

# Heat Transfer Simulation in Materials Processing

*This briefing is a summary of the webcast of the same name on May 22, 2014.*

## Introduction

Engineers in the aerospace, defense, heavy manufacturing, automotive, and medical product industries develop processes to produce materials with specific mechanical and physical properties. COMSOL Multiphysics is used to analyze conjugate heat transfer problems that are relevant to material processing, and investigate the effects of fluid flow on the quenching of hot components. The analyses show the effects of conduction, convection, and radiation for sufficiently high processing temperatures. COMSOL is a true multiphysics code, and does not iterate between physics — it solves them simultaneously. COMSOL Multiphysics is a full-featured code that does everything from laminar flow, to linear structures, to simple electromagnetics. To it, various modules can be added that enable considerable capabilities. The particular module of interest to this topic is the Heat Transfer Module.

## Technical Overview

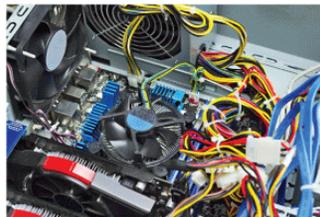
This work describes how to solve one complex problem using COMSOL Multiphysics in heat transfer analysis. The solution was developed for conjugate heat transfer analysis for both a gas quench and a liquid quench for metal parts.

Conjugate heat transfer analyses are developed across a wide range of applications. In electronics, the aerospace industry has many conjugate heat transfer problems; other applications include biomedical and the metals industry.

What is a conjugate heat transfer analysis? It refers to the ability to compute conductance of heat through the solid, and couple that with conductive heat transfer within the fluid. Cooling electronics components is an excellent example of this, and it's needed to maintain safe operation of the electronics and extend the operating life of those electronics.

For turbines, the industry continues to seek more effective gas turbines. Server inlet temperatures will continue to increase. These high temperatures apply more load to the turbine parts; thus, they are looking to improve their cooling systems, and that will be necessary if they want to extend the life of the components for the entire operating temperature. The turbine provides an opportunity to look at conjugate heat transfer in two separate flows: the hot gas heating the metal itself, and then the cooling system, which is seeking to pull the heat out.

Electronics



Blood Vessel Cooling



Turbines



Heat Treatment of Metals

In the medical industry, blood vessel cooling is extremely important. Researchers are developing medical treatments that are going to locally increase tissue temperatures, so they're seeking to understand heat transfer that occurs around these heated tissues. Blood flow is a very important part of cooling in this area. One could simply use heat transfer coefficients developed for a cylindrical duct, but they typically provide less than accurate solutions. Solving directly for conjugate heat transfer from the tissue to the blood flow is going to dramatically improve the accuracy of this type of problem.

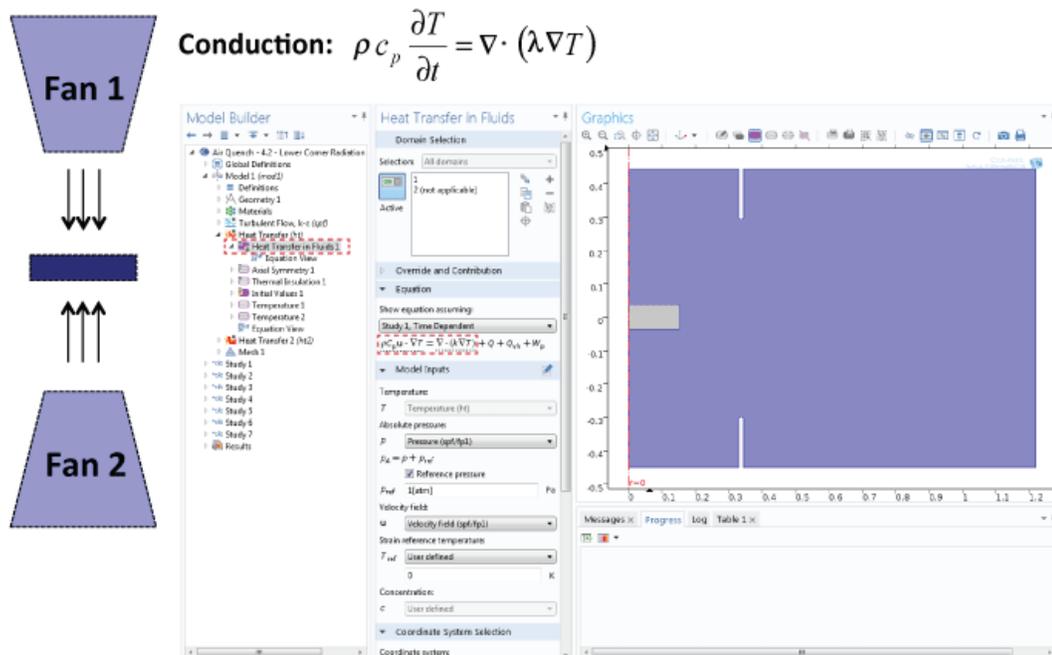
### Quenching of Metals

In the case focused on in this work — quenching of metals from elevated temperatures — it's required to develop the specific microstructural features that are needed for the described properties. The conjugate heat transfer problem develops between the quenching

media, which is either a gas or a liquid, and the hot metal being dumped into the quenching media. An accurate solution will help provide a better prediction of the mechanical properties of that metallic part.

The mechanical properties of metal depend on the microstructure — the strength, ductility, and toughness can all be altered based on the composition of the metal and the temperature history of the part. The forming process is typically done at an elevated temperature, and the quenching operation seeks to control the cooling to develop a specific set of mechanical properties. These cooling rates may vary dramatically within whatever part is being quenched, and those variations can produce unwanted mechanical properties within the part.

To estimate the cooling within a part, metallurgists typically analyze their quenching operation using heat transfer coefficients (HTCs). These HTCs are often developed through a combination of experience and



**Figure 1.** In the experimental setup, heat will transfer from the hot pancake to the surrounding air to cool the pancake. For this transient case, the conduction will be driven by the density, specific heat, and thermal conductivity of the air. In addition, the temporal and spatial gradients of the temperature ( $T$ ) are shown in the equations.

experimentation, and are specified as being constant with time as the part cools down. This assumption can produce inaccuracies across the solution as the cooling varies with temperature of the part. Additional difficulties arise when HTC's are developed for simple geometries, and then one attempts to apply them into a complex shape. These experimentally developed HTC's may not represent what cooling is actually being experienced by the complex part.

Finally, the method used to develop these HTC's from experimental data can develop unique solutions, and they correlate well with the experimental data, yet they fail to operate successfully when looking at a more complex part.

### Focus on Multiphysics

The primary focus of the work with COMSOL Multiphysics was to provide a tool that could calculate accurate HTC's for complex geometries, and use those

in current design practices. The first step toward achieving this goal was to develop an accurate heat transfer modeling methodology of the quenching process. COMSOL Multiphysics provides all of the heat transfer capabilities needed to complete these complex analyses of both air and oil quenching.

The goal of this development was to obtain HTC's that are functions of both time and position. Based on the current practices, these HTC's would be constant within a segment of the surface. The process would also permit the user to define the number of segments and the minimum segment size. Based on this input, the algorithm would optimize segmentation of the surface for which the HTC's had been calculated by minimizing the standard deviation between the results calculated in COMSOL and the constant HTC approximations necessary within the current design practice.

But the first step is to develop an accurate heat transfer solution. COMSOL includes the ability to

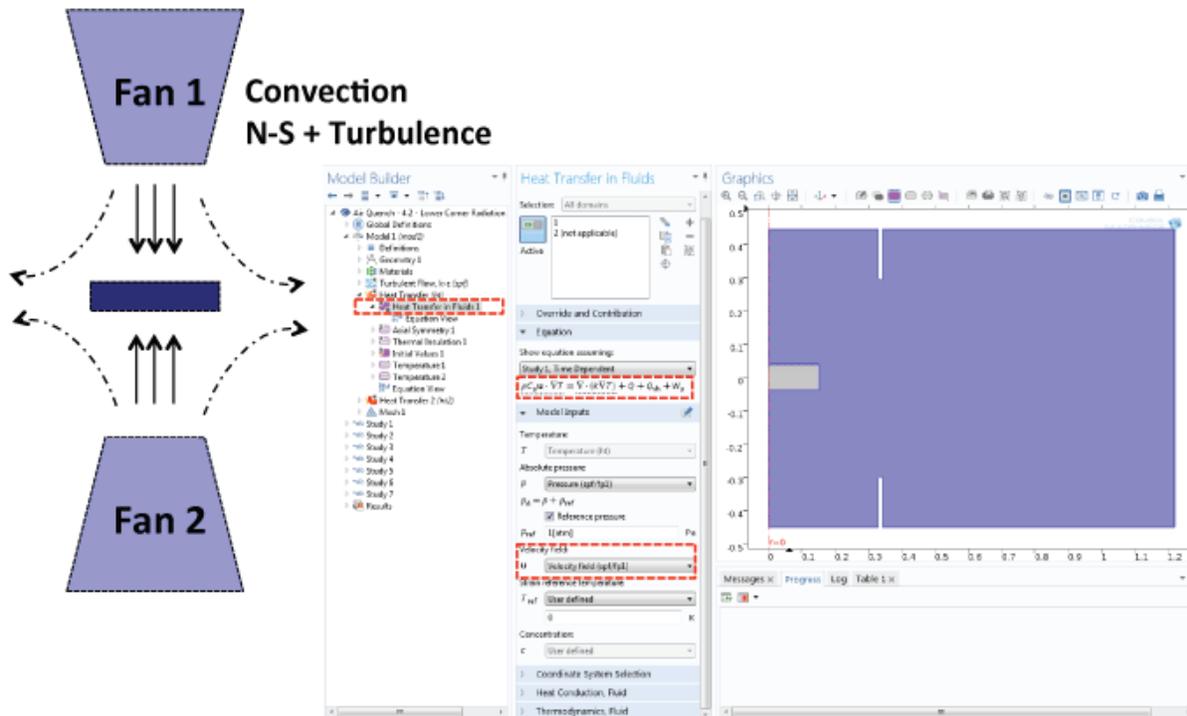


Figure 2. In the full heat transfer equation, the velocity of the flow is included. The velocity provided comes from the solution of the Navier-Stokes equation for the turbulent flow using the k-epsilon model available in the Heat Transfer Module.

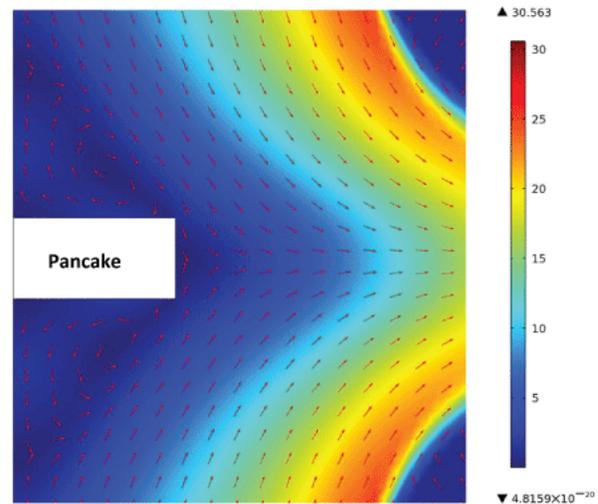
model conduction, convection, and radiation. The experimental setup being modeled is a cylindrical sample called a pancake, which sits between two fans. The heat will transfer from the hot pancake to the surrounding air to cool the pancake. For this transient case, the conduction will be driven by the densities, specific heat, and thermal conductivity in the air (Figure 1). In addition, one can see the temporal and spatial gradients of the temperature, which is  $T$  in the equation, within the solution.

The full equation that COMSOL uses also includes conductive effects. To include these conductive effects, the analysis must solve the turbulent flow around the pancake, which is turbulent in this case. In the full heat transfer equation, the velocities of the flow are included (Figure 2). The velocity comes from the solution of the Navier-Stokes equation of the turbulent flow, in this case using the K epsilon model available in the Heat Transfer Module. This analysis directly couples the heat transfer and the Navier-Stokes solution to provide the most accurate solution for this problem.

As part of this coupling, the velocities in the heat transfer analysis are defined as the velocities from the Navier-Stokes equation. The final heat transfer mechanism to be included in this is the radiation due to the temperature of the pancake prior to quenching. In this case, we solve for the radiation in the ambient environment by adding a surface ambient radiation mode to the model tree. The radiation flux of the pancake surface is a function of the emissivity of the surface, the ambient temperature, and the temperature of the pancake. By including this radiation node in the model tree, the radiation equations are then automatically and directly coupled into the solution with the conduction and the convection rates.

### Importance of Velocity

Once the geometry and the boundary conditions are included in the model, then the problem can be



**Figure 3.** In this axisymmetric model of the airflow around the pancake, the color spectrum represents the magnitude of the air velocity in m/s. The arrows show the flow direction. The two fans above and below the pancake blow air at it. This condition produces a complex flow field that COMSOL Multiphysics solves. The velocities shown here include the effects of changing air density due to heating.

solved in COMSOL Multiphysics. Although the primary variable of interest is the temperature in this problem, the gas flow velocities will need to be calculated as part of the solution.

The two fans mounted above and below the pancake blow air at the pancake. This condition produces a complex flow that COMSOL Multiphysics solves (Figure 3). The velocities shown in the figure show the effects of the change in the air density due to heating, which is an important part of this multiphysics problem.

To validate the analysis, a series of quenching experiments was run using the pancake with the thermocouples. There are 11 thermocouples embedded in the pancake, which is on the quenching rack above the oil bath (Figure 4). The leads from the thermocouple to the data acquisition system are coming out of the pancake. Each one of those leads then leads into a thermocouple, which is mounted within the pancake itself.

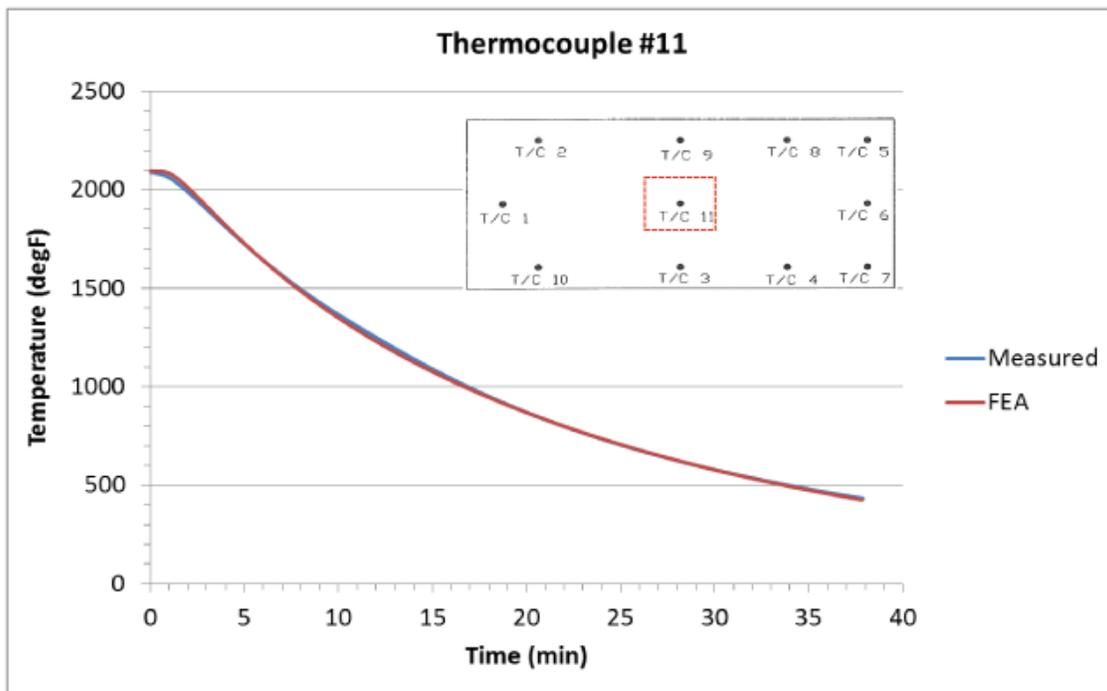
Many of the thermocouples are located near the surface of the pancake. The data shown in



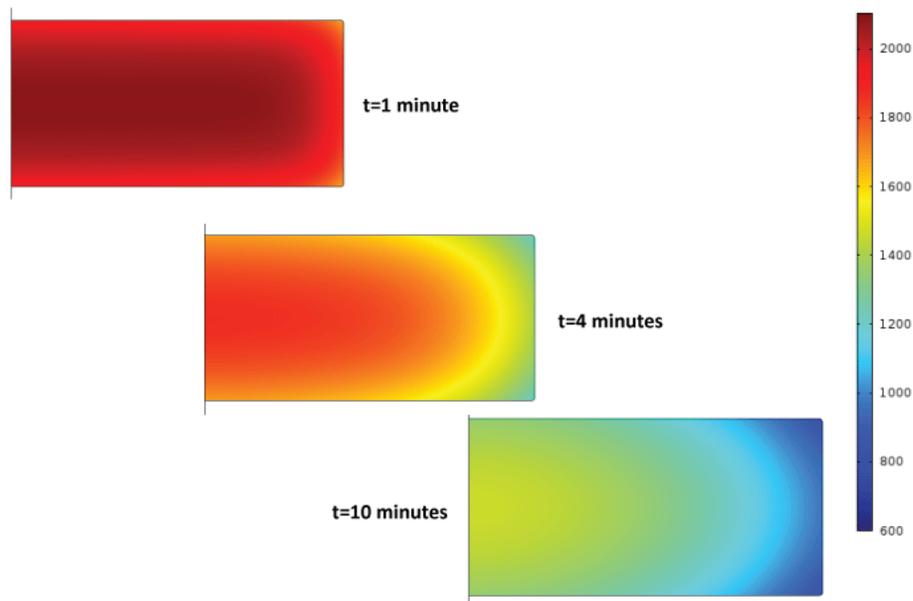
**Figure 4.** The actual test pancake with the 11 thermocouples embedded into it. The pancake sits on the quenching rack above the oil bath. Leads are seen coming from the thermocouples for data acquisition during quenching. The pancake has thermocouples located throughout the volume.

Figure 5 is from thermocouple 11, which is located mid-thickness and at half-radius of the pancake. This location was chosen because it includes both the effects of the heat transfer at the surface, and the conductive effects throughout the shell of the pancake. In experiments, you can only get as much data as you put thermocouples into a part, whereas COMSOL provides the ability to look at the temperature data at thousands of nodes — or at least as many nodes as are within the model.

The spatial variation of temperature is shown in Figure 6 at three different time slices: one minute, four minutes, and ten minutes. Initially, the temperature is relatively uniform with a slight decrease in temperature in the corners of the pancake. It is at these points where the airflow convects the maximum amount of heat away. As the quenching time progresses, these corners cool more rapidly than the rest of the pancake. These results provide



**Figure 5.** This plot shows the temperature in degrees Fahrenheit as a function of time from the start of air quench to approximately 40 minutes. The blue line shows the measured data from the thermocouple, and the red line shows the finite element results.



**Figure 6.** The spatial variation of temperature at three different times is shown here. Initially, the temperature is relatively uniform with a slight decrease in temperature near the corners of the pancake. At these points, the airflow convects the maximum amount of heat away. As the quench time progresses, these corners cool more rapidly than the rest of the pancake.

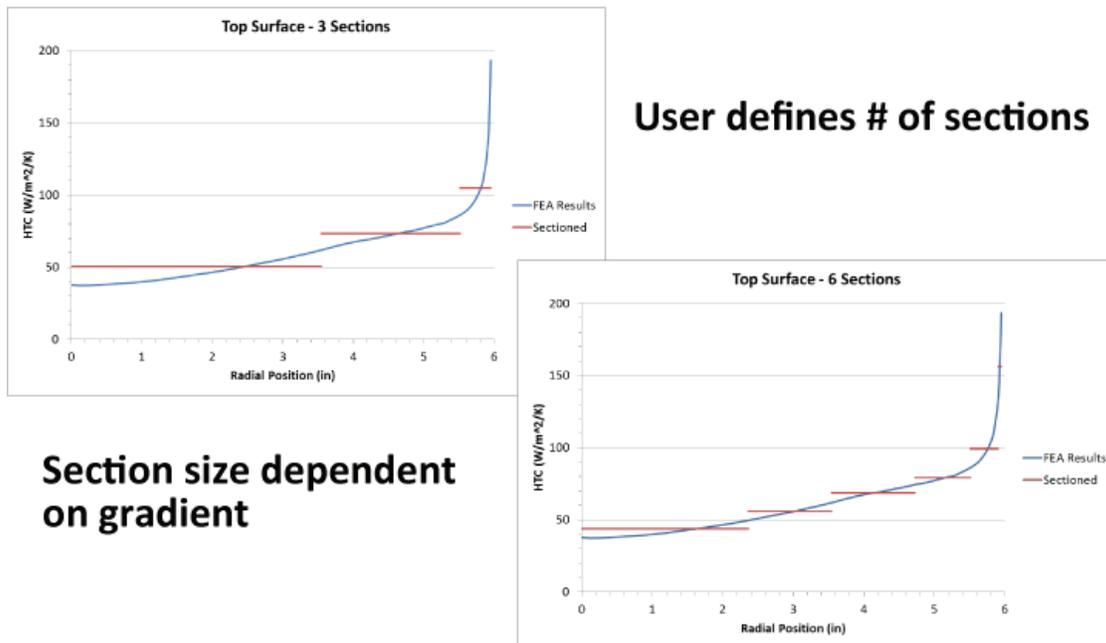
a clear understanding of temperature distribution throughout the entire pancake.

### Calculating HTCs

The primary objective of this work was to determine the HTCs on the surface of the pancake, which required an algorithm to calculate the HTCs. The data presented in Figure 7 shows the variation of the HTC along the top of the pancake. The blue lines show the values calculated directly from COMSOL Multiphysics. The HTCs take the largest value at the corner of the pancake. The red lines show the results of an algorithm that calculated the constant HTC over a section of the surface. Within this algorithm, the user can specify the number of sections per surface, with the user defining three surfaces in the top left plot and six surfaces on the bottom right plot. This algorithm sections the surface based on the gradient, and the regions with the highest gradient have the smallest sections. The algorithm then seeks to minimize the standard deviation between the constant HTC value and the COMSOL Multiphysics results.

Currently, the design method used in this work specifies that the HTC values be held constant over time. However, the results from COMSOL show that these HTC values actually do vary quite significantly with time. In these results, the HTCs are calculated in three different segments. The case considered here segments the top section of the pancake into three sections. Data shows that the HTC can increase by approximately 40% during the quenching process. The results from COMSOL show that the constant HTC approximation introduces some air into the results.

Although the primary goal of these analyses was to provide HTC values for additional analysis, COMSOL can actually solve all the way through the mechanical problem because of its ability to solve multiphysics problems. If the solid mechanics were included in this analysis, then information on plastic deformation during quenching could be obtained. COMSOL can go directly to the solids mechanical analysis and conduct, simultaneously, the stress and strain calculations as well as the HTC calculation.



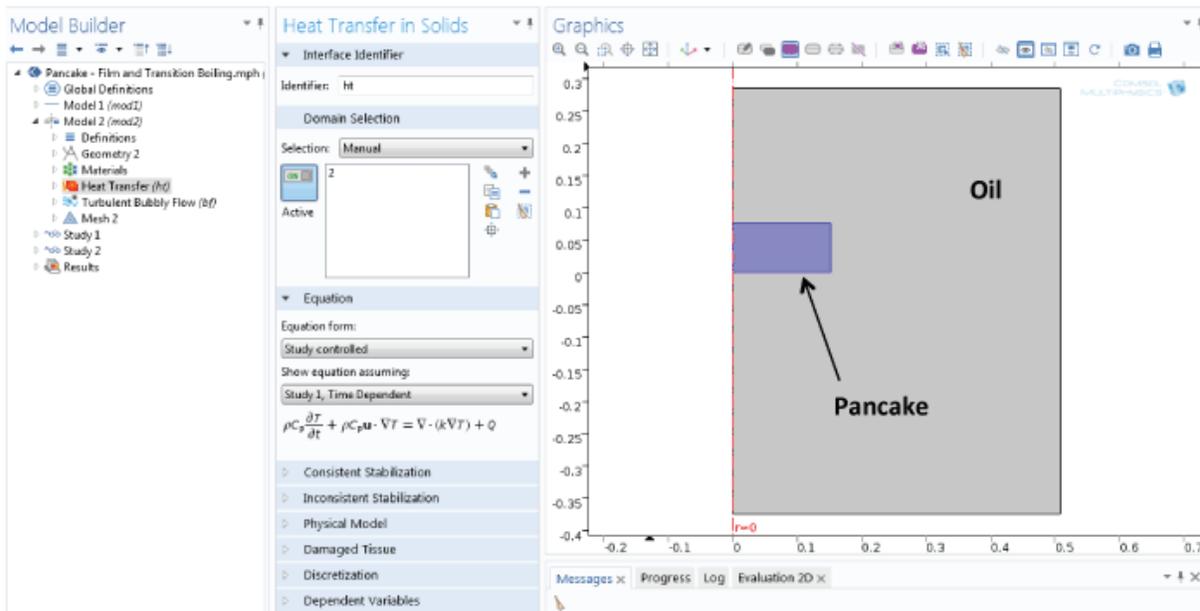
**Figure 7.** The data presented in these two plots shows the variation of the heat transfer coefficient along the top surface of the pancake. The blue line shows the values calculated directly from COMSOL Multiphysics. The HTC takes the largest values at the corner of the pancake. The red line shows the results of an algorithm that calculates a constant HTC over a section of the surface. Within this algorithm, the user can specify the number of sections per surface, with the user defining three surfaces in the top left plot, and six surfaces on the bottom right plot.

In the oil quenching case, the same hot pancake used in the air quenching study is lowered into an oil bath, and the quenching oil is pumped around the tank to maximize cooling. In COMSOL, the geometry window shows the pancake in blue and the oil region in gray (Figure 8). This model was analyzed in both two and three dimensions. As part of this problem, the oil flow around the pancake due to the pumping system was calculated. These results show the magnitude of the velocity of the oil at one section through the oil tank. This section includes the outlet from the pump to the cooling system.

These analyses were developed and solved in the Navier-Stokes equations with inputs from the tank pump. In addition to velocity magnitude, COMSOL can also provide streamline data. The streamlines are shown throughout the volume of the tank. The oil enters the tank through nine

nozzles located under the pancake, and exits at the outlet at the back right of the tank. Once the pancake has been placed into the tank, it circulates oil to the point of boiling. This process develops over time through the following stages.

Initially, the surface is cooled by a combination of natural and forced convection. Once the oil close to the surface heats to a sufficient point, small bubbles will start to form at the onset of boiling. The number and size of these bubbles will grow through a series of different stages until stable film boiling has developed. Each of these stages represents a different boundary condition on the surface of the pancake. A key component of this phase of the work was the development of a set of partial differential equations (PDEs) that describes the major stages of boiling, and then applying them within COMSOL Multiphysics. The ability



**Figure 8.** In this COMSOL Multiphysics interface, on the right side, the geometry window shows the region for the pancake in blue and the oil region in gray. The model was analyzed in both two and three dimensions. As part of this problem, the oil flow around the pancake due to the pumping system was calculated.

to define these PDE boundary conditions proves significant to this particular problem, and shows the flexibility of the software.

As with the air quench, the results from the COMSOL analyses were compared with experimental data from the oil quench. The data shows that for the first 80 to 100 seconds, the results compare favorably with the experimental data. For longer times, these results suggest that the PDEs developed to represent these later stages of boiling need to be proven.

Gas quenching was done using the capabilities native to COMSOL. The oil quenching used user-defined boundary equations. The heat transfer values were calculated using a model, and it was shown that the HTCs do vary with time.

### Author/Presenter

Kyle C. Koppenhoefer is a principal at AltaSim in Columbus, OH. He has extensive experience in a wide range of computational mechanics applied to the medical products, automotive, petrochemical, and defense industries. He has considerable expertise in advanced finite element procedures and their application to the analysis of structures subjected to multiphysics loading. He obtained his B.S. in Engineering Science and Mechanics from Virginia Polytechnic Institute, and his M.S. and PhD. in Civil Engineering from the University of Illinois.



View the original webcast at:  
[www.techbriefs.com/webinar210](http://www.techbriefs.com/webinar210)