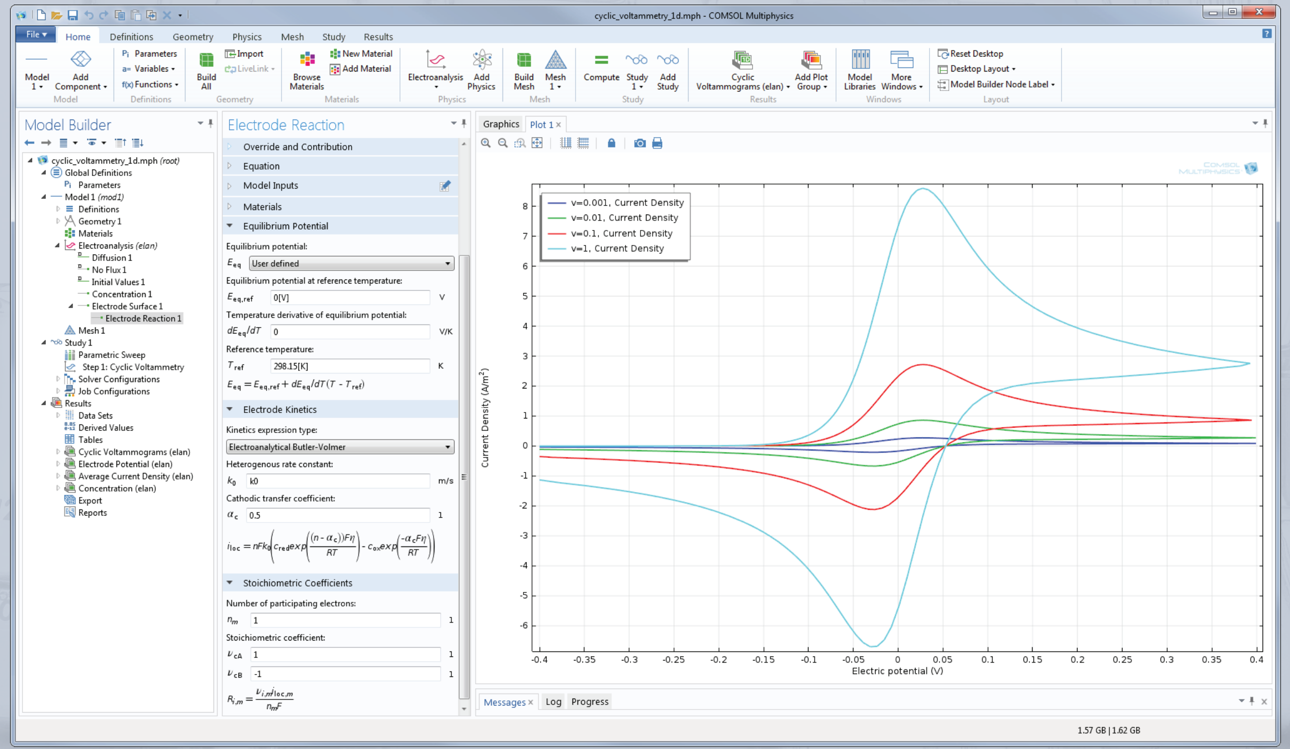
Dedicated interfaces in the Electrochemistry Module enable the definition of voltammetry, amperometry, potentiometry, electrochemical impedance, and coulometry studies. Exchange current densities and activation overpotentials can then be determined from the combined experiment and simulation results.

The Electrochemistry Module permit 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

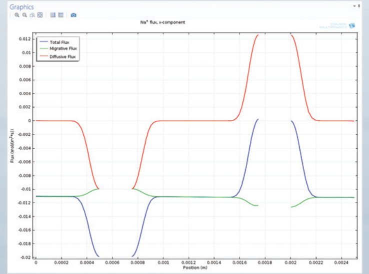
Cyclic voltammetry is a common technique for electrochemical analysis. During simulation, the potential at the working electrode is swept over a voltage range while the current is recorded. This waveform provides information about the reactivity and mass transport properties of an electrolyte.

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

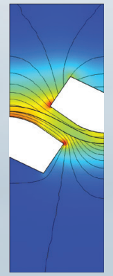
ELECTRODIALYSIS

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, biomedical engineering.



CHLOR-ALKALI MEMBRANE CELL

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



Optimization Module

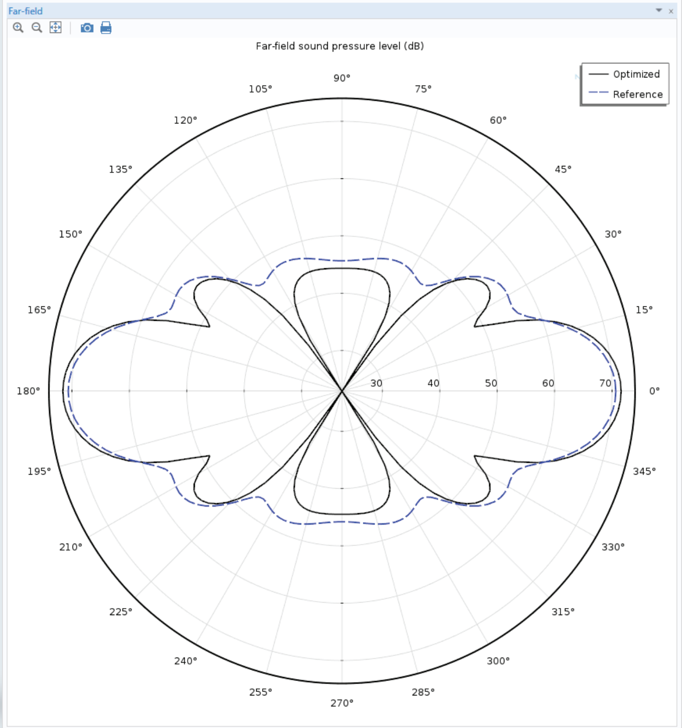
No matter the engineering discipline, once you have a model of a product or a process, you want to improve upon it. The Optimization Module can be used throughout the COMSOL Multiphysics product family; it is a general interface for computing optimal solutions to engineering problems. Any model input, 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.

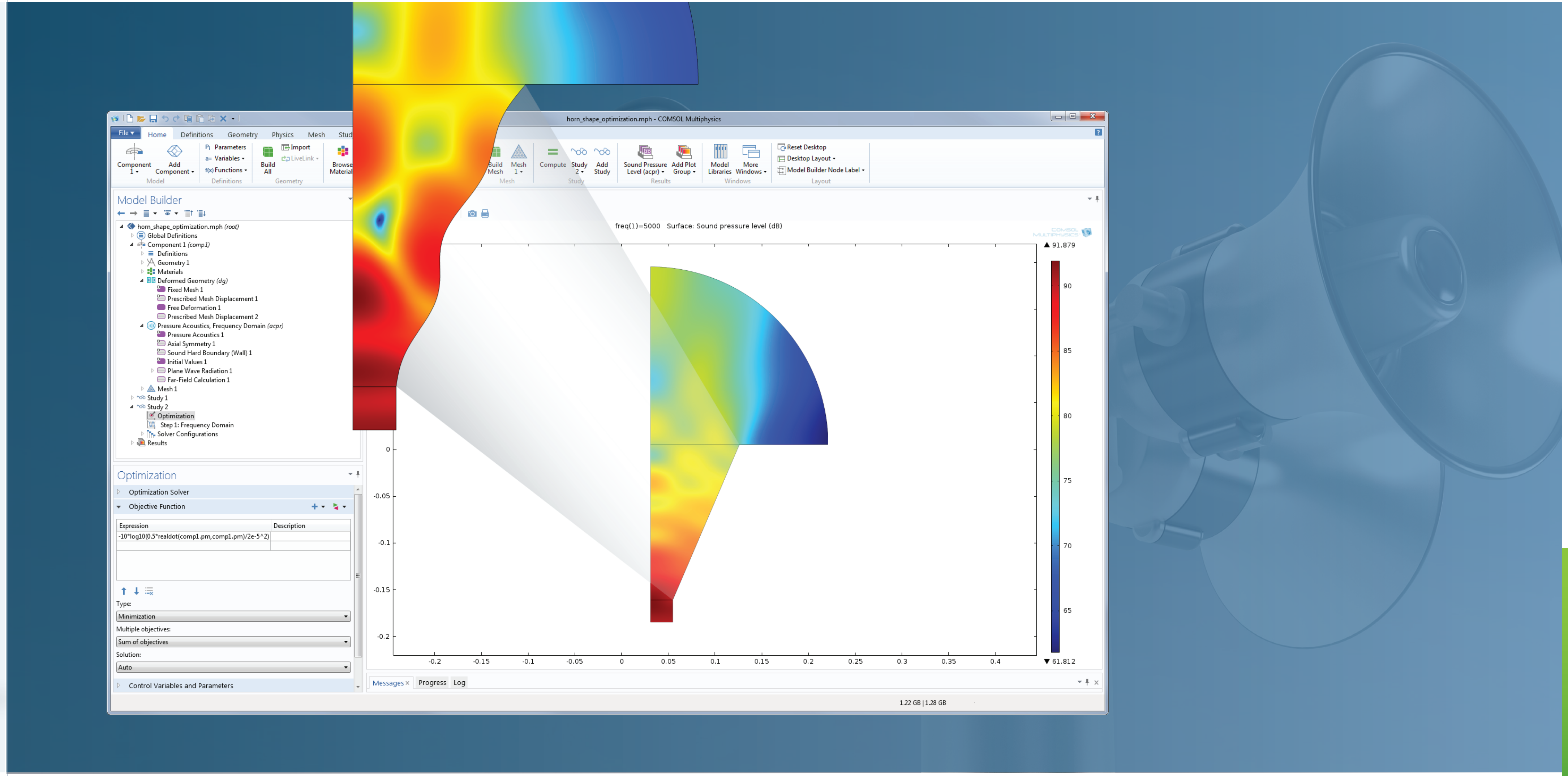
Both derivative-based and derivative-free methods are available in order to cover a wide range of optimization problems. The following derivative-based methods are available: SNOPT, MMA, and Levenberg-Marquardt. The following derivative-free methods are available: coordinate search, Monte Carlo, Nelder-Mead, 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.

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.





Expression	Description
-10*log10(0.5*realdot(comp1.pm,comp1.pm)/2e-5^2)	

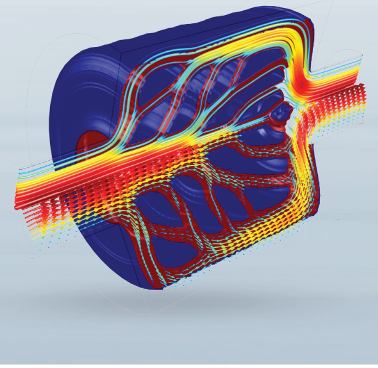
Type: Minimization

Multiple objectives: Sum of objectives

Solution: Auto

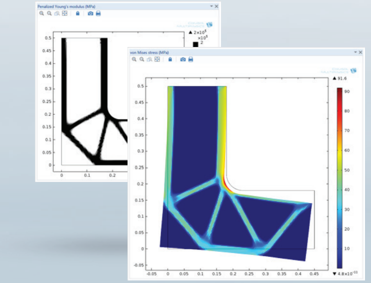
PLACING OF CATALYST IN A REACTOR

A chemical solution is pumped through a catalytic reactor where a solute species reacts in contact with the catalyst surface. The model aims to maximize the total reaction rate of the solute for a given total pressure difference across the bed by finding an optimal catalyst distribution. Shown is the catalyst distribution (as the height), direction of flow (streamlines) and concentration distribution (color plot).



TOPOLOGICAL OPTIMIZATION

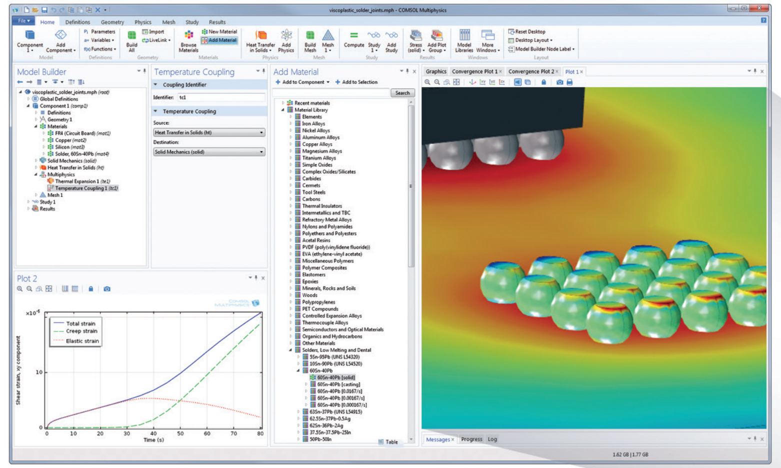
The SIMP model (Solid Isotropic Material with Penalization) for structural topology optimization is used to minimize weight given a minimum stiffness of 40% the value computed from the fully solid design. Such a design reduces both weight and costs while maintaining the specified load carrying capacity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



Material Library

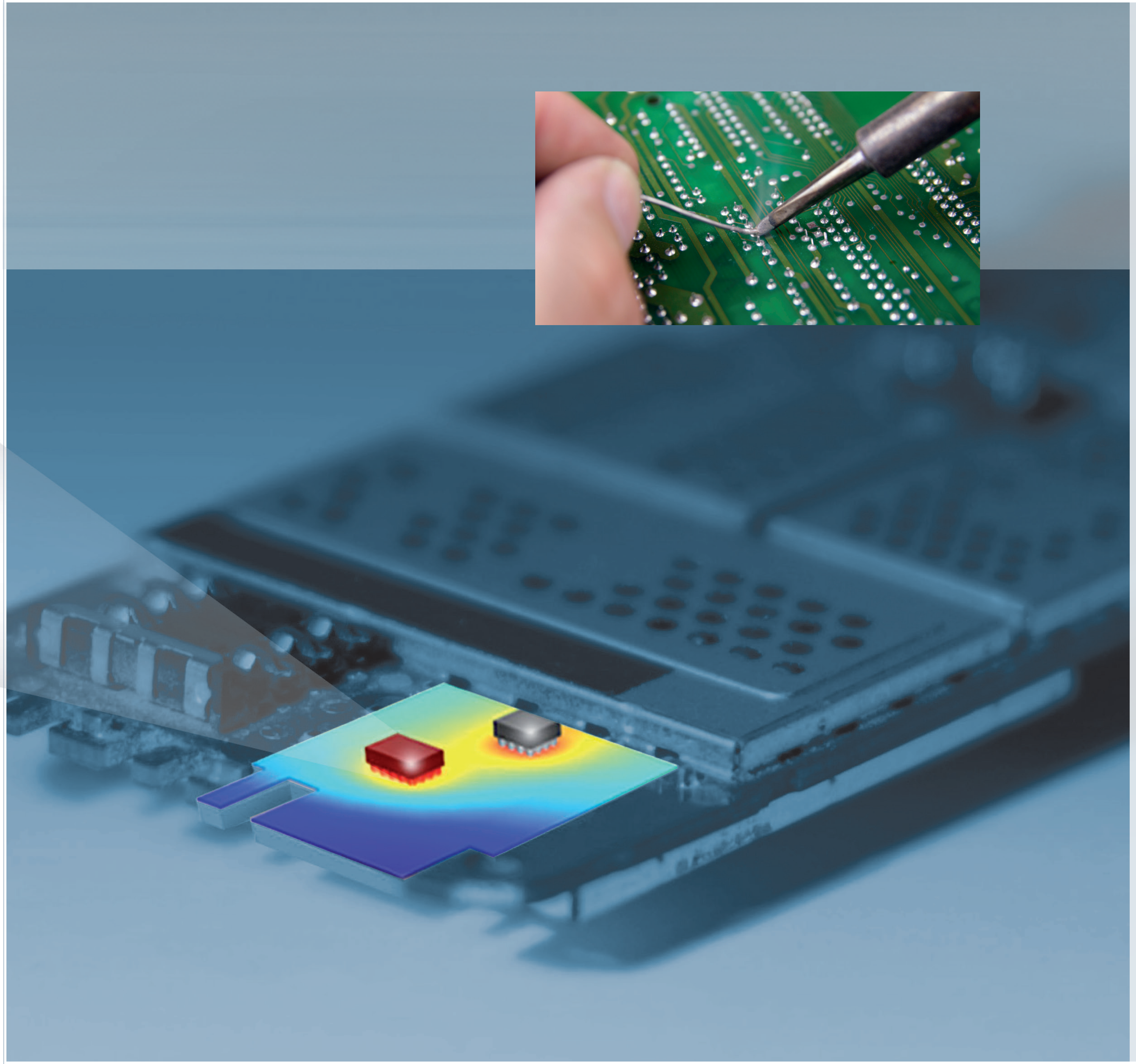
COMSOL Multiphysics gives you complete control over definitions and use of your material properties through the Model Builder and Material Browser. The Material Browser allows you to manage all your model's materials in one place and can be complemented by the Material Library.

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.



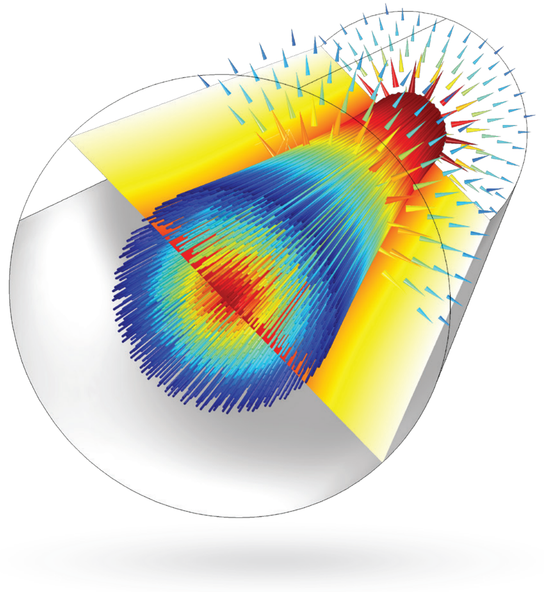
AVAILABLE MATERIALS

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

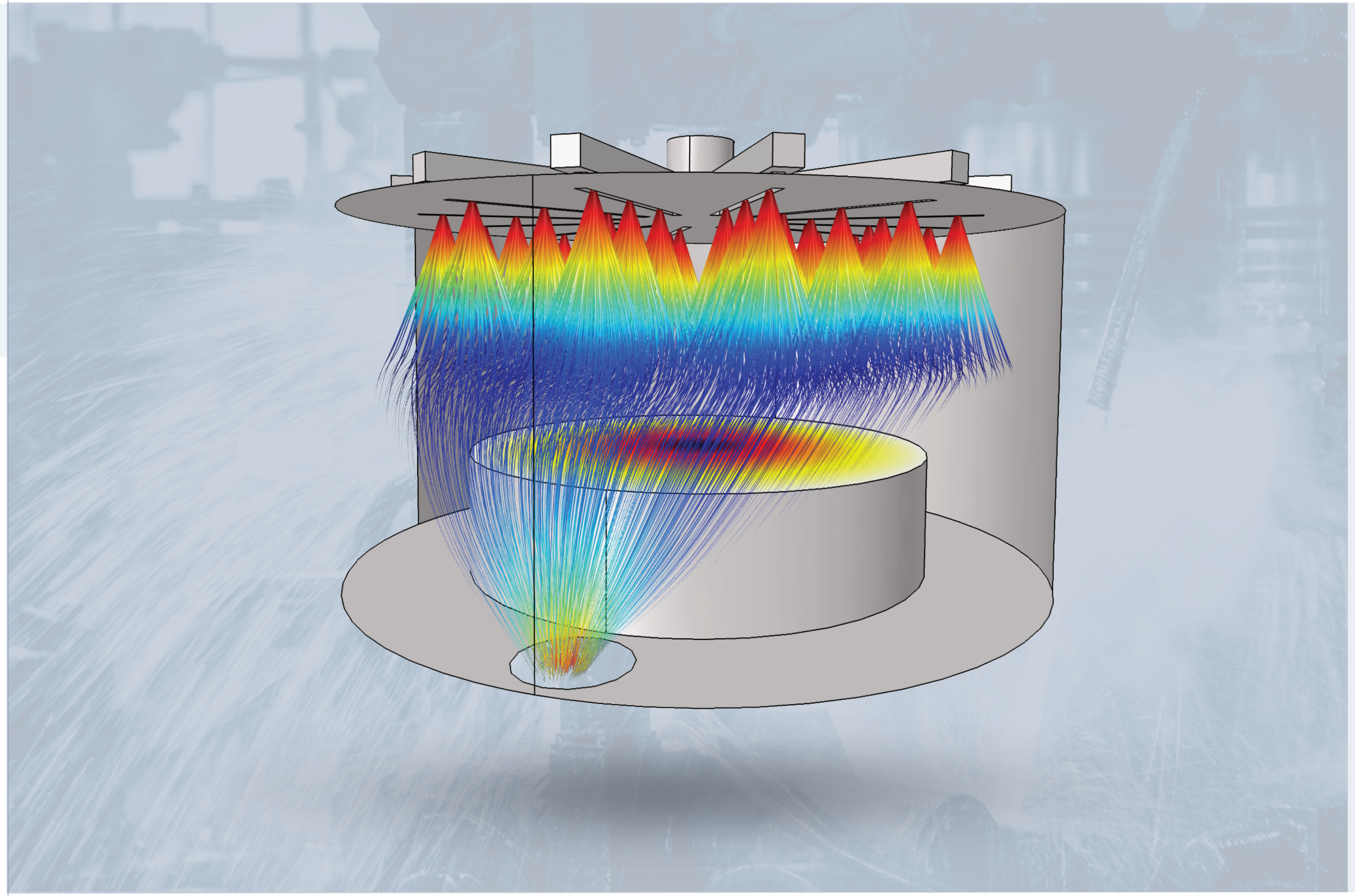
Particle Tracing Module

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.



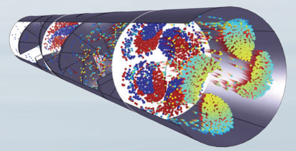
PARTICLE-FIELD INTERACTIONS
The Charged Particle Tracing interface can be used to model the divergence of a beam of electrons diverging due to its own space charge. An iterative procedure allows the particle trajectories and the beam potential to be computed in a self-consistent manner. In this image, 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.



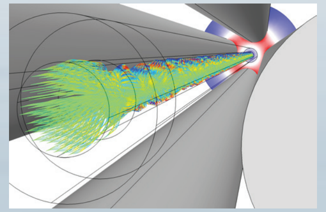
CVD CHAMBER
The Particle Tracing Module offers many ways to release particles. Particles may be released from an inlet surface or within a domain, and their initial positions may either be based on the finite element mesh or prescribed via a user-defined density function. It is also possible to release 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.

HIGHLIGHTS

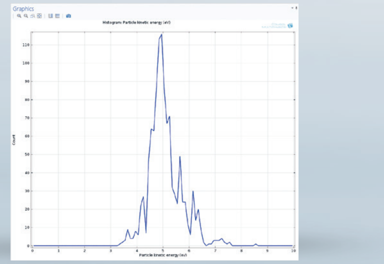
- Acoustophoresis
- Aerosol dynamics
- Beam physics
- Brownian motion
- Classical mechanics
- Dielectrophoresis and Magnetophoresis
- Electric propulsion
- Fluid flow visualization
- Fluid-particle interactions
- Geometrical optics in continuous media
- Ion energy distribution function visualization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



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.



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® geometry engine, the CAD Import Module enables 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®, Inventor®, AutoCAD®, Creo™ Parametric, 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 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.

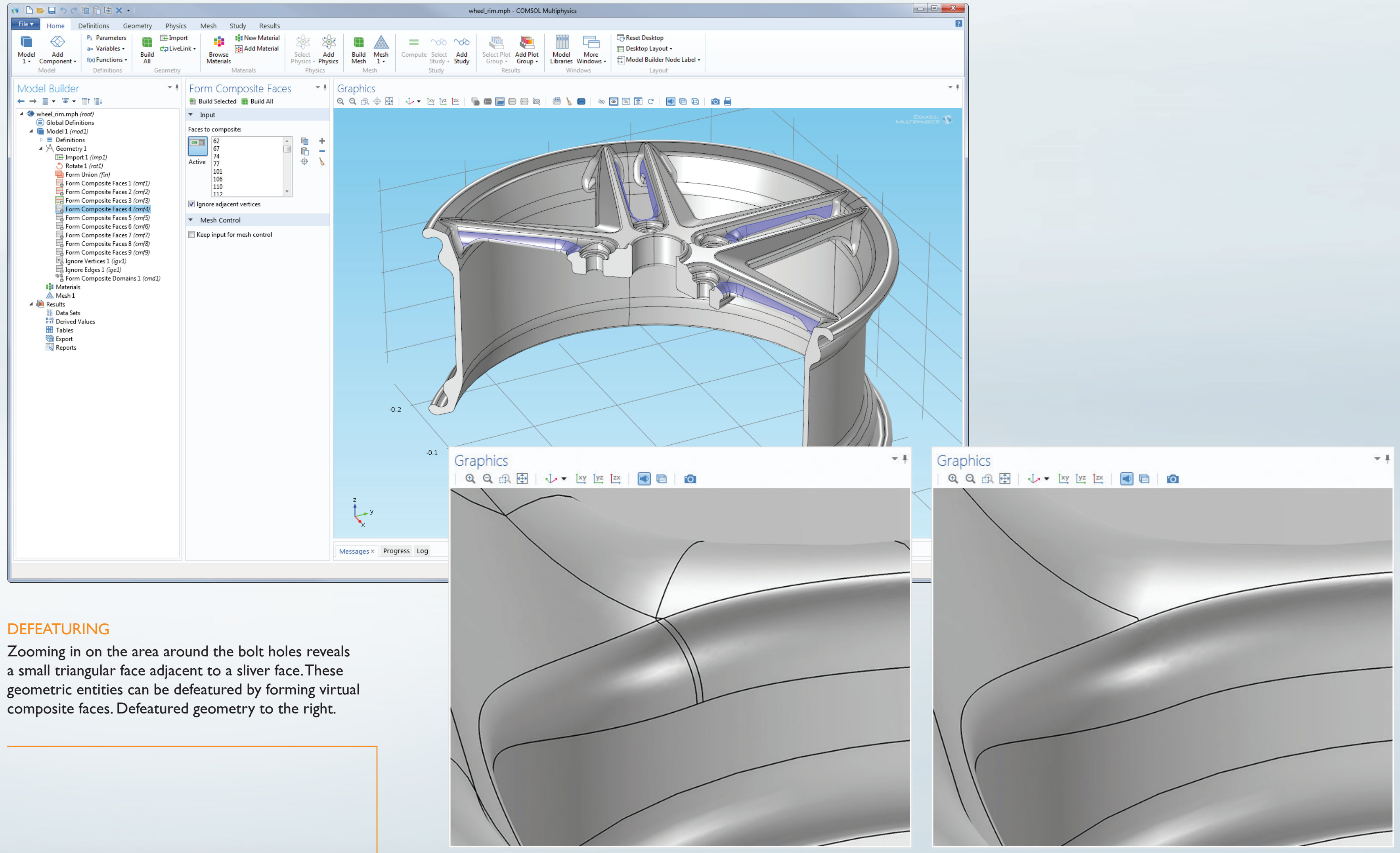
SPATIAL

Powered by PARASOLID

SUPPORTED CAD FILE FORMATS

ACIS® (.sat, .sab, .asat, .asab)
Autodesk Inventor® (.ipt, .iam)
CATIA® V5 (.CATPart, .CATProduct) *
Creo™ Parametric (.prt, .asm)
IGES (.iges, .igs)
Parasolid® (.x_t, .xmt_txt, .x_b, .xmt_bin)
Pro/ENGINEER® (.prt, .asm)
SolidWorks® (.sldprt, .sldasm)
STEP (.step, .stp)

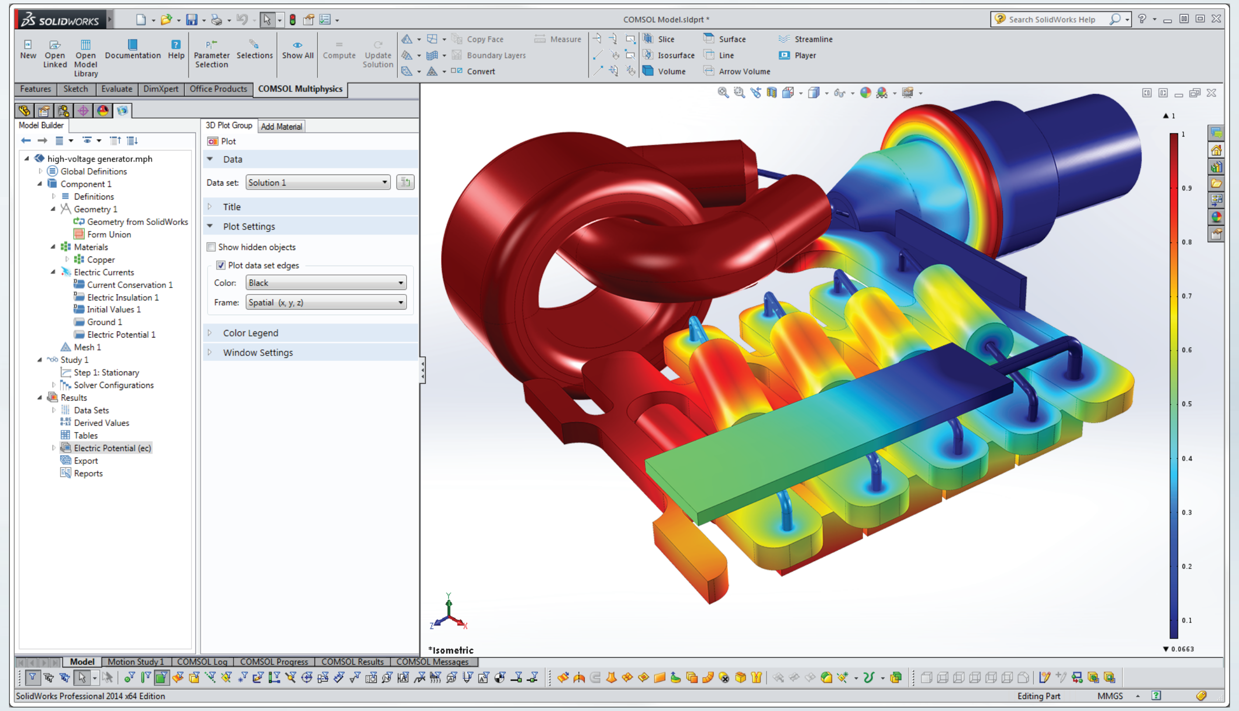
* Requires File Import *for* CATIA® V5.



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 to the right.

LiveLink™
for SolidWorks®



ONE WINDOW INTERFACE

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

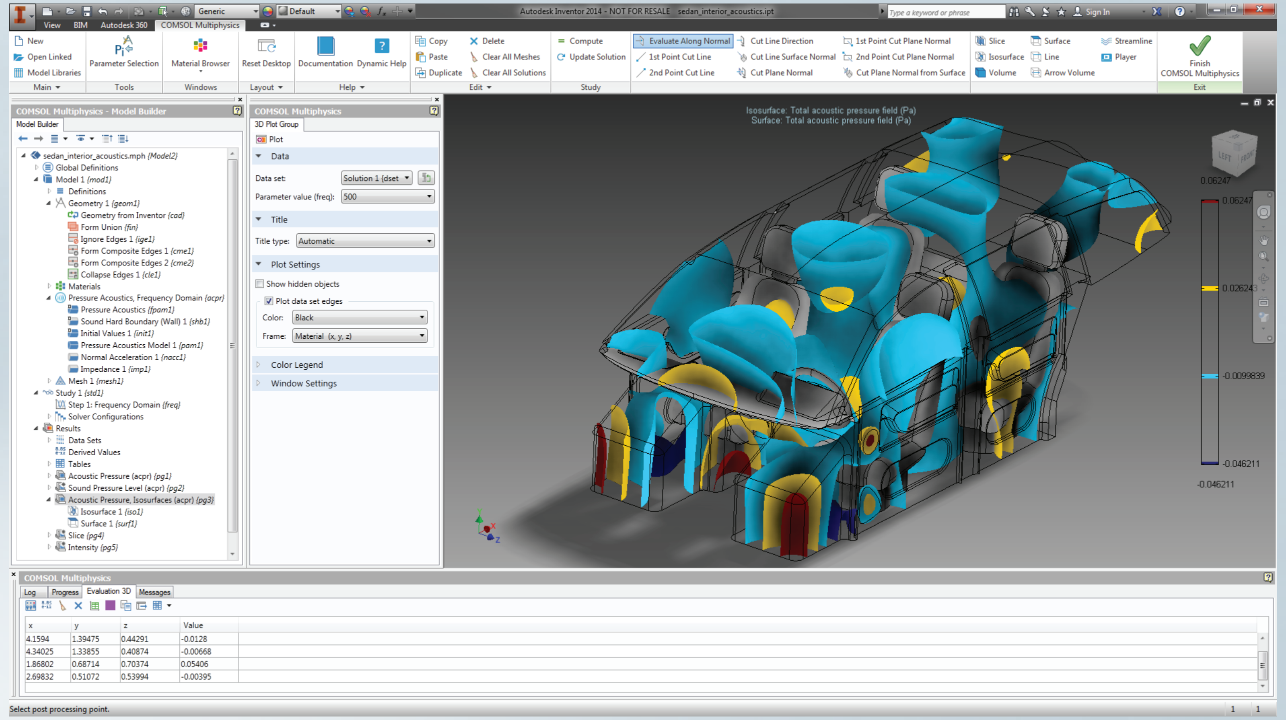
HIGH-VOLTAGE GENERATOR

Simulation of the electromagnetic field in a high-voltage generator in an X-ray device. The design is created in SolidWorks® 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.

SOLIDWORKS
Solution Partner

LiveLink™
for Inventor®



ONE WINDOW INTERFACE

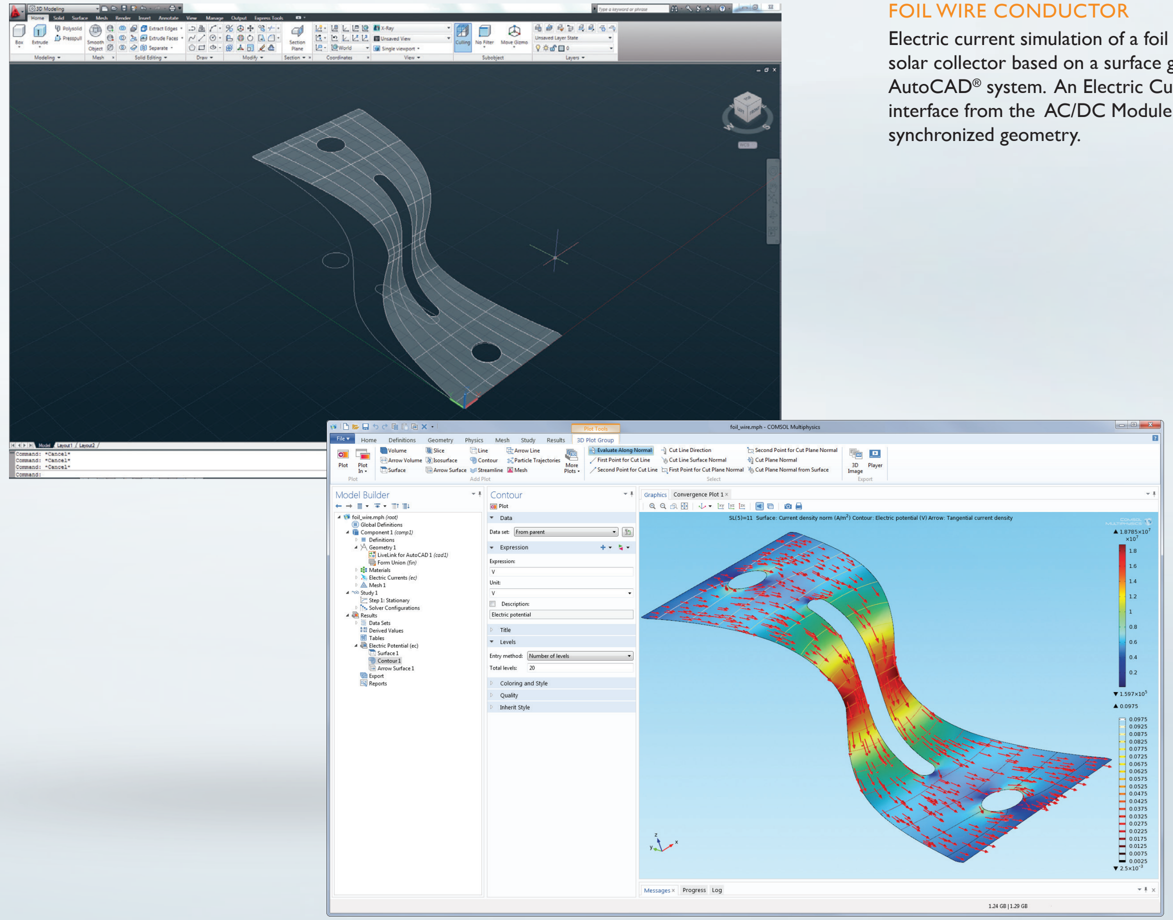
The LiveLink™ interface for the Inventor® CAD system has been extended with a One Window Interface where Inventor® users can stay inside its environment and perform their multiphysics analysis directly from it. All of the COMSOL Multiphysics modeling tools are available through the One Window interface, 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 Inventor® CAD system while the simulation is set up and performed using COMSOL Multiphysics.

AUTODESK
Authorized Developer

LiveLink™
for AutoCAD®

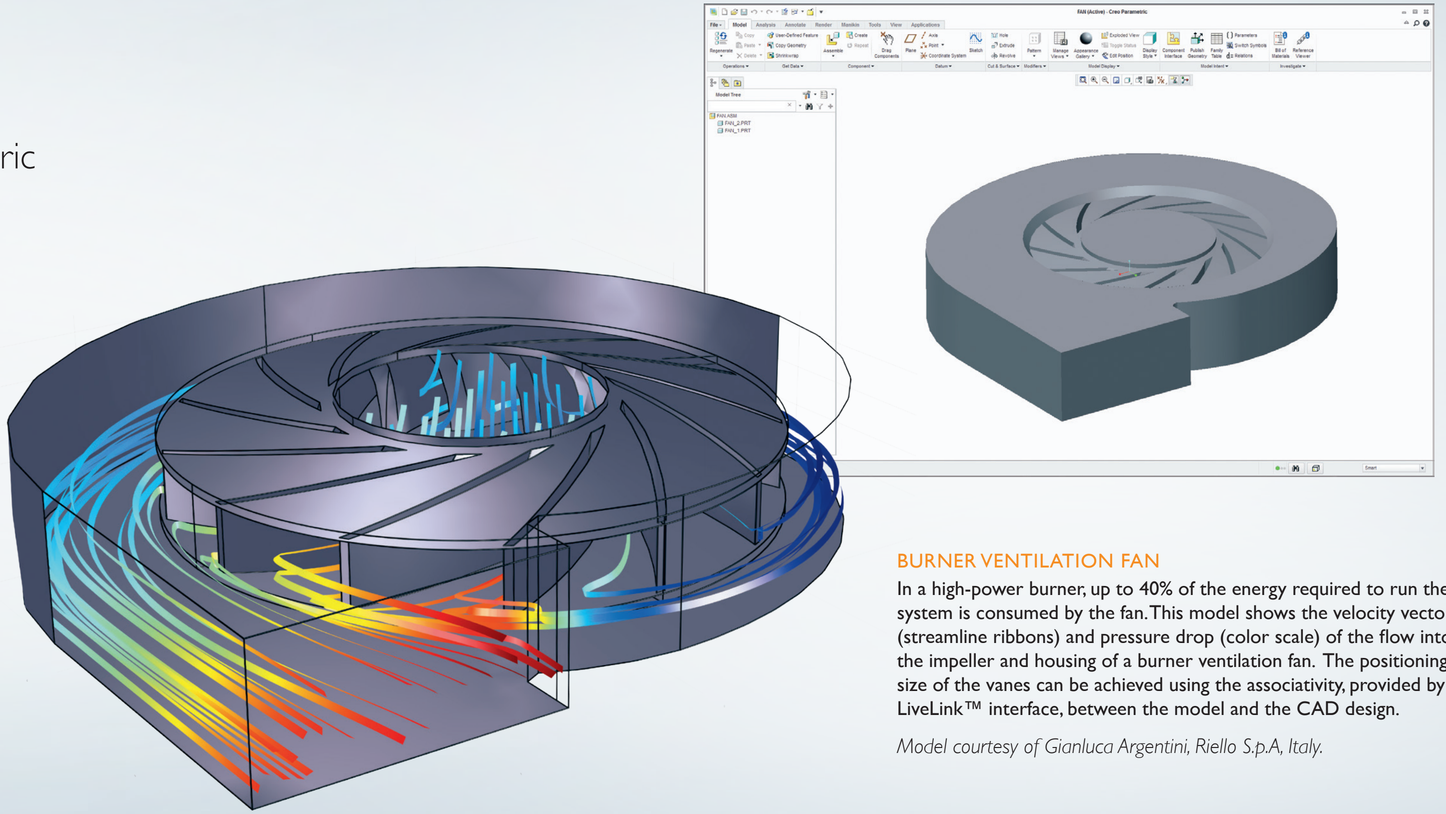


FOIL WIRE CONDUCTOR

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

AUTODESK
Authorized Developer

LiveLink™
for Creo™ Parametric



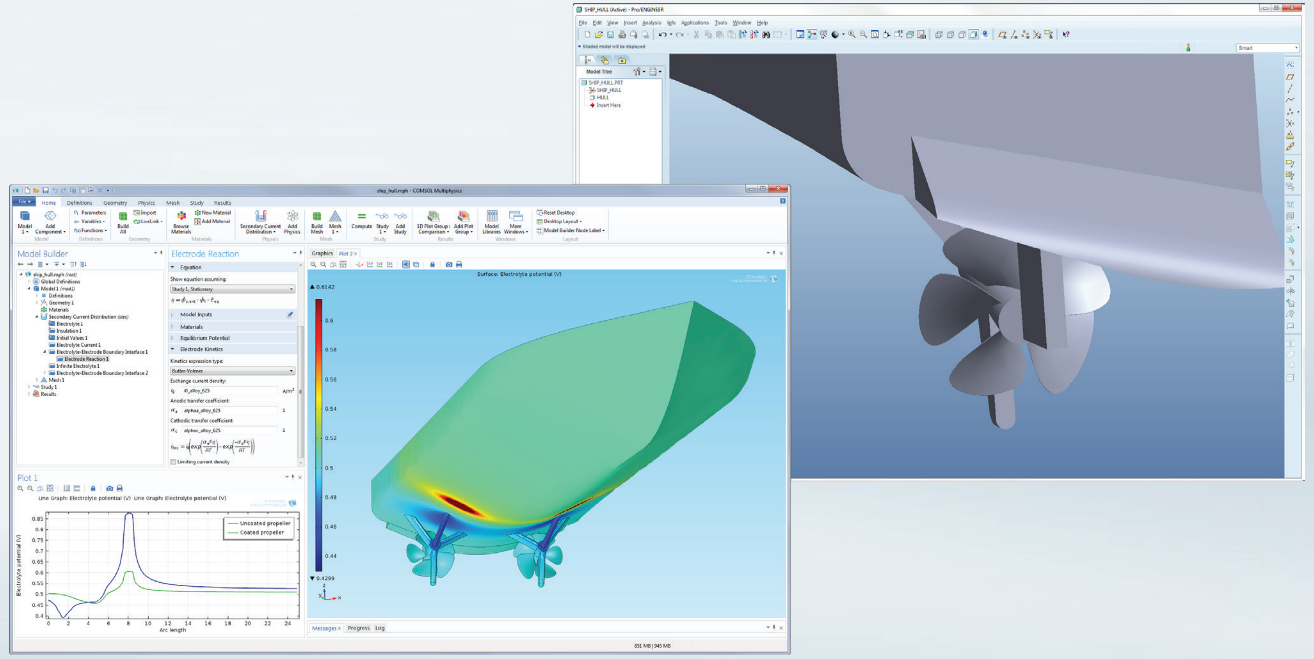
BURNER VENTILATION FAN

In a high-power burner, up to 40% of the energy required to run the system is consumed by the fan. This model shows the velocity vector (streamline ribbons) and pressure drop (color scale) of the flow into the impeller and housing of a burner ventilation fan. The positioning and size of the vanes can be achieved using the associativity, provided by the LiveLink™ interface, between the model and the CAD design.

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

PTC® PartnerAdvantage

LiveLink™
for Pro/ENGINEER®



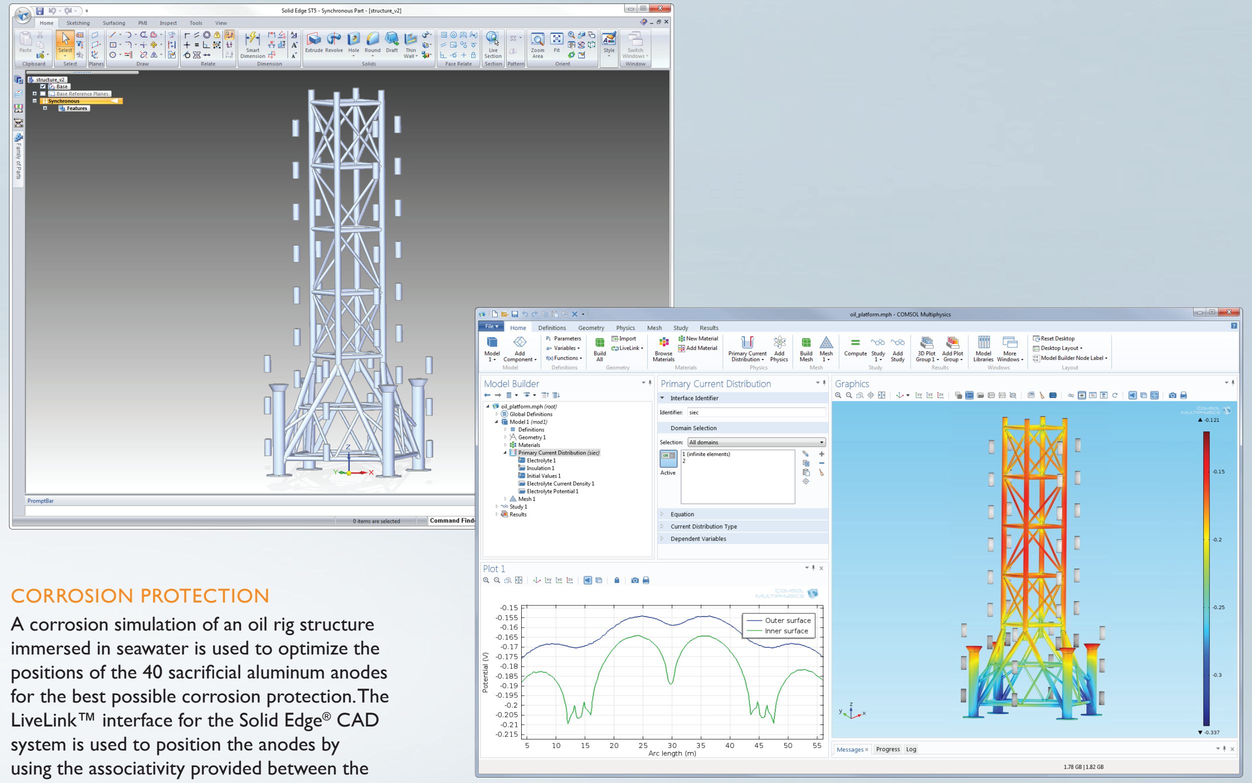
CORROSION MITIGATION

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

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®



CORROSION PROTECTION

A corrosion simulation of an oil rig structure immersed in seawater is used to optimize the positions of the 40 sacrificial aluminum anodes for the best possible corrosion protection. The LiveLink™ interface for the Solid Edge® CAD system is used to position the anodes by using the associativity provided between the model and the CAD geometry.

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

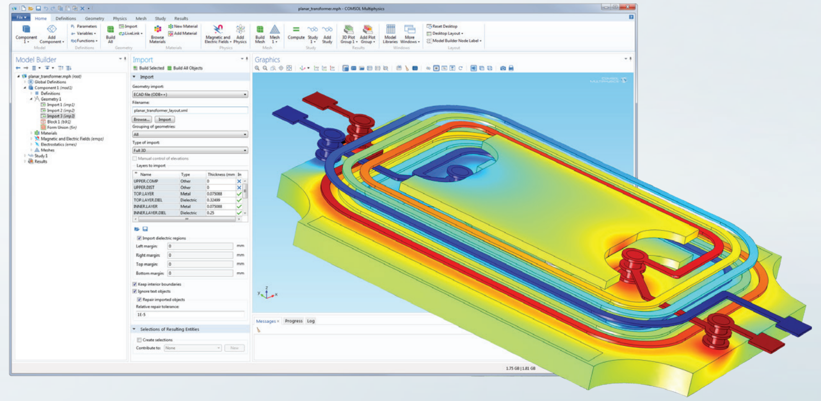
Using the ECAD Import Module, you can bring your ECAD files into COMSOL Multiphysics where the 2D layouts are automatically converted to 3D CAD models. This opens up the world of modeling for simulating, among other applications, the components in integrated circuits, systems of connected MEMS devices,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

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

The layout is automatically extruded and converted to a 3D CAD model for use in any kind of COMSOL Multiphysics simulations with any combination of add-on products. The 3D geometry model can further take part in general solid modeling operations in COMSOL Multiphysics. When combined with the CAD Import Module 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

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

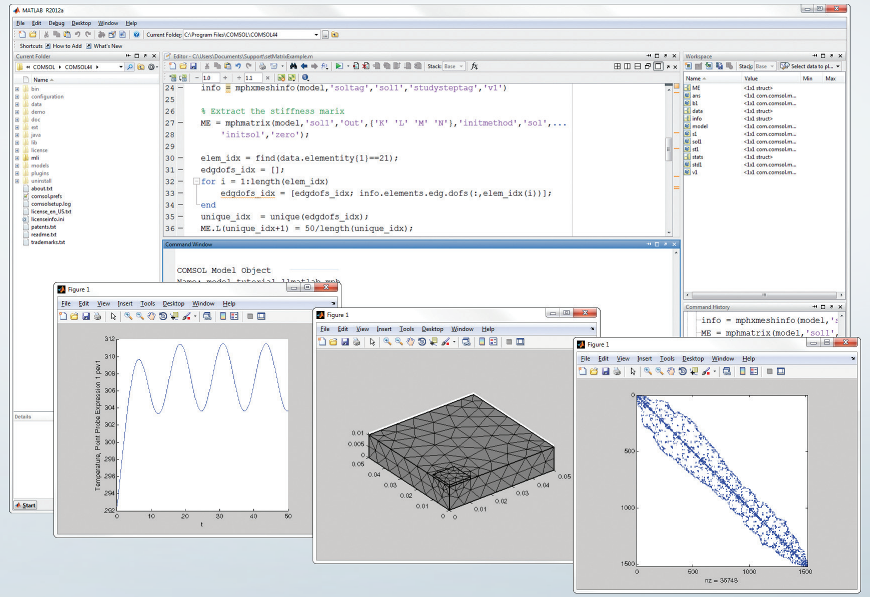


POWER ELECTRONICS
The simulation results show the electric potential on the surface of a planar transformer. The layout is imported from an ODB++(X) file using the ECAD Import Module and is converted to a 3D geometry model.

LiveLink™
for MATLAB®

LiveLink™ *for* MATLAB® allows the user to control a model both from MATLAB® and from the COMSOL Desktop - at the same time. This streamlines the workflow and you can use MATLAB® 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® 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 on state-space matrix format for incorporating in control systems
- Call MATLAB® functions from the COMSOL Desktop



```
info = mphxmeshinfo(model,'soltag','sol1','studysteptag','v1')

% Extract the stiffness marix
ME = mphmatrix(model,'sol1','Out',{'K' 'L' 'M' 'N'},'initmethod','sol',...
    'initsol','zero');

elem_idx = find(data.elementity{1}==21);
edgdofs_idx = [];
for i = 1:length(elem_idx)
    edgdofs_idx = [edgdofs_idx; info.elements.edg.dofs(:,elem_idx(i))];
end
unique_idx  = unique(edgdofs_idx);
ME.L(unique_idx+1) = 50/length(unique_idx);
```

```
COMSOL Model Object
```

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® workspace, where the state-space model has been run in the MATLAB® environment. The figures show: temperature vs. time, the finite element mesh, and the sparsity pattern of the stiffness matrix.

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

LiveLink™
for Excel®

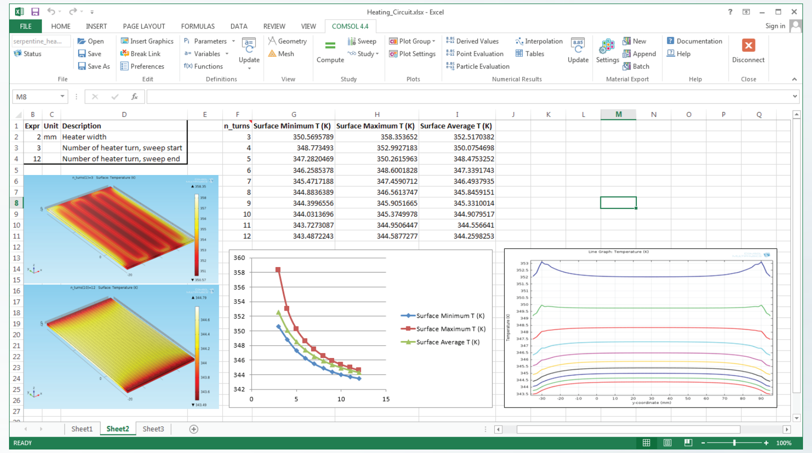
Extend your modeling capacity by running COMSOL Multiphysics simulations from a spreadsheet with LiveLink™ for Excel®. Parameters and variables defined and modeled in COMSOL Multiphysics are instantly available in Microsoft® Excel® and automatically synchronized to your COMSOL model.

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® files for parameter and variable lists in the COMSOL Desktop environment. LiveLink™ for Excel® requires Excel® 2007, 2010 or 2013 for Windows®.

SUPPORTED FILE FORMATS

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



Expr	Unit	Description
2	mm	Heater width
3		Number of heater turn, sweep start
12		Number of heater turn, sweep end

n_turns	Surface Minimum T (K)	Surface Maximum T (K)	Surface Average T (K)
3	350.5695789	358.353652	352.5170382
4	348.773493	352.9927183	350.0754698
5	347.2820469	350.2615963	348.4753252
6	346.2585378	348.6001828	347.3391743
7	345.4717188	347.4590712	346.4937935
8	344.8836389	346.5613747	345.8459151
9	344.3996556	345.9051665	345.3310014
10	344.0313696	345.3749978	344.9079517
11	343.7273087	344.9506447	344.556641
12	343.4872243	344.5877277	344.2598253

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® for easy access to parameters, variables, functions, geometry, mesh, solvers, and results.

User Support

✓ Software Subscription

You automatically receive 12-month access to software updates and technical support with your COMSOL purchase. This assures you not only that your license stays current but that you also benefit from the fast-paced development of new functionality, new solvers, and new models. Additionally, the subscription program provides license administration services to install COMSOL Multiphysics 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 COMSOL with tools such as CAD (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

The online COMSOL Community lets you network and share modeling tips and tricks with thousands of users and across all scientific and engineering disciplines. You can join the discussion forum, check out the Blog, download models, view user presentations, and access the latest news.

✓ 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 Multiphysics 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 Choices

A variety of license options meets the requirements of any user, no matter what the size of the organization or the computer setup:

Named Single User License: One physical person designated to COMSOL by name can run a session of the software. This designated user may be replaced on a temporary or permanent basis provided that only one individual is designated to COMSOL as the user at any given time. A Named Single User License may not be used or accessed over a network.

CPU-Locked Single User License: You can install the software on one computer and different users can take turns using COMSOL on that computer, one session at a time. A CPU-Locked Single User License may not be used or accessed over a network.

Floating Network License: 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. Cluster computing is supported on Windows® Compute Cluster Server 2003, Windows HPC Server 2008, and Red Hat® Enterprise Linux® 5. Cloud computing is supported on Amazon Elastic Compute Cloud™ (Amazon EC2™).

Class Kit License: 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.

Supported platforms: Windows®, Linux®, and Macintosh

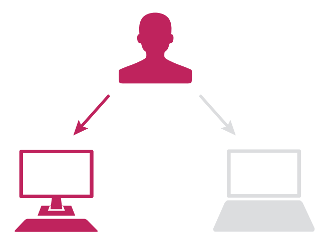
High-Performance Computing: All license options include support for multicore computing with no limit on the number of cores. The Floating Network License includes additional support for cluster and cloud computing with no limit on the number of compute nodes.

Visit us at www.comsol.com/contact for more information.

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

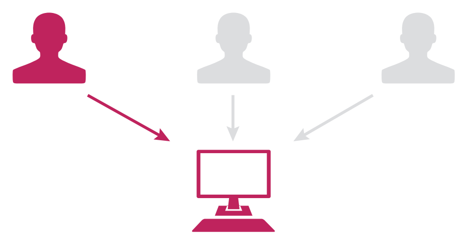
For detailed license conditions, please refer to the COMSOL Software License Agreement.

Named Single User License
Single Designated User



*Access or use over a network is not permitted

CPU-Locked Single User License
Single Designated Computer



*Access or use over a network is not permitted

Floating Network License
Multiple Users

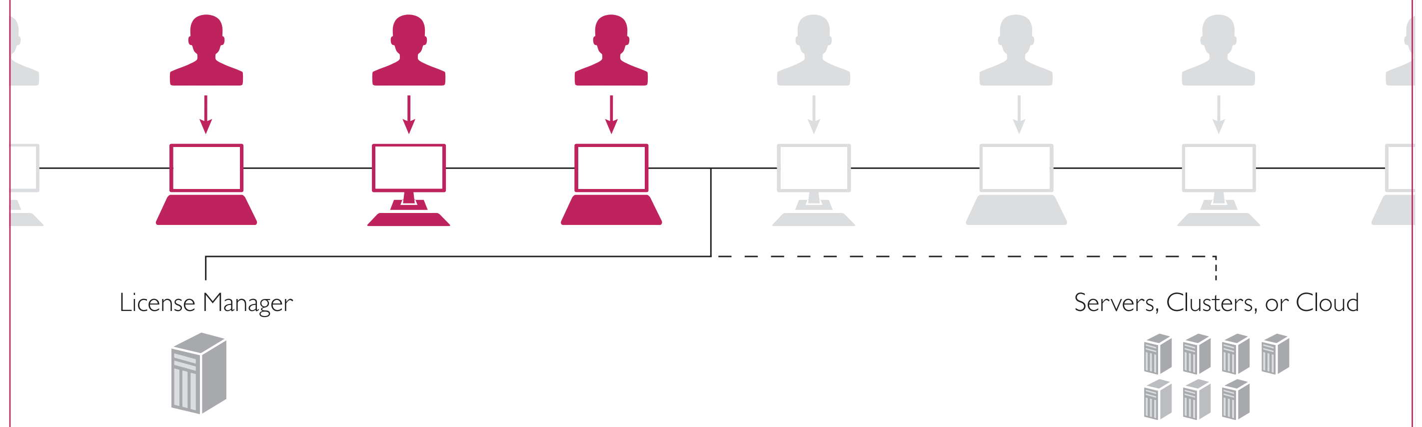


*Diagram depicts a Floating Network License enabled for 3 concurrent users

All Licenses Perpetual, 20% yearly support and upgrades www.comsol.com

COMSOL MULTIPHYSICS®

PRODUCT BOOKLET

VERSION 4.4

COMSOL