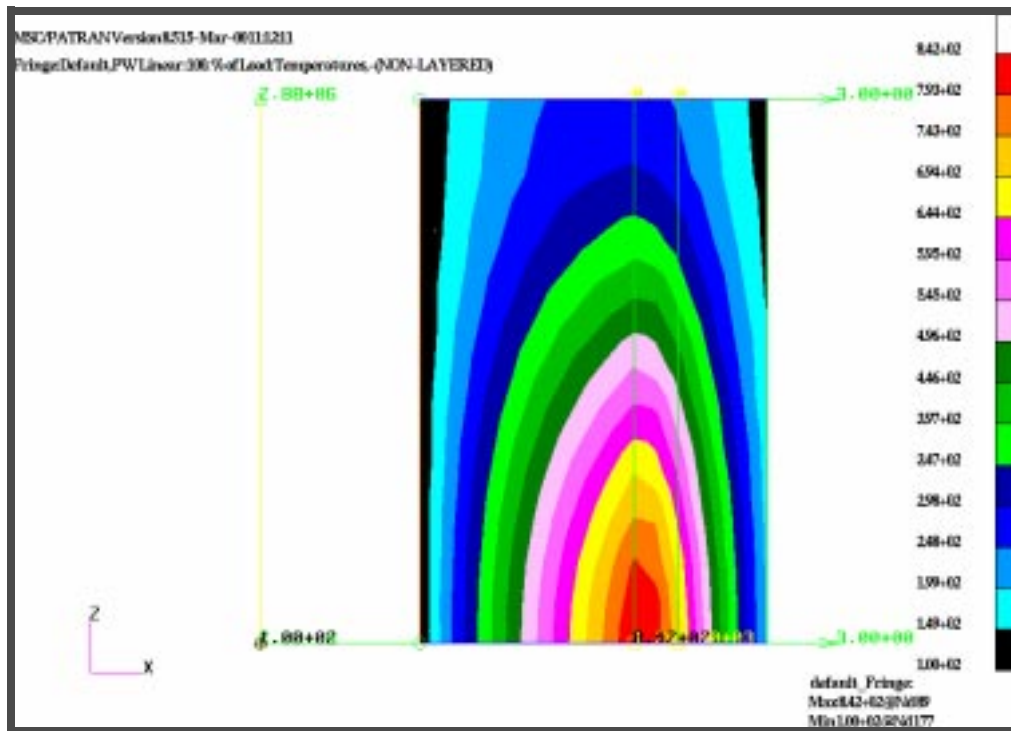


WORKSHOP 7

Axisymmetric Flow in a Pipe





Model Description:

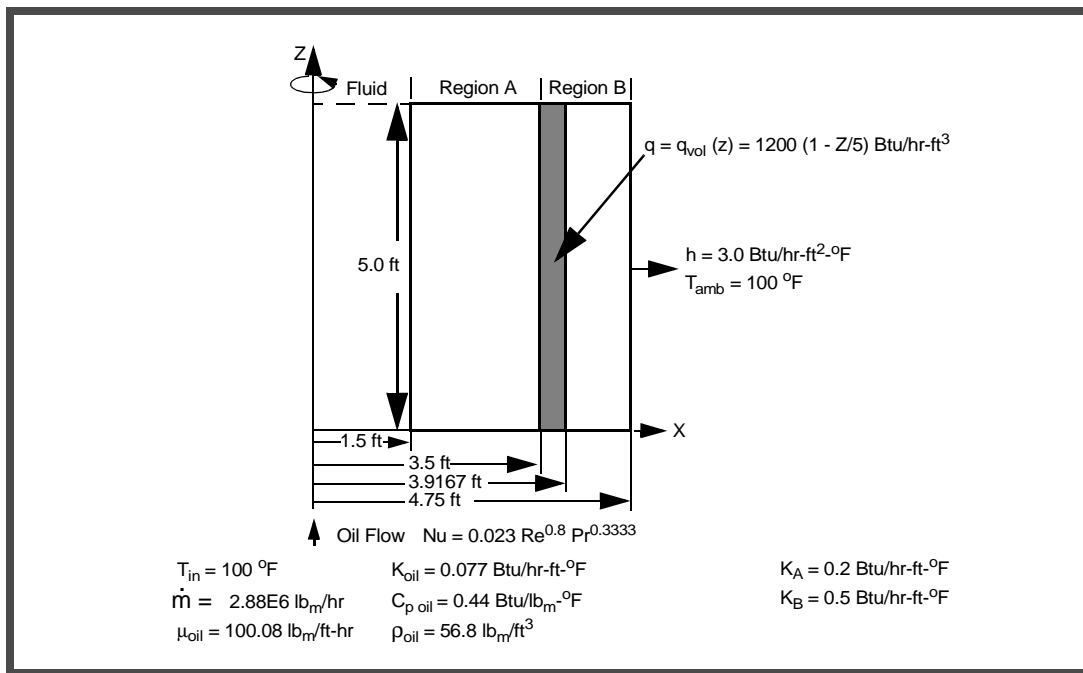
In this example we will analyze an axisymmetric structure for its temperature distribution. We will use the MSC.NASTRAN CTRIAX6 axisymmetric element (in its 3 node configuration) as the heat conduction element.

The basic geometry is detailed in the figure above. A section of pipe consisting of composite materials is divided into two different material regions. Region A is from radius 1.5 feet to 3.5 feet. Region B is from radius 3.5 feet to 4.75 feet. The overall pipe section is 5.0 feet long with an inside diameter of 3 feet and an outside diameter of 9.5 feet.

Oil flows through the interior with an inlet temperature of 100 °F and a mass flow rate of 2.88E6 lb_m/hr. The forced convection heat transfer coefficient between the oil and wall is calculated by MSC.NASTRAN using the following relationship:

$Nu = 0.023 Re^{0.8} Pr^{0.3333}$. Thermal conductivity properties for Region A and Region B are 0.2 and 0.5 Btu/hr-ft-°F. Volumetric internal heat generation occurs in the subregion of Region B (Specifically from radius 3.5 feet to 3.9167 feet), and varies based on Z location. The heat generation is $1200 * (1-Z/5)$ Btu/hr-ft³, where Z is given in units of feet. Free convection to an ambient temperature of 100 °F is applied to the exterior surface of the structure through a heat transfer coefficient of 3.0 Btu/hr-ft²-°F.

Figure 7.1





Suggested Exercise Steps:

- Create a new database called **ex7.db**
- Using thermal analysis, create a geometry representing a pipe divided up into two region.
- Perform meshing on the Fluid Curve and Pipe Surfaces using one way bias mesh seed.
- Mesh the rest of the solid.
- Merge all coincident nodes using Equivalence action in patran.
- Define all material properties accordingly.
- Using 2D Axisym Solid to define the element's properties and 1D solid to represent the Flow Tube.



Exercise Procedure:

1. Open a new database. Name it **ex7.db**

File/New...

New Database Name:

ex7

OK

The viewport (PATRAN's graphics window) will appear along with a *New Model Preference* form. The *New Model Preference* sets all the code specific forms and options inside MSC.PATRAN.

In the *New Model Preference* form set the *Analysis Code* to **MSC.Nastran**

Tolerance:

◆ **Based on Model**

Analysis Code:

MSC/NASTRAN

Analysis Type:

Thermal

OK

2. Create the Geometry.

◆ **Geometry**

Action:

Create

Object:

Curve

Method:

XYZ

Vector Coordinates List:

<0 0 5>

Origin Coordinates List:

[0 0 0]

Click on the **Bottom View** icon for working with axisymmetric geometries.



Bottom View

Apply

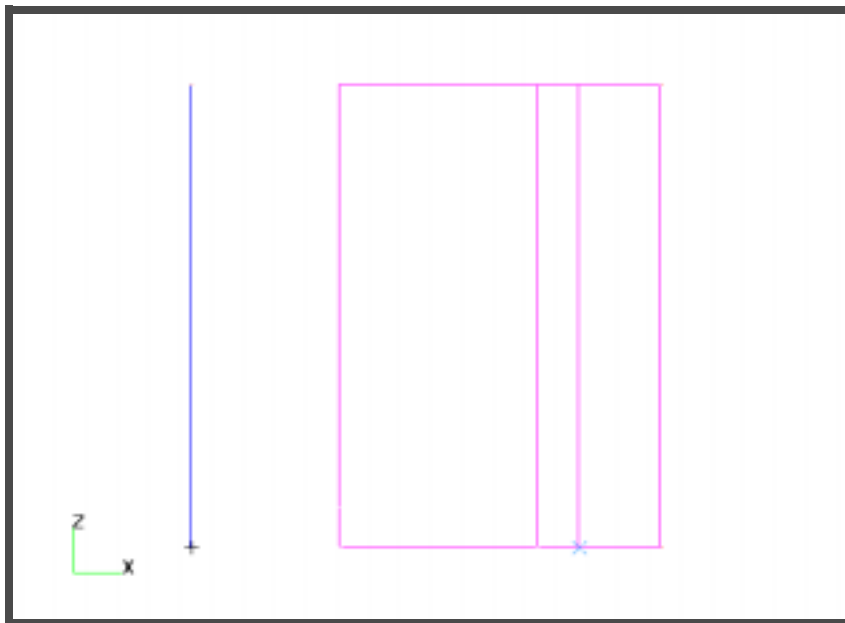
◆ **Geometry**

Action:
Object:
Method:
Surface ID List:
Vector Coordinates List:
Origin Coordinates List:

Surface ID List:
Vector Coordinates List:
Origin Coordinates List:

Surface ID List:
Vector Coordinates List:
Origin Coordinates List:

Your model should look like the following figure.



3. Mesh the Fluid Curve and Pipe Surfaces.

◆ **Finite Elements**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Mesh Seed"/>
<i>Type:</i>	<input type="text" value="One Way Bias"/>
<i>Number:</i>	<input type="text" value="10"/>
<i>L2/L1:</i>	<input type="text" value="2.0"/>
<i>Curve List:</i>	<input type="text" value="Curve 1 Surface 1.4 3.2"/>
<input type="button" value="Apply"/>	

◆ **Finite Elements**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Mesh"/>
<i>Type:</i>	<input type="text" value="Surface"/>
<i>Global Edge Length:</i>	<input type="text" value="0.25"/>
<i>Element Topology:</i>	<input type="text" value="Tria3"/>
<i>Surface List:</i>	<input type="text" value="Surface 1:3"/>
<input type="button" value="Apply"/>	

◆ **Finite Elements**

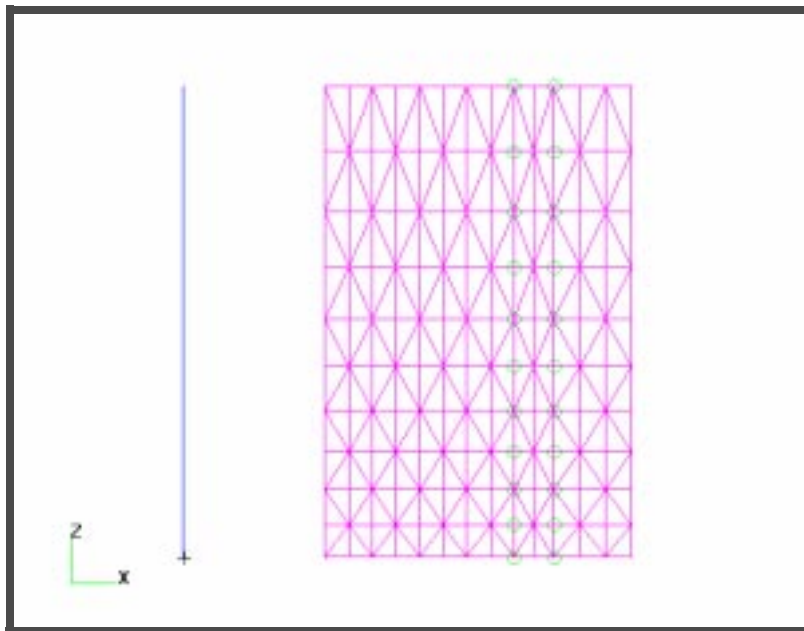
<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Mesh"/>
<i>Type:</i>	<input type="text" value="Curve"/>
<i>Global Edge Length:</i>	<input type="text" value="0.25"/>
<i>Element Topology:</i>	<input type="text" value="Bar2"/>
<i>Curve List:</i>	<input type="text" value="Curve 1"/>
<input type="button" value="Apply"/>	

4. Remove Coincident Nodes.

◆ **Finite Elements**

<i>Action:</i>	<input type="text" value="Equivalence"/>
<i>Object:</i>	<input type="text" value="All"/>
<i>Type:</i>	<input type="text" value="Tolerance Cube"/>
<i>Equivalencing Tolerance:</i>	<input type="text" value="0.005"/>

Your model should look like the following figure.



5. Specify Material Properties.

◆ **Materials**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Isotropic"/>
<i>Method:</i>	<input type="text" value="Manual Input"/>
<i>Material Name:</i>	<input type="text" value="mat_a"/>

<i>Constitutive Model:</i>	<input type="text" value="Solid Properties"/>
<i>Thermal Conductivity:</i>	<input type="text" value="0.2"/>

Apply

Cancel

Material Name:

mat_b

Input Properties...

Constitutive Model:

Solid Properties

Thermal Conductivity:

0.5

Apply

Cancel

Material Name:

oil

Input Properties...

Constitutive Model:

Fluid Properties

Thermal Conductivity:

0.077

Specific Heat:

0.44

Density:

56.8

Dynamic Viscosity:

100.08

Apply

Cancel

6. Define Element Properties.

◆ **Properties**

Action:

Create

Object:

2D

Type:

Axisym Solid

Property Set Name:

pipe_a

Input Properties...

Material Name:

m:mat_a

OK

Select Members:

Add

Apply

◆ Properties

Action:

Object:

Type:

Property Set Name:

Input Properties...

Material Name:

OK

Select Members:

Add

Apply

◆ Properties

Action:

Object:

Type:

Property Set Name:

Input Properties...

Material Name:

Hydraulic Diam. at Node:

OK

Select Members:

Add

Apply

Surface 1

Create

2D

Axisym Solid

pipe_b

m:mat_b

Surface 2 3

Create

1D

Flow Tube

oil

m:oil

3.0

Curve 1

7. Define a Spatial Field.

◆ **Fields**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Spatial"/>
<i>Method:</i>	<input type="text" value="PCL Function"/>
<i>Field Name:</i>	<input type="text" value="qvol_z"/>
<i>Scalar Function:</i>	<input type="text" value="1200*(1.0-'Z'/5.0)"/>
	<input type="button" value="Apply"/>

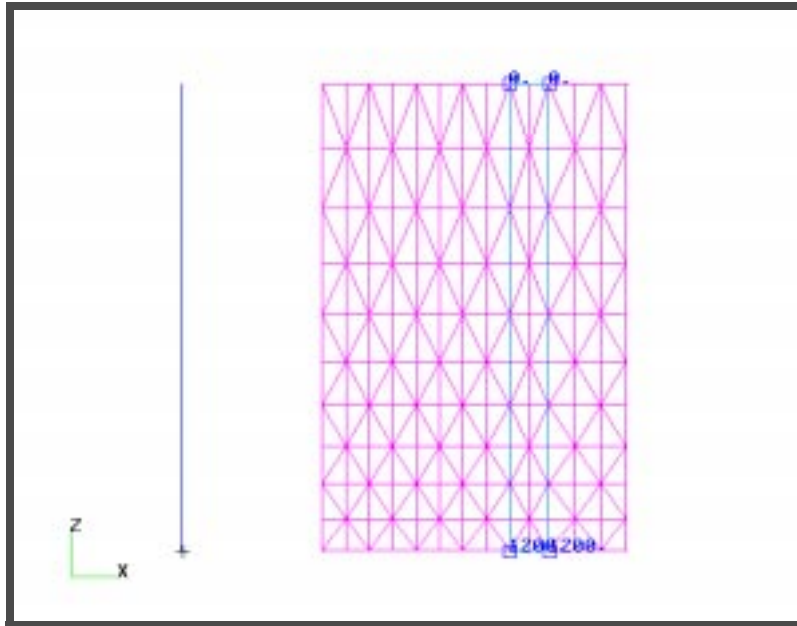
8. Apply a Volumetric Heat Load.

◆ **Load/BCs**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Applied Heat"/>
<i>Type:</i>	<input type="text" value="Element Uniform"/>
<i>Option:</i>	<input type="text" value="Volumetric Generation"/>
<i>New Set Name:</i>	<input type="text" value="qvol"/>
<i>Target Element Type:</i>	<input type="text" value="2D"/>
	<input type="button" value="Input Data..."/>
<i>Volumetric Heat Generation:</i>	<input type="text" value="f:qvol_z"/>

<i>Geometry Filter:</i>	◆ Geometry
<i>Select Surfaces:</i>	<input type="text" value="Surface 2"/>

Your model should look like the following figure.



9. Apply Free Convection.

◆ **Load/BCs**

Action:

Create

Object:

Convection

Type:

Element Uniform

Option:

To Ambient

New Set Name:

conv

Target Element Type:

2D

Input Data...

Surface Option:

edge

Edge Convection Coef:

3.0

Ambient Temperature:

100

OK

Select Application Region...

Geometry Filter:

◆ **Geometry**

Click on the **Edge View** icon.



Edge

Select Surfaces:

Surface 3.2

Add

OK

Apply

10. Define Inlet Temperatures of the Fluid.

◆ Load/BCs

Action:

Create

Object:

Temp (Thermal)

Type:

Nodal

New Set Name:

inlet_temp

Input Data...

Boundary Temperature:

100

OK

Select Application Region...

Geometry Filter:

◆ Geometry

Select Geometry Entities:

Point 1

Add

OK

Apply

11. Define Coupled Flow Tube.

Apply a fluid-structure coupling between the oil and inner wall of the pipe.

◆ Load/BCs

Action:

Create

<i>Object:</i>	<input type="text" value="Convection"/>
<i>Type:</i>	<input type="text" value="Element Uniform"/>
<i>Option:</i>	<input type="text" value="Coupled Flow Tube"/>
<i>New Set Name:</i>	<input type="text" value="coup_ftube"/>
<i>Target Element Type:</i>	<input type="text" value="1D"/>
<i>Region 2:</i>	<input type="text" value="2D"/>
Input Data...	
<i>Form Type:</i>	<input type="text" value="Advanced"/>
<i>Mass Flow Rate:</i>	<input type="text" value="2.88e6"/>
<i>Heat Transfer Coefficient:</i>	<input type="text" value="0.023"/>
<i>Formula Type Option:</i>	◆ $h=k/d*coef*Re**Expr*Pr**$
<i>Reynolds Exponent:</i>	<input type="text" value="0.8"/>
<i>Prandtl Exponent, Heat In:</i>	<input type="text" value="0.3333"/>

Select Application Region...

<i>Geometry Filter:</i>	◆ Geometry
<i>Select Curves:</i>	<input type="text" value="Curve 1"/>

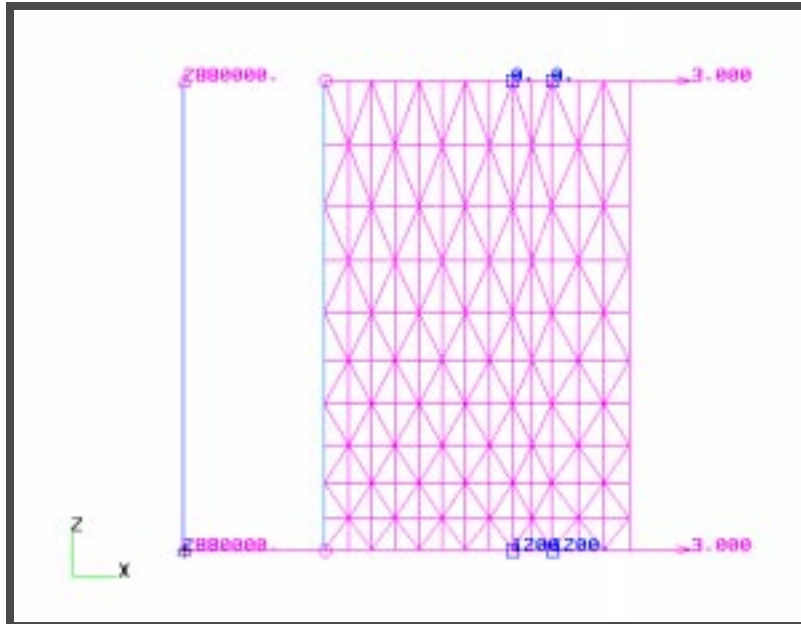
Active List

<i>Select Surfaces or Edges:</i>	<input type="text" value="Surface 1.4"/>
----------------------------------	--

Click on the **Edge View** icon.



Your model should look like the following figure.



12. Perform the Analysis.

◆ **Analysis**

Action:

Analyze

Object:

Entire Model

Method:

Analysis Deck

Job Name:

ex7

Apply

An MSC.Nastran input file called **ex7.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:

13. Submit the input file to MSC.NASTRAN for analysis.
 - 13a. To submit the MSC.Patran **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran ex7.bdf scr=yes**. Monitor the run using the UNIX **ps** command.
 - 13b. To submit the MSC.Nastran **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran ex7 scr=yes**. Monitor the run using the UNIX **ps** command.
14. When the run is completed, edit the **ex7.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.

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

16. Proceed with the Reverse Translation process, that is, attaching the ex7.xdb results file into MSC.Patran. To do this, return to the Analysis form and proceed as follows:

◆ **Analysis**

Action:

Attach XDB

Object:

Result Entities

Method:

Local

Select Results File...

Select Results File

ex7.xdb

OK

Apply

17. Display the Results.

◆ **Results**

Select Results Cases:

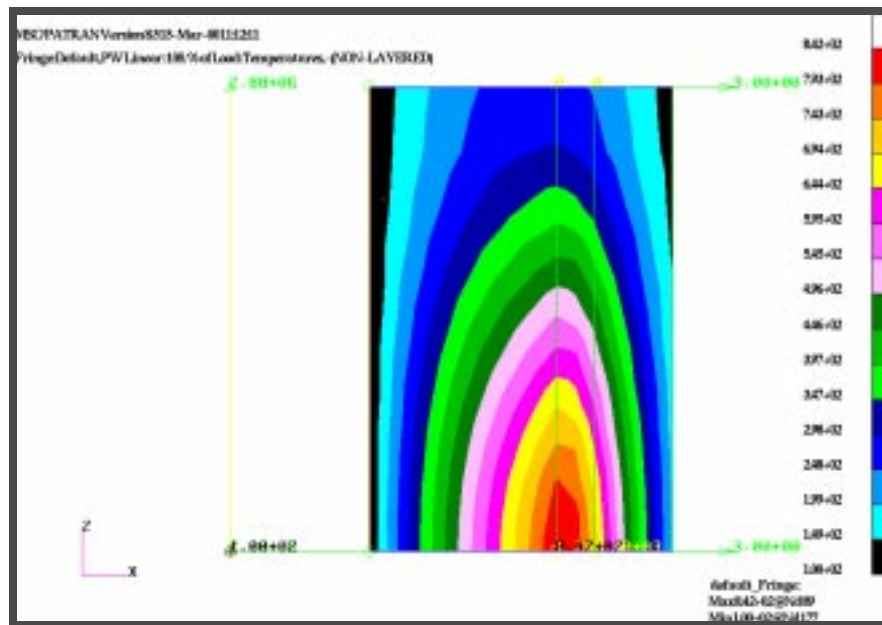
Default, PW Linear: 100. % of Load

Select Fringe Result:

Temperatures

Apply

Your model should look like the following figure.



The maximum temperature occurs near the internal heat generation region with a temperature of 842.3°F. The fluid temperature remains constant at 100 °F because of the massive flow rate at 2.88E6 lb_m/hr.

We can check the energy balance on this model as follows:

Total heat = 2.91246E4 Btu/hr (from the OLOAD RESULTANT of the F06 file)

Sum of the heat on the column under Free Convection = 2.5828E4 Btu/hr

Sum of the heat on the column under Forced Convection = 3.297E3 Btu/hr

Sum of the heat on the above two columns = 2.9125E4 Btu/hr, which is equal to the input heat of 2.91246E4 Btu/hr.

An assumption of a 1-D fluid element is that temperature gradients within the fluid are only significant along the axial direction. With such a large diameter flow tube, this assumption is probably being misused in this particular problem. The application of the flow tube boundary convection relationship also implies fully developed flow, yet, over only a 5 foot section and with a 3 foot diameter, this is also a very crude approximation. In essence, what we are saying, is that this example serves to illustrate coupled convection in an axisymmetric environment, application of spatial heat loads, and use of convection correlation equations, rather than fluid physics.

Quit MSC.Patran when you have completed this exercise.