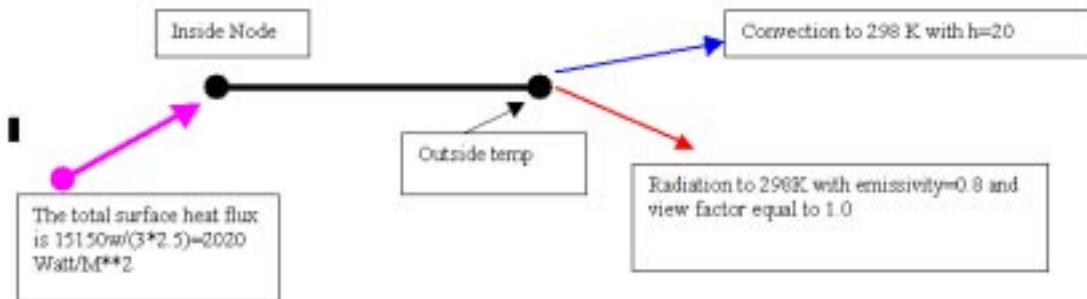


## APPENDIX B

# *Analytical Solution for a Simple Radiation to Space Problem*



### Objectives:

- Create a new database.
- Create the solid model
- Assign the thermal loads
- Submit the model for analysis



## Suggested Exercise Steps:

- Create a new database and name it **furnance.db**.
- Create a surface of the wall and then extrude it to generate a solid model.
- Generate mesh seeds on the edges of the solid.
- Create a finite element model using the available mesh seeds.
- Define material and element properties.
- Apply the convection conditions to front surface of the furnance wall.
- Apply radiation and heat flux to the back side of the model.
- Submit the model to MSC.Nastran for analysis.
- Review results.



## Exercise Procedure:

1. Open a new database called **furnance.db**.

### File/New...

*New Database Name*

furnance

OK

In the New Model Preferences form set the following:.

### New Model Preference

*Tolerance*

◆ Default

*Analysis Code:*

MSC/NASTRAN

*Analysis Type:*

Thermal

OK

Change to a front view by selecting the **Iso1 View** button on the toolbar.



Iso1 View

Whenever possible click  **Auto Execute** (turn off).

2. Create the surfaces of the furnance wall.

### ◆ Geometry

*Action:*

Create

*Object:*

Surface

*Method:*

XYZ

*Reference Coordinate Frame*

Coord 0

*Vector Coordinates List*

<3, 2.5, 0>

*Origin Coordinates List*

[0, 0, 0]

Apply

---

Extrude the surface to create a solid wall with thickness of 0.15

◆ **Geometry**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Solid"/>
<i>Method:</i>	<input type="text" value="Extrude"/>
<i>Reference Coordinate Frame</i>	<input type="text" value="Coord 0"/>
<i>Translation Vector</i>	<input type="text" value="&lt;0, 0, 0.15&gt;"/>
<i>Surface List</i>	<input type="text" value="[Surface 1"/>

3. Create a finite element model

Generate two mesh seeds on the edge of the wall.:

◆ **Finite Element**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Mesh Seed"/>
<i>Method:</i>	<input type="text" value="Uniform"/>
<b>Element Edge Length Data</b>	◆ <b>Number of Elements</b>
<i>Number =</i>	<input type="text" value="2"/>

Choose the lower right hand curve

<i>Curve List</i>	<input type="text" value="Solid 1.2.1"/>
-------------------	--

Create mesh seed on the two longer edges of the solid

<b>Element Edge Length Data</b>	◆ <b>Element Length (L)</b>
<i>Number =</i>	<input type="text" value="0.2"/>
<i>Curve List</i>	<input type="text" value="Solid 1.4.3 1.1.2"/>

Create finite elements using the mesh seeds generated in the previous steps.

◆ **Finite Element**

<i>Action:</i>	<b>Create</b>
<i>Object:</i>	<b>Mesh</b>
<i>Method:</i>	<b>Solid</b>
<i>Surface List</i>	<b>Solid 1</b>
<b>Apply</b>	

4. Define a material using the specified thermal conductivity.

◆ **Materials**

<i>Action:</i>	<b>Create</b>
<i>Object:</i>	<b>Isotropic</b>
<i>Method:</i>	<b>Manual Input</b>
<i>Material Name</i>	<b>mat</b>
<b>Input Properties</b>	
<i>Thermal Conductivity =</i>	<b>1.2</b>
<b>OK</b>	
<b>Apply</b>	

5. Apply the material properties to the furnace wall

◆ **Properties**

<i>Action:</i>	<b>Create</b>
<i>Object:</i>	<b>3D</b>
<i>Type:</i>	<b>Solid</b>
<i>Property Set Name</i>	<b>solid</b>
<b>Input Properties</b>	
<i>Material Name</i>	<b>mat</b>
<b>OK</b>	

---

Select Members

Solid 1

Add

Apply

6. Assign the thermal loads to the wall

The front surfaces of the solid is undergoing convection.

◆ Loads/BCs

Action:

Create

Object:

Convection

Type:

Element Uniform

New Set Name

convection

Target Element Type:

3D

Input Data

Convection Coef

20

Ambient Temperature

298

OK

Select Application Region

Geometry Filter

◆ Geometry

Choose the front faces of solid

Select Surfaces or Edges

Surface 1.6

Add

OK

Apply

Heat flux and radiation is being applied to the back surface.

◆ Loads/BCs

Action:

Create

Object:

Applied Heat

Type:

Element Uniform

<i>New Set Name</i>	<input type="text" value="heat_flux"/>
<i>Target Element Type:</i>	<input type="text" value="3D"/>
<b>Input Data</b>	
<i>Form Type:</i>	<input type="text" value="Basic"/>
<i>Surface Option:</i>	<input type="text" value="Bottom"/>
<i>Bottom Surf Heat Flux</i>	<input type="text" value="2020"/>
<b>OK</b>	

**Select Application Region**

<i>Geometry Filter</i>	◆ <b>Geometry</b>
<i>Select Solid Faces</i>	<input type="text" value="Solid 1.5"/>
<b>Add</b>	
<b>OK</b>	
<b>Apply</b>	

Specify the parameters for the radiation on the back surface.

◆ **Loads/BCs**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Radiation"/>
<i>Type:</i>	<input type="text" value="Element Uniform"/>
<i>New Set Name</i>	<input type="text" value="radiation"/>
<i>Target Element Type:</i>	<input type="text" value="3D"/>
<b>Input Data</b>	
<i>Top Surf Emissivity</i>	<input type="text" value="0.8"/>
<i>Top Surf Absorptivity</i>	<input type="text" value="0.8"/>
<i>Ambient Temperature</i>	<input type="text" value="298"/>
<i>View Factor</i>	<input type="text" value="1"/>
<b>OK</b>	

**Select Application Region**

---

*Geometry Filter*

*Select Solid Faces*

**Add**

**OK**

**Apply**

◆ **Geometry**

**Solid 1.6**

7. Click on the **Analysis** radio button on the **Top Menu Bar** and complete the entries as shown here:

◆ **Analysis**

*Action:*

**Analyze**

*Object:*

**Entire Model**

*Type:*

**Analysis Deck**

**Translation Parameters**

*Data Output:*

**XDB and Print**

**OK**

**Solution Type**

*Solution Type:*

◆ **STEADY STATE ANALYSIS**

**Solution Parameters**

*Default Init Temperature*

**300**

**Radiation Parameters**

*Stefan-Boltzmann Constant:*

**5.67e-8**

**OK**

**OK**

**OK**

**Apply**

An MSC.Nastran input file called **furnance.bdf** will be generated. This process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green.

## Submitting the Input File for Analysis:

8. Submit the input file to MSC.Nastran for analysis.
  - 8a. To submit the MSC.Patran **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran furnance.bdf scr=yes**. Monitor the run using the UNIX **ps** command.
  - 8b. To submit the MSC.Nastran **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran furnance scr=yes**. Monitor the run using the UNIX **ps** command.
9. When the run is completed, edit the **furnance.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing **WARNING** messages indicate modeling errors.

---

10. **MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.**

11. Proceed with the Reverse Translation process, that is, attaching the **furnance.xdb** results file into MSC.Patran. To do this, return to the **Analysis** form and proceed as follows:

◆ **Analysis**

<i>Action:</i>	<b>Attach XDB</b>
<i>Object:</i>	<b>Result Entities</b>
<i>Method:</i>	<b>Local</b>
<b>Select Results File</b>	
<i>Select Results File</i>	<b>furnance</b>
<b>OK</b>	
<b>Apply</b>	

12. When the translation is complete and the Heartbeat turns green, bring up the **Results** form.

Choose the **Default** result case, and plot the result by selecting **Temperature** in the Select Fringe Result.

◆ **Results**

<i>Action:</i>	<b>Create</b>
<i>Object:</i>	<b>Quick Plot</b>
<i>Select Result Cases</i>	<b>Default</b>
<i>Select Fringe Result</i>	<b>Temperature</b>
<b>Apply</b>	