COMSOL APPLICATION NOTES

SIMULATION-DRIVEN DESIGN *for the Thermal Management of Buildings*

COMSOL

TABLE OF CONTENTS

INTRODUCTION

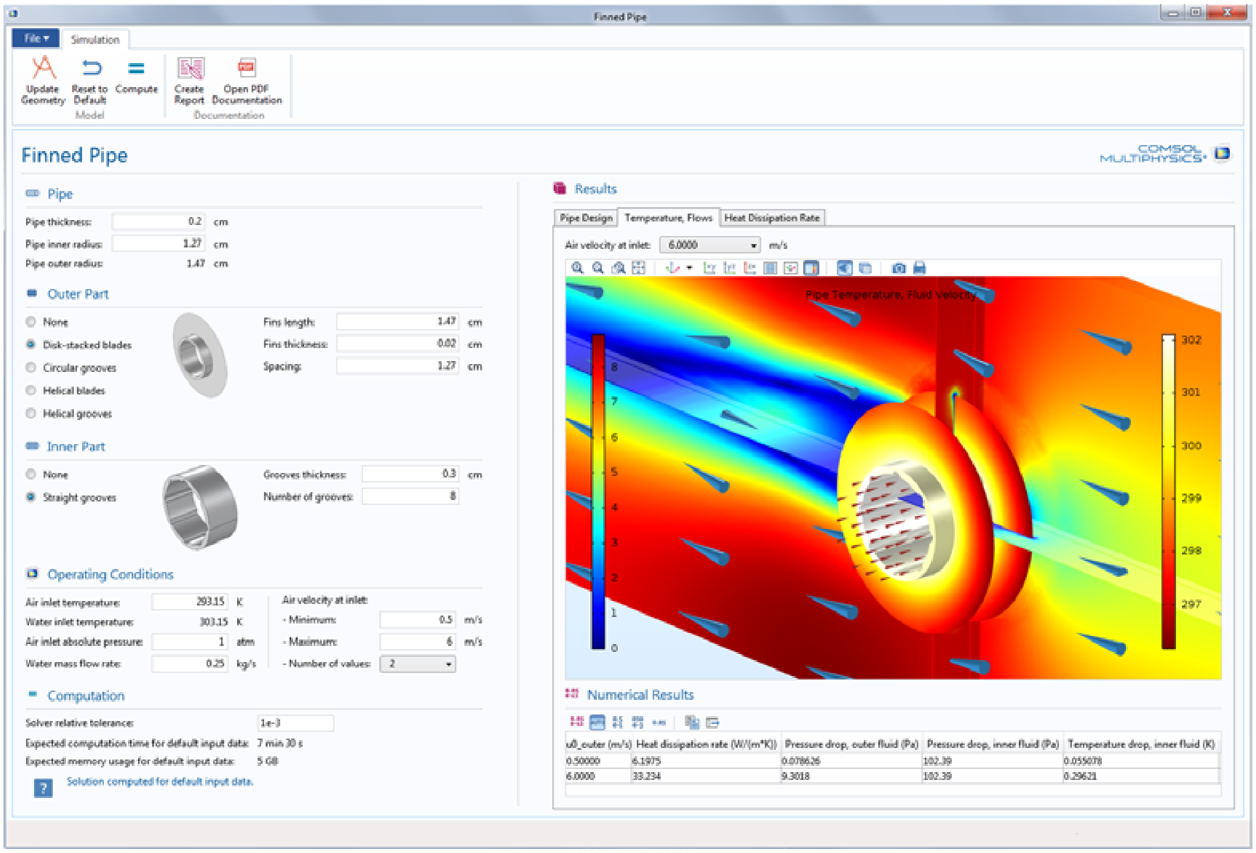
Thermal management (also called temperature management, thermal control, or temperature control depending on the application) is the control of temperature through the use of various monitoring devices and cooling or heating methods. It allows for human comfort and the operability of most systems, ranging from devices used in everyday life to industrial installations. Mathematical modeling and numerical simulation are invaluable in attaining these objectives and critical in meeting time-to-market goals and compliance with all applicable building codes. Simulation results can also inform building information modeling (BIM) processes, increasing their efficiency and quality.

In using simulation for thermal management, investigating heat transfer, while necessary, is not enough to fully understand how changes in a design will affect its overall performance. A simulation must also simultaneously account for the physics that cause the temperature variations and their effects in order to serve as a comprehensive design solution.

This application note presents a selection of thermal management applications relevant to building design, for which COMSOL Multiphysics® software is used to evaluate design concepts.

For the applications presented, the relevant physics and modeling approach are discussed. The examples presented here include phenomena such as thermal expansion, radiation, and convective cooling. The capabilities of the software and its add-on modules can be applied to essentially any application.

These multiphysics modeling and application design capabilities have become essential for optimizing the thermal performance of any device or process. Furthermore, simulation apps based on multiphysics models, such as the Finned Pipe demo app shown in Figure 1, and the COMSOL Server™ product allow simulation specialists to extend their knowledge and the capabilities of the COMSOL® software to colleagues and customers worldwide. Through easy-to-use interfaces, app users can run complex analyses and engineer efficient thermal control in buildings.



Finned Pipe

Pipe

Pipe thickness: 0.2 cm
Pipe inner radius: 1.27 cm
Pipe outer radius: 1.47 cm

Outer Part

○ None
● Disk-stacked blades
○ Circular grooves
○ Helical blades
○ Helical grooves

Fins length: 1.47 cm
Fins thickness: 0.02 cm
Spacing: 1.27 cm

Inner Part

○ None
● Straight grooves

Grooves thickness: 0.3 cm
Number of grooves: 8

Operating Conditions

Air inlet temperature: 293.15 K
Water inlet temperature: 303.15 K
Air inlet absolute pressure: 1 atm
Water mass flow rate: 0.25 kg/s

Air velocity at inlet:
- Minimum: 0.5 m/s
- Maximum: 6 m/s
- Number of values: 2

Computation

Solver relative tolerance: 1e-3
Expected computation time for default input data: 7 min 30 s
Expected memory usage for default input data: 5 GB
Solution computed for default input data.

Air velocity at inlet: 6.0000 m/s

Numerical Results

u0_outer (m/s)	Heat dissipation rate (W/(m*K))	Pressure drop, outer fluid (Pa)	Pressure drop, inner fluid (Pa)	Temperature drop, inner fluid (K)
0.50000	6.1975	0.078626	102.39	0.055078
6.0000	33.234	9.3018	102.39	0.29621

FIGURE 1: Simulation app based on a model of a finned pipe. For more information about the app, see Finned Pipe model in the Application Gallery available on comsol.com.

THERMAL MANAGEMENT OF BUILDINGS

The engineering design objective of thermal management factors into essentially any application where efficiency, cost, safety, and reliability are relevant to improving usability, longevity, and the overall human experience. Although approximate control of the temperature differential is sufficient for some designs, more precise thermal management is needed in others depending on the intended use for the space.

Openings in the building envelope are an important thermal management design consideration. In ongoing work by Schellen et. al. (ref. R1), numerical simulation is used to investigate an approach to window glazing that can help preserve the authentic character of a historic building. Through coupled heat, air, and moisture (HAM) modeling, the performance of a double glazing with a ventilated cavity was determined (Figure 2). In historic buildings, replacing older, single-pane windows (Figure 2a) with typical double-pane windows with plastic frames (Figure 2b) ruins authenticity. By investigating the glazing schemes (Figure 2c) using numerical simulation, the appearance may be preserved through using alternative replacement panes. Simulation results for the inside ventilation glazing scheme (Figure 2d) present relative humidity for an outdoor ventilated window section, which corresponds to the risk of condensation forming.

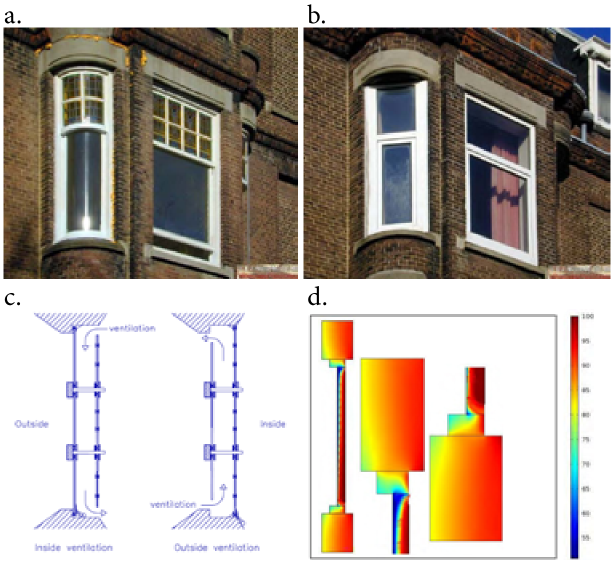


FIGURE 2: Numerical simulation of the performance of different glazing schemes in a historic building. Photo credits: E. J. Nusselder (a) and (b), Schellen et al. (c) and (d).

ROLE OF HEAT TRANSFER IN BUILDING DESIGN

Heat transfer is involved in almost every physical process, and its study is therefore of vital importance for many aspects of building design.

Heat transfer can be a key design consideration in the design of floors. For example, thermal design is indispensable in the design of thermally active flooring, where heat transfer occurs between fluid in the tubing, a thermally active slab, the floor finishes, and finally into the enclosed space. A schematic (Figure 3a) and residential application (Figure 3b) of radiant flooring. The underlying heat pipes can be seen using infrared imaging (Figure 3c).

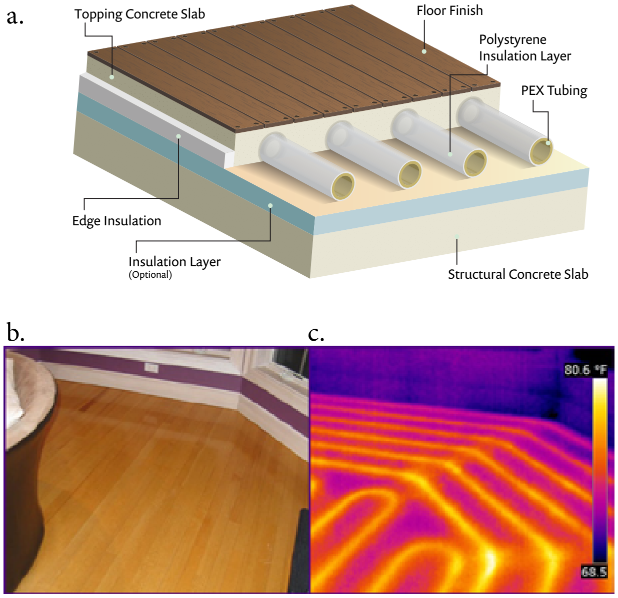


FIGURE 3: A schematic of a radiant flooring design (a). Photograph of a wood flooring (b) and infrared imaging of the underlying heat pipes (c) are shown. Image credit (b and c): Massachusetts Home Inspections.

Heat transfer often appears together with, or as a result of, other physical phenomena. The thermal dependence of material properties must be considered for an accurate description of most processes. For example, thermal stress must be included in structural mechanics simulations where temperature variations induce relevant deformations.

To assess the need for or performance of thermal insulation, the temperature extremes are usually known and the heat transfer rate, also referred to as heat loss, needs to be determined. In the thermal design of buildings and other enclosures, the main objective is therefore to minimize the heat transfer rate by choosing, for example, materials with low thermal conductivity. By considering the geometrical configuration of a design, insulation may be placed strategically in order to avoid the formation of thermal bridges or areas of relatively high rates of heat transfer.

In addition to insulation, heat exchangers can also be used to provide a given heat transfer rate between two domains. Heat transfer enhancement is obtained by optimizing the flow patterns of the fluids and the exchange properties of the solid surfaces.

In many applications, the goal is to prevent the temperature of one domain from going above or below a critical value. Such control may be necessary, for example, to avoid the melting of a hot surface or the cracking of a building's foundation due to thermal cycling.

Often, the design objective of maintaining a specific temperature range can be achieved using methods such as injecting a cooling airflow over a surface, adding thermal barrier coatings, or embedding cooling/heating passages into a surface.

To optimize process efficiency and explore new designs, numerical simulation and computational applications have earned an increasingly important role in the thermal management of buildings, reducing the time, risk, and cost associated with experimental trials.

Building specialists at Vahanen Group in Finland (ref. R2), for example, are using multiphysics simulation to determine if building renovations, such as removing a damaged heat pipe or replacing existing insulation with expanded polystyrene, would be enough to protect the foundation from damage. Figure 4a shows the geometry of the building used by Vahanen Group to develop a model to assess the risk of damage to the foundation. The temperature distribution in the unrenovated building for extreme winter conditions endured over 50 years is shown in Figure 4b.

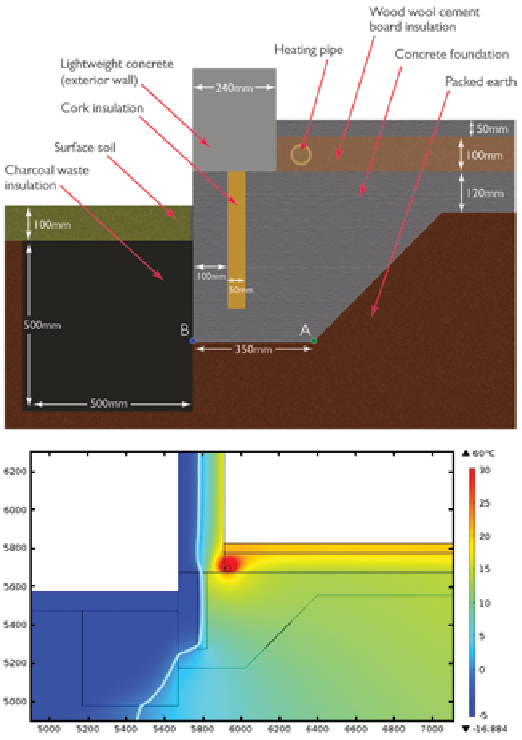


FIGURE 4: Geometry of the foundation (a) and temperature distribution (b) for extreme weather conditions.

THE MECHANISMS OF HEAT TRANSFER

Thermal control requires understanding the different heat transfer processes as they pertain to the design of buildings. Conduction and convection occur through opaque material elements such as roofs and walls in addition to transparent elements, such as windows and doors. Solar radiation can heat up a space through transparent windows, while convection occurs in indoor air through ventilation. Heat generation by occupants and their activities must also be considered. This section covers some of the definitions and mechanisms of heat transfer that form the foundation for modeling and simulation of thermal management.

The amount of energy transferred per unit time, or the heat transfer rate q' (W), depends on the underlying physical mechanisms that define the mode of transfer, depicted in Figure 6. Rate equations provide the heat flux q" (W/m^2), which is the heat transfer rate per unit area in the plane or surface perpendicular to the direction of transfer.

Conduction is the physical mechanism resulting from the transfer of energy due to molecular vibration only, and occurs in any medium when a temperature gradient exists. The rate equation is known as Fourier's law:

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

This law says that the heat flux is proportional to the temperature gradient, multiplied by the thermal conductivity k (W/m$^2\cdot$ K).

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

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

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

Finally, all objects emit energy in the form of electromagnetic waves. Radiation occurs between opaque surfaces at different temperatures in the absence of a participating media. The net rate of radiative heat transfer from a surface is the difference between its emissive power and the irradiation it receives:

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

where ϵ (dimensionless) is the emissivity, σ (W/m^2 · 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 7 (ref. R3) demonstrates the extent to which each heat transfer mechanism contributes to the total heat flux in a numerical model of an attic with an HVAC duct running through the center.

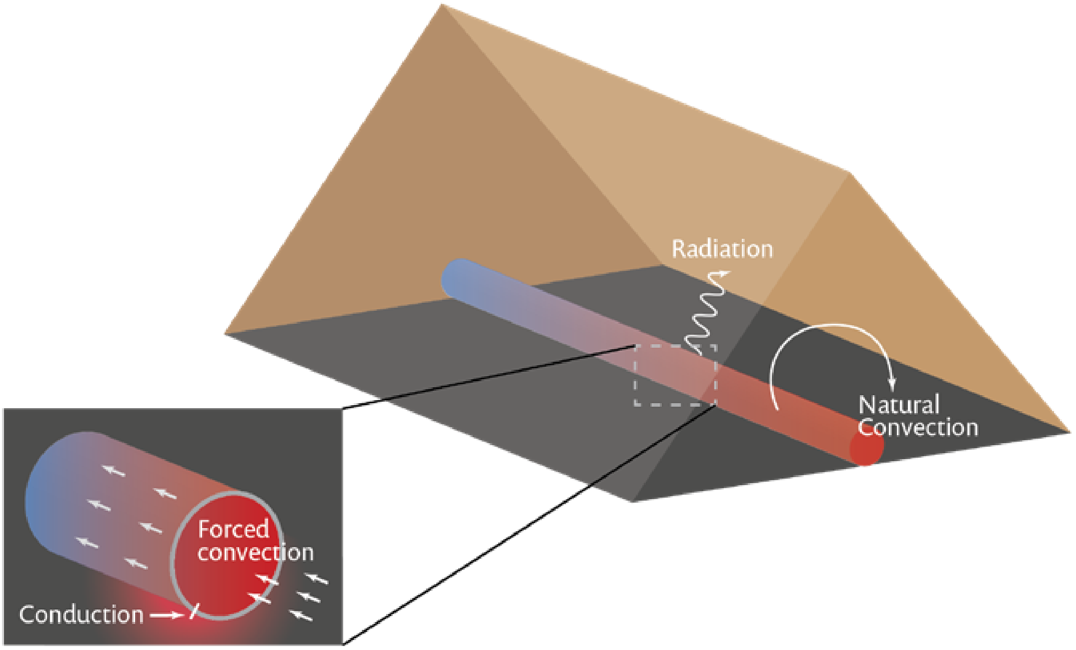


FIGURE 6: Conduction, convection, and radiation contribute to heat transfer between an HVAC duct in an attic and the surrounding air and enclosure.

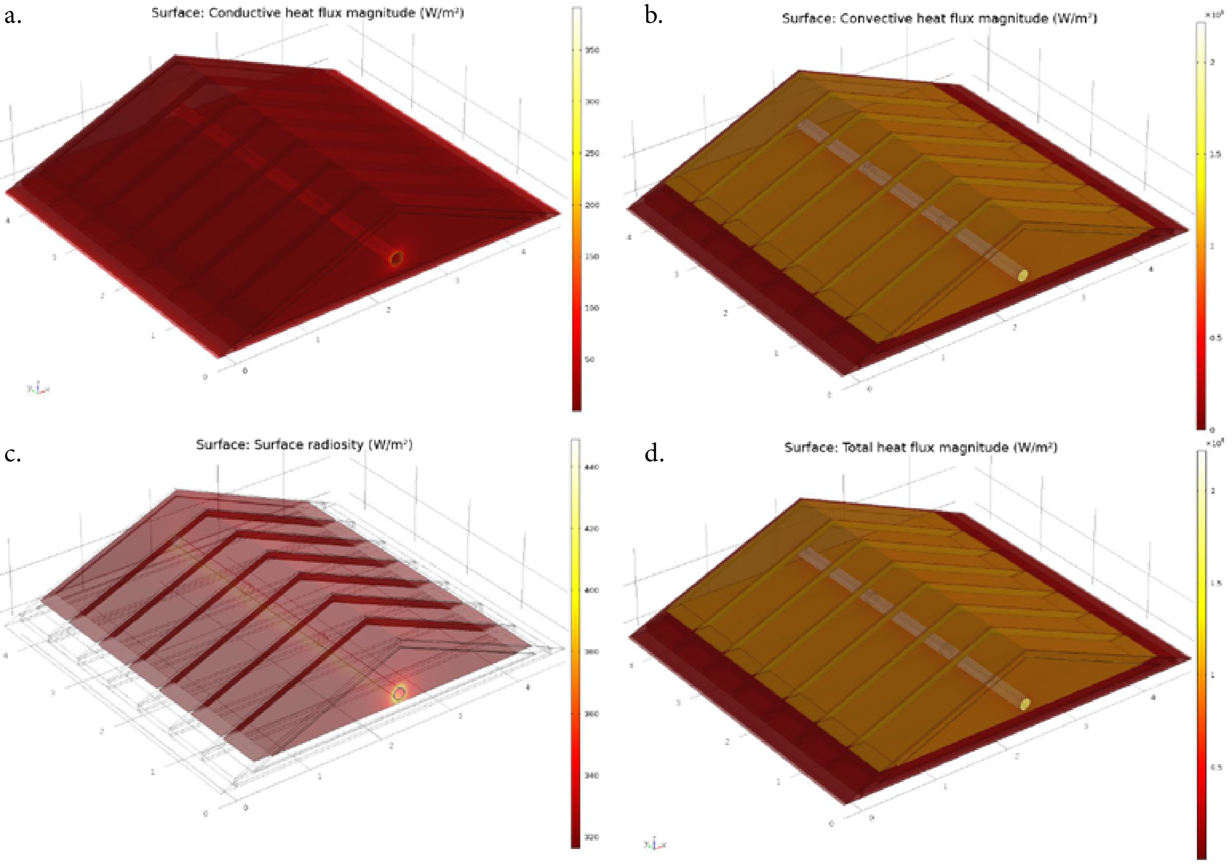


FIGURE 7: Model of an attic with an HVAC duct. Results show the contribution of conductive heat flux (a), convective heat flux (b), and surface radiosity (c) to the total heat flux (d). Image credits: Liu, et.al, Fraunhofer Center for Sustainable Energy Systems.

NUMERICAL SIMULATION FOR BUILDING AND HVAC DESIGN

The design of building structures used to depend largely on the experience and intuition of practiced engineers. The work of building and HVAC design engineers is now greatly aided by numerical simulation. For these applications, simulation software must be able to accurately model heat transfer in complicated structures made of several thermal bridges and insulation in which conduction, convection, and radiation may occur.

For example, an integrated approach to numerical simulation was used by researchers at the University of Florence (ref. R4) to evaluate both thermal and acoustic performance of a thermal bridge connecting a concrete beam and a balcony for varying insulation methods (Figure 8a). The total heat flux (Figure 8b) shows a steel reinforcement rod producing a higher thermal bridge effect. The acoustic analysis (Figure 8c) revealed poorer performance due to the steel rod and thermal insulation method compared to other test cases.

In numerical models of building thermal performance, heat transfer is often coupled to airflow and moisture transport, with airflow promoting heat and moisture transfer. In addition to moisture damage and the health risks due to mold growth, increased moisture content can result in additional heat loss. Evaporation and condensation affect the heat balance significantly, due to the large amount of energy stored as latent heat. As such, relevant models must be able to represent multiphysics phenomena by properly coupling fluid flow, heat transfer, and mass transport.

COMSOL Multiphysics® can be used to simulate building and HVAC designs through several add-on products. The Heat Transfer Module, for example, may be used for the modeling of heat and moisture transport. The CFD Module and Pipe Flow Module may be combined with the Heat Transfer Module for the analysis of porous media flow. By using the Acoustics Module, sound propagation through a structure or enclosure, including pipes or ducts, can also be evaluated. Adding the Structural Mechanics Module enables the evaluation of thermal stress in building structures and materials.

A wide range of options are available for using numerical simulation to better understand and optimize the thermal performance of building and HVAC designs, beyond the selection of examples presented in this section. All modules included in the COMSOL® product suite may be coupled together.

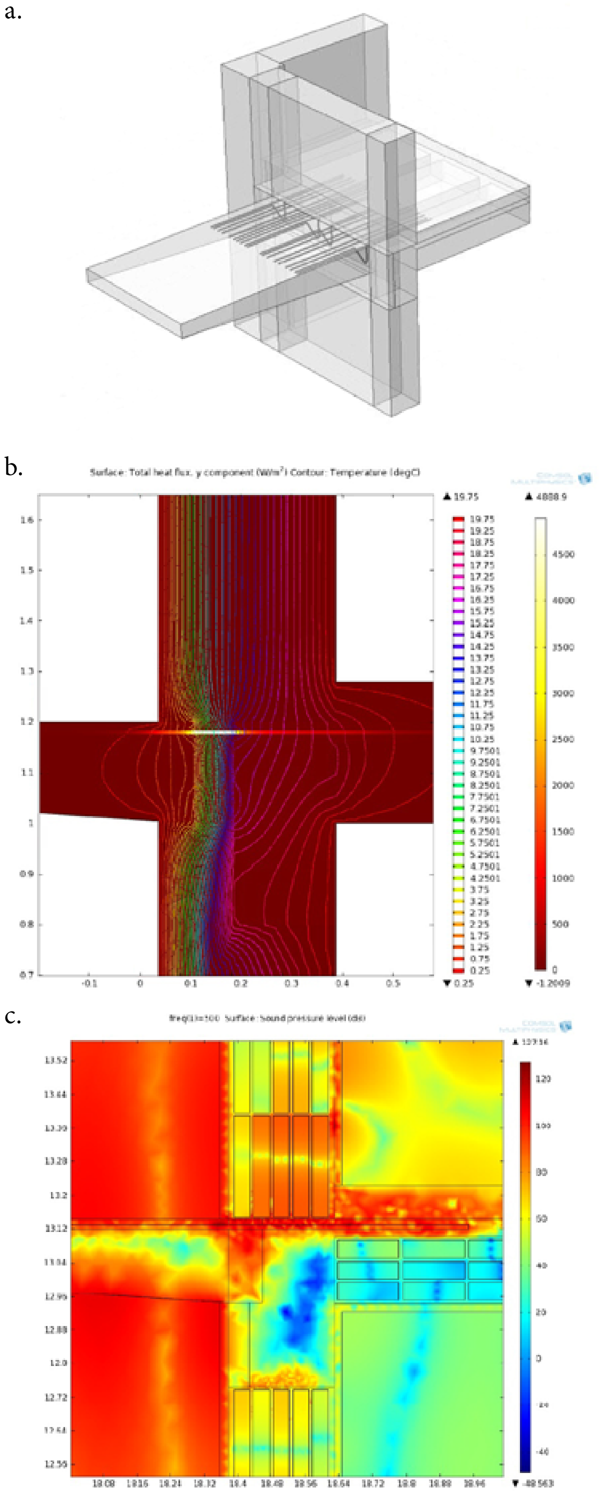


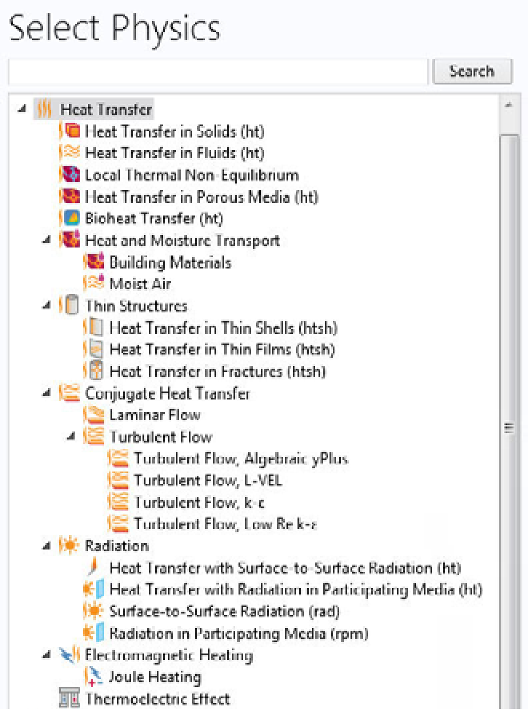
FIGURE 8: Heat transfer and sound propagation were evaluated for the thermal bridge geometry (a). The total heat flux (b) and sound pressure level (c) are shown. Image credits: C. Balocco and E. Marmonti, University of Florence.

ANALYZING HEAT TRANSFER IN THE BUILT ENVIRONMENT WITH COMSOL MULTIPHYSICS®

The Heat Transfer Module is an add-on product to COMSOL Multiphysics® software that extends the capabilities for modeling conductive, convective, and radiative heat transfer in combination with other physics. A temperature field can be modeled together with velocity, pressure, moisture, or radiation; for example, to improve the accuracy of thermal management simulations. Figure 9 shows the list of physics interfaces available when using the Heat Transfer Module and no further add-on products. Interfaces with predefined single and coupled physics make it straightforward to model many phenomena.

Many physics interfaces are available for transient and stationary study types in 1D, 2D, and 3D in addition to axisymmetric components with cylindrical coordinates in 1D and 2D. A brief description of each physics interface is provided at right, and more detail is available in the software. Figure 10 presents a model of a wood frame wall, demonstrating the use of several of the interfaces shown in the table. This 2D stationary model computes condensation risk within a wall composed of multiple materials.

For relevant heat transfer interfaces, standard weather data is available for the accurate modeling of outdoor conditions. The data is compiled from more than 6000 stations worldwide, and includes the annual, monthly, and hourly averaged variations of the temperature, relative humidity, absolute pressure, and wind velocity.



- Heat Transfer
  - Heat Transfer in Solids (ht)
  - Heat Transfer in Fluids (ht)
  - Local Thermal Non-Equilibrium
  - Heat Transfer in Porous Media (ht)
  - Bioheat Transfer (ht)
  - Heat and Moisture Transport
    - Building Materials
    - Moist Air
  - Thin Structures
    - Heat Transfer in Thin Shells (htsh)
    - Heat Transfer in Thin Films (htsh)
    - Heat Transfer in Fractures (htsh)
  - Conjugate Heat Transfer
    - Laminar Flow
    - Turbulent Flow
      - Turbulent Flow, Algebraic yPlus
      - Turbulent Flow, L-VEL
      - Turbulent Flow, k-ε
      - Turbulent Flow, Low Re k-ε
  - Radiation
    - Heat Transfer with Surface-to-Surface Radiation (ht)
    - Heat Transfer with Radiation in Participating Media (ht)
    - Surface-to-Surface Radiation (rad)
    - Radiation in Participating Media (rpm)
  - Electromagnetic Heating
    - Joule Heating
  - Thermoelectric Effect

FIGURE 9: Heat transfer physics interfaces available in COMSOL Multiphysics®.

MULTIPHYSICS INTERFACES

CONJUGATE HEAT TRANSFER

Combines all features from the Heat Transfer and Single-Phase Flow physics interfaces to describe heat transfer in solids and fluids, as well as nonisothermal flow in fluids. These interfaces are available for laminar and turbulent flow.

HEAT AND MOISTURE TRANSPORT

Combines heat and moisture transport in building materials and moist air by taking into account heat and moisture storage, latent heat effects, and liquid and convective transport of moisture.

JOULE HEATING

Combines the Electric Currents physics interface with any of the Heat Transfer physics interfaces to provide the capabilities for modeling Joule heating.

LOCAL THERMAL NONEQUILIBRIUM

Implements a macroscale model designed to simulate heat transfer in porous media, where the temperatures into the porous matrix and the surrounding or infiltrating fluid are not in equilibrium.

THERMOELECTRIC EFFECT

Implements the Peltier-Seebeck-Thomson effects (the direct conversion of temperature differences to electric voltage or vice versa).

SINGLE PHYSICS INTERFACES

HEAT TRANSFER IN SOLIDS

Describes heat transfer by conduction. It can also account for heat flux due to translation in solids and solid deformation.

HEAT TRANSFER IN FLUIDS

Accounts for conduction and convection in gases and liquids, as well as for viscous dissipation and work done by pressure changes.

HEAT TRANSFER IN POROUS MEDIA

Combines conduction in a porous matrix and in the fluid contained in the pore structure with the convection of heat by the flow of the fluid.

HEAT TRANSFER IN FRACTURES

Models heat transfer by conduction and convection in thin porous structures when temperature differences across the thickness of the material are negligible. Surface-to-surface radiation is also modeled.

HEAT TRANSFER IN THIN SHELLS

Describes heat transfer by conduction in thin solid structures. Surface-to-surface radiation is also modeled.

HEAT TRANSFER IN THIN FILMS

Describes heat transfer by conduction and convection in thin fluid structures.

HEAT TRANSFER WITH SURFACE-TO-SURFACE RADIATION

Surface-to-surface radiation is combined with heat transfer in fluids or solids as well as porous media, including conduction and convection.

RADIATION IN PARTICIPATING MEDIA

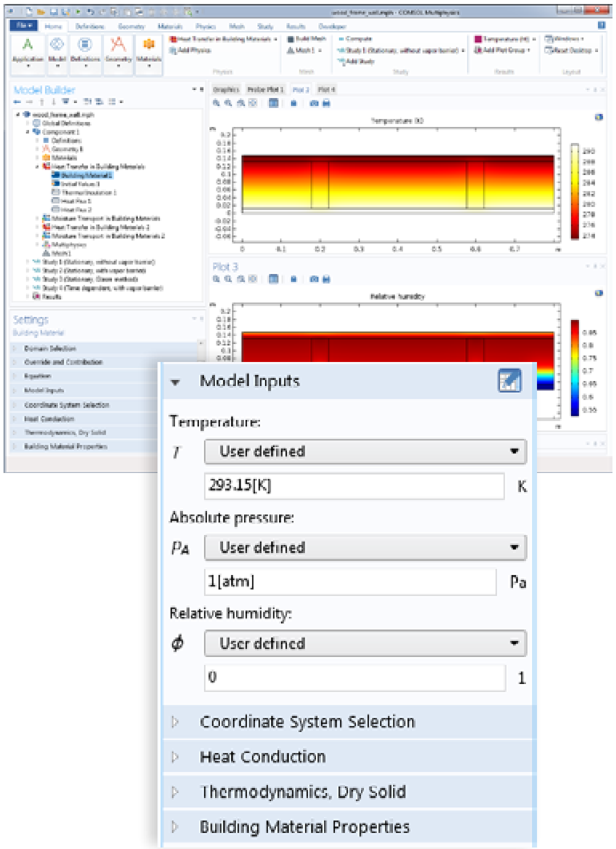
Models radiative heat transfer inside participating or semitransparent media.

HEAT TRANSFER WITH RADIATION IN PARTICIPATING MEDIA

Models radiative heat transfer inside participating or semitransparent media. Whereas the Radiation in Participating Media interface requires the temperature field as model input, this physics interface computes it.

SURFACE-TO-SURFACE RADIATION

Models the radiative transfer of energy between boundaries and external heat sources, where the medium does not participate in the radiation.



Model Inputs

Temperature:

T: User defined

293.15[K] K

Absolute pressure:

pA: User defined

1[atm] Pa

Relative humidity:

ϕ: User defined

0 1

FIGURE 10: Model inputs for a wood frame wall simulation. For more information, see the Condensation Risk in a Wood-Frame Wall model in the Application Gallery.

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 in the software. The powerful graphics, which allow for visualization of phenomena such as the temperature distribution and heat flux, along with structural stresses, fluid flow, or electric current variables, is especially useful for thermal management. In Figure 11, the postprocessing capabilities of the software are used to examine both fluid flow and heat transfer in a model of a finned pipe, an apparatus common in heating and cooling applications. Using a combination of surface, multislice, and arrow volume plots, results show the temperature distribution in the pipe domain as well as the velocity magnitude in the surrounding air domain.

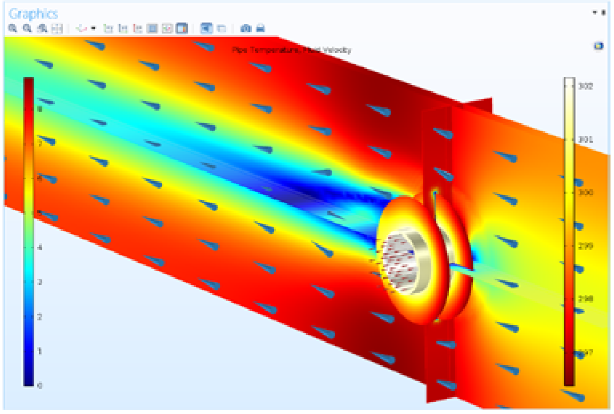


FIGURE 11: Simulation results for a model of a finned pipe.

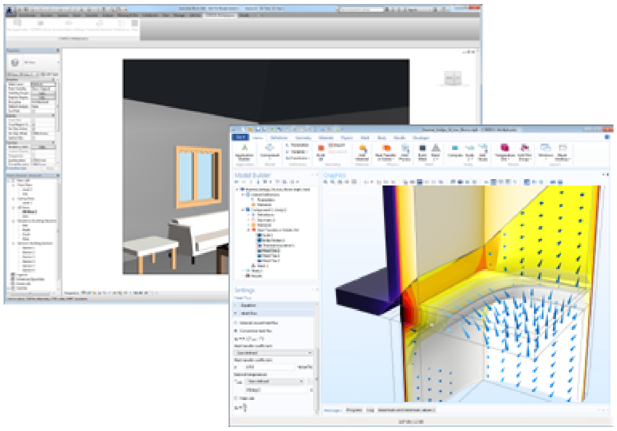


FIGURE 12: By using LiveLink™ products, it is possible to synchronize a model geometry in another software package with COMSOL Multiphysics® software.

Plots may be produced over the full simulation time, or a specific time frame or step, which provides great flexibility and 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 (like extreme temperatures or the average temperature in a part of a model) while solving.

The ability to interface with other software provides flexible workflow options for thermal management simulations, which are summarized below. With LiveLink™ products, such as LiveLink™ for Revit® shown in Figure 12, you can use external programs to design and update your geometry, set up and run your model, or share results generated with the COMSOL® software. Figure 12 shows simulation results from a study of heat conduction in a structure separating two floors of a building from the external environment.

In the following sections, examples demonstrate the use of the software for the accurate modeling of heat transfer in building structures and are compared to standard benchmarks. Also discussed is the numerical validation of innovative techniques for improving energy efficiency including the use of phase change materials and energy piles.

INTERFACING ADD-ON PRODUCTS

CAD IMPORT MODULE

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

LIVELINK™ *for* MATLAB®

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

LIVELINK™ *for* REVIT®

Allows for bidirectional geometry synchronization between Autodesk® Revit® software and COMSOL Multiphysics®. The import of 3D CAD file formats is also supported.

BENCHMARK EXAMPLE: THERMAL LOSSES THROUGH OPENINGS IN THE BUILDING ENVELOPE

The thermal performance of a building is an important consideration throughout the entire design or renovation process. Minimizing the thermal losses through all openings of a building is a design objective that is greatly aided by simulation.

Two test cases implemented in the software are presented in this section and serve as benchmarks for modeling thermal losses in buildings. The international standard ISO 10077-2:2012 applies to the thermal performance of windows, doors, and shutters. Simulation accuracy can be assessed by comparing results against standards provided. Calculated values are provided for the thermal characteristics of frame profiles, for example, in order to validate simulation software.

The thermal performance of a window is highly dependent on the insulation provided by its frame. A frame contains many cavities that may be categorized as unventilated, slightly ventilated, or well ventilated depending on their connection with the interior and exterior media, separated by the window. As an example, Figure 13 shows a wood frame containing thermal barriers made of polyamide and ethylene propylene diene monomer (EPDM) gaskets to waterproof the window.

Simulation allows for the comparison of the wood frame in Figure 13 with other designs made from an assembly of materials, such as aluminum, wood, polyamide, or PVC, as well as various geometries. The heat transfer rate in the cavities is represented by an equivalent thermal conductivity, which accounts for conduction, convection, and radiation.

In addition to the temperature profiles across the window shown in Figure 14, the thermal conductance, defined as the heat transfer rate through the window divided by the difference between the exterior and interior temperatures, was computed and found to be 0.284 W/(m·K). In this case, the computed value for thermal conductance varies from the expected value by 0.35%, a difference that is about 10 times more precise than required.

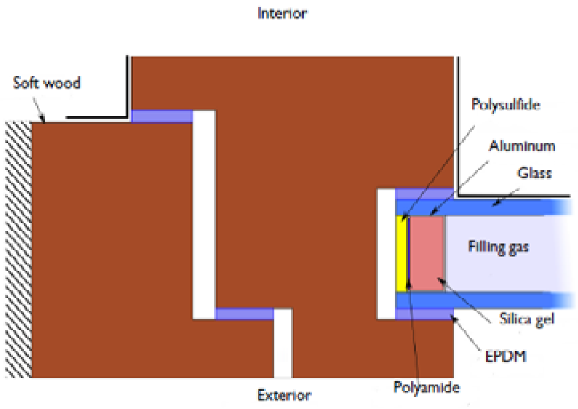


FIGURE 13: Geometry of a wood frame with a glazing. For more information, see the Glazing Influence on Thermal Performances of a Window model in the Application Gallery.

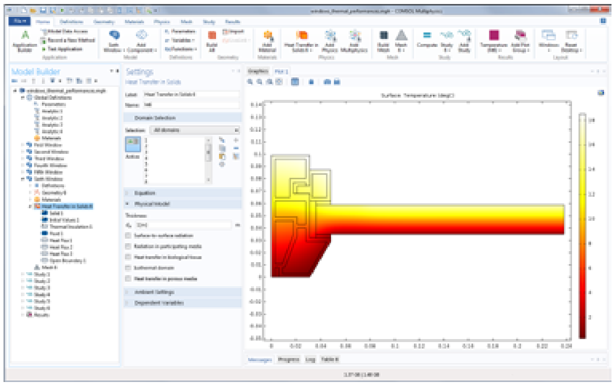


FIGURE 14: Temperature distribution in a PVC frame. For more information, see the Thermal Performances of Windows model in the Application Gallery.

The influence of glazing can also be investigated by computing the convective flux in the gas that fills the gap between the two glass panels of a window. The temperature profiles and thermal conductance can then be compared with a model using an insulation panel in the gap. Figure 15 shows the temperature profiles obtained for a wood frame window with an insulation panel and thermal conductance of 0.345 W/(m·K) compared with a wood frame window with traditional glazing and thermal conductance of 0.483 W/(m·K). Both computed values are within 0.5% of the standard values, where the tolerance norm is 3%.

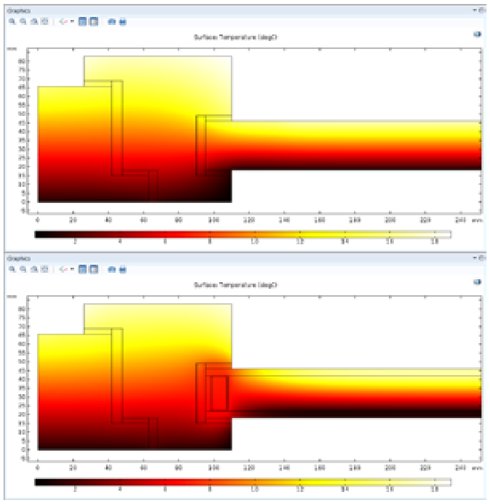


FIGURE 15: Temperature distribution in a wood-framed window with an insulation panel (at top) and traditional glazing (at bottom).

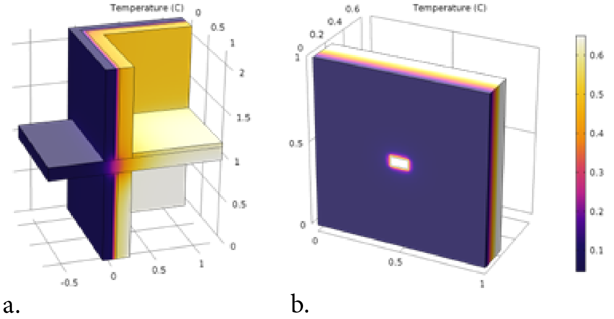
BENCHMARK EXAMPLE: MODELING OF THERMAL BRIDGES

Thermal bridges occur within a building structure where a high thermal conductivity is present in one region, allowing a heat flux significantly larger than in the surrounding parts. This happens, for example, when metallic layers or rods are embedded in an insulation layer for structural purposes. Even if the size of the thermal bridge is small compared to the surface area of the rest of the structure, the influence on heat flux distribution can be significant.

The lowest inside surface temperature and the heat fluxes through surfaces constitute good performance indicators for thermal bridges, as they measure energy losses and the condensation risk.

The European standard EN ISO 10211:2007 provides test cases for the validation of numerical methods for the modeling of thermal bridges in building structures. The COMSOL Multiphysics® software successfully passes all of the test cases described by the standard and as such, is classified as a three-dimensional steady-state high-precision method.

Figure 16 shows two simulation examples featuring a thermal bridge. The model at left represents a building structure separating two floors from the external environment. The thermal bridge, where larger heat losses are observed, is the horizontal block with high conductivity that separates the two floors. In the model at right, the thermal bridge is made up of an iron bar embedded in an insulation layer. For simulation, the use of mesh refinement at the intersection of the bar with the insulation layer, where the temperature gradient is large, reduces computation time compared to using a fine mesh everywhere.



a.

b.

FIGURE 16: Temperature throughout a thermal bridge (a) and an insulation layer (b). For more details, see the 3D Structure Between Two Floors model and the 3D Iron Bar Through Insulation Layer model in the Application Gallery.

When the relative size of the thermal bridge becomes very small compared to the rest of the structure, the software provides specific features for the cost-effective modeling of layers and rods with high or low conductivity, without adding complexity to the meshed geometry. The Thin Layer and Thin Rod features of the Heat Transfer Module offer settings to define thermal properties of materials located on thin boundaries and conductive rods located in larger geometries, respectively.

BENCHMARK EXAMPLE: MODELING HEAT AND MOISTURE TRANSPORT IN BUILDING MATERIALS

Most building components are made of porous materials, such as wood or concrete, which necessitates measures to control condensation as part of the design. Reasons for controlling condensation include preventing mold growth and the structural degradation of building materials.

Condensation formation depends on the relative indoor and outdoor conditions, characterized by temperature, relative humidity, pressure, wind, and precipitation. The design of the building and the ability of a building component to trap or transfer moisture in its liquid and gas phases affect the rate of degradation and the service life of the building material.

The European standard EN ISO 15026 defines the equations that describe the phenomena considered in the assessment of coupled heat and moisture transport in hygroscopic building materials. The equations allow a transient quantification of heat and moisture transfer through porous materials.

Numerical simulation based on these equations may be used as a validation step for a design obtained by good practice rules. This is an alternative to the Glaser method, which only accounts for moisture transfer by vapor diffusion and does not account for moisture storage or the dependence of heat and moisture fluxes on vapor pressure.

The equations given by the standard (European standard EN ISO 15026) are implemented in the software within the Heat and Moisture Transport multiphysics interface.

Vapor is transferred through building materials by diffusion, depending on the material permeability. The combined molecular and Knudsen diffusion flux is expressed as a function of the gradient of the vapor partial pressure and permeability.

The liquid phase flow is driven by the capillary pressure gradient, which can be more conveniently expressed as a function of the moisture content gradient. The heat flux is due to conduction and a heat source results from the moisture flux, essentially because of phase change. As an example and benchmark, the temperature and moisture storage function distributions through a 20-m concrete structure after 7, 30, and 365 days are shown in Figure 17. The outdoor and indoor spaces have initial temperatures of 30°C and 20°C, and relative humidity values of 95% and 50%, respectively.

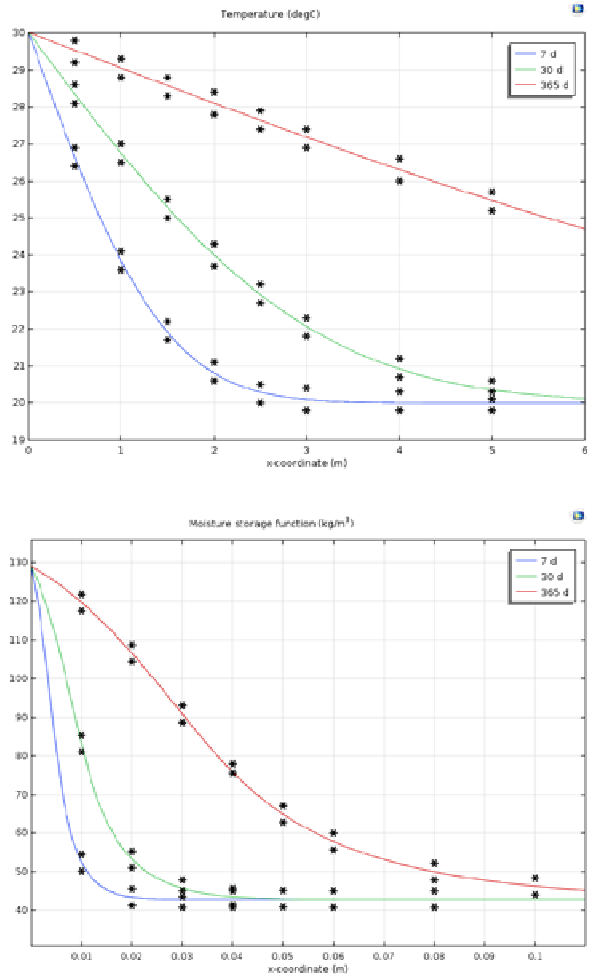


FIGURE 17: Temperature and moisture storage distribution through a 20-m concrete structure. More information can be found in the Heat and Moisture Transport in a Semi-Infinite Wall model in the Application Gallery.

APPLICATION EXAMPLE: MODELING THE IMPACT OF SOLAR IRRADIANCE ON THE CHEMICAL DEGRADATION OF PAINTED WALL HANGINGS IN A HISTORIC INTERIOR

The software was applied to simulate heat transfer with surface-to-surface radiation in the rear salon of a historic interior (ref. R5). Different degrees of chemical degradation were observed on wall hangings, possibly due to temperature variation.

The purpose of the model built with COMSOL® software is to predict surface temperature and surface radiosity of the wall hangings based on the outdoor temperature, solar irradiation, and building properties. The predicted thermal conditions indicate possible scenarios that could have induced chemical degradation of the paintings.

Figure 18 depicts the back side of the first floor of the historic Hofkeshuis (the annex on the ground floor is a modern addition) in Almelo, the Netherlands. Built around 1775, it features paintings that date from 1778. The simple and similar pigmentation of the paintings, their original hanging, and few cleaning interventions make it possible to use the degree of the formation

FIGURE 18: First floor of the Hofkeshuis. The first three windows from the left are those of the rear salon. (Photo of P. van Galen, National Museum of Cultural Heritage, item number 162,675. CC-BY-SA 4.0.)

of lead soaps as an internal marker for the chemical degradation of the oil paint.

Microscopic image analysis of paint samples revealed that the degree of saponification throughout the paintings differs, and since temperature and relative humidity can strongly influence chemical degradation, the researchers sought to characterize this connection.

The model includes heat transfer with surface-to-surface radiation, as well as an external (solar) radiation source. It combines conductive heat transfer through the building envelope, convective heat transfer through indoor air, nonisothermal flow in the indoor air, and radiant heat transfer. The outdoor temperature and solar irradiance were based on meteorological data collected at the building site. Material properties for brick, concrete, glass, and air were obtained from the Material Library available in the software.

The measured indoor and outdoor air temperature and surface irradiation from July 1 to 21, 2013 were applied to predict surface temperatures in one of the warmest months of the research period. The results were compared with the observed degree of saponification at various positions on the paintings. Figure 19 visualizes the surface temperature of the north wall: Incoming solar irradiation affected a large area of the wall, but the surface temperature differences on the wall were small (1–2°C). Further studies will include a turbulence model for indoor airflow and substitution of some estimations for material properties with higher resolution values.

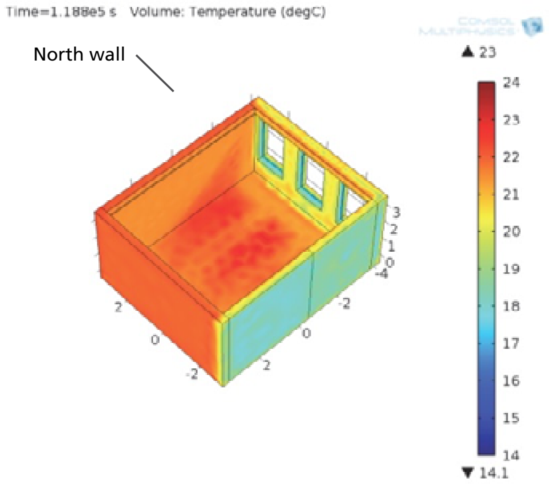


FIGURE 19: Simulation results depicting the surface temperature throughout the rear salon of the Hofkeshuis on July 2, 2013, at 9 a.m. (Huijbregts et al.).

APPLICATION EXAMPLE: MODELING PHASE CHANGE MATERIALS FOR A LATENT HEAT STORAGE SYSTEM

With the increasing number of high-performance building materials available, new techniques have been developed to improve the energy efficiency of building air conditioning systems for heating and cooling.

Phase change materials (PCMs) often have a narrow temperature range where high energy storage or release occurs during melting and solidification, for example. The energy storage density due to latent heat is significantly higher when compared to sensible heat storage. Therefore, PCMs can efficiently store energy through phase change.

PCMs absorb heat as the temperature increases and release heat when the temperature decreases, for example, when progressing through day to night temperatures. Shifting when the energy is consumed can result in better matching between supply and demand, and increased efficiency.

PCMs are generally embedded inside building materials such as concrete and can be used in combination with a layer of silica aerogel on the exterior side of a wall. The latter material is used for its low thermal conductivity and high solar energy transmittance, which ensures that solar radiation energy stored in the PCM during the day is released to the building interior with low heat loss during the night. Figure 20 (ref. R6) shows that the variability of the heat flux at the interior surface of a wall containing PCMs is damped compared to those obtained for a reference wall. In this example, phase change occurs at 15°C in the phase change material.

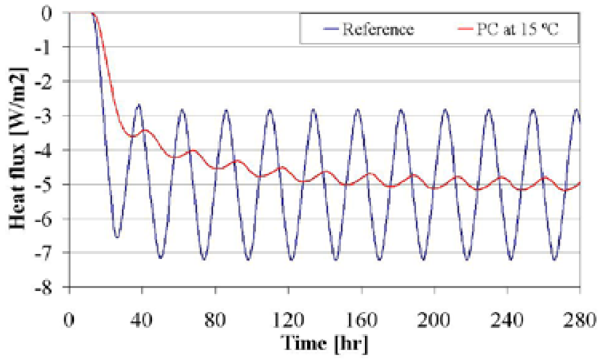


FIGURE 20: Comparison of heat flux at the interior wall surface for a wall with the phase change material (red) and for a reference wall (black). Image courtesy Baghban et. al.

The Phase Change Material feature in the Heat Transfer Module can be coupled with conduction and convection. In this model, the apparent heat capacity formulation is used to account for the phase change. Coupling this feature with the Laminar Flow interface from the CFD Module allows the modeling of natural convection into the liquid phase by adding the corresponding body force term. In addition, adaptive mesh refinement can be used to refine the mesh along the phase change front, as this is the region where the results are strongly dependent upon mesh size.

The modeling of PCMs for a latent heat energy storage system (LHESS) in a domestic water heater is presented in Figures 21 and 22 (ref. R7) and serves as an example of including convection in the melt fraction in the model. The temperature profile over time (Figure 22, top), as well as the velocity profile (Figure 22, bottom) are modeled using a quadratic temperature discretization and account for both conduction and convection. The solid phase is shown in dark blue in Figure 22, top.

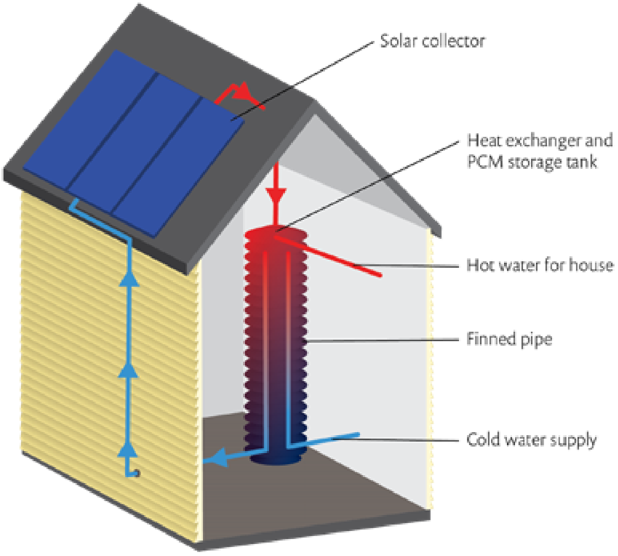


FIGURE 21: A domestic water heater with a latent heat energy storage system (LHESS) based on PCMs.

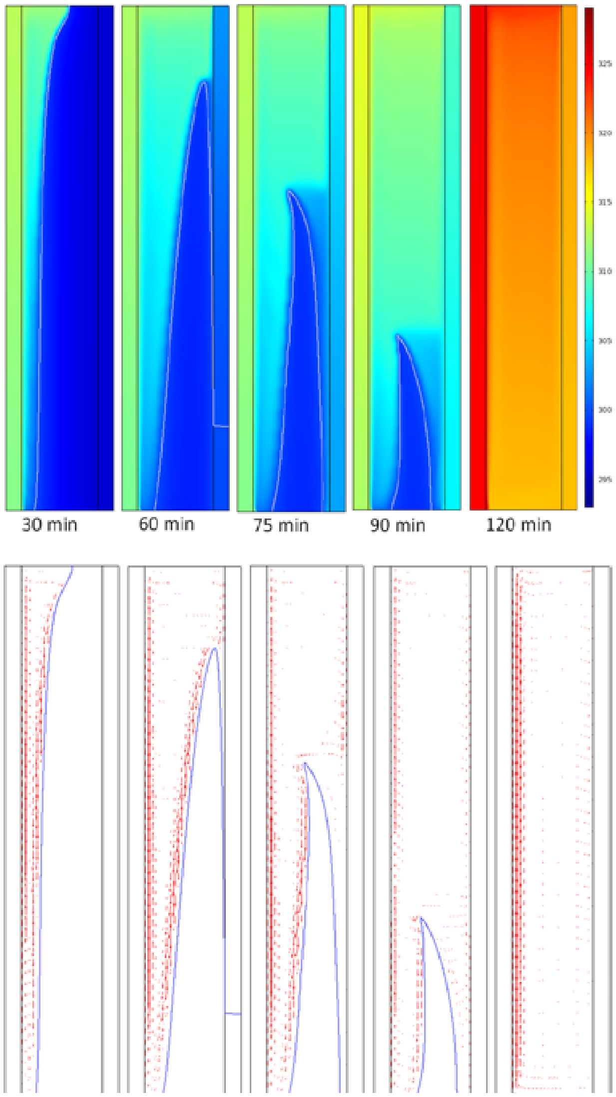


FIGURE 22: Evaluation of the melting behavior of RT25, a commercial PCM from Rubitherm GmbH, in a domestic water heater. Image courtesy Groulx et al.

APPLICATION EXAMPLE: MODELING HEAT EXCHANGERS AND ENERGY PILES

Ground heat storage, also referred to as seasonal thermal energy storage, is an efficient method of long-term energy storage for thermal management in buildings, where heat captured during warm expanses of time is stored for use during colder periods. Several systems may be combined in practice for a reduction of up to 50% of CO_2 emissions for new buildings.

Energy storage relies on either a closed loop ground heat exchanger or an open loop groundwater heat exchanger. A typical closed loop installation consists of a borehole containing two small-diameter tubes in which a fluid is circulating, linked at the bottom with a U–bend. This type of installation can be built into the foundation piles of a building to form energy piles, as shown in Figure 23 (ref. R8).

The software allows the coupled geomechanical, thermal, and hydraulic modeling of energy piles and the poroelastic adjacent ground. The hydraulics and geothermics are strongly coupled by the addition of a porous convection term in the heat equation and by the temperature dependency of the density and viscosity used in defining the hydraulic conditions. The variations in temperature lead to thermal expansion or contraction that induces strain. The pore pressure affects the mechanical strain tensor, while the hydraulics equation is modified to account for deformation, a coupling process performed automatically by the software.

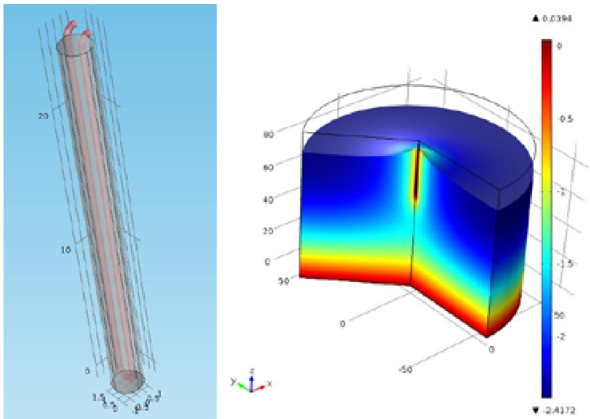


FIGURE 23: Left: Concrete foundation pile (gray) and heat exchanger pipe (red). Right: Temperature distribution from an axisymmetric 2D steady-state simulation. Image courtesy E. Holzbecher et. al.

SUMMARY

The contents of this application note presented how the thermal management of buildings can be achieved through simulation-led design of the building envelope and its constituent materials. COMSOL Multiphysics® software and the add-on products include built-in physics and multiphysics interfaces, providing the ability to readily model heat transfer, moisture transport, structural mechanics, and many other physical effects.

By adding numerical simulation to your workflow for building design, it is possible to test new concepts and gain insight into the thermal performance of a proposed structure or retrofit before investing the time and expense to build. Improving energy efficiency and human comfort, while reducing the cost associated with constructing and maintaining the built environment, are all potential benefits of using numerical simulation to evaluate and improve thermal performance.

REFERENCES

R1. Schellen, H.L. Heat, Air, and Moisture (HAM) Modeling of Historic Windows. Excerpt from the Proceedings of the 2012 COMSOL Conference in Milan.

R2. Sekki, P. Using Multiphysics Simulation to Prevent Building Damage. COMSOL News 2014.

R3. Liu et al. Numerical Investigation of Heat Transfer in an Attic Duct Model. Excerpt from the Proceedings of the 2015 COMSOL Conference in Boston.

R4. Balocco, C. and E. Marmonti. Modeling Thermal Bridging at Interface Conditions: Analysis of Solutions for Reducing Thermal Bridges Effects on Building Energy Consumption. Excerpt from the Proceedings of the 2012 COMSOL Conference in Milan.

R5. Huijbregts et al. Computational Modelling of the Impact of Solar Irradiance on Chemical Degradation of Painted Wall Hangings in a Historic Interior. Excerpt from the Proceedings of the 2014 COMSOL Conference in Cambridge.

R6. Baghban et al. Numerical Simulation of a Building Envelope with High Performance Materials. Excerpt from the Proceedings of the COMSOL Conference 2010 Paris.

R7. Groulx et al. Natural Convection Driven Melting of Phase Change Material: Comparison of Two Methods. Excerpt from the Proceedings of the 2012 COMSOL Conference in Boston.

R8. Holzbecher, E. Energy Pile Simulation – An Application of Thermo-Hydro-Mechanical Modeling. Excerpt from the Proceedings of the 2014 COMSOL Conference in Cambridge.

ADDITIONAL RESOURCES

User Story Gallery

Read how simulation experts use COMSOL® software to resolve design challenges in the building industry.

- Shake, Rattle, and Roll
  Norwegian Geotechnical Institute (NGI)
- Multiphysics Analysis Helps Preserve the Past
  CompuToolAble and Eindhoven University of Technology
- Better Ways to Heat and Cool Buildings
  Fraunhofer ISE
- Optimizing the Performance of Complex Building Façades
  Newtecnic

Application Gallery

Use these tutorials and demo app files as a starting point for your own modeling work in thermal management of buildings.

- Glazing Influence on Thermal Performances of a Window
  Application ID: 16075
- Thermal Performances of Windows
  Application ID: 16077
- Thermal Bridges in Building Construction - 3D Structure Between Two Floors
  Application ID: 12573
- Thermal Bridges in Building Construction - 3D Iron Bar Through Insulation Layer
  Application ID: 12575
- Heat and Moisture Transport in a Semi-Infinite Wall
  Application ID: 39001
- Fluid Damper
  Application ID: 8568
- Thermal Performances of Roller Shutters
  Application ID: 16073
- Condensation Risk in a Wood-Frame Wall
  Application ID: 43871

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 search
  100+ building-related results

COMSOL Blog

Browse our blog to get inspiration and guidance for your own simulation work on buildings.

- An Analysis of Caustic Surface Generation at the Vdara® Hotel
- Bringing Multiphysics into Architecture
- Evaluating the Instability of a Space Arc Frame
- Simulating Thermal Performance in Windows
- Testing the Safety of Reinforced Concrete Beams with an App
- Thermal Modeling of the Air Flow Inside and Around Your House

Contact Information

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

Contact technical support at: comsol.com/support www.comsol.com

COMSOL