14  Axisymmetric Solid Analysis

14.1 Cylinder with given temperature and convection

14.1.1 Introduction

Consider a thick walled cylinder with a given internal temperature and convection at its external radius. The analysis of heat transfer in a thick wall infinite cylinder is a one-dimensional problem, and the analytic solution is known [1]. The cylinder has inner and outer radii of 9.375 and 13.40 inches, respectively, and is made of a material with a thermal conductivity of 1.736e-4 BTU/in-s-F. The inner and outer temperatures are 500 and 45 degree F, respectively, with and outer convection coefficient of 2.89e-4 BTU/F-s-in². The goal is to determine the inner temperature distribution, and the required heat flow through the wall. This problem provides a chance to verify ones knowledge of SW Simulation, and to illustrate some optional features. To access some optional features you must have a part open, or prepare to open the first part with File→New→Part→OK.

14.1.2 SW Simulation thermal study

Next, enter SW Simulation by selecting its managed icon, 📀. In SW Simulation:

1. Right click on the Part name→Study to open the Study panel.

2. Insert a Study name (say FE_T_Cyl_h), select Thermal from the pull down Analysis type list, and pull down the Mesh type list options.

3. Any of the three mesh types (solid or two shell options) can be used. For a faster 2D solution select Shell mesh using mid-surfaces. (At this point a new user will not know which mid-plane will be selected: an axial slice (yes) or a radial slice.)

14.1.3 Boundary conditions

The above choice of a Shell mesh using mid-surfaces defines the methods used to apply the boundary conditions. Since part faces are used to locate the mid-surface, the boundary conditions must be specified on the faces of the part. By way of comparison, if the alternate shell mesh by defined surfaces had been selected the boundary conditions would have to be specified on the edges of the faces picked to define the mesh. Begin the mid-surface mesh boundary conditions with the inner cylindrical (red) face:

1. In the SW Simulation manager right click on Thermal Loads→Temperature.
2. When the Temperature panel appears right click on the inner face to insert it as the Selected Entities.
3. Under Temperature set the units as degrees Fahrenheit and insert a value of 500.
4. Hit Preview (eyeglass icon) to verify your assignment, click OK.
Next apply the convection conditions of the outer cylindrical (light green) face:

1. In the SW Simulation manager right click on **Thermal Loads → Convection**.

2. When the **Temperature panel** appears rotate the part until you can see the light green face.

3. **Right click** on the outer face to insert it as the **Selected Entities**.

4. Under **Convection Parameters** select **English units**, set the temperature to 45 F, and the surface free convection coefficient (h) to \(2.89 \times 10^{-4}\) BTU/F·s·in\(^2\).

5. Click **Preview**, and then **OK**.
14.2 Define the material

The homogeneous (location independent) conducting material is not in the standard materials library, so its properties must be user defined:

1. In the manager tree, under Mid-surface Shell, right click on the Part name → Apply/Edit Material to open the Material panel.
2. In the Material panel, Source → Custom defined, Units → English, Type → Linear Elastic Isotropic (direction independent).
3. Assign a Material name, say User Defined.
4. In the Thermal conductivity row enter a value of 1.736e-4 BTU/in-s-F. That should be sufficient for this study. However, SW Simulation also demands the value of the density in case you do a transient study. Since the density is unknown enter a fake value, say unity.

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
<th>Value</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>E</td>
<td>Elastic modulus</td>
<td></td>
<td>psi</td>
</tr>
<tr>
<td>NUxy</td>
<td>Poisson’s ratio</td>
<td></td>
<td>NA</td>
</tr>
<tr>
<td>Gxy</td>
<td>Shear modulus</td>
<td></td>
<td>psi</td>
</tr>
<tr>
<td>DENS</td>
<td>Mass density</td>
<td>1.0000</td>
<td>lb/in3</td>
</tr>
<tr>
<td>SIGXT</td>
<td>Tensile strength</td>
<td></td>
<td>psi</td>
</tr>
<tr>
<td>SIGMC</td>
<td>Compressive strength</td>
<td></td>
<td>psi</td>
</tr>
<tr>
<td>SIGYLD</td>
<td>Yield strength</td>
<td></td>
<td>psi</td>
</tr>
<tr>
<td>ALPX</td>
<td>Thermal expansion coeff</td>
<td></td>
<td></td>
</tr>
<tr>
<td>K</td>
<td>Thermal conductivity</td>
<td>1.736e-4</td>
<td>BTU/in-s-F</td>
</tr>
</tbody>
</table>

Here the material property is known to four significant figures. That is high for experimental material property measurements. To be consistent, the results of the analysis should not be reported with a larger number of significant digits (although they frequently are).
14.3 Meshing

For this 1D problem only a few elements are needed in the radial direction. Thus, the default mesh will be generated:

1. In the manager tree, right click Mesh→Create to open the Mesh panel.
2. In the Mesh panel accept the default element size and transition controls, click OK. Do not check “Run study after meshing”. You should always check the mesh first.

3. Visually inspect the mesh. Having about 20 elements in the radial direction should be fine. The solution in the circumferential direction should be constant, so the number of elements in the second does not matter (but they do cost). Save the SW Simulation files.

14.4 Temperature solution

The thermal analysis is ready for solution. Right click on the Study name→Run. After the equation solver reports a successful calculation the post-processing results can be reviewed and plotted.

14.5 Post processing

14.5.1.1 Temperatures

Begin the study review by examining the temperature distribution:

1. In the SW Simulation manager tree click on Temperature and the double click Plot-1. The default plot is a continuous color contour.

2. For an alternate format right click in the graphics area and pick Edit Definition. In the Thermal Plot panel pick a Filled, Discrete for the Fringe Type, click OK.
3. The discrete contours appear. The convection surface temperature is about 96

![Graph showing convection surface temperature]

4. Graphs can provide more detail in selected regions. Graph the radial temperature first:
   1. Right click in the graphics window, pick List Selected.
   2. Select the bottom edge, click Update to see max, min, and average values
   3. In List Selected panel, pick Plot to open the Edge Plot
The graph agrees very well with the logarithmic analytic solution. A reasonable estimate could have been obtained with a single element hand solution to help validate the temperature result.

### 14.5.1.2 Heat flux

The heat flux is a vector quantity obtained from the scalar temperature. In this case it must be in the radial direction (plot the vector form to see that) so just the values are shown here in Figure 14-1.

![Figure 14-1 Radial heat flux magnitudes](image-url)