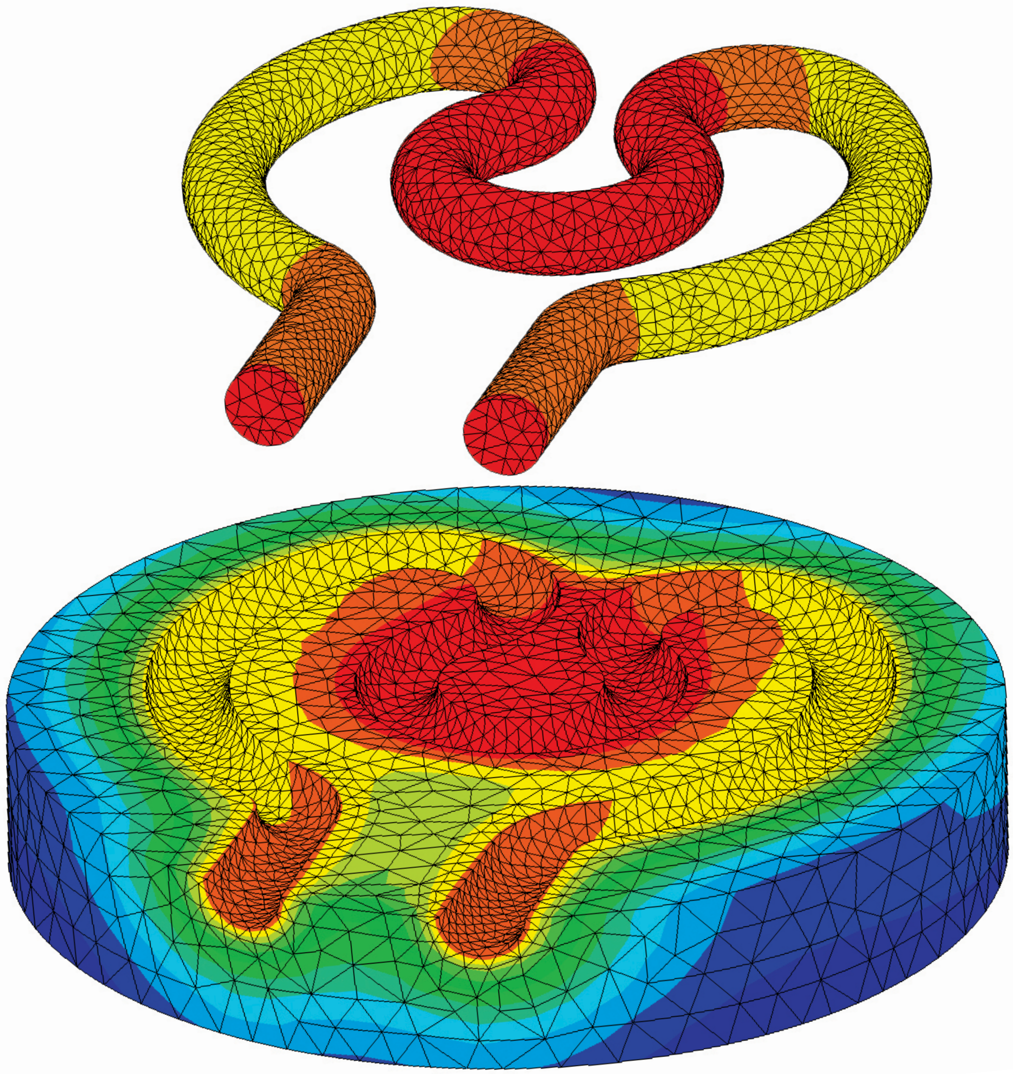
# Thermal Analysis with SolidWorks® Simulation 2012

Paul M. Kurowski



Design Generator Inc.

SDC PUBLICATIONS
Schroff Development Corporation

Better Textbooks. Lower Prices.
www.SDCpublications.com

Visit the following websites to learn more about this book:

SDC Publications

amazon.com

BARNES & NOBLE

Google books

# 2: Hollow plate

## *Topics covered*

- Heat transfer by conduction
- Heat transfer by convection
- Different ways of presenting results of thermal analysis
- Convergence analysis in thermal problems
- Solid elements in heat transfer problems
- Shell elements in heat transfer problems

## *Project description*

We'll conduct thermal analysis of simple models to study the effects of discretization error and the use of different types of elements. In this chapter we use our expertise in structural analysis gained from "**Engineering Analysis with SolidWorks Simulation 2012**". We use HOLLOW PLATE TH, similar to the model from this introductory textbook where it is used in structural analysis examples.

Open model HOLLOW PLATE TH and review the two configurations: *01 solid* where the model is represented as a solid body and *02 shell* where the model is represented as a surface body. Stay in the *01 solid* configuration and create a thermal study called *01 solid*. Apply the prescribed temperature boundary conditions as shown in Figure 2-1; these prescribed temperatures will induce heat flow from hot to cold.

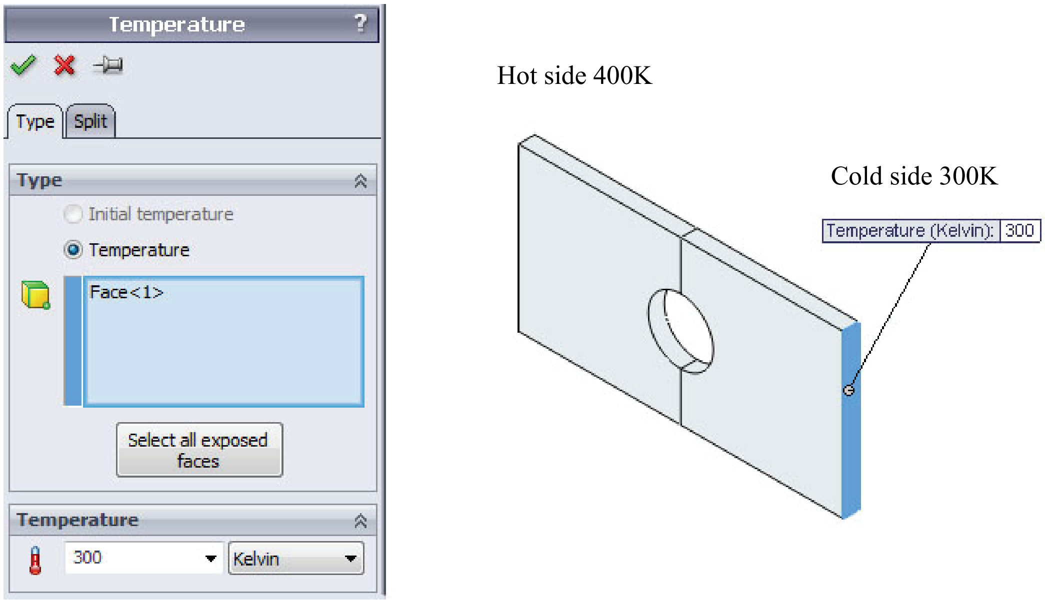



Figure 2-1: Prescribed temperatures boundary conditions applied to the cold side.

*No convection is defined anywhere in the model meaning that all faces are insulated except where temperature boundary conditions are defined.*

Repeat the definition of the prescribed temperature on the hot side where it is 400K. Mesh the model with a coarse mesh as shown in Figure 2-2.

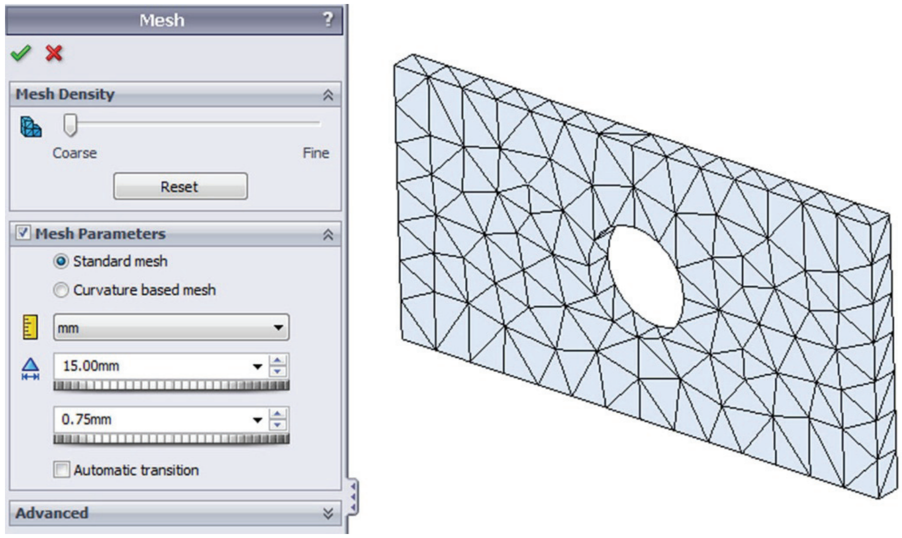



Figure 2-2: Coarse mesh created with 15mm element size

*Use Standard mesh; do not use Draft quality elements. This mesh is coarse; it is used only as the first step in the convergence process.*

Run the solution and create two plots: **Temperature** and **Resultant Heat Flux**, probe results as shown in Figure 2-3 and Figure 2-4.

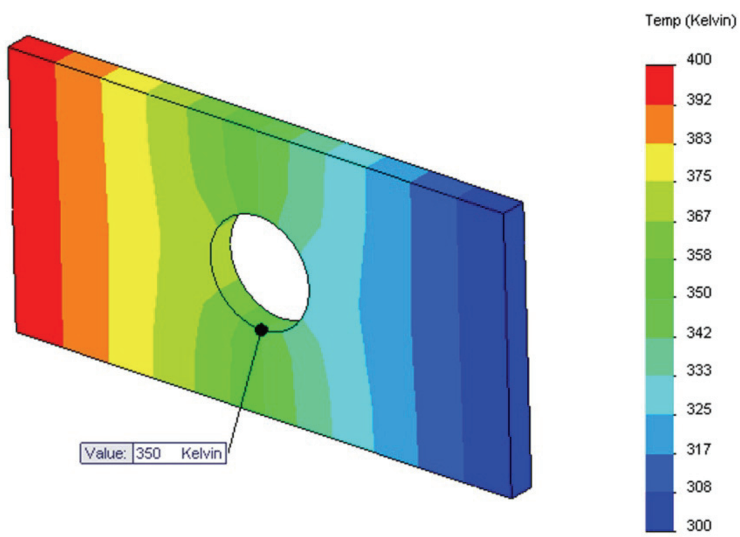



Figure 2-3: Temperature results: 350K in the probed location.

*Probe at the vertex created by the split line.*

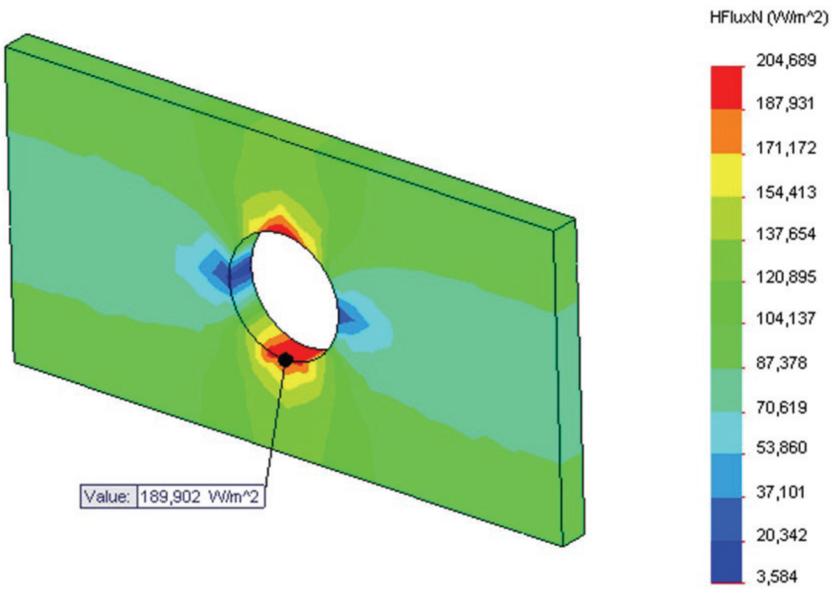



Figure 2-4: Resultant heat flux results: 189902W/$m^2$ in the probed location.

*Probe at the vertex created by the split line.*

Repeat the analysis using a mesh of default size 5.72mm (study *02 solid*) and a fine mesh with element size 2mm (study *03 solid*). Optionally try meshing the model with 1mm element size but be prepared for a long run. Notice that while temperature results are almost insensitive to mesh refinement, the resultant heat flux changes with mesh refinement as shown in Figure 2-5.

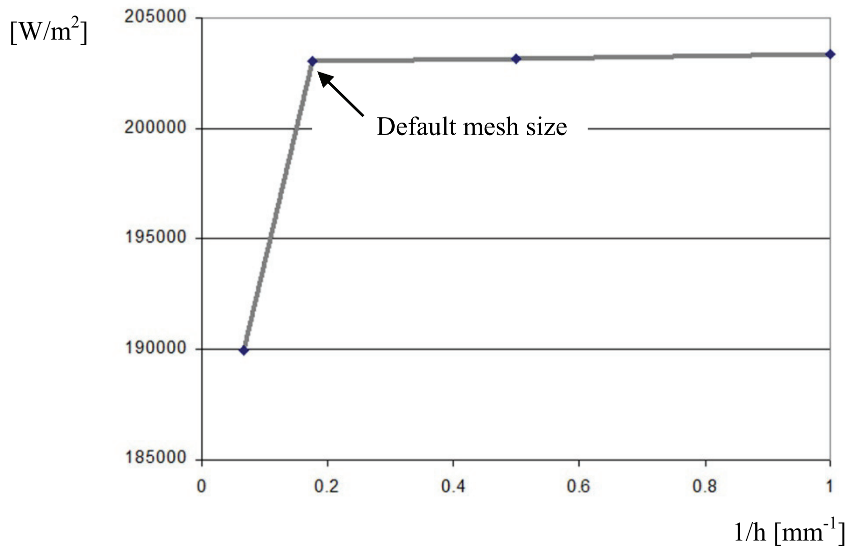



Figure 2-5: Resultant heat flux in the probed location as a function of the inverse of element size.

*Four points are connected by line only to visually enhance the graph.*

As the graph in Figure 2-5 demonstrates, the heat flux converges to a finite value and the default element size produces acceptable results. What should also be noticed is that every given mesh introduces artificial thermal resistance; the coarser the mesh the larger that added resistance is and that causes lower heat flux. This is the effect of discretization error in direct analogy to structural models where artificial stiffness is added to model by discretization (meshing).

Copy study *02 solid* (the one with default mesh) into study *05 solid convection.* We'll use it to demonstrate convective heat transfer out of the plate. Delete prescribed temperature on the cold size. In its place define **Convection** as shown in Figure 2-6.

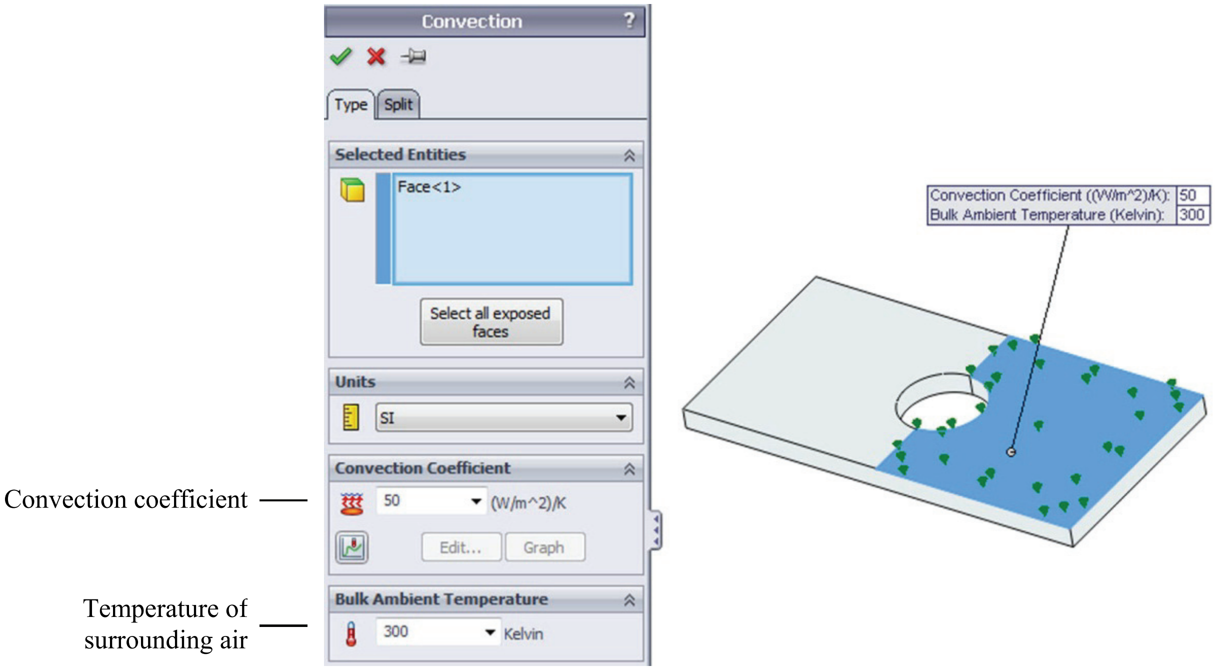
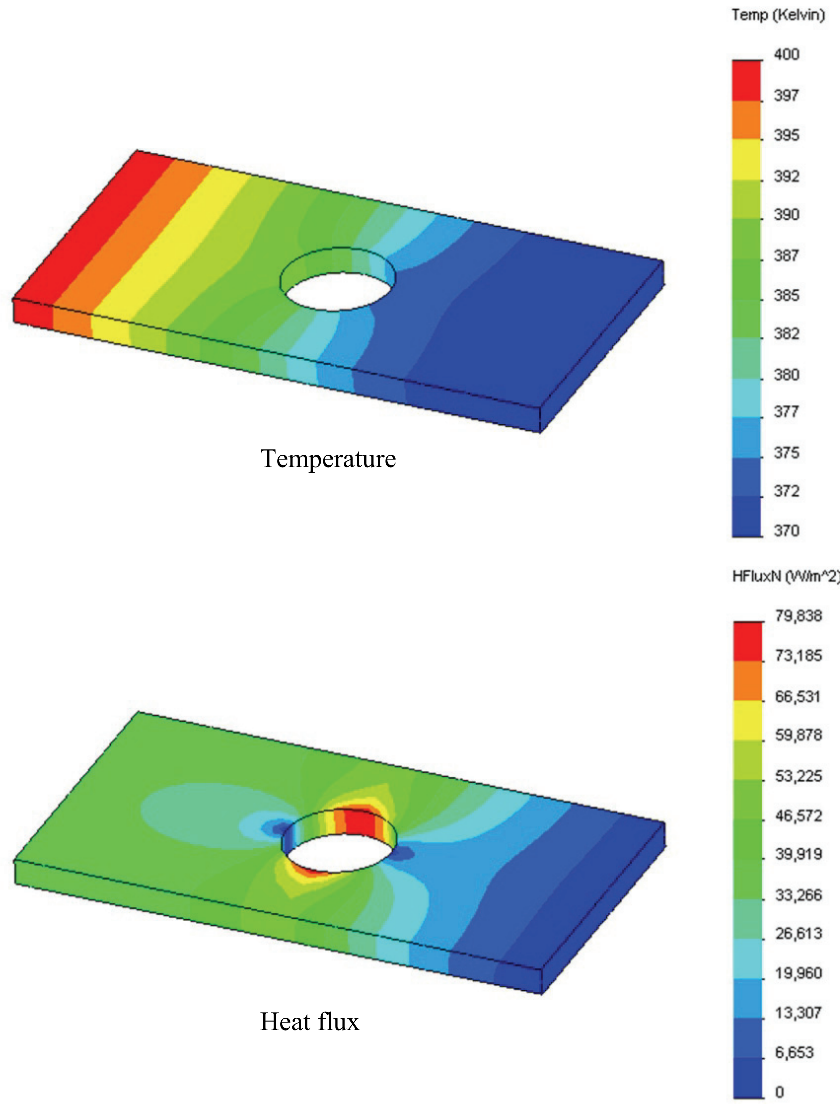



Figure 2-6: Definition of convection on ½ of one side.

*As always, definition of convection requires a convection coefficient and bulk temperature which is the temperature of the fluid surrounding the model.*

*A convection coefficient of* $50 W/m^2/K$ *corresponds to free convection with air.*

Run the solution of study *05 solid convection* and obtain results as shown in Figure 2-7.




<u>Figure 2-7: Temperature and heat flux results in the presence of convection.</u>

*Temperature is a scalar value and can be only plotted using a fringe plots above. Heat flux is a vector value; presenting it as a fringe plot shows only the magnitude but not the direction of heat flow.*

Modify the heat flux plot into a vector plot as shown in Figure 2-8.

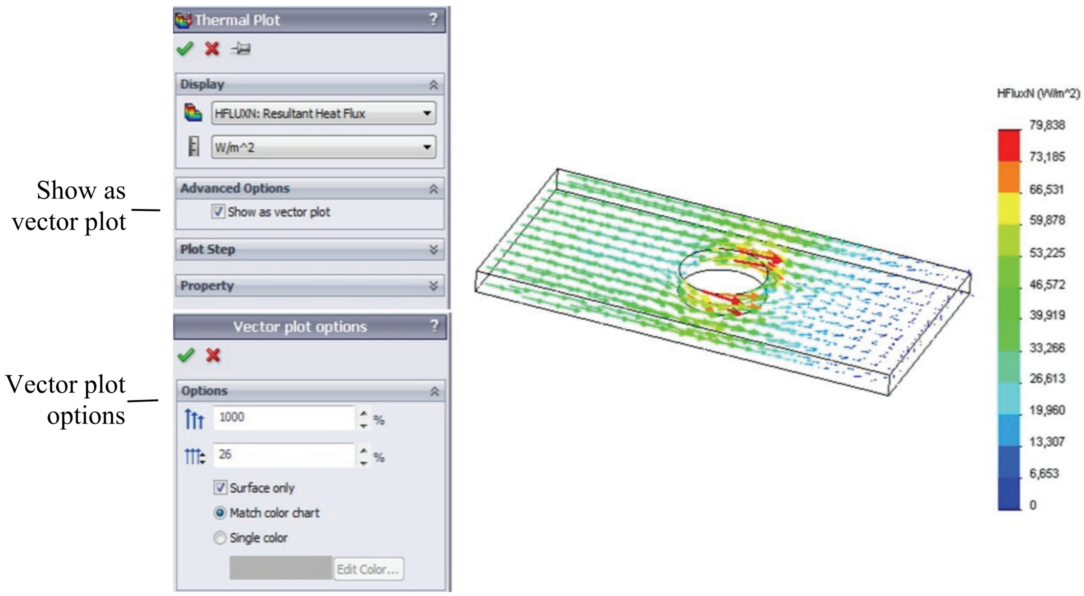



Figure 2-8: Heat flux results presented as a vector plot display the direction of heat flow.

*Rotate the plot as required to see vectors "coming out" of the model through the face where convection conditions have been defined.*

You may want to review the results of other studies with heat flux results shown as a vector plot.

Having demonstrated analogies between the convergence process in structural and thermal analyses using solid elements, we'll now study the use of shell element in thermal analysis.

Stay with model HOLLOW PLATE TH but switch to *02 surface* configuration. Create a thermal study *06 surface convection*. Define the surface thickness as shown in Figure 2-9.

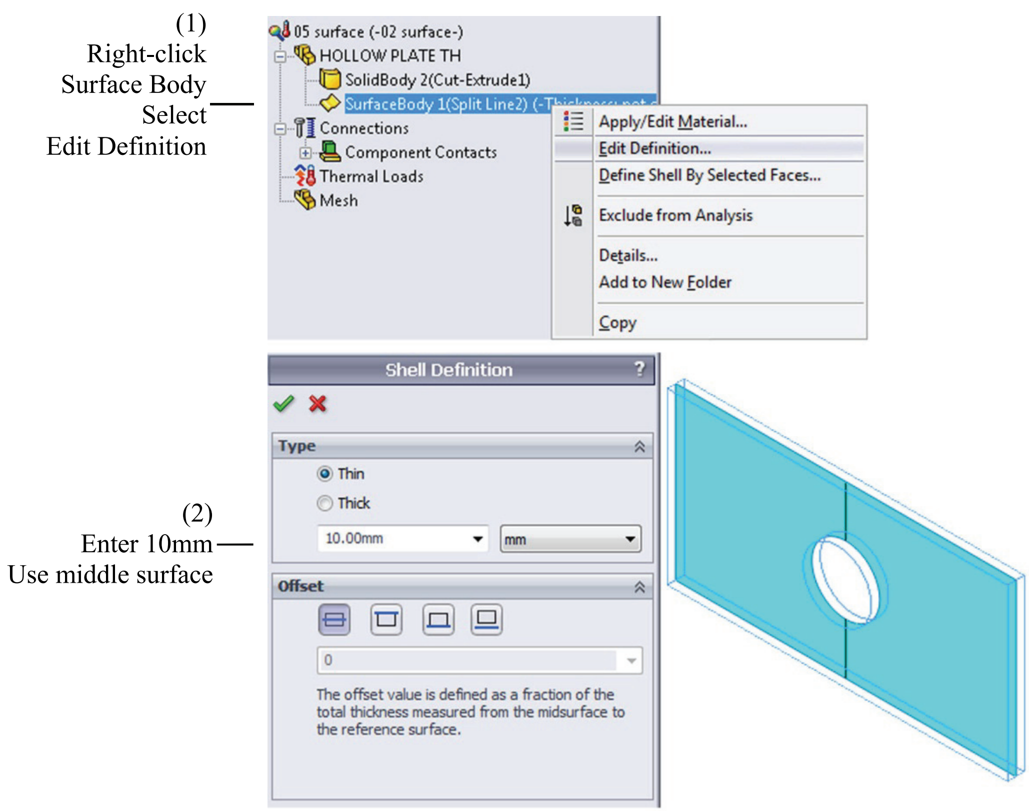



Figure 2-9: Definition of surface thickness.

*The definition of thickness is required because thickness is not specified in the surface CAD model. Thermal analysis does not distinguish between Thin and Thick shell element formulation.*

Follow steps in Figure 2-10 to eliminate the **Solid Body** from the analysis.

Right-click
Solid Body
Select
Exclude from Analysis

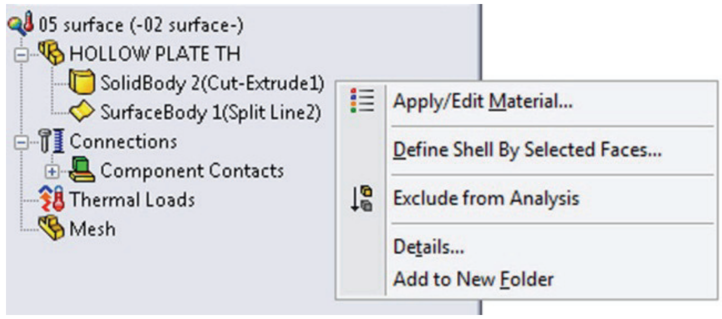



Figure 2-10: Exclusion of the Solid Body from the analysis.

*Even though the Solid Body was made invisible in the CAD model, it still forms a part of the CAD geometry and needs to be excluded from the analysis.*

Apply thermal boundary conditions as in study *05 solid convection*: a prescribed temperature of 400K on the hot side (the edge) and convection to the side face as shown in Figure 2-6. Use the default element size to mesh the model into shell elements, obtain a solution and present **Temperature** and **Heat Flux** results using fringe plots (Figure 2-11).

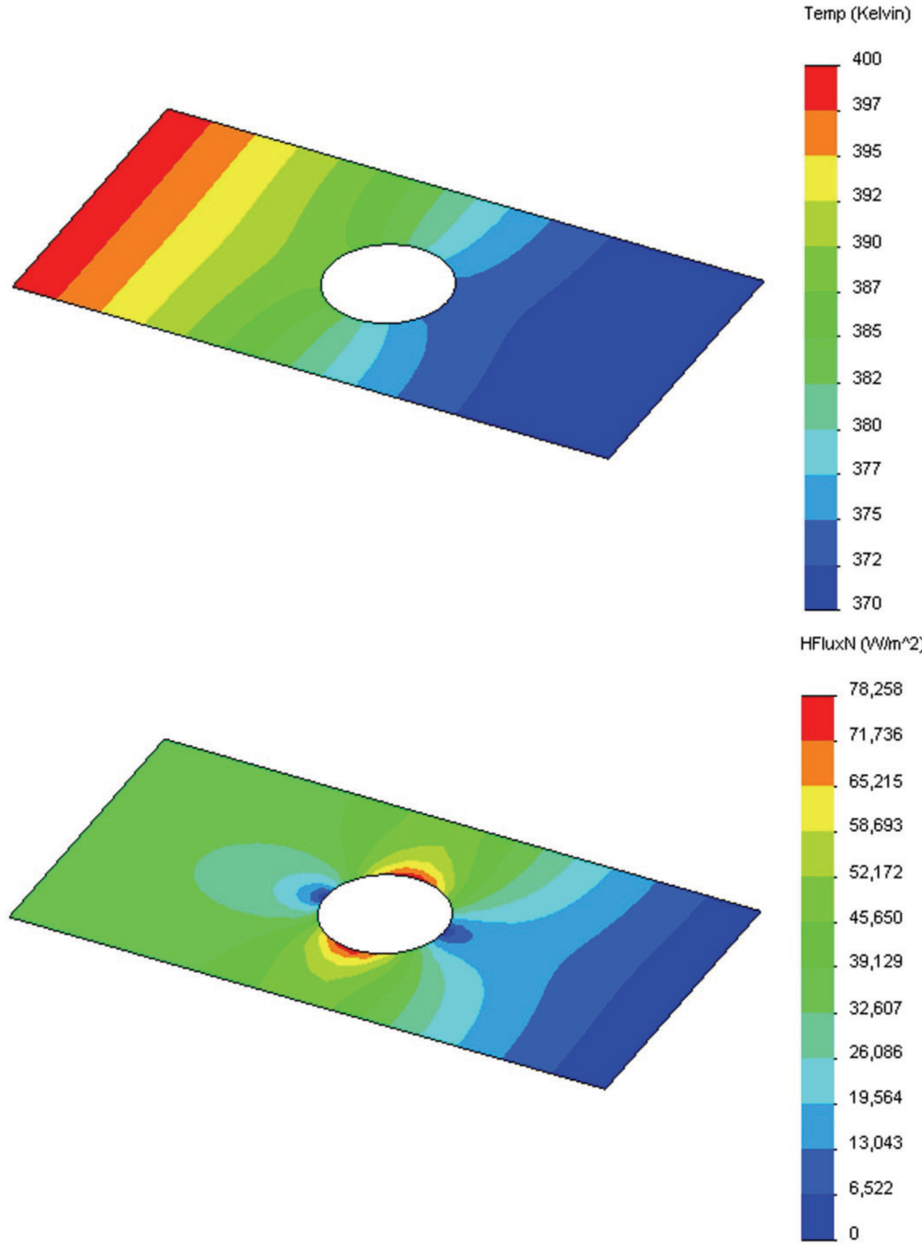



Figure 2-11: Temperature and heat flux results from the shell element model presented as fringe plots.

*These results are very similar to the results produced by the solid element model.*

While fringe plots show little difference between results obtained from the solid element and shell element models, vector plots do show important differences (Figure 2-12).

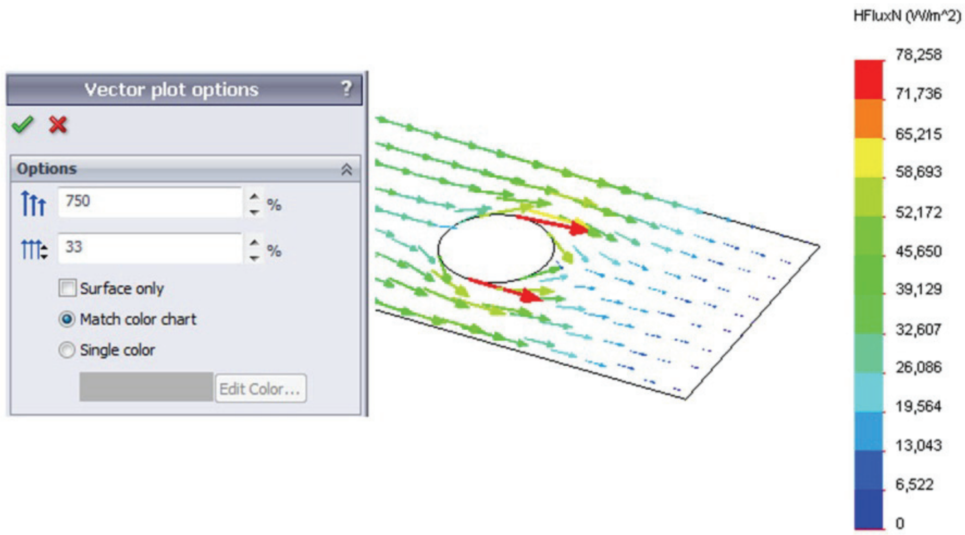



Figure 2-12: Heat flux results in the shell element model presented with vector plots.

*Rotate the plot to see that heat flux vectors do not "come out" of the model.*

In structural analysis shell elements differentiate between top and bottom and different stress results are read on opposite sides. In thermal analysis shell elements do not differentiate between sides. Therefore, the shell element model cannot show to which side heat escapes from the model. As shown in Figure 2-12, heat flux vectors "fade away" without "coming out" of the model.

Since in thermal analysis shell elements do not differentiate between sides, you cannot apply different thermal boundary conditions to the sides of face meshed with shell elements.