1.6 Mapping Temperatures to a NASTRAN Model

What will be learned:

- Importing a finite element mesh created outside of Thermal Desktop.
- Mapping temperatures from a Thermal Desktop model to another type of model.

Prerequisites:

- 1.1 Setting Up a Template Drawing

This tutorial maps temperatures from a coarse model of a plate with two holes in it to a different mesh of the same model that has much finer detail. This situation is very common when the stress analyst has a very detailed model for a part, but the thermal model must be made coarse so as to get the part integrated into the entire thermal model.

CRTech would like to give a special thanks to Jim Braley for providing the sample NASTRAN models for this tutorial.

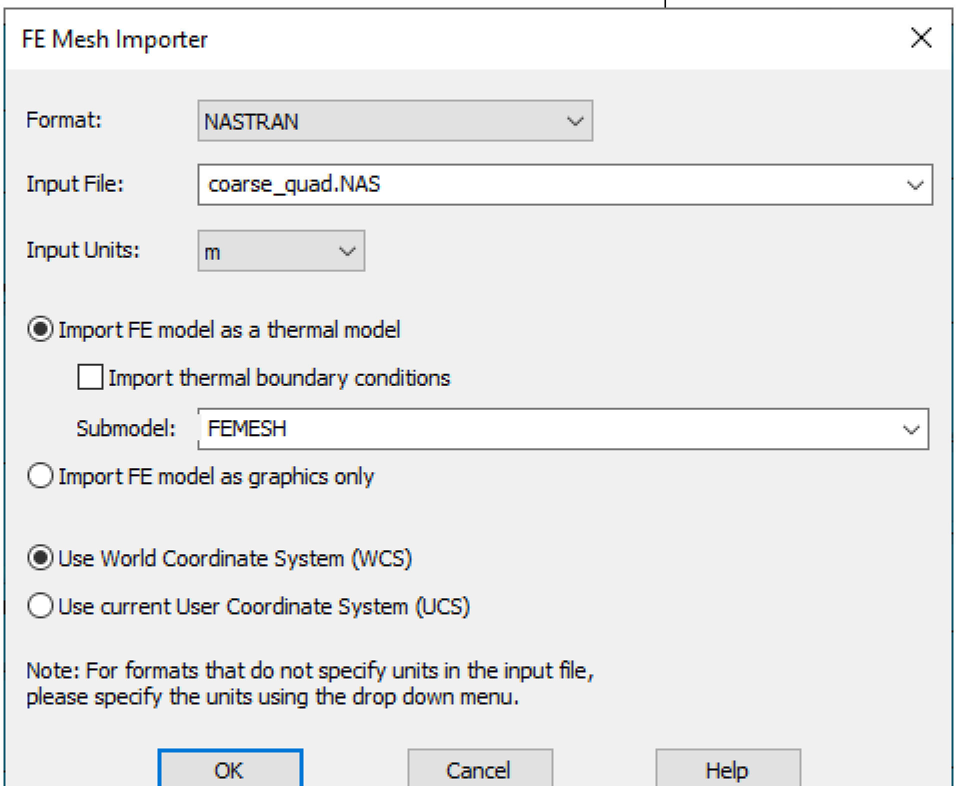
Mapping Example	
1. Copy the template `thermal.dwg` file created in the first tutorial to the `\Tutorials\Thermal Desktop - legacy\mappingExample` directory. **Note**: Be sure to hold the <**Ctrl**> key down if dragging the template file icon to the new directory so that the file is copied, rather than moved. In addition to the copied template drawing, there are two existing files in the `mappingExample` folder: ▪ `coarse_quad.nas` ▪ `fine_quad.nas` The two files will be imported into the model during the exercise. 2. Rename the copied template file to `coarse`. 3. Start Thermal Desktop by double clicking on the `coarse` drawing file icon in the `mappingExample` directory.	
4. Select **View** > **3D Views** > **Top**. The UCS icon reflects the new orientation.	Change the view in the drawing area to the top view.

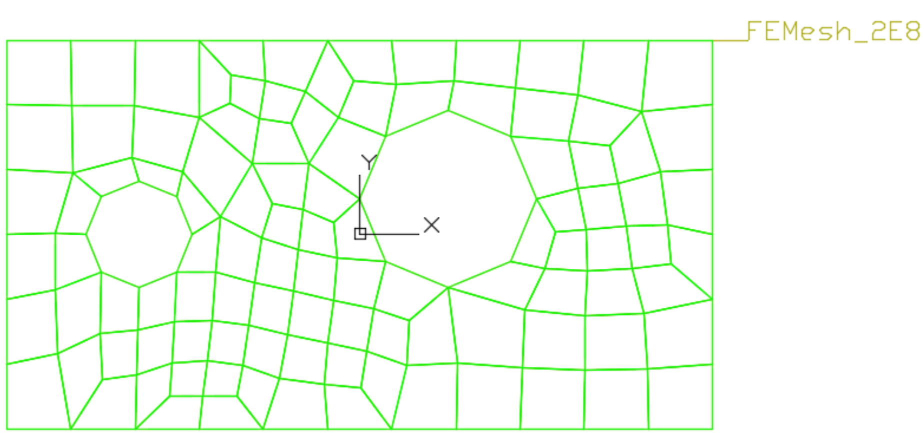
Mapping Example (Continued)

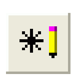
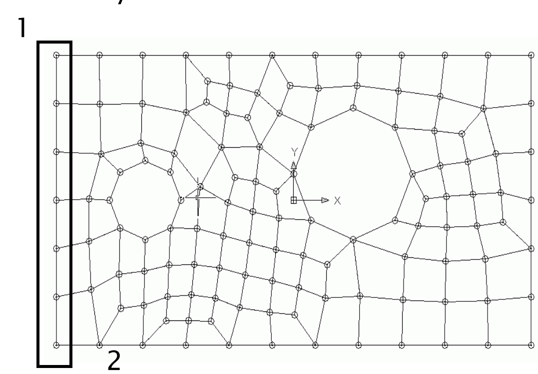
5. Select **Thermal** > **Import** > **Create FE Mesh Importer**.
6. The **FE Mesh Importer** dialog box appears.
7. If it is not already filled in, change the input file name to be **coarse_quad.NAS**.
8. Change the submodel to **MAIN**.
9. Select **OK** to close the dialog box.

The next step imports the coarse_quad.NAS file into the drawing. When the import process is complete, a model of a plate with two holes in it will be displayed.

The model should look similar to the example below.





Mapping Example (Continued)	
10. Select **Thermal** > **Preferences** 11. Select **Graphics Visibility** tab 12. Check the **TD/RC Nodes** box 13. Select **OK** to close the **User Preferences** dialog.	
14. Select the nodes on the left edge of the model by drawing a selection box from points 1 to 2, as shown in the example to the right. **Note**: In this case, selection order is not important in that only the nodes are to be selected rather than the nodes and any other objects that may be partially enclosed by the selection box. Remember that selecting objects from top to bottom will only select those items fully enclosed by the selection box whereas selecting objects from bottom to top will include items not fully enclosed by the selection box into the selection set. 15.  or **Thermal** > **Edit.** The **Node - Mulit Edit Mode** dialog box appears. (next page)	Some boundary conditions must be created so the model can be run and some gradients obtained. The nodes on the left edge of the model are to be designated as boundary nodes. 
16. Select **Override calculations by elements/surfaces.** The **Type** fields activate. 17. Select **boundary.** 18. Select **OK** to close the dialog box. A **Thermal Desktop/AutoCAD** dialog box appears asking for confirmation of the node changes. 19. Confirm the changes to close the dialog box. The node shapes change to show their new designation	

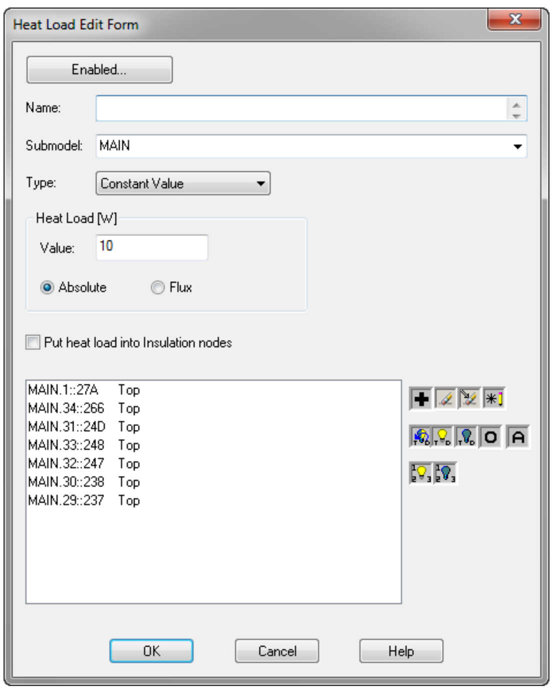
Mapping Example (Continued)

20. Select the nodes on the right edge of the model by drawing a selection box from points 3 to 4, as shown in the example to the right.

Note: As in the selection of the nodes on the left edge, selection order (top to bottom versus bottom to top) is not important in this case as only the nodes on the right edge are being selected in this step.

21.  or select **Thermal > FD/Fem Network > Heat Load on Nodes**.

    The **Heat Load Edit Form** dialog box appears.



22. Highlight the current value in the **Heat Load [W] Value** field and type **10**.

23. Select **OK** to close the dialog box.

A heat load of 10 watts is to be applied to the nodes on the right edge of the model.

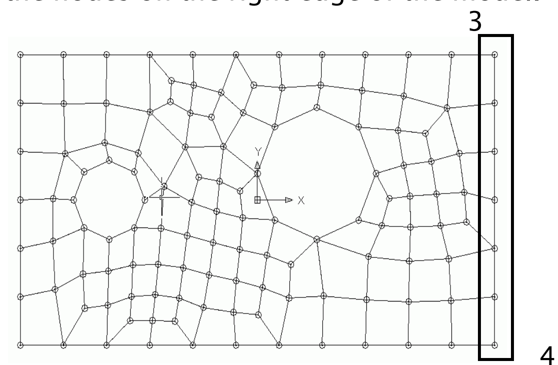
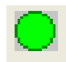


Figure 1-7: Selection Points

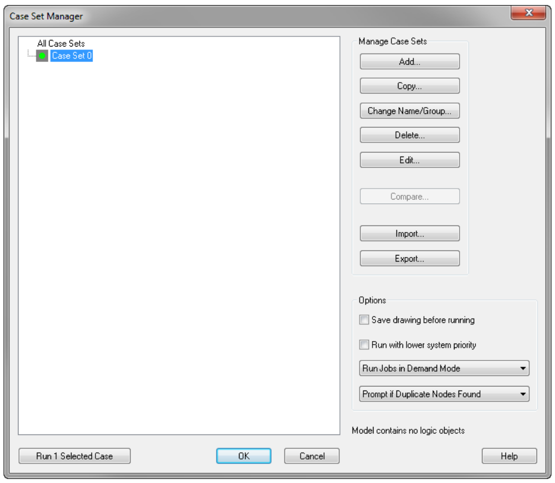
Note: Because of the top view of the model, once the heat load has been applied it appears the nodes change color from green to red. If the model is rotated to look at from another angle, small red arrows are displayed which give the illusion of the red nodes from the top view.

If the model is rotated back to a top view, the nodes appear white in color again, with small spots of red.

Mapping Example (Continued)

24.  or **Thermal** > **Case Set Manager**.

The **Case Set Manager** dialog box appears.



25. Click on **Edit.**

The **Case Set Information** dialog box appears.

Use the **Case Set Manager** to process a steady state solution on the problem.

The primary purpose of the **Case Set Manager** is to allow the user to set up different thermal analysis cases and to have the calculations made: from doing radiation calculations to creating and running the SINDA model to post-processing temperatures with the click of a single button. When the Run 1 Selected Case button is clicked, Thermal Desktop will first calculate the radiation conductors and heating rates for all of the tasks set up for the current Case Set. Nodes and conductors are then computed and output. A SINDA model is then built and run. And finally, the temperature results are displayed mapped onto the thermal model in color.

The user may set up different Case Sets to be steady state or transient analyses. Each Case Set may have different start and stop times for transient runs. The user can also have different SINDA Logic, property databases or aliases, or even different symbol values

26. Click on the **Calculations** tab.

Steady State in the **Solution Type** field is already selected.

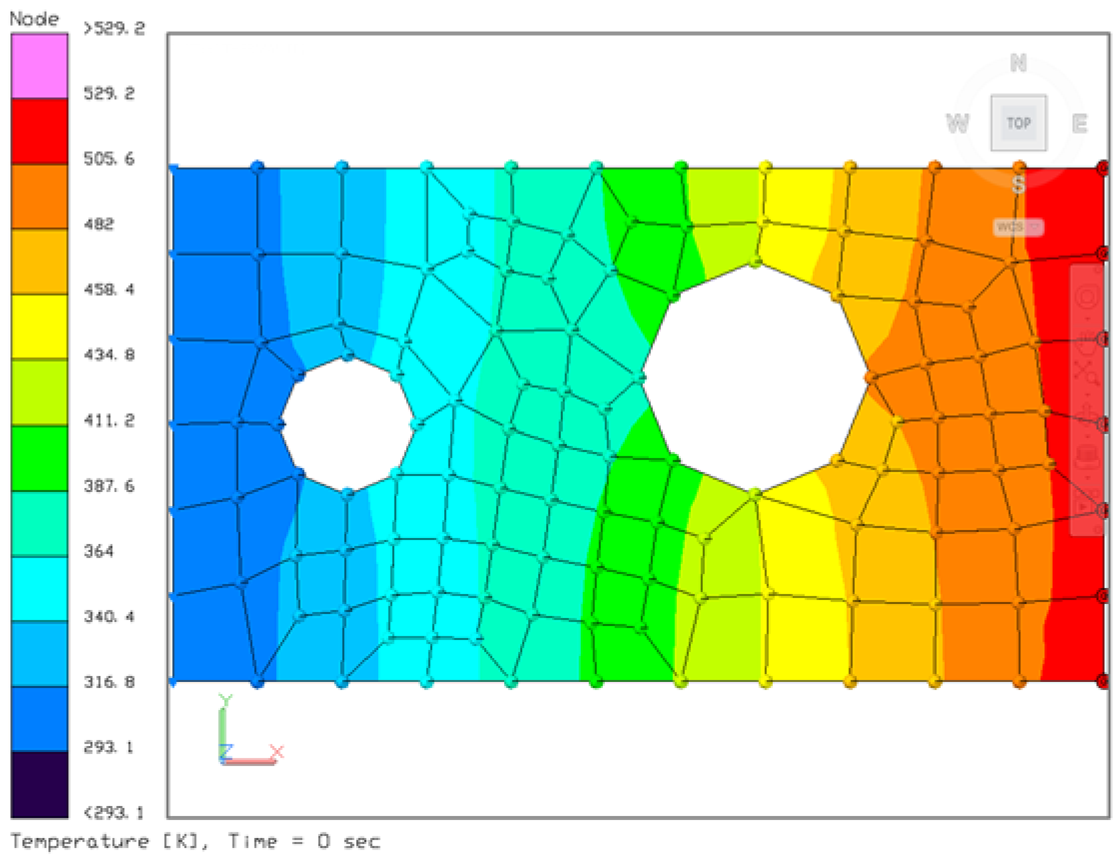
27. Select **OK** to close the dialog box and return to the **Case Set Manager** dialog box.

28. Click on **Run 1 Selected Case**.

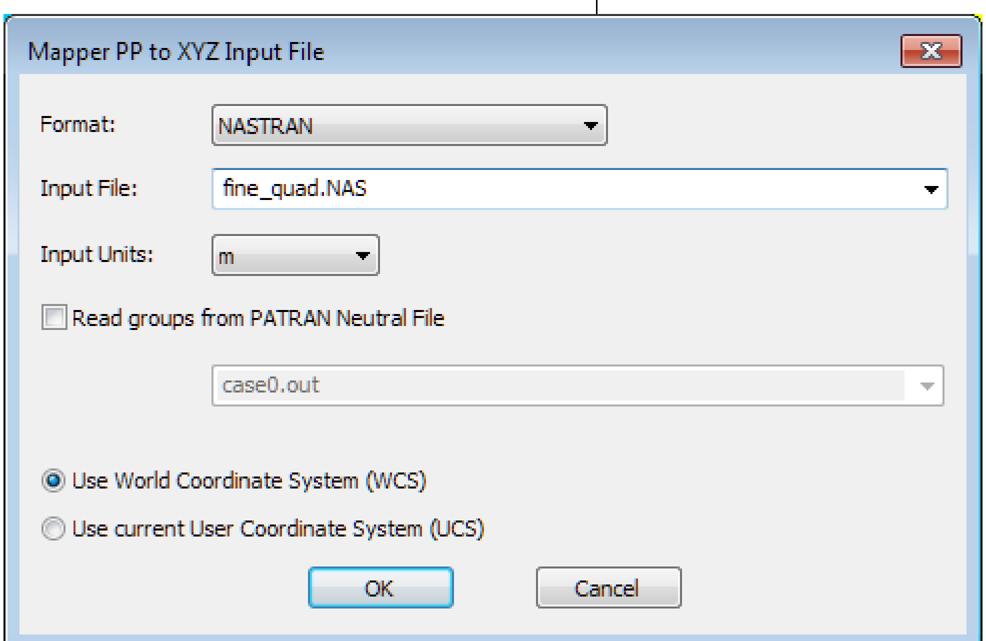
Mapping Example (Continued)

The solution is processed. When complete, the data displays on the screen along with a **SINDA/FLUINT Run Status** dialog box confirming successful completion of the run.

29. Click on **OK** to close the dialog box.



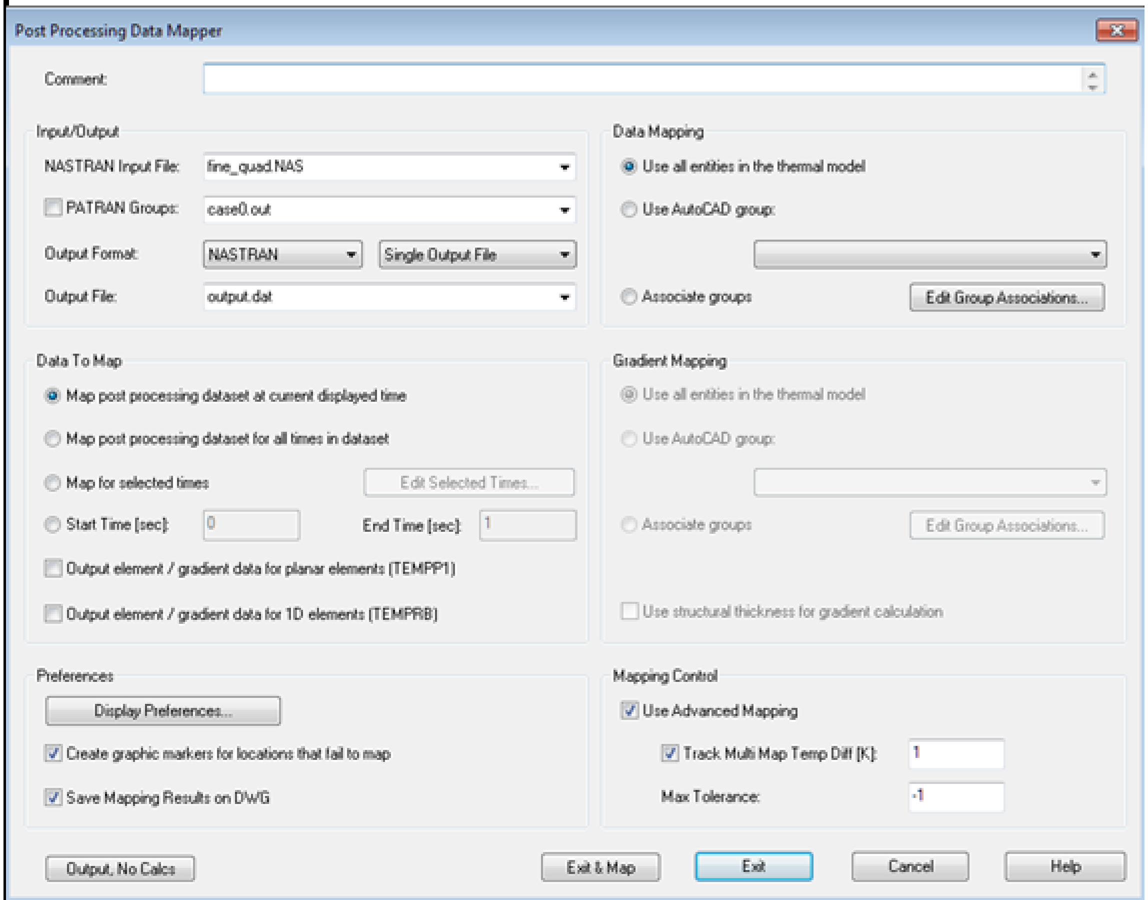
Mapping Example (Continued)	
30. Select **Thermal** > **Export** > **Post Processing Data Mapper**. The **Mapper PP to XYZ Input File** dialog box appears.	The stress analyst has added a more detailed model titled fine_quad.nas. and needs the temperatures for the nodes in this model in order to perform thermal stress calculations.
31. Choose **NASTRAN** in the **Format** field.	The Map Object allows the Thermal Desktop user to preview the stress model in side Thermal Desktop, align the two models, and map the postprocessed data to the stress model.
32. Choose **fine_quad.nas** in the **Input File** field drop down menu.	The available format options are NASTRAN, ANSYS, FEMAP, and I-deas. The user may choose to browse if the desired input file is not in the current directory.
33. Ensure that **Use World Coordinate System (WCS)** is selected	The user may choose to align the stress model to the World Coordinate System (WCS) or the User Coordinate System (UCS)



Mapping Example (Continued)

34. Select **OK** to close the dialog box.

    The **Post Processing Data Mapper** dialog box appears.



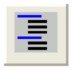
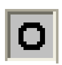
35. Highlight the current value in the **Output File** field and type **temps.out**.

36. Select **Exit & Map** to close the dialog box and map the data.

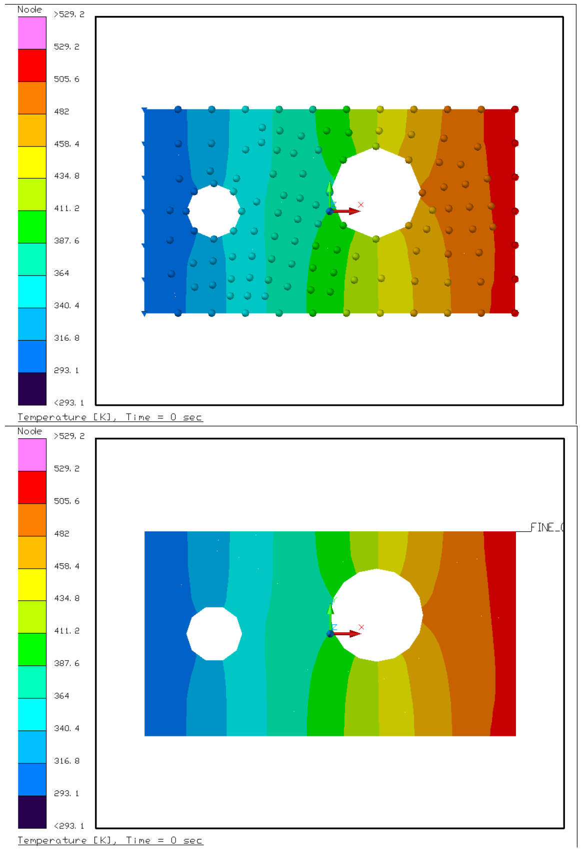
The Use Advanced Mapping option automatically finds the best mapping. If more than one object is found for a location, then a notice will be provided if the temperature difference is greater than the Track Multi Map Temp Dif value.

The user can choose to map the data immediately using Exit & Map, or may simply Exit to manipulate the mapper graphical object (e.g. align the mapper to the thermal model)

The file temps.out contains mapped temperatures in the format necessary for them to be included into the Stress model. The engineer must cut and paste these into the proper place in the NASTRAN model in order to perform the thermal stress calculations.

Mapping Example (Continued)	
37.  or **Thermal** > **Model Browser**. The **Model Browser** window appears on the left side of the screen. 38. Choose **List By** > **Mesh Displayers/PP Mappers/BCM/Cutting Planes** from the Model Browser menu bar. 39. Select **MapperPP** from the Mesh Display-ers tree. 40.  or **Display** > **Turn Visibility Off**. The mapper turns off and the coarse finite element model is visible. Note the shape of the contours and the holes. 41.  or **Display** > **Only** on the Model Browser menu bar. The finite element model has been turned off (notice the nodes are gone) and the mapper is turned back on. Note that the shape of the holes has been refined, but the contours are the same.	After mapping is complete, the mapped data is displayed on the mapper. Any locations in the external file (the stress model) that did not get mapped will be displayed as grey.

Mapping Example (Continued)



Finite element thermal model based on coarse mesh

Mapper based on fine mesh

Figure 1-7: Compare Mapper to

42. Select **File** > **Exit**.

Note: Thermal Desktop can also be exited using the Windows Close button (X) in the upper right corner of the screen.

A **Thermal Desktop/AutoCAD** dialog box appears asking to save the drawing changes.

43. Select **Yes**.