

COMSOL APPLICATION NOTES

SIMULATION-DRIVEN DESIGN

*for the Thermal Management
of Electromagnetic Devices*

TABLE OF CONTENTS

INTRODUCTION	3
ENGINEERING OBJECTIVES	
Thermal Management in Electromagnetic Devices	4
Heat Transfer Applications	4
The Mechanisms of Heat Transfer	5
NUMERICAL SIMULATION	
Numerical Simulation for Electronic Design	6
Physics Interfaces for Modeling Heat Transfer	7
Single-Physics Interfaces	8
Multiphysics Interfaces	9
Interfacing Add-On Products	10
MODELING EXAMPLES	
Nonisothermal Turbulent Flow over a Flat Plate	10
Nonisothermal Laminar Flow in a Circular Tube	11
Optimization of a Thermo-Optic Silicon Photonic Switch	11
Heat Transfer and Fluid Dynamics of a Flat Heat Pipe	12
The Superconducting Magnets of the Large Hadron Collider	12
Temperature Compatibility of Implantable Medical Devices	13
SIMULATION APPLICATION EXAMPLES	
Teaching Heat Transfer and Fluid Dynamics with Simulation Applications	14
Simulating Custom Capacitors with a Family of Applications	15
An Application for Simulating Heat Transfer in Graphite Foils	16
CLOSING REMARKS	17
ADDITIONAL RESOURCES	18

INTRODUCTION

Simply stated, thermal management factors into the design of all electronic devices. The use of numerical simulation to achieve design goals associated with reliability, energy and cost efficiency, safety, and user experience is increasing with device performance and competition.

The simulation projects presented in this application note span diverse systems that employ electrical currents. In these cases and in most engineering applications, the concurrent investigations of heat transfer mechanisms and factors in a system that cause temperature variations lead to a better understanding of the impact of design changes on a product's performance.

In this application note, the capabilities and features of the COMSOL Multiphysics® software and its add-on products for analyzing heat transfer and multiphysics behavior in devices are described as they are used across the case studies under consideration. The use of physics interfaces,

couplings, and postprocessing functionality to study phenomena such as convective cooling, Joule heating, phase change, and thermal expansion is discussed.

In addition to physics modeling capabilities, the ability to turn multiphysics models into simulation applications, allowing for greater access to numerical simulation, is also highlighted. Simulation applications, such as the one shown in Figure 1, enable simulation specialists to deploy their simulation results to collaborators or customers without compromising their intellectual property or requiring the recipient to be familiar with the COMSOL® software.

The examples found within this application note are for illustrative purposes only. The capabilities of COMSOL Multiphysics are applicable to a seemingly infinite number of applications and design needs and allow any number of physics couplings and behavior types to be studied.

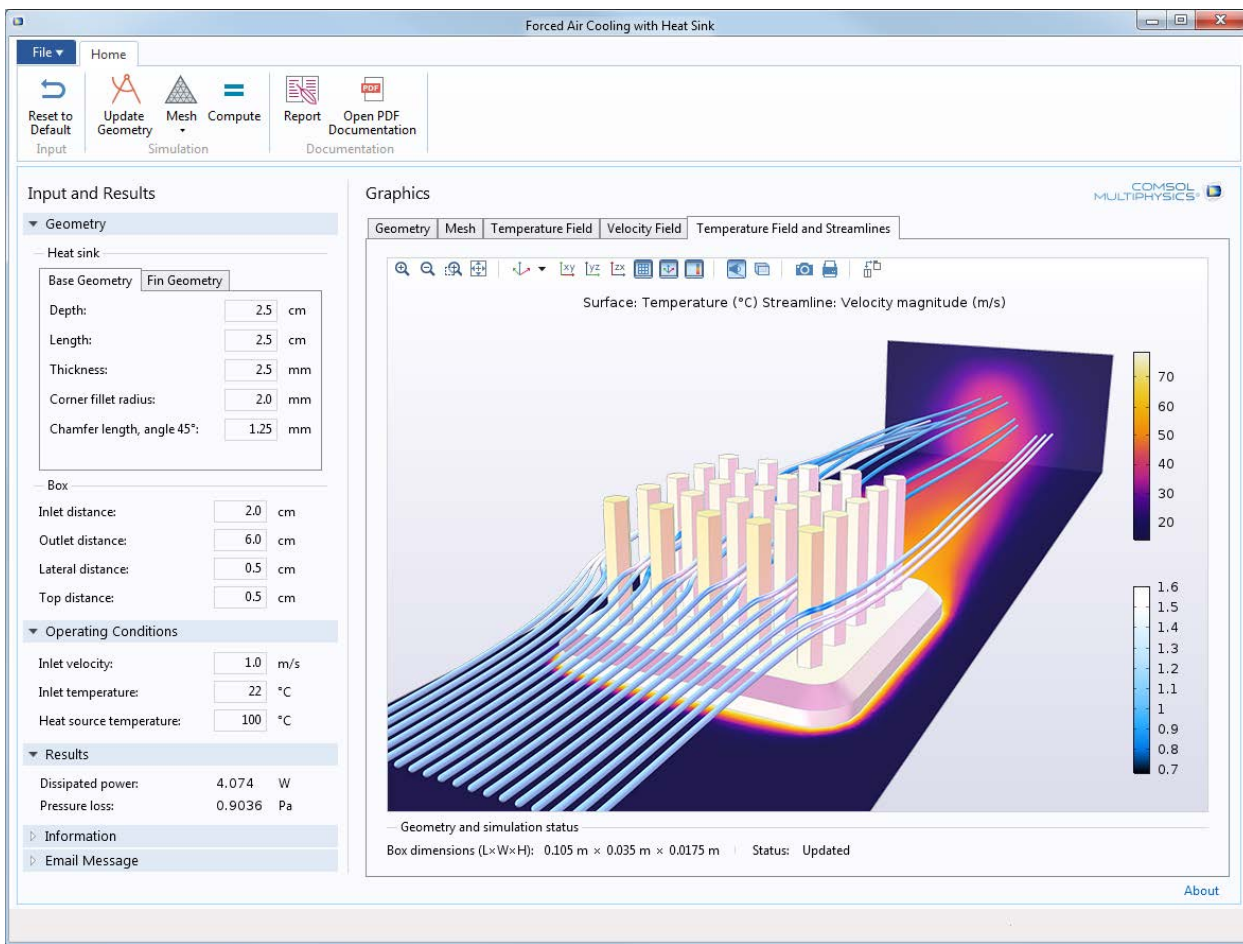


FIGURE 1: This simulation of a heat sink allows for calculation of heat dissipation and pressure loss. In the corresponding simulation application, air velocity and heat source properties can be adjusted, as well as the number and type of pins in the heat sink. For more information about the simulation application, see the Heat Sink model in the Application Gallery available on [comsol.com](https://www.comsol.com).

THERMAL MANAGEMENT IN ELECTROMAGNETIC SYSTEMS AND DEVICES

When it comes to the thermal management of devices, engineering objectives primarily involve efficiency, safety, reliability, user experience, and cost reduction. Furthermore, across the design of all types of electronic devices, recurring themes include considering material properties, maintaining proper current levels, and avoiding thermal fatigue.

Previously, the design of electronic devices depended largely on the experience accumulated through expensive and time-consuming experimental campaigns (see Ref. R1). The work of design engineers is now greatly aided by the use of numerical simulation. COMSOL Multiphysics® supports studies of thermal management in electromagnetic systems and devices by enabling the accurate representation of multiphysics phenomena by properly coupling, for example, fluid flow, heat transfer, and mechanical behavior.

Such simulations for cooling and thermal behavior can be accomplished with the help of several add-on products. The Heat Transfer Module can be used alone or with the AC/DC Module, where the effect of electric current levels can also be evaluated. The Heat Transfer Module can also be combined with the CFD Module or Pipe Flow Module to analyze fluid flow.

For example, to understand and predict quench events, researchers at CERN used numerical simulation to develop a multiphysics, multiscale, and multirate representation of the electrodynamics and thermal evolution of the event (see Ref. R2). In Figure 2, the temperature distribution through the superconducting coils of the magnets of the Large Hadron Collider at CERN is shown.

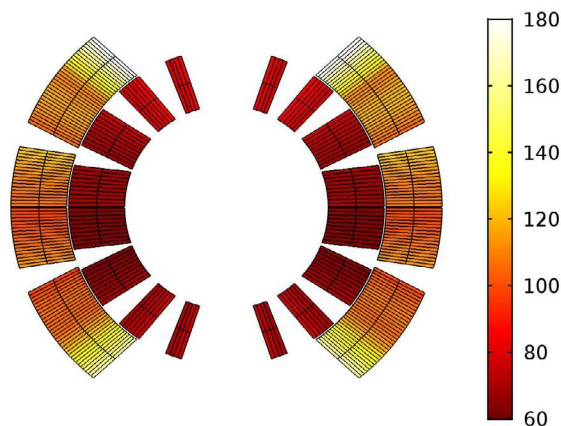


FIGURE 2: Temperature distribution (K) in the superconducting coils of the Large Hadron Collider after a 500-ms quench event.

HEAT TRANSFER APPLICATIONS

Heat transfer often appears together with, or as a result of, other physical phenomena. To accurately simulate such cases, the model needs to account for coupled effects and the thermal dependence of material properties.

For example, structural mechanics simulations where temperature variations induce relevant deformations must include thermal stress. Such deformations can result in the warping or breaking of small, crucial parts such as wires, bonds, and connectors.

There are many applications where thermal loading must be considered. They include a range of part designs, temperature control methods, and manufacturing processes. For example, it is necessary to weigh thermal loading when considering heat dissipation in small consumer products, controlling thermal calibration for sensitive machinery, or increasing the conversion efficiency of a photovoltaic cell.

As one example, engineers at Cornell Dubilier (see Ref. R3) use simulation to test the electrical and thermal properties of custom capacitor designs and select the best design for the application (Figure 3).

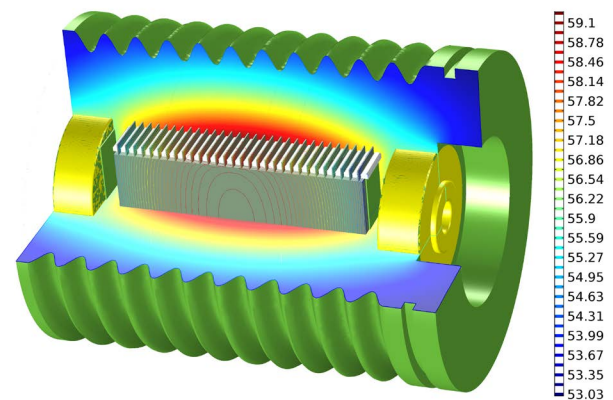


FIGURE 3: Cutaway view of the internal temperature distribution of a high-voltage, corona-free mica capacitor (CDE Type 297) operating at 12 kHz, 20 kVrms.

Considering the geometry and material properties of a design can give engineers insight into heat dissipation rates and the maximum temperature tolerated by individual components. This information allows engineers to decide whether parts can withstand the heat load for their projected lifetime, or if cooling methods must be implemented or adjusted.

Cooling can be achieved using strategies such as adding an airflow over a surface, embedding cooling or heating channels into a surface, or introducing a heat pipe. Heat sinks and fans are frequently used in consumer devices, while heat pumps or thermoelectric coolers are more commonly used in larger electronic and electromechanical systems.

For another example, in the design of a power steering unit, thermal loading in an electrical control unit (ECU) for a wiring harness can have significant impact on the performance of the entire vehicle. Engineers at FZB (see Ref. R4) use multiphysics simulation to characterize heat transfer through the ECU.

Figure 4 shows the temperature distribution throughout an ECU and rotor assembly. Intent on avoiding overheating, engineers at FZB rely on numerical analysis to confirm the feasibility of a design or part choice and to suggest necessary changes. Their model is also used to predict a system's thermal performance when the product is installed on the vehicle.

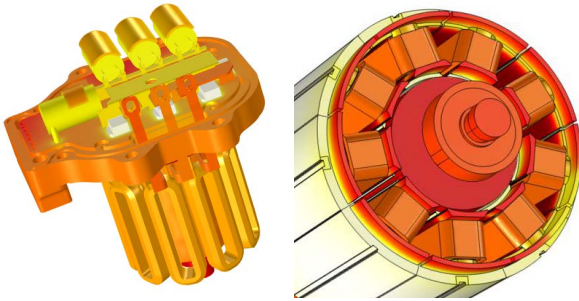


FIGURE 4: Thermal profile of the electrical control unit (ECU) and rotor assembly (left), and stator and rotor assembly (right) for a wiring harness in a power steering unit. Image credit: FZB Technology

THE MECHANISMS OF HEAT TRANSFER

The design of electromagnetic systems and devices often requires consideration of multiple heat transfer processes and how each affects performance. For example, heat generation due to Joule heating can create a rise in temperature that must be monitored and carefully controlled to avoid part failure. This section briefly describes the mechanisms of heat transfer that form the foundation behind modeling and simulation of thermal problems in the COMSOL Multiphysics® software.

Conduction and convection are usually the primary mechanisms that govern heat transfer throughout a process or device. Conduction occurs through solid components such as motors and wires, while convection occurs through fluid channels such as open space for air flow around a heat source. Convection is often forced by way of cooling fans, and parts such as heat sinks take advantage of convection to transfer energy away from temperature-sensitive areas.

The heat flux q' (W), depends on the underlying physical mechanisms that define the mode of transfer, depicted in Figure 5. Rate equations provide the heat flux q'' (W/m^2), the heat transfer rate per unit area in the plane or surface perpendicular to the direction of transfer.

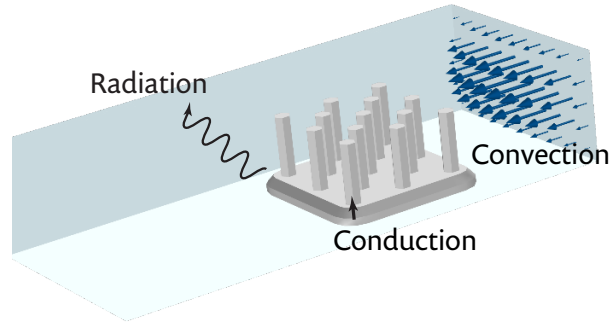


FIGURE 5: Conduction, forced convection, and radiation contribute to heat transfer between a heat sink and the surrounding air.

Conduction results from the transfer of energy due to electron movements and molecular vibration and occurs in a stationary solid or fluid medium when a temperature gradient exists. Fourier's law says that the heat flux is proportional to the temperature gradient, multiplied by the thermal conductivity k ($W/m^2\cdot K$).

$$q'' = -k\nabla T$$

Convective cooling at a surface is induced by fluid motion and results from the transfer of energy by bulk or macroscopic motion of the fluid. Forced or free (natural) convection is considered when the fluid flow is caused by external means or buoyancy forces, respectively. The rate equation known as Newton's law of cooling is used as a boundary condition for convection problems. The parameter h ($W/m^2\cdot K$) is the convection heat transfer coefficient:

$$q'' = -h(T_s - T_\infty)$$

where T_s (K) and T_∞ (K) are the surface and fluid temperatures.

Lastly, radiation can contribute to the heat transfer of a device and occurs between opaque surfaces at different temperatures with or without a participating media. The net rate of radiative heat transfer from a surface is obtained by taking the difference between its emissive power and the irradiation it receives:

$$q'' = -(\epsilon\sigma T_s^4 - \alpha G)$$

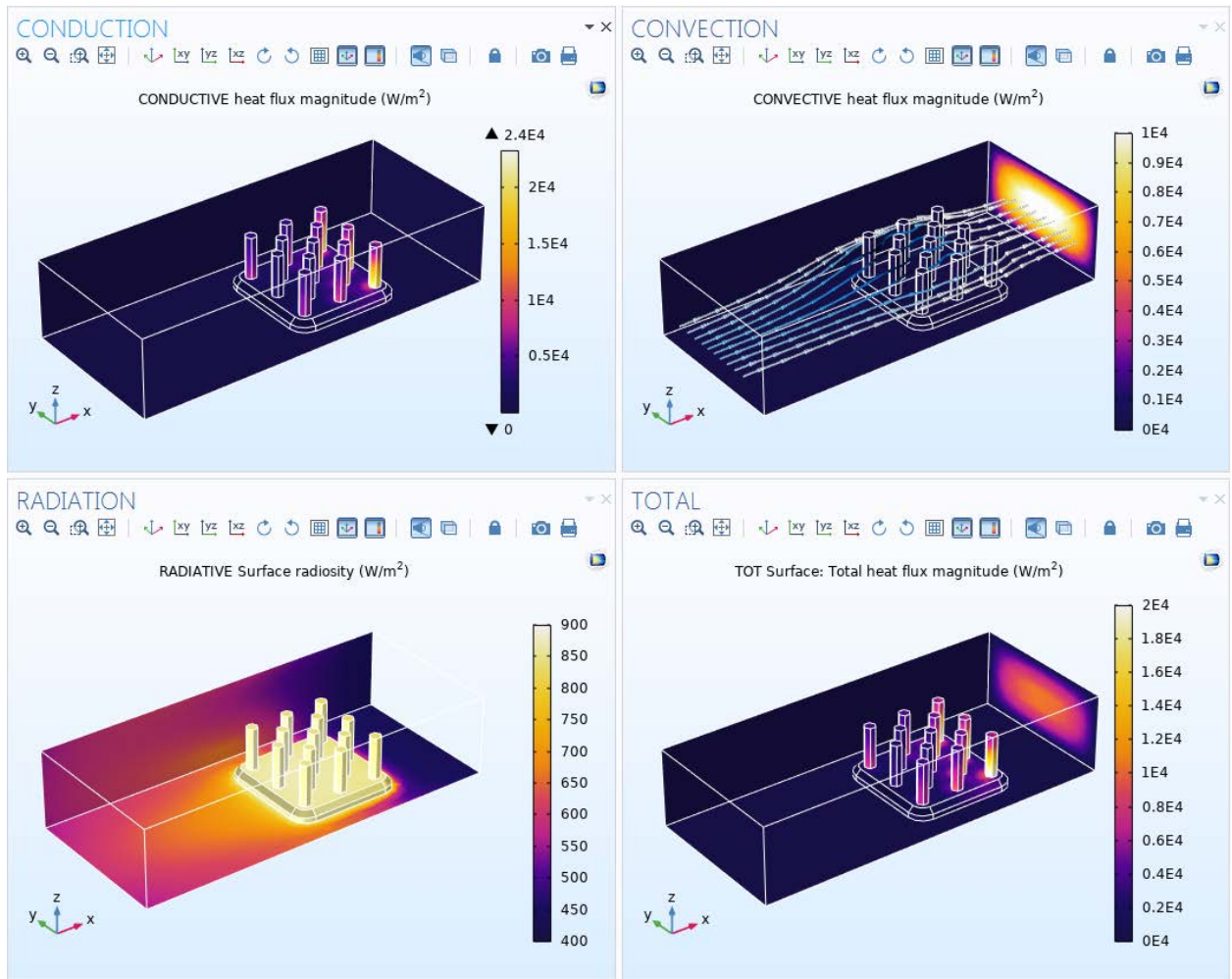


FIGURE 6: Simulation results show the contributions of conductive heat flux (top left), convective heat flux (top right), and surface radiosity (bottom left) to the total heat flux (bottom right) of a heat sink.

where ϵ (dimensionless) is the emissivity, σ ($\text{W}/\text{m}^2\cdot\text{K}^4$) is the Stefan-Boltzmann constant, T_s (K) is the surface temperature, α (dimensionless) is the absorptivity, and G (W/m^2) is the irradiation.

These three modes of heat transfer can coexist and should be considered in most thermal management applications. Figure 6 demonstrates the extent to which each heat transfer mechanism contributes to the total heat flux in a numerical model of a heat sink.

NUMERICAL SIMULATION FOR ELECTRONIC DESIGN

Numerical simulation has become a widely used tool for the design of electromagnetic devices. By developing realistic multiphysics models, the need for expensive and time-consuming physical prototyping can be significantly reduced — which greatly benefits time to market of today's high-tech products.

For example, multiphysics modeling is used in the development of manufacturing processes for high-end battery systems at NEXT ENERGY EWE Research Centre for Energy Technology at the University of Oldenburg, Germany (see Ref. R5).

Lithium-ion batteries, which have a tendency to degrade at high temperatures, must be designed to adhere to stringent regulations guaranteeing their resistance to expected temperature fluctuations.

While it is well known that manufacturing processes such as welding would greatly increase the temperature within a battery, the extent to which such elevated temperatures could propagate within and compromise a cell was not known. Scientists at the center began to investigate whether the expected high welding temperatures could create enough temperature propagation within cells to cause irreversible damage, such as electrolyte decomposition and capacity loss.

First, experimental data was collected from a prismatic

lithium cell when it was subjected to short-term thermal stress, which later would be used to validate the mathematical model. The team then created a 3D replica of a commercial prismatic lithium cell in Autodesk® Inventor® software and imported this into the COMSOL Multiphysics® software.

They modeled heat transfer by conduction due to an external heat source at different positions on the cell corresponding to different manufacturing processes and the natural convective cooling on other areas of the cell surface. Numerical results show the temperature distribution 60-s after a 50-W heat stamp is applied to the housing surface of the lithium-ion cell (Figure 7, top). Temperature propagation after tab welding at 1100°C in the lithium ion cell is also determined (Figure 7, bottom).

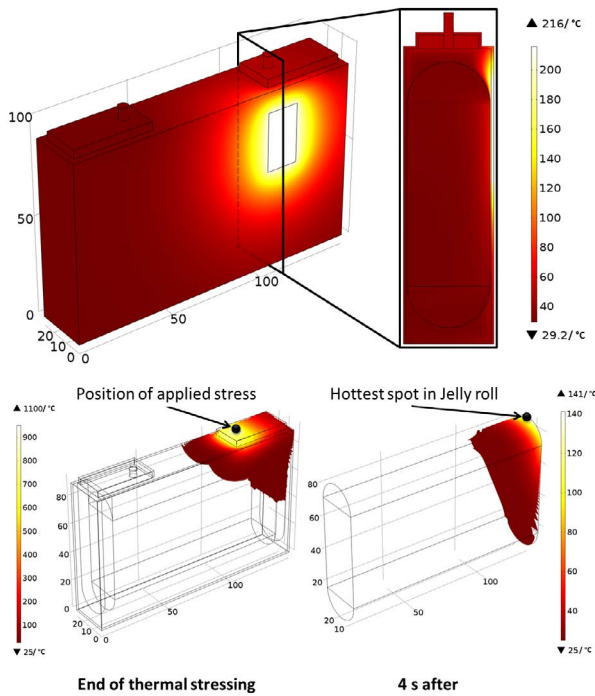


FIGURE 7: Numerical results show the hottest points within the lithium-ion battery cell (top) and the propagation of temperature to the cell interior over time (bottom). Image credit: G. Liebig and L. Komisyska, NEXT ENERGY EWE Research Centre for Energy Technology.

8 shows the list of physics interfaces available when using COMSOL Multiphysics with the Heat Transfer Module and no other add-on products.

Several physics interfaces are available for transient and stationary study types in 1D; 2D; and 3D; as well as axisymmetric components with cylindrical coordinates in 1D and 2D. For example, Figure 9 presents a model of a microthermoelectric cooler (μ TEC) developed at Bell Labs Alcatel-Lucent (see Ref. R6), which employs the Heat Transfer in Solids physics interface as well as the Electric Currents physics interface.

Select Physics

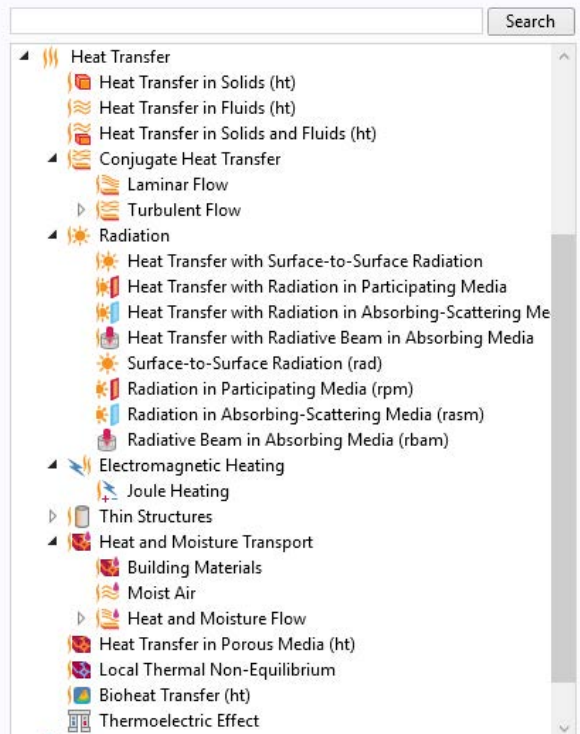


FIGURE 8: Heat transfer interfaces available in the COMSOL® software with the Heat Transfer Module add-on product.

PHYSICS INTERFACES FOR MODELING HEAT TRANSFER

COMSOL Multiphysics® can be used to simulate heat transfer in electromagnetic designs. The Heat Transfer Module allows for modeling all fundamental mechanisms including conductive, convective, and radiative heat transfer. For example, to improve the accuracy of thermal management simulations, a temperature field can be modeled in combination with velocity, pressure, moisture, or electromagnetics. Figure

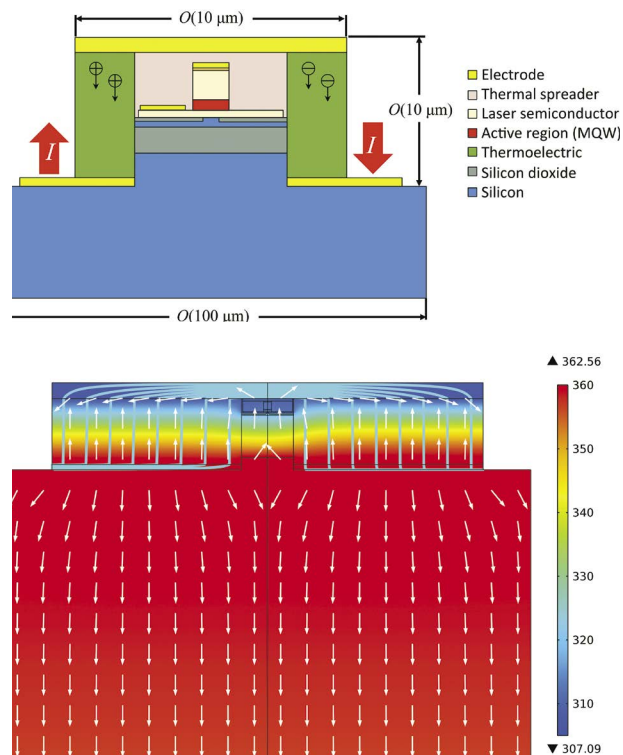


FIGURE 9: Top: Architecture of a telecommunication laser with an integrated μ TEC. Bottom: Simulation showing temperature (surface plot), current density (streamlines), and heat flux (arrows). Image credit: Bell Labs Alcatel-Lucent.

The Material Library in COMSOL Multiphysics® contains material properties and data for 2650 materials, including thermal insulators, semiconductors, and piezoelectric materials. For relevant heat transfer interfaces, standard weather data are also available for the accurate modeling of outdoor conditions, relevant for devices such as transformers, telecommunications cables, and undersea or desert equipment. The data are compiled from more than 8000 stations worldwide and include the annual and monthly variations of the temperature, relative humidity, absolute pressure, and wind velocity.

SINGLE-PHYSICS INTERFACES

HEAT TRANSFER (IN SOLIDS AND FLUIDS)

Accounts for heat transfer in solids and fluids by conduction, convection, and surface-to-ambient radiation.

SURFACE-TO-SURFACE RADIATION

Models surface-to-surface radiation on diffuse surfaces, mixed diffuse-specular surfaces, and semitransparent layers. Accounts for mutual irradiation and external radiation sources, like the sun.

RADIATION IN PARTICIPATING MEDIA

Models radiation in semitransparent media that emits, absorbs, and scatters radiation.

RADIATION IN ABSORBING-SCATTERING MEDIA

Models radiation in nonemitting semitransparent media that absorbs and scatters radiation.

RADIATIVE BEAM IN ABSORBING MEDIA

Models attenuation due to absorption of radiative beams, like lasers in a nonemitting, nonscattering media.

HEAT TRANSFER IN SHELLS, IN FILMS, AND IN FRACTURES

Accounts for heat transfer in solids and fluids by conduction, convection, and surface-to-ambient radiation in thin structures.

BIOHEAT TRANSFER

Provides dedicated user interface to model heat transfer by conduction, convection, and radiation in biological tissue.

MULTIPHYSICS INTERFACES

CONJUGATE HEAT TRANSFER

Combines Heat Transfer and Single-Phase Flow physics interfaces to model heat transfer in solids and fluids for laminar and turbulent flow.

HEAT TRANSFER WITH SURFACE-TO-SURFACE RADIATION

Combines Heat Transfer and Surface-to-Surface Radiation physics interfaces to model heat transfer by conduction, convection, and surface-to-surface radiation.

HEAT TRANSFER WITH RADIATION IN PARTICIPATING MEDIA

Combines Heat Transfer and Radiation in Participating Media physics interfaces to model heat transfer by conduction, convection and radiation in semitransparent media.

HEAT TRANSFER WITH RADIATION IN ABSORBING-SCATTERING MEDIA

Combines Heat Transfer and Radiation in Absorbing-Scattering Media physics interfaces to model heat transfer by conduction, convection, and radiation in semitransparent media.

HEAT TRANSFER WITH RADIATIVE BEAM IN ABSORBING MEDIA

Combines Heat Transfer and Radiation in Absorbing-Scattering Media physics interfaces to model heat transfer by conduction, convection, and radiation in semitransparent media.

JOULE HEATING

Combines Heat Transfer and Electric Currents physics interfaces to model resistive heating, including temperature dependency of material properties. With additional products you can also analyze the effects of induction heating and microwave heating.

BUILDING MATERIALS AND MOIST AIR

Combines Heat Transfer and Moisture Transport physics interfaces to model coupled heat and moisture transport in building material or air, accounting for latent heat effects, heat, and moisture storage, and transport.

HEAT AND MOISTURE FLOW

Combines Single-Phase Flow, Heat Transfer, and Moisture Transport physics interfaces to model flow with coupled heat and moisture transport.

LOCAL THERMAL NONEQUILIBRIUM

Combines two heat transfer physics interfaces to model heat transfer in porous media in cases where the solid and fluid system is not in equilibrium. Accounts for heat transfer at the fluid-solid interface at pores.

THERMOELECTRIC EFFECT

Combines Heat Transfer and Electric Currents physics interfaces to model Peltier-Seebeck-Thomson effects.

Interpretation of the simulation results is necessary in order to understand the underlying physical processes of a system. This is aided by the postprocessing and visualization tools available within the COMSOL® software. The powerful graphics tools, which allow for the visualization of a wide range of phenomena such as temperature distribution, heat flux, structural stresses, fluid flow, or electric current variables, are especially useful for thermal management. For example, in cable design, it is important to account for capacitive, inductive, and thermal effects in the cable parts to keep current buildup and losses in check. The model shown in Figure 10 includes the Heat Transfer in Solids physics interface, along with a Magnetic Fields interface from the AC/DC Module add-on product. The postprocessing capabilities of the COMSOL software are used to illuminate both the thermal profile and magnetic field in a model of a submarine cable. For more details, see the Cable Tutorial Series in the Application Gallery.

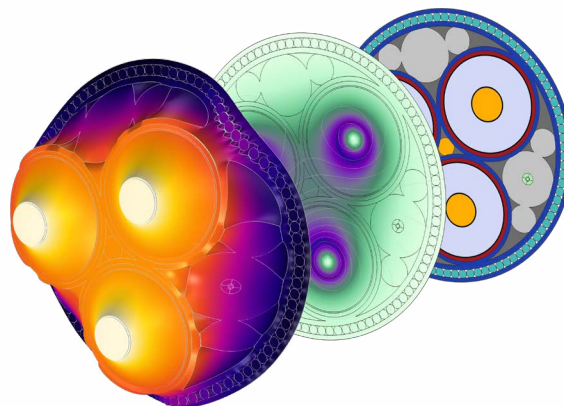


FIGURE 10: A visualization of temperature (left) and magnetic flux density norm (center) in the cross section (right) of an industrial-scale cable.

Plots may be produced over the full simulation time or a specific time frame or step, which provides flexibility, better efficiency, and control over how results are presented. Probes can also be placed in the computational domain to monitor the evolution of some quantity while solving.

The ability to interface with other software provides flexible workflow options for thermal management simulations, which are summarized below. By using LiveLink™ products, for example, it is possible to synchronize a model geometry in another software package with the COMSOL Multiphysics® software, as in the case of the model of the electronic enclosure shown in Figure 11. For more details about the model, see Forced Convection Cooling of an Enclosure with Fan and Grille in the Application Gallery.

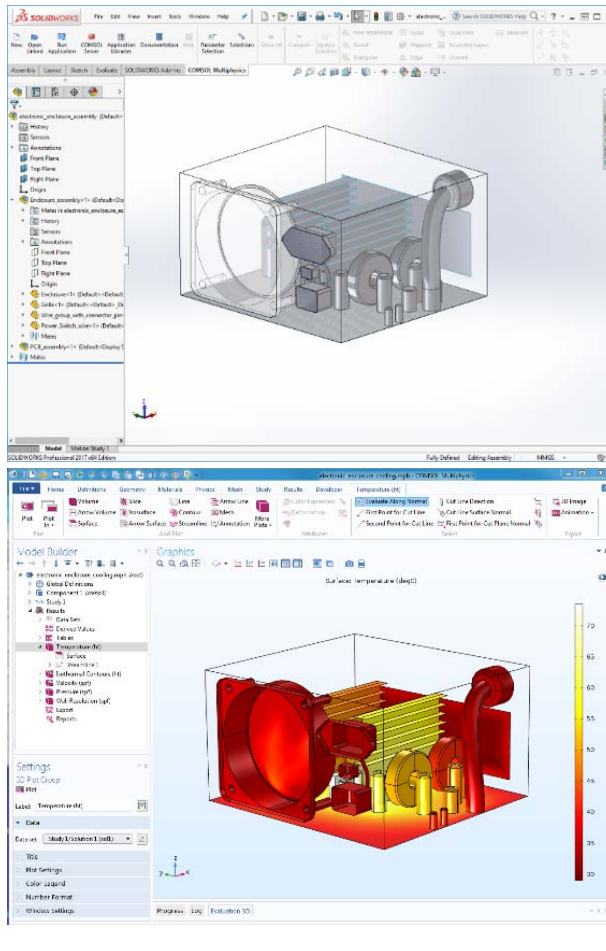


FIGURE 11: Bidirectional synchronization of a CAD design and COMSOL® model of an electronic enclosure by way of the LiveLink™ for SOLIDWORKS® add-on product.

INTERFACING PRODUCTS

CAD IMPORT MODULE

Allows for the import of file formats supported by major CAD packages as well as various native file formats.

ECAD IMPORT MODULE

Import ECAD files, such as GDSII, ODB++, and NETEX-G file formats, into COMSOL Multiphysics®.

LIVELINK™ for MATLAB®

Allows for bidirectional linking between the COMSOL Multiphysics® software models, MATLAB® software, and Simulink® software.

LIVELINK™ PRODUCTS

Allows for bidirectional geometry synchronization between COMSOL Multiphysics® and CAD systems such as SOLIDWORKS®, Creo® Parametric™, and Inventor®.

MODELING EXAMPLES

The COMSOL Application Gallery provides model examples in all engineering fields; among them heat transfer, fluid flow and electromagnetics. These applications and model files include fully documented, step-by-step tutorial and demo applications that serve as a starting point for your own modeling work. The first two examples here cover well-known benchmarks that verify the correctness from simulation in COMSOL Multiphysics.

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

Autodesk, the Autodesk logo, and Inventor are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. MATLAB and Simulink are registered trademarks of The MathWorks, Inc. SOLIDWORKS is a registered trademark of Dassault Systèmes SolidWorks Corp. Creo and Parametric are trademarks or registered trademarks of PTC Inc. or its subsidiaries in the U.S. and in other countries.

NONISOTHERMAL TURBULENT FLOW OVER A FLAT PLATE

The model Nonisothermal Turbulent Flow over a Flat Plate, a verification example provided within the Application Libraries available in the COMSOL Multiphysics® software, is a physical description that can be thought of as a starting point with relevance for many cooling applications. In this model, the heat transfer coefficient obtained from a conjugate heat transfer simulation is verified against reference values based on Nusselt number correlations found in the literature (see Ref. R1) In Figure 12, the comparison between numerical results and correlations shows an especially close match over the plate. For more details, see the Nonisothermal Turbulent Flow over a Flat Plate model in the Application Gallery.

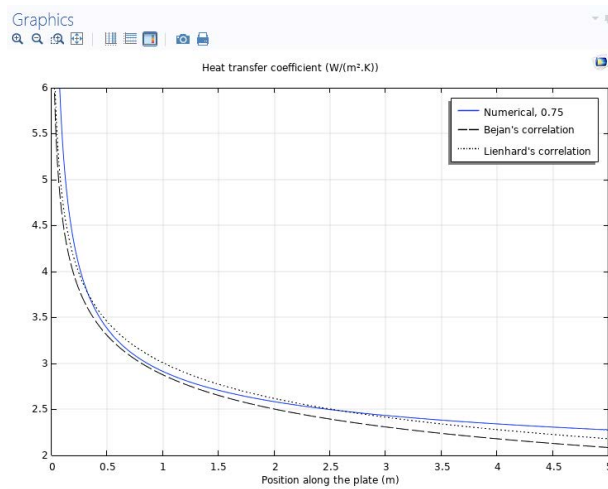


FIGURE 12: A comparison of the computed heat transfer coefficient (solid line) with the heat transfer coefficient estimation based on Nusselt number correlations.

NONISOTHERMAL LAMINAR FLOW IN A CIRCULAR TUBE

Another physical description relevant to many applications is that of nonisothermal laminar flow in a circular tube. Also, this model validates the heat transfer coefficient obtained from the simulation against experimentally determined Nusselt number correlations (see Ref. R1). Simulation results show the velocity field (Figure 13, top left) and the temperature field (Figure 13, top right) through the cylinder. Comparison of the computed heat transfer coefficient with correlations (Figure 13, bottom) is excellent. For more details, see the Nonisothermal Laminar Flow in a Circular Tube model in the Application Gallery.

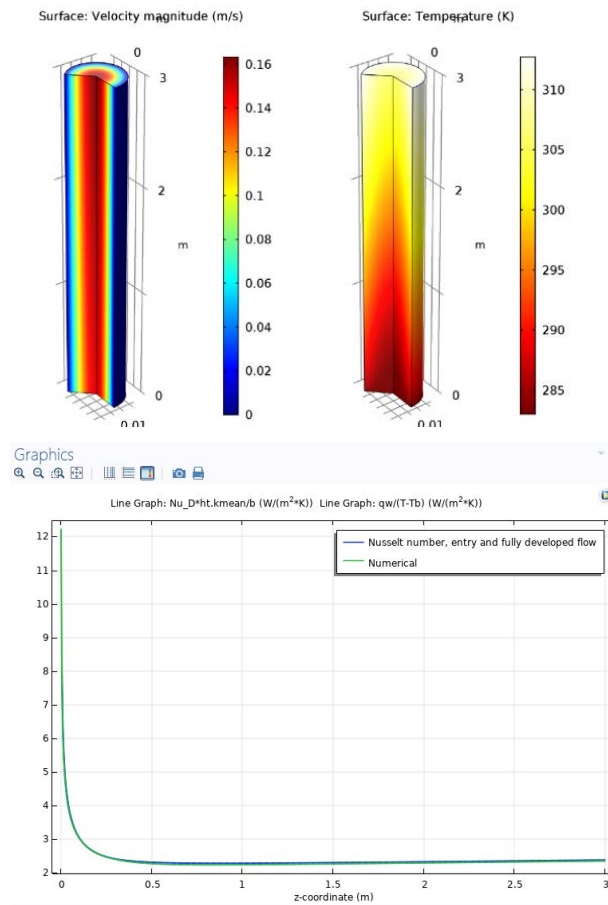


FIGURE 13: Nonisothermal laminar flow in a circular tube. Velocity (top left), temperature (top right), and the heat transfer coefficient comparison (bottom) are shown.

The comparison of the computed heat transfer coefficient with the experimentally determined Nusselt number shows that the local Nusselt number provides excellent approximation over the entire cylinder. Where the Nusselt number variations are small represents the region where velocity and temperature profile are fully developed.

OPTIMIZATION OF A THERMO-OPTIC SILICON PHOTONIC SWITCH

In an electrical switching matrix, an optical signal is converted to an electrical signal for routing, then converted back to optical signals before being sent out in another optical fiber. An all-optical switch, on the other hand, does not require additional electrical-optical conversion stages, making it an attractive choice.

Engineers at Huawei Technologies Canada (see Ref. R7) enlist numerical simulation to optimize thermo-optic silicon photonic switches for the communications and high-performance computing industries. The ongoing research aims to minimize power consumption and maximize switching speed to meet the ever-growing demand for fast transmission of data.

Including a thermal undercut in the design can improve the energy efficiency of a device, but it can also decrease switching speed and add additional steps to the manufacturing process. For this reason, thermal undercut is suited only for certain applications. Engineers at Huawei turned to multiphysics simulation to quantify the performance of various thermo-optic designs, both with and without thermal undercut, as shown in Figure 14.

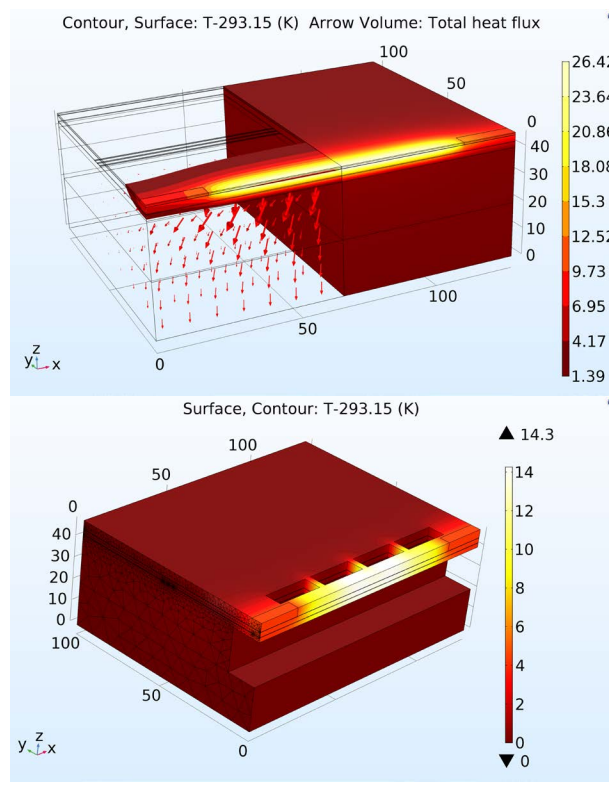


FIGURE 14: Model of a thermo-optic switch without (top) and with (bottom) thermal undercut.

HEAT TRANSFER AND FLUID DYNAMICS OF A FLAT HEAT PIPE

Heat pipes are used in a wide range of applications, from microelectronic systems to large cooling devices in the space industry. They often play a role in thermal management of the system. When in contact with a hot surface, the liquid inside the heat pipe turns into vapor, absorbing latent heat. The vapor then travels along the pipe to the cold side, where it becomes liquid again and releases latent heat. Taking advantage of this phase change results in extremely efficient heat transfer between the cold side and the hot side. A heat pipe design could reach an effective thermal conductivity 250 times greater than copper.

The flat heat pipe model shown in Figure 15 includes a rectangular copper geometry with a vapor chamber and internal wick. Flow inside the vapor chamber is induced by the difference in the saturation pressure between the evaporator and condenser sides, and the resultant mass flow creates a heat source/sink term due to latent heat release.

The model studies nonisothermal laminar flow and includes convective and conductive heat transfer in the vapor chamber, alongside evaporation and condensation. Simulation results include the velocity field inside the vapor chamber and the temperature distribution throughout the heat pipe. For more details, see the Flat Heat Pipe model in the Application Gallery.

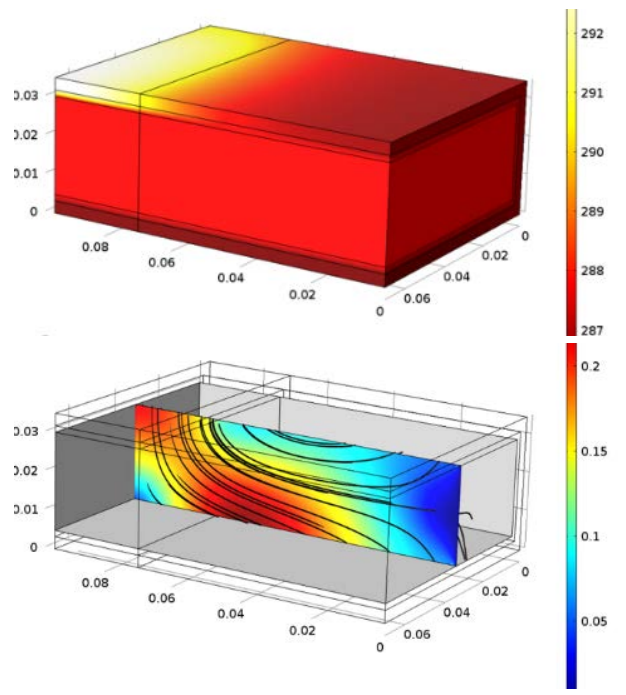


FIGURE 15: Temperature throughout the heat pipe (top) and velocity field in the vapor chamber (bottom).

SUPERCONDUCTING MAGNETS OF THE LARGE HADRON COLLIDER

High-field dipole magnets, operated at currents as high as 12 kA and reaching magnetic fields of 8.33 T, are responsible for keeping particles circling through the Large Hadron Collider (LHC). To maintain the magnet cables, shown in Figure 16, in the superconducting state, the magnets are cooled down to 1.9 K, which is colder than the temperature of outer space. This, in principle, allows the current to continuously circulate in the magnet coils without resistive losses. However, in practice, portions of the magnet coil may revert to a resistive state at times.

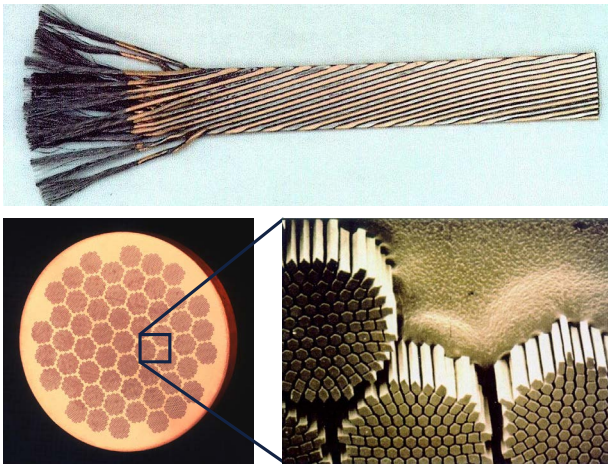


FIGURE 16: High-current superconducting magnets in the LHC are achieved through cables made of superconducting microfilaments embedded in a copper matrix.

To understand and prevent quench events, researchers at CERN (see Ref. R2) used numerical simulation to develop a multiphysics, multiscale, and multirate representation of the electrodynamics and thermal evolution of such an event. Figure 17, left, shows a cross section of the main dipole of the Large Hadron Collider, where red and blue domains represent the superconducting coils keeping the particles in a circular trajectory. The gray domain represents the iron yoke. Figure 17, right, shows the temperature distribution throughout the magnet sections after a 500-millisecond quench event.

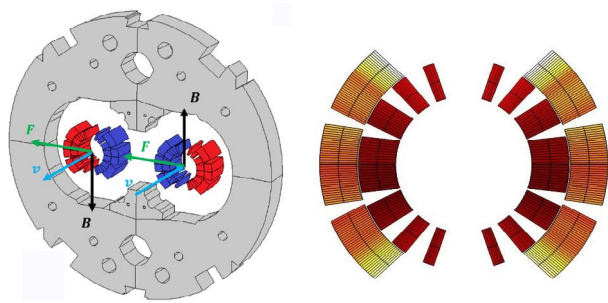


FIGURE 17: A cross section of the main dipole of the LHC (left). Temperature distribution (K) in the coil after a 500-ms quench event (right).

TEMPERATURE COMPATIBILITY OF IMPLANTABLE MEDICAL DEVICES

It is no small task to create a device that takes the place of the human heart and circulates blood throughout the human body — not to mention a device that is biocompatible. At St. Jude Medical (now Abbott), multiphysics simulation is used to model the detail of left ventricular assist devices (LVADs) to improve the outlook and quality of life for patients with heart failure (see Ref. R8).

Freddy Hansen, senior research and development engineer, created simulations of fluid flow, electromagnetics, and heat transfer to improve LVAD biocompatibility, hemocompatibility, and immunocompatibility of the device in the body.

In one simulation, the heat resulting from wireless transfer of energy was modeled and compared to often-cited experimental data for validation (Figure 18).

Along with CFD simulations of bloodflow throughout the pump and simulations that ensure durability of the remote controller, multiphysics simulation has proven to be integral to the development of the LVAD. The latest innovations show much promise for improved quality of life of patients in the future.

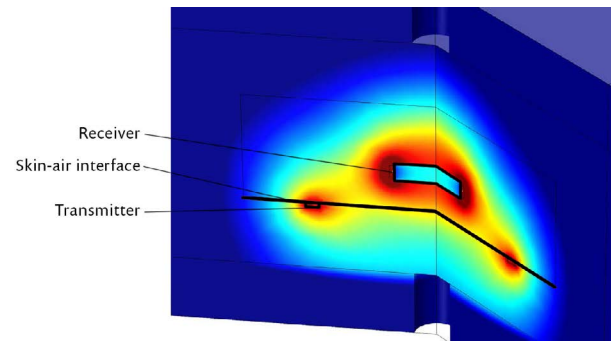


FIGURE 18: Model of heating induced in the body through magnetic power transfer. Results show power density distribution in the tissue and surrounding air.

SIMULATION APPLICATION EXAMPLES

The Application Builder, available in COMSOL Multiphysics® can be used to create a simulation application from a multiphysics model — that is, an interface that hides the complexity of the underlying model while allowing the user to perform the specified simulation.

A simulation application extends the power of a multiphysics model to colleagues, collaborators, or customers. With an application, people other than the simulation expert can contribute and quickly test different cases of a problem without having an intimate knowledge of the simulation.

Simulation applications can also be compiled into standalone executables with COMSOL Compiler™. Such compiled applications can be deployed to anyone, anywhere, without the user needing a COMSOL license to run the application. For deployment throughout organizations, simulation applications can be uploaded to a local installation of COMSOL Server™, which makes it possible to administrate and launch applications through a web interface.

TEACHING HEAT TRANSFER AND FLUID DYNAMICS WITH SIMULATION APPLICATIONS

Innovation can often require breaking the mold, especially in the classroom. At the University of Hartford, Dr. Ivana Milanovic, professor of mechanical engineering, is using numerical simulation and applications to bring inquiry-based learning to her fluid mechanics and heat transfer courses (see Ref. R9).

The applications let students view and control designated inputs, settings, and outputs, and a simplified interface enables them to take advantage of the software's powerful tools without needing to be experts or before becoming experts.

Applications also add value to the University of Hartford's industry-sponsored research and increase students' appeal to potential employers, because many aerospace and manufacturing-related companies in the school's home state of Connecticut use simulation software.

Simulation applications can also help provide a dynamic 3D view of the system when textbooks are limited to immobile, 2D images. A simulation application (Figure 19) created by a student at the University of Hartford gives the thermal and flow profile of a glass of cold water heated to room temperature.

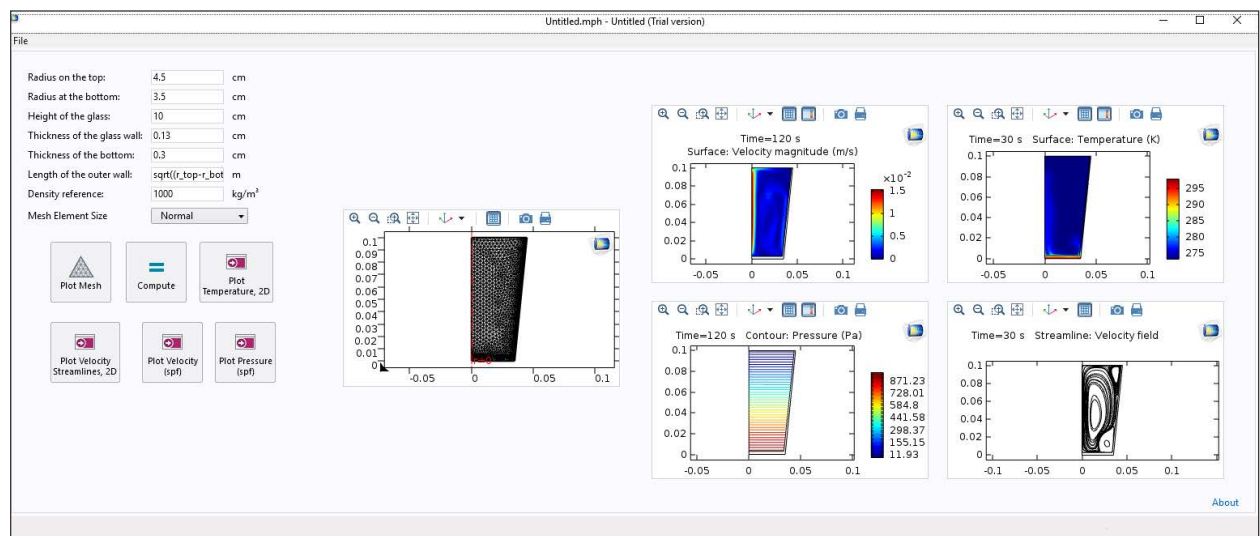


FIGURE 19: In this [simulation application](#) created by a student at the University of Hartford, the user can study nonisothermal flow in a glass of cold water heated to room temperature. This gives outputs for fluid velocity, temperature, and pressure in the glass.

SIMULATING CUSTOM CAPACITORS WITH A FAMILY OF APPLICATIONS

Engineers at Cornell Dubilier (see Ref. R3) used multiphysics simulation to model custom electrolytic capacitors (Figure 20) in order to determine the heat resulting from passing current through the aluminum foils of the windings.



FIGURE 20: Aluminum electrolytic capacitors with windings composed of aluminum foils and cellulosic separators that exhibit thermal anisotropy.

In optimizing heat generation and heat dissipation, the complex materials of the capacitor must be accurately represented. Passing current through the aluminum foils of the windings results in Joule heating. Too much heat dramatically shortens the capacitor lifetime, which is cut in half each time the capacitor's temperature is 6-10 degrees higher than the maximum. One capacitor can easily include as many as six materials, some of which have anisotropic properties.

Sam Parler, research director, created simulations of a metallized polypropylene film capacitor (Figure 21, top) and an aluminum electrolytic capacitor (Figure 21, bottom), both approximately 76x120 mm and dissipating 5 watts in a 45°C environment. In creating a simulation of heat transfer through the capacitors, he could predict device performance before prototyping, enabling a more economical and efficient design process.

Sam Parler found simulation applications to be especially useful in sharing knowledge across the organization. After multiphysics models of his designs, Parler and his team converted the models into simulation applications that are ready to be deployed to design engineers and manufacturing sites to assist in the design process.

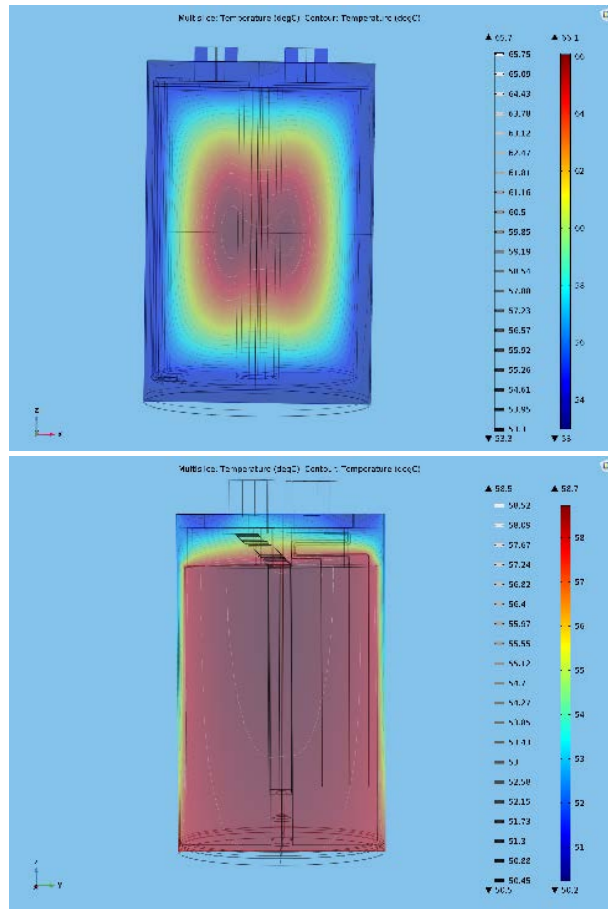


FIGURE 21: A thermal comparison of a metallized polypropylene film capacitor (top) and an aluminum electrolytic capacitor (bottom).

In one simulation application for a power film capacitor, a design engineer can enter the film width, film length, surface resistances, and transition region location into the interface to determine the capacitance and resistance of a segment of the metal film (Figure 22).

In another, the user can calculate power density for the metal film in a cylindrical capacitor. It also predicts the core temperature distribution, including throughout the tabs and terminals, taking into account the customer's operating conditions, such as ripple current, ambient temperature, and air velocity (Figure 23). A third application is used to calculate the effective series inductance of a single-tab film capacitor (Figure 24).

The ability to build multiphysics models and simulation applications has streamlined their capacitor design process and sped up the development of customized capacitors for customers across the world.

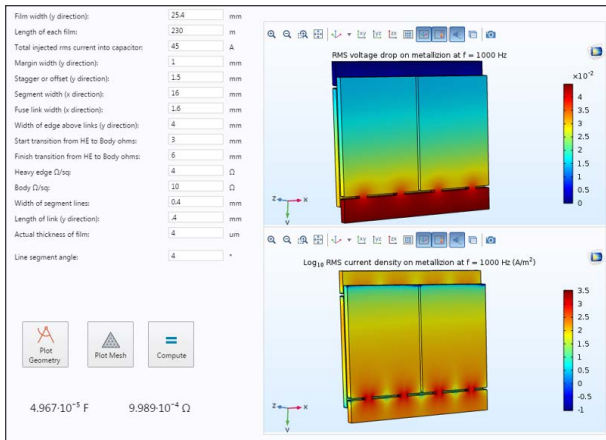


FIGURE 22: A design application for a power film capacitor used to determine capacitance and resistance.

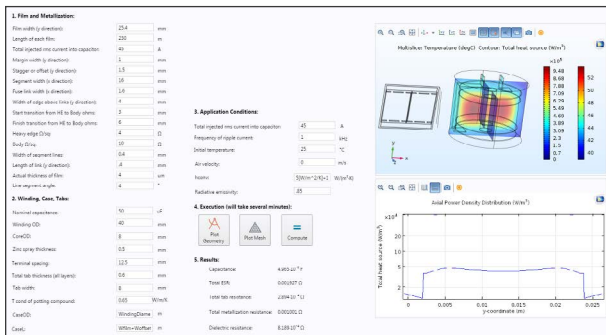


FIGURE 23: A simulation application that predicts the core temperature distribution and the power density for the film of a cylindrical capacitor with tabs and terminals.

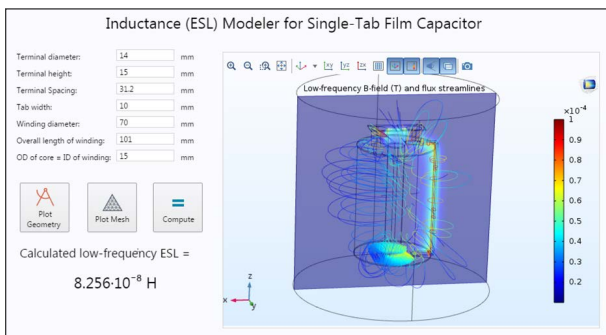


FIGURE 24: An application used to calculate effective series inductance (ESL) of a single-tab film capacitor.

AN APPLICATION FOR SIMULATING HEAT TRANSFER IN GRAPHITE FOILS

GrafTech, a leader in the graphite industry, is using simulation applications to improve the sales process and spread knowledge throughout the organization.

High-fidelity multiphysics simulation with COMSOL Multiphysics® was especially valuable in capturing the proper representation of heat transfer in devices with graphite because of challenges in modeling graphite’s highly orthotropic ratios. Ignoring this level of complexity was simply not an option.

Pierre Hatte, sales director at GrafTech, finds value in using simulation applications because they allow a prospective customer to compare heat transfer throughout different configurations, much like a customer can try on a suit before buying it and be confident that it will fit their needs (see Ref. R10).

One such application, SpreaderCalc (Figure 25), was built by simulation specialists at GrafTech and compares heat transfer among the graphite foils that dissipate heat in consumer electronics. Customers can adjust parameters to see exactly how their adjustments will affect heat transfer throughout their device. Using a local installation of COMSOL Server™, the application is accessed over the company’s intranet through a web browser.

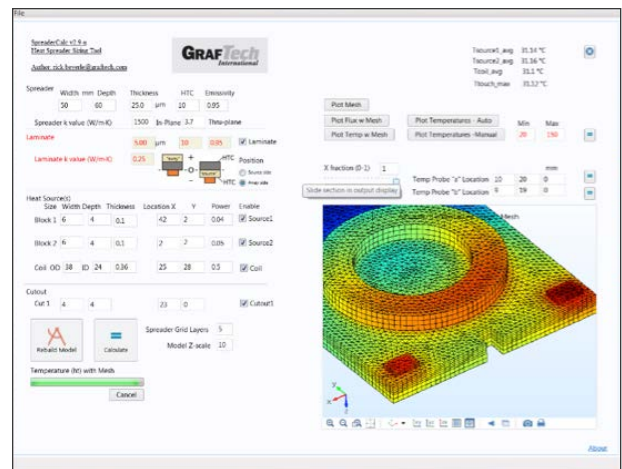


FIGURE 25: The SpreaderCalc application compares heat transfer among the graphite foils that dissipate heat in consumer electronics.

SUMMARY

As we've seen throughout the examples presented in this application note, thermal management factors into the design of all electronic devices. In these cases and in most engineering applications, the concurrent investigations of heat transfer mechanisms and factors in a system that cause temperature variations led to a better understanding of the impact of design changes on a product's performance.

In this application note, we saw how the capabilities and features of the COMSOL Multiphysics® software and its add-on products can be used to analyze heat transfer and multiphysics behavior in devices. Physics interfaces, couplings, and postprocessing functionality are used to study phenomena such as convective cooling, Joule heating, phase change, and thermal expansion.

Multiphysics simulation tools allow for immediate adjustments of a design and creation of simulation applications, streamlining communication between collaborators and increasing the number of people who can meaningfully contribute to a product.

COMSOL Multiphysics supports product innovation in the thermal management of electromagnetic devices through its powerful numerical simulation capabilities and the accessibility of high-fidelity couplings of fluid flow, heat transfer, and mechanical behavior. Add-on modules and interfacing products provide tools, solvers, and application libraries for specific modeling needs.

Further, COMSOL Multiphysics and its Application Builder allow simulation engineers to turn their complex simulation into a presentation-ready and collaboration-friendly application and extend the power of simulation throughout an organization. With COMSOL Compiler™ and COMSOL Server™, simulation applications can be deployed to colleagues and customers worldwide.

REFERENCES

- R1. A. Bejan et al., *Heat Transfer Handbook*, John Wiley & Sons, 2003.
- R2. "Analyzing the LHC Magnet Quenches", CERN, Switzerland, *Multiphysics Simulation* 2017.
- R3. "Accelerating Custom Capacitor Design with Simulation Apps", Cornell Dubilier, South Carolina, *COMSOL News* 2017.
- R4. "Revving up Electrohydraulic Power Steering with Virtual Prototyping", FZB Technology, *COMSOL News* 2017.
- R5. "How Fast Do Elevated Temperatures Reach the Cell Interior?" NEXT ENERGY EWE Research Centre for Energy Technology, Oldenburg, Germany, *COMSOL News* 2017.
- R6. "Meeting High-Speed Communications Energy Demands Through Simulation", Bell Labs, Ireland, *Multiphysics Simulation* 2014.
- R7. "Designing Energy-Efficient Photonic Switches to Support Growing Network Traffic", Huawei Technologies, Canada, *Multiphysics Simulation* 2016.
- R8. "Enhancing Performance and Safety of Medical Implantable Devices with Multiphysics Simulation", St. Jude Medical, *Multiphysics Simulation* 2016.
- R9. "Simulation Apps Bring STEM to Life", University of Hartford, *Multiphysics Simulation* 2017.
- R10. "Industrial Applications of Carbon and Graphite for Thermal Management", Graftech, *Multiphysics Simulation* 2016.

ADDITIONAL RESOURCES

User Story Gallery

Read how simulation experts use the COMSOL® software to resolve design challenges in the thermal management of electromagnetic systems and devices:

- » **Increasing Productivity and Revenue with Computational Applications**
AltaSim Technologies, COMSOL News 2016
- » **Thermal Characterization of an Electronic Device with a Custom App**
BE CAE & Test, COMSOL News 2016
- » **Medtronic Advances Ablation Technology with Multiphysics Simulation**
Medtronic, Multiphysics Simulation 2015
- » **Increasing Lifespans of High-Power Electrical Systems**
ABB, Multiphysics Simulation 2015

Application Gallery

Use these tutorials and demo application files as a starting point for your own modeling work in thermal management of electromagnetic devices:

- » **Heat Sink**
Application ID: 8574
- » **Cable Tutorial Series**
Application ID: 43431
- » **Forced Convection Cooling of an Enclosure with Fan and Grille**
Application ID: 6222
- » **Nonisothermal Flow over a Flat Plate**
Application ID: 56591
- » **Nonisothermal Laminar Flow in a Circular Tube**
Application ID: 55861
- » **Flat Heat Pipe**
Application ID: 43841
- » **Thermoelectric Cooler**
Application ID: 30611
- » **Condensation Detection in an Electronic Device with Transport and Diffusion**
Application ID: 14397
- » **Electronic Chip Cooling**
Application ID: 47721
- » **Convection Cooling of Circuit Boards – 3D Natural Convection**
Application ID: 449

Technical Papers and Presentations

Browse papers and presentations given at COMSOL Conferences to learn more about products and processes designed by your peers:

- » **Heat Transfer and Phase Change**
130+ electromagnetic devices-related resources

COMSOL Blog

Browse our blog to get inspiration and guidance for your own simulation work in thermal management:

- » **Modeling Cables in COMSOL Multiphysics®: 6-Part Tutorial Series**
- » **Predict Thermal Drift in Microwave Filters Using Multiphysics Simulation**
- » **Using Apps to Optimize Induction Heating for Food Processing**
- » **Capturing Eddy Current Losses in a Permanent Magnet Motor Design**
- » **Comparing 2 Approaches for Modeling Electronic Chip Cooling**

Contact Information

For inquiries about how to use the numerical simulation and application design capabilities in the COMSOL® software for your unique modeling scenario, visit comsol.com/contact.

Contact technical support at: comsol.com/support

www.comsol.com

