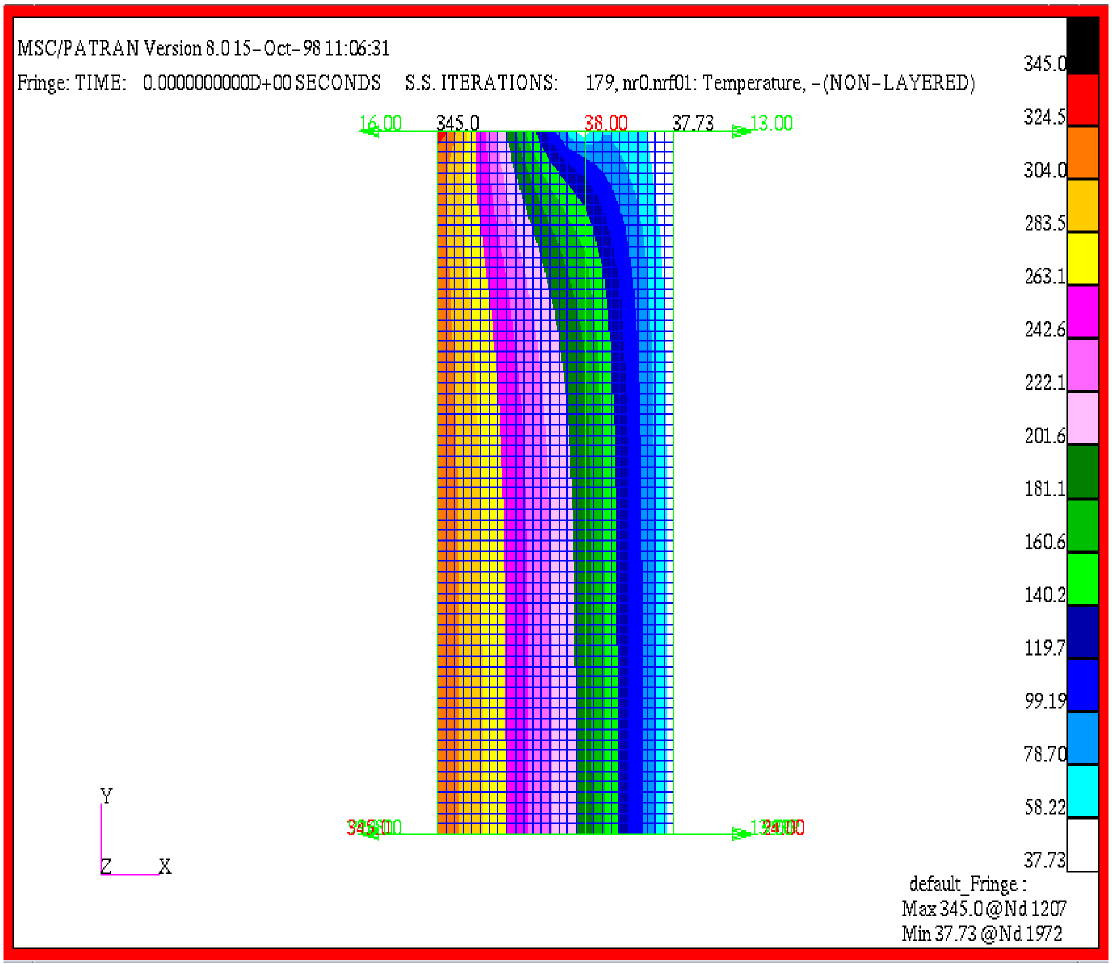
WORKSHOP 7

An Oven Window Design



Objective:

- Model a 2D planar slice of an oven window.
- Learn how to initiate and use **Utilities**.

Model Description:

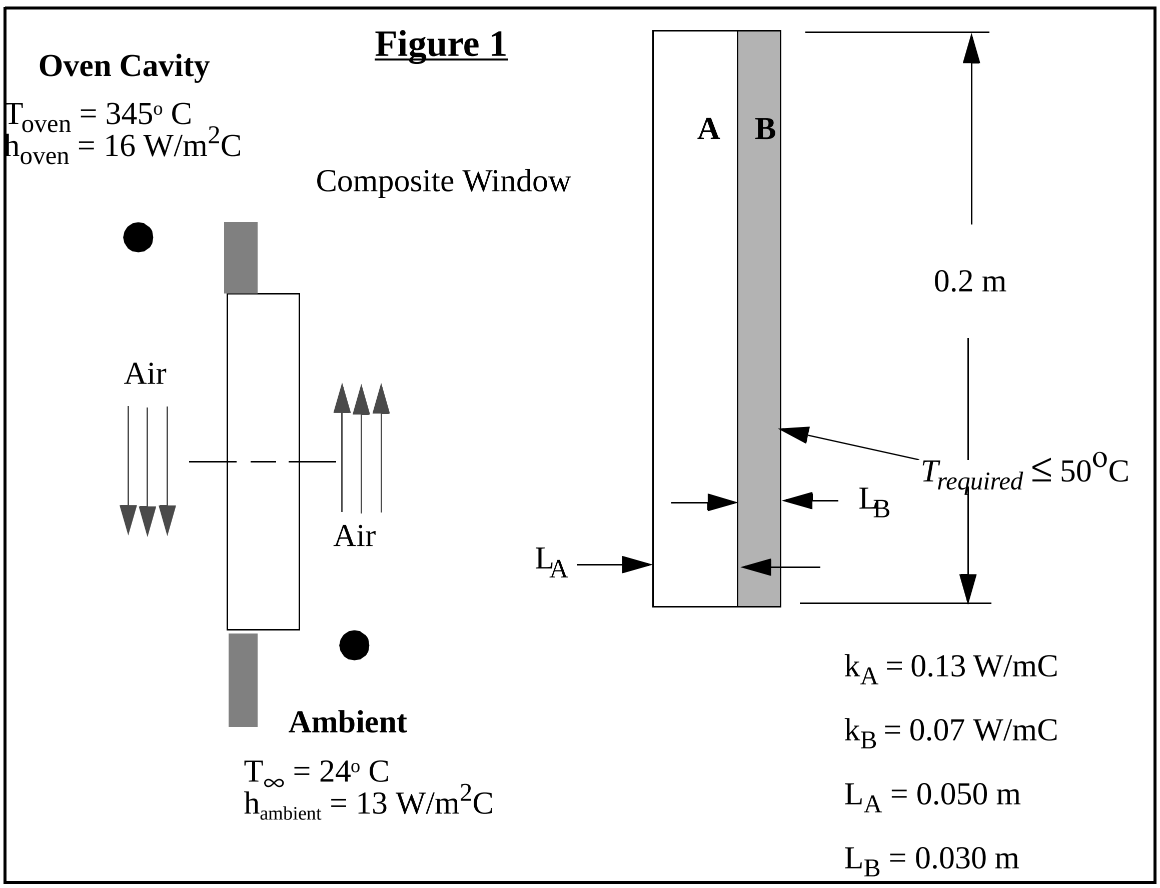
In this exercise you will model a 2D planar section of an oven window. You will learn how to initiate and use **Utilities** which facilitate this task.

A manufacturer of appliances is proposing a self cleaning oven design that uses a composite window separating the oven cavity from room air. The composite consists of two high temperature plastics (A & B) whose physical and thermal attributes are shown below. The combined convection/ linearized radiation heat transfer parameters for inside and outside of the oven are also shown. (Note: Radiation will be linearized and is include in the heat transfer coefficient). The design specification for safe operation requires an outside oven temperature of 50^{O}C or less.

The following assumptions can be made for the model:

- Steady-state conditions exist.
- The oven door can be modeled as a 2-dimensional slice.
- Contact resistance is negligible.
- Each plastic is homogeneous with constant properties.

Figure 1



Information on Utilities:

Utilities refer to a set of tools which facilitate the use of MSC.Patran. These tools are supplied with MSC.Patran. In version 9.0 they are located on each CD ROM.

Utilities are written or supplied by MSC.Software software developers, applications engineers, and anyone within MSC who has a good idea for improving MSC.Patran functionality. Sometimes **Utilities** are the vehicle for implementing an improvement which for organizational reasons will not be officially implemented within a reasonable release horizon.

Utilities are written in PCL, MSC.Patran Command Language. Since **Utilities** are developed from the personal initiative of individuals and not as part of the MSC.Software corporate software development strategy, they are not subjected to any formal quality assurance testing. Hence, they are supplied by MSC.Software as a courtesy but they are officially not supported by MSC.Software. Most **Utilities** are supplied with the authors name, an e-mail address, and telephone number. If you have a problem with a **Utilities** tool you may contact the author if ownership data is available. You may report suspected or identified problems with **Utilities** to the MSC.Patran support line but no obligation to fix the **Utilities** problem is incurred by MSC.Software. That being said, **Utilities** are generally reliable and quite handy. Most intermediate and advanced user of MSC/PATRAN install and use **Utilities**.

Load the MSC.Patran CD in the CD-ROM drive and mount the CD-ROM drive Installation instructions are listed in “Installing PCL Utilities and MSC.Software Institute Files on Unix”, p. 43 of “MSC.Patran Installation and Operations Guide”. Instructions for Windows NT are found on p. 65.

If the user has installed MSC.Patran with the “FULL” install option utilities are loaded automatically. If user selects “CUSTOM” installation, then PCL Utilities must be selected as an option under the MSC.Patran Core Applications.

When loaded (installed) **Utilities** are initiated by copying the **p3epilog.pcl** file from **<P3_HOME path>/shareware/msc/unsupported/utilities** (e.g., /patran/patran3/shareware/msc/unsupported/utilities/p3epilog.pcl), into a users home directory (for user-by-user access) or the **P3_HOME** directory (for a system wide access). Once the **p3epilog.pcl** file is in place **Utilities** is available as a pick on the *Menu Bar* after re-starting MSC.Patran.

Exercise Overview:

- Create a new database named **exercise_07.db.** Set *Approximate Maximum Model Dimension* to **0.20,** and the *Analysis Code* to **MSC/THERMAL**.
- Create two surfaces which define the oven window geometry.
- Mesh the surfaces with an **IsoMesh**, *Global Edge Length* of **0.003**.
- Create two fluid nodes **9998** and **9999** for the oven interior and ambient conditions respectively.
- **Equivalence** the nodes at the mating surface edges.
- Define the two material properties for the plastics.
- Apply element properties to the elements using the defined materials. These are **Thermal 2D** elements.
- Use the Fields Form to define the temperature distribution at the interior pane upper edge.
- Apply temperature and convection boundary conditions.
- Visualize and verify the convection LBC's using **Utilities/ Thermal/Thermal BC Display...**
- Prepare and submit the model for analysis specifying that it is a **2D Plane Geometry** model and that the **Weakly Nonlinear Solution** solver will be used for analysis.
- Read the results file and plot results.
- Check the results against the requirement of **50^{o}C**.
- **Quit** MSC.Patran.

Exercise Procedure:

Open a new database

1. Open a new database named **exercise_07.db**.

Within your window environment change directories to a convenient working directory. Run MSC.Patran by typing **p3** in your xterm window.

Next, select **File** from the *Menu Bar* and select **New …** from the drop-down menu. Assign the name **exercise_07.db** to the new database by clicking in the *New Database Name* box and entering **exercise_07**.

Select **OK** to create the new database.

File	
New ...	
New Database Name	**exercise_07**
OK	

MSC.Patran will open a Viewport and change various *Main Form* selections from a ghosted appearance to a bold format. When the New Model Preferences form appears on your screen, set the *Approximate Maximum Model Dimension* to **0.20,** and the *Analysis Code* to **MSC/THERMAL**. Select **OK** to close the New Model Preferences form.

Approximate Maximum Model Dimension:	**0.20**
Analysis Code	**MSC/THERMAL**
OK	

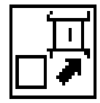
Create the oven window surfaces

2. Create two surfaces which define the oven window geometry.

Select the **Geometry** *Applications radio button*. Create a surface using the following *Action*, *Object*, and *Method*. Click in the appropriate list boxes to edit the default values and change them to values listed below.

◆ Geometry	
Create/Surface/XYZ	
Vector Coordinate List	**<0.050 0.2 0>**
Apply	

Turn on labels using the *Show Labels* icon.



Create the second surface from the same *Action/Object/Method* but change the *Vector Coordinate List* to **<0.030 0.2 0>**. Modify the *Origin Coordinates List* by clicking in the list box and selecting **Point 4** from the viewport.

◆ **Geometry**

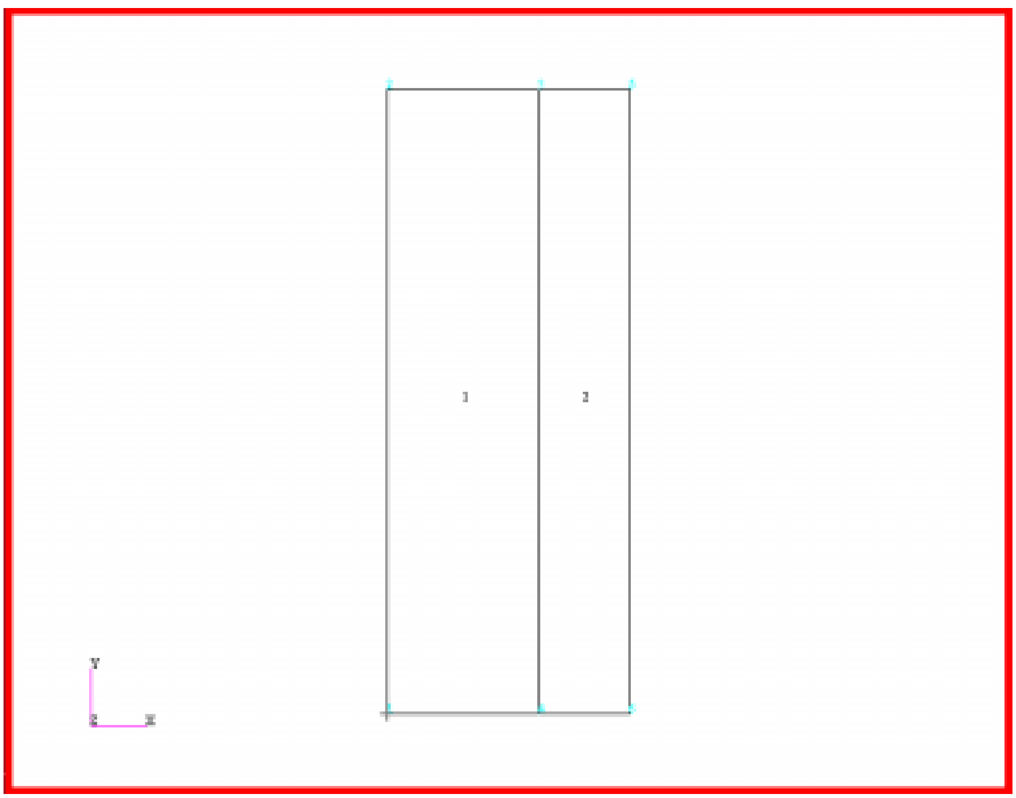
Create/Surface/XYZ

☐ **Auto Execute**

Vector Coordinate List	**<0.030 0.2 0>**
Origin Coordinates List	**<Select Point 4 from viewport>**

Apply

The model will appear as shown below.



IsoMesh the surfaces

3. Mesh surfaces with an IsoMesh, global edge length of **0.003**.

Select the **Finite Elements** *Applications radio button*. Set the *Action*, *Object*, and *Type* to **Create/Mesh/Surface**. Change the *Global Edge Length* to **0.003** and select both surfaces for inclusion in the *Surface List*.

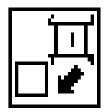
◆ Finite Elements

Create/Mesh/Surface

Global Edge Length	**0.003**
Surface List	**<drag a rectangle around both surfaces>**

Apply

Turn off labels using *Hide Labels* icon.



Create an ambient node

4. Create two fluid nodes **9998** and **9999** for the oven interior and ambient conditions respectively.

Using the Finite Elements form create a boundary nodes which are not associated with geometry. The node numbers are **9998** and **9999**. Locate the nodes at **[-0.03 0 0]** and **[0.11 0 0]**, to the left and right of model.

The spatial location of the boundary nodes is irrelevant to the analysis; but, these locations facilitate display and verification of LBC's.

◆ Finite Elements

Create/Node/Edit

Node ID List	**9998**
☐ **Associate with Geometry**	
Node Location List	**[-0.03 0 0]**
Apply	
Node Location List	**[0.11 0 0]**
Apply	

Increase the display size of nodes to facilitate the application of boundary condition. Use either **Display/Finite Elements** or the associated *Toolbar Node Size* icon to change the node size. The model should now appear as shown below.

Display

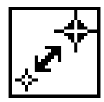
Finite Elements...

Node Size **6**

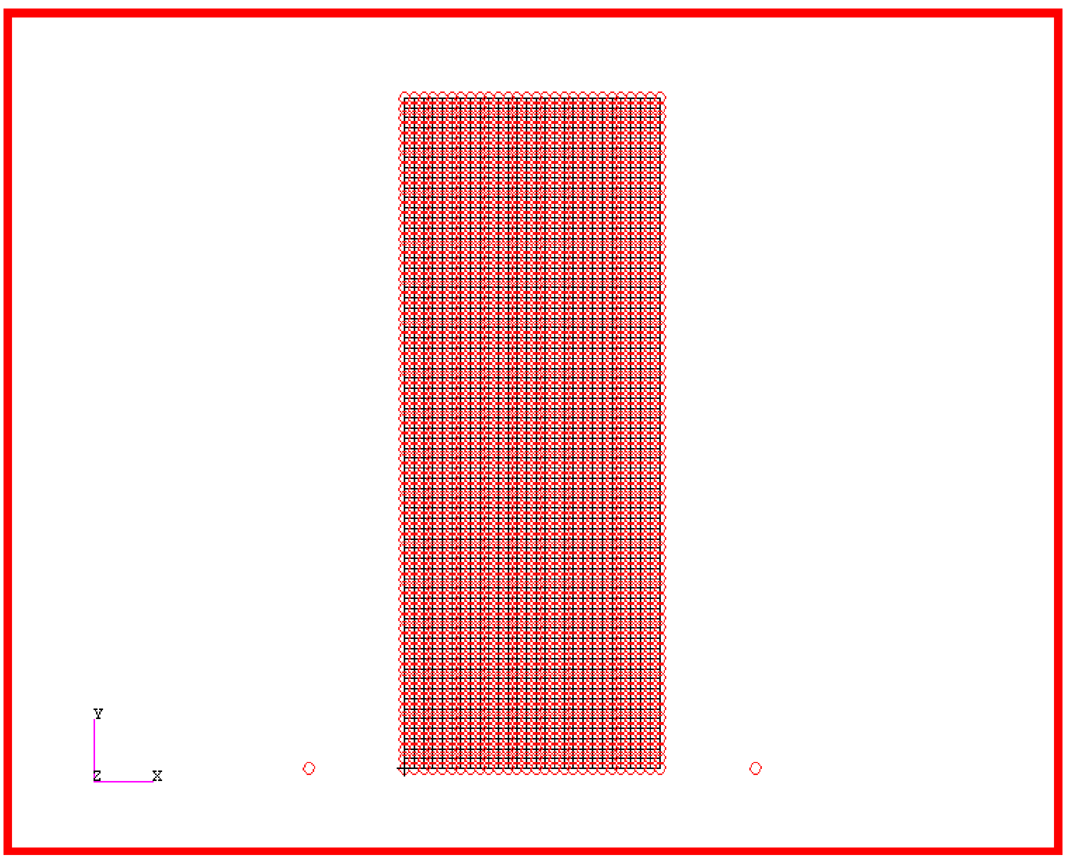
Apply

Cancel

or,



The display should now appear as shown below.



5. Equivalence the nodes at the mating surface edges.

Equivalence nodes

Using the Finite Elements form set the *Action*, *Object*, and *Method* to **Equivalence/All/Tolerance Cube** and select **Apply** to eliminate duplicate nodes created at geometric entity edges.

◆ **Finite Elements**

Equivalence/All/Tolerance Cube

Apply

6. Define the two material properties for the plastics.

Define two materials

Since this will be a steady state analysis, thermal conductivity is the only material property used in the solution. Thermal conductivity values are provided in Figure 1; however, the Input Options form also requires data for *Density* and *Specific Heat*. Enter a value of 1.0 in each of these fields.

The **Apply** button is selected from within the Input Options form. The form does not close upon hitting **Apply**. This is a convenient, if unintended, feature since one needs only to enter a new material name in *Material Name* and proceed with entering new material data in the Input Options form.

After each **Apply** the new material should appear in the *Existing Materials* list box on the Materials form.

◆ Materials

Create/Isotropic/Manual Input

Material Name	**ka**

Input Properties...

Thermal Conductivity =	**0.130**
Density =	**1.0**
Specific Heat =	**1.0**

OK

Apply

Material Name	**kb**
Thermal Conductivity =	**0.07**
Density =	**1.0**
Specific Heat =	**1.0**

OK

Apply

7. Apply element properties to the elements using the defined materials. These are **Thermal 2D** elements.

Apply element properties

Select the **Properties** *Applications radio button*. Set the *Action*, *Dimension*, and *Type* to **Create/2D/Thermal 2D**. Enter *Property Set Name* **interior_pane**. Select the *Input Properties...* box. Click in the *Material Name* box and select **ka** from the *Material Property Sets* list box. Select **OK** to close the form.Click in the *Select Members* box and choose Surface 1 from the default viewport. Select **Add** then **Apply** in the Element Properties form to complete the element property definition.

◆ Properties

Create/2D/Thermal 2D

Property Set Name **interior_pane**

Input Properties...

Material Name **<select ka>**

OK

Select Members **<select Surface 1 in the viewport>**

Add

Apply

Perform the same steps for Surface 2 using, **exterior_pane**, for the *Property Set Name*, and select **kb** for the *Material Name* from the *Material Property Sets* list box.

◆ Properties

Create/2D/Thermal 2D

Property Set Name **exterior_pane**

Input Properties...

Material Name **<select kb>**

OK

Select Members **<select Surface 2 in the viewport>**

Add

Apply

8. Define the temperature distribution at the interior pane upper edge.

Create a spatial field

Select the **Fields** *Applications radio button*. Set the *Action, Object,* and *Method* to **Create/Spatial/Tabular Input**. Enter *Field Name* **edge_T**. Select *Input Data*... and enter 2 data pairs **0.0, 345.0** and **0.05, 38.0** via the *Input Scalar Data* box. Select **OK** and **Apply** to finish the definition.

◆ Fields

Create/Spatial/Tabular Input

Field Name:	**edge_T**

Input Data...

Data:	**<Select Cell X 1>**
Input Scalar Data:	**0.0 <CR>**
Input Scalar Data:	**0.05 <CR>**
Data:	**<Select Cell Value 1>**
Input Scalar Data:	**345.0 <CR>**
Input Scalar Data:	**38.0 <CR>**

OK

Apply

9. Apply temperature and convection boundary conditions.

Apply boundary conditions

Begin applying boundary conditions. Select the **Loads/BCs** *Applications radio button*. Create a fixed **345^{o}C** nodal boundary temperature named **oven**. In the Input Data form define the fixed temperature. In the Select Applications Region form pick node **9998** located to the left of the window.

◆ Loads/BCs

Create/Temperature/Nodal

Option:	**Fixed**
New Set Name	**oven**

Input Data...

Fixed Temperature | **345.0**

OK

Select Application Region...

Geometry Filter | ◆ **FEM**

Select Nodes | **<select node 9998>**

Add

OK

Apply

Repeat the steps for a fixed **24^{o}C** boundary temperature named **ambient**. In the Select Applications Region form pick node **9999** located to the right of the oven window.

New Set Name | **ambient**

Input Data...

Fixed Temperature | **24.0**

OK

Select Application Region...

Select Nodes | **<select node 9999>**

Add

OK

Apply

Repeat steps for fixed edge temperature distribution using spatial field **edge_T**. Apply the distribution to the upper Geometry edge of the interior pane.

New Set Name | **edge**

Input Data...

Select Spatial Field...

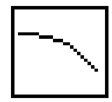
Fixed Temperature | **<Select spatial field edge_T>**

Close

OK

Select Application Region...

Geometry Filter	◆ **Geometry**
Select Menu	**curve or edge icon**



Select Geometric Entities	**<Select upper edge of interior pane, Surface 1.2>**

Add

OK

Apply

Create the heat transfer coefficient boundary conditions with the **Template, Convection** option, set name **oven_convection**, and a heat transfer coefficient of **16.0 W/^{o}C-m^{2}**. Apply the boundary condition to the left most oven window surface(edge) as shown in Figure1 with fluid node **9998**.

◆ **Loads/BCs**

Create/Convection/Element Uniform

Option:	**Template, Convection**
New Set Name	**oven_convection**
Target Element Type	**2D**

Input Data...

In the Input Data form provide the heat transfer coefficient and fluid node.

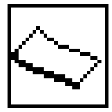
Convection Coefficient	**16.0**
Fluid Node ID	**<Select Node 9998>**

OK

Select Application Region...

In the Select Applications Region form select the left facing surface (edge) of the oven window. Switch to the **Select an Edge** icon in the *Select Menu* form. When selecting the edges the edge chosen will be highlighted.

Geometry Filter	◆**Geometry**
Select Menu	**Edge icon**



Select Surfaces or Edges	**<select the left facing oven window edge as shown in Figure 1>**

Add

OK

Apply

Repeat these steps for a New Set Name **air_convection** with a heat transfer coefficient of **13.0 W/^{o}C-m^{2}** applied to the right most oven window surface(edge) as shown in Figure 1 with fluid node **9999**.

New Set Name	**air_convection**
Target Element Type	**2D**

Input Data...

Convection Coefficient	**13.0**
Fluid Node ID	**<Select Node 9999>**

OK

Select Application Region...

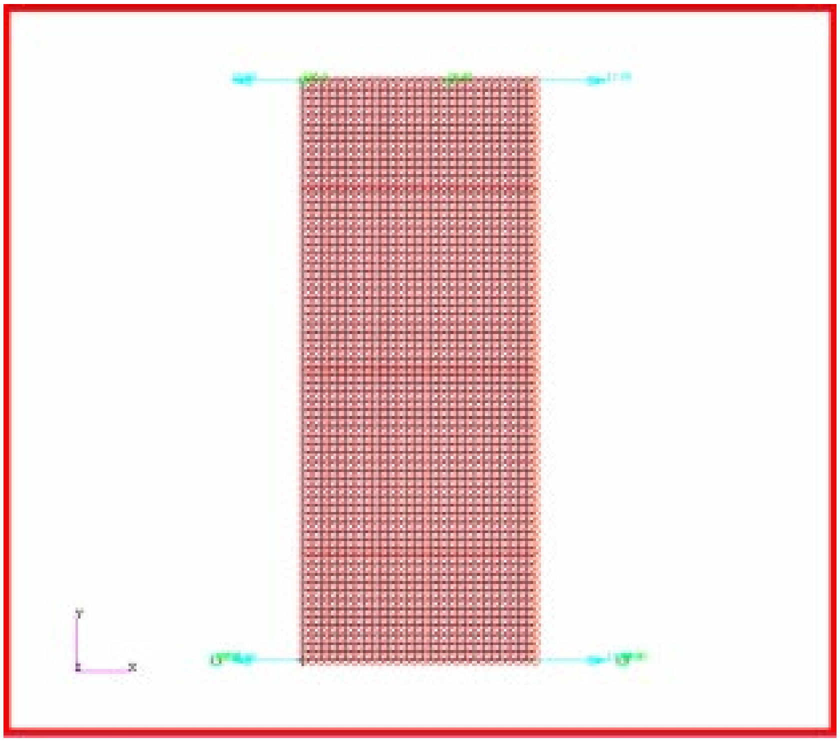
Select Surfaces or Edges	**<select the right facing oven window edge as shown in Figure 1>**

Add

OK

Apply

With boundary conditions applied the model should appear as shown below.



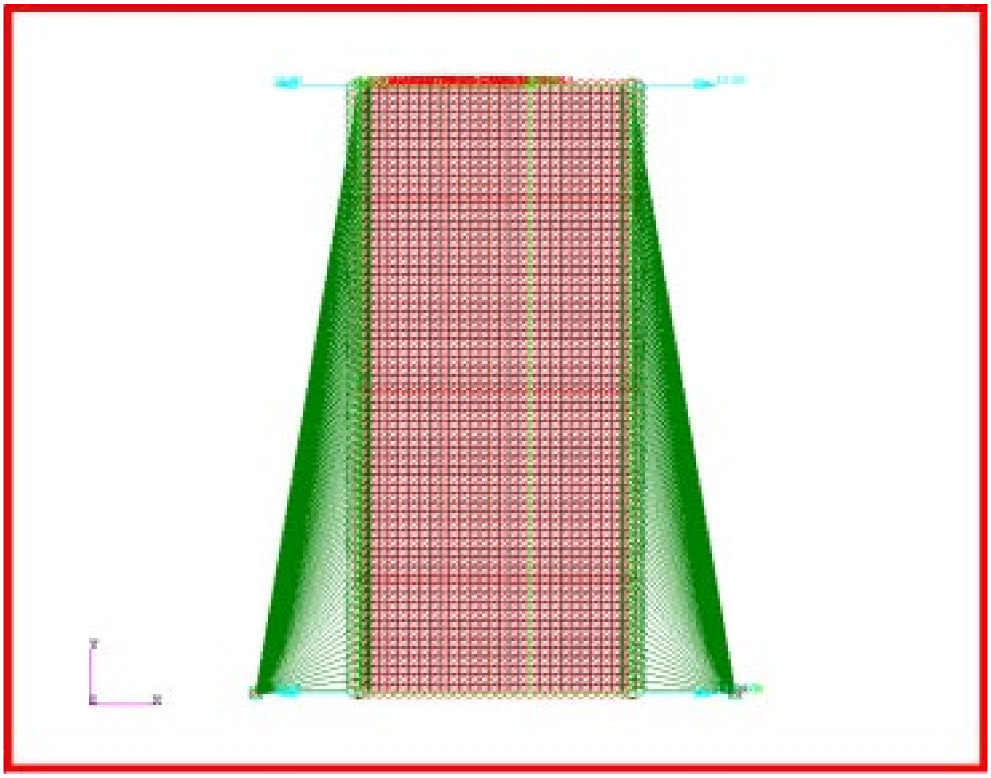
Use Shareware to verify LBC's

10. Visualize and verify the convection LBC's using **Utilities/ Thermal Tools/Thermal BC Display...**

Shareware contains various utilities for facilitating model creation and LBC's verification. Verify your convective coupling by drawing a vector from the centroid of each element to the associated fluid node using **Utilities/ Thermal/Thermal BC Display...**

Utilities
Thermal
Thermal BC Display...
OK
Apply

The model should appear as shown below.

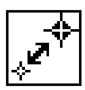


Use **Clear** and **Close** in the Thermal BC's form to revert to a normal display.

Clear

Close

Reduce the node size using the *Node Size* icon.



11. Prepare and submit the model for analysis.

Prepare and run analysis

Select the **Analysis** *Applications radio button* to prepare the analysis. Select the parameter forms reviewing and changing the settings as shown below. The analysis is submitted by selecting **Apply** in the Analysis form.

◆ Analysis

Analyze/Full Model/Full Run

Translation Parameters...

Model Dimensionality	◆ **2D Plane Geometry, XY Co-ordinates (unit Thickness in Z)**

OK

Solution Parameters...

Calculation Temperature Scale	◆ **Celsius**
Solver Option	**1, Weakly Nonlinear Solution**

OK

Output Requests...

Units Scale for Output Temperatures	◆ **Celsius**

OK

Apply

Read and plot results

12. Read results file and plot results.

From within MSC.Patran the only indication that the analysis has successfully finished is the existence of an **nrX.nrf.01** results file in a subdirectory one level below your working directory.

P3 was initiated from a working directory which contained the **exercise_07.db** database. Applying the analysis created a new subdirectory with the same name as the *Job Name*; **exercise_07**/. By using **Read Result** in the Analysis form and Selecting **Results File...** you can filter down to the *Job Name* subdirectory and check for the existence of a results file.

◆ **Analysis**

Read Results/Result Entities

Select Results File...

Directories	**<path>/exercise_07**

Filter

Available Files	**nr0.nrf.01**

OK

Select Rslt Template File...

Files	**pthermal_1_nodal.res_tmpl**

OK

Apply

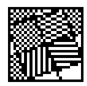
To plot the results to posted FEM use the **Results** *Application radio button*.

◆ Results

Create/Quick Plot

Select Result Case	**TIME: 0.0000000000D+00 S...**
Select Fringe Results	**Temperature,**

Select the *Fringe Attributes* icon.



Display:	**Element Edges**
Label Style...	
Label Format:	**Fixed**
Significant figures	**4 <use slider bar>**

OK

Apply

The model should now appear as shown on the front panel of this exercise.

What is the exterior temperature of the oven window? Is it at or below the required maximum of **50^{o}C**?

13. **Quit** MSC.Patran

Quit MSC.Patran

To stop MSC.Patran select **File** on the *Menu Bar* and select **Quit** from the drop-down menu.