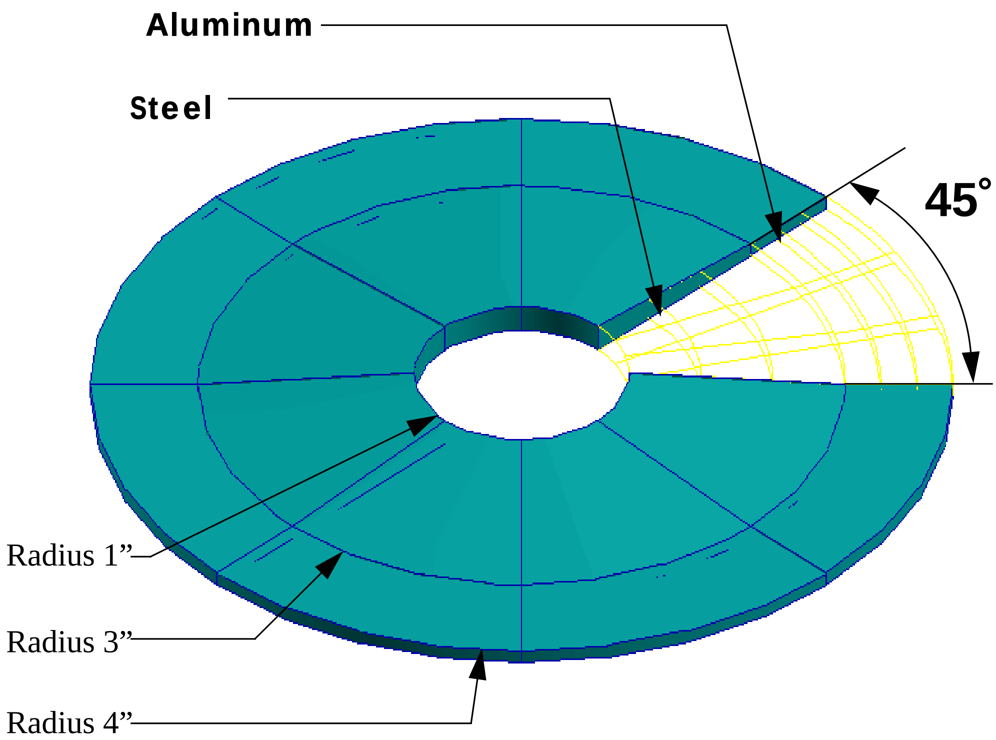
LESSON 13

Spatial Variation of Physical Properties



Objective:

- To model the variation of physical properties as a function of spatial coordinates.

Model Description:

In this exercise you will create a portion of a circular plate which has a hole at its center. Due to the model's symmetry only a 45˚ slice of the plate will be modeled. You will also create spatially varying material and physical properties.

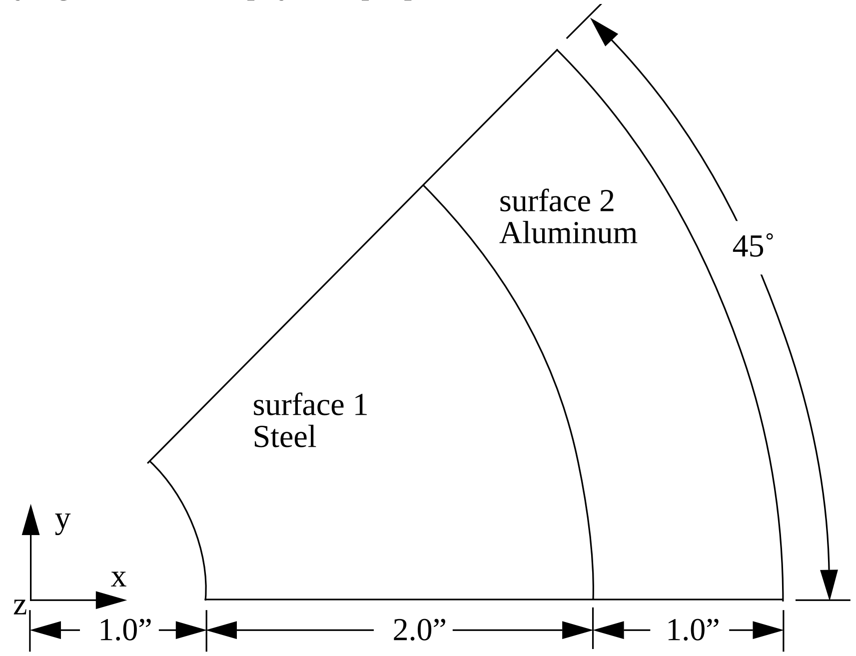


Figure 13-1

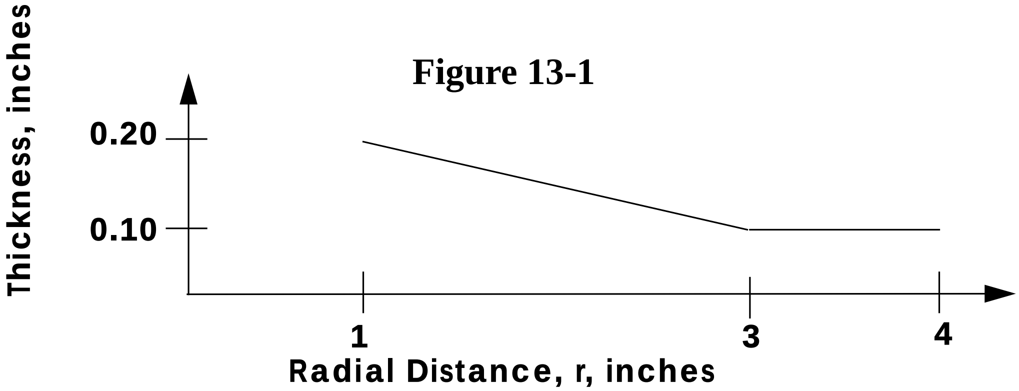


Table 13-1

Analysis Code:	MSC/NASTRAN	
Element type:	Quad4	
Element Global Edge Length:	0.5	
Material Constant Description	Steel	Aluminum
Modulus of Elasticity, E (psi)	30E6	10E6
Poisson's Ratio, ν	0.30	0.20
Density, ρ (lb-sec2/in4)	0.0007324	0.0002588

Suggested Exercise Steps:

- Create a new database named **circular_Plate.db**.
- Change the Tolerance to **Default** and the Analysis Code to MSC/NASTRAN.
- Create the geometry that represents the 45˚ slice of the circular plate shown in Figure 13-1.
- Create the finite element mesh using the information listed in Table 13-1.
- Create a cylindrical coordinate frame whose origin is located at [0,0,0] and whose R-, T-, Z-axis are aligned with the X-, Y-, Z-axes respectively of the global coordinate system.
- Using the cylindrical coordinate frame, define a spatially varying field named **thickness_spatial**, that represents the model's thickness. Verify the field by displaying an XY-plot.
- Create the Isotropic Steel and Aluminum material properties using the material constants shown in Table 13-1.
- Inspect the constitutive (stiffness) matrices, C_{ijkl}, of each material type.
- Create the model's element properties assigning the material type and element thickness to the correct region of the model. Use the names **prop_1** and **prop_2** for your element property definitions.
- Verify that the spatial variation of the element thickness has been assigned correctly to your model by rendering a scalar plot of the thickness.

Exercise Procedure:

1. Create a **New Database** and name it `circular_Plate.db`.

File/New Database...

New Database Name	**circular_plate**
OK	

2. Change the *Tolerance* to **Default** and the *Analysis Code* to **MSC/NASTRAN** in the *New Model Preferences* form. Verify that the *Analysis Type* is **Structural**.

New Model Preference

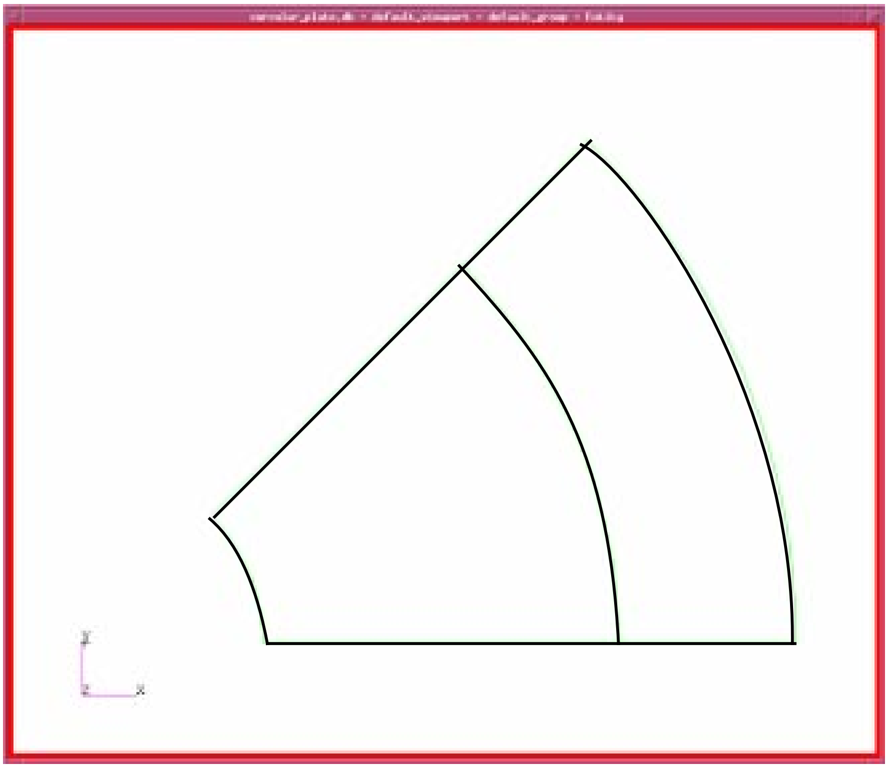
Tolerance	◆ **Default**
Analysis Code:	**MSC/NASTRAN**
Analysis Type	**Structural**
OK	

Create the Circular Plate model

3. Create the geometry that represents the 45˚ slice of the circular plate shown in Figure 13-1.

Create the 45 degree slice of the circular plate by creating two adjacent surfaces that lie in the global xy-plane. The two surfaces meet along the material boundary. See Figure 13-1 of this exercise for the required dimensions.

When you are finished your model should look like the one shown in the figure below.



Mesh the Model

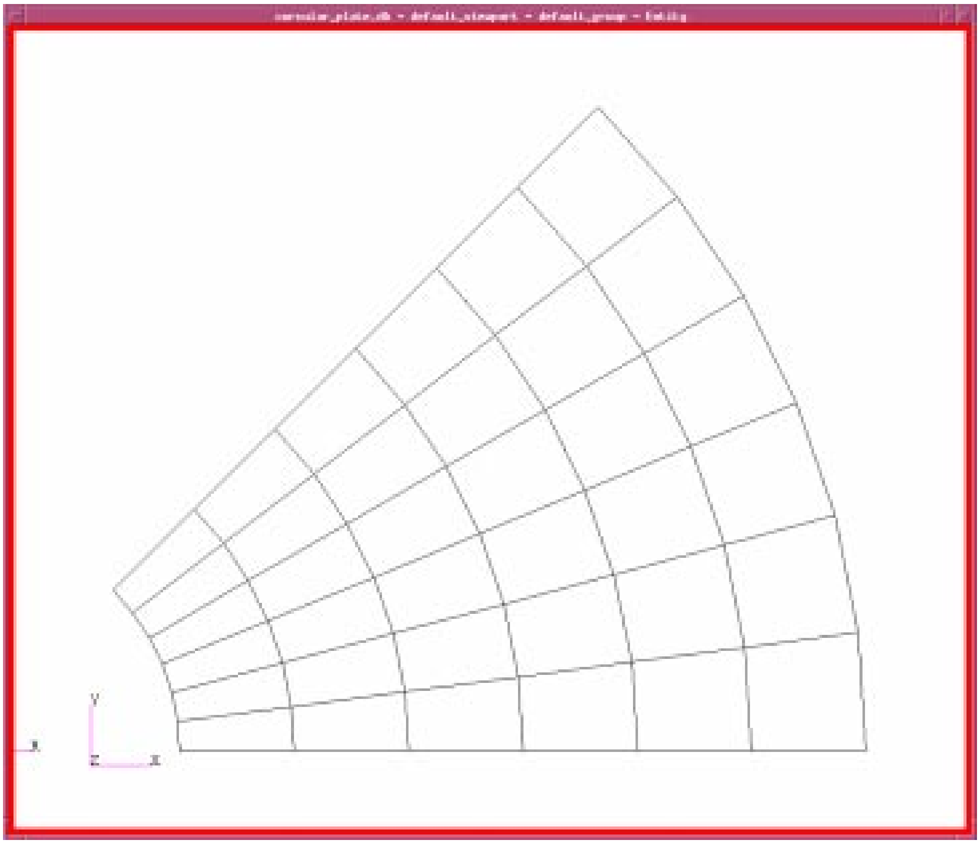
4. Create the finite element mesh using the information listed in Table 13-1.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Global Edge Length	**0.5**
Element Topology	**Quad 4**
Surface List	**Surface 1, 2**

Apply

Your model should appear like the one shown below.



5. Create a cylindrical coordinate frame whose origin is located at [0,0,0] and whose R-, T-, Z-axis are aligned with the X-, Y-, Z-axes respectively of the global coordinate system.

Create a Cylindrical Coordinate Frame

◆ **Geometry**

Action:	**Create**
Object:	**Coord**
Method:	**3Point**
Type:	**Cylindrical**
Origin	**[0, 0, 0]**
Point on Axis 3	**[0, 0, 1]**
Point on the Plane 1-3	**[1, 0, 0]**

Apply

Create a Tabular Spatial Scalar Field

6. Using the cylindrical coordinate frame, define a spatially varying field named **thickness_spatial**, that represents the model's thickness. Verify the field by displaying an XY-plot.

In MSC/PATRAN, the Physical property spatial variations are specified using spatial fields. In this exercise, you will create a tabular spatial scalar field to describe the variation of the plate's thickness as a function of the radial distance.

◆ **Fields**

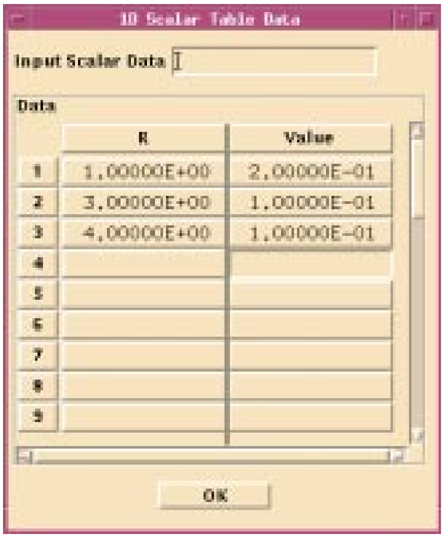
Action:	**Create**
Object:	**Spatial**
Method:	**Tabular Input**
Field Name	**thickness_spatial**
Coordinate System	**Coord 1**
Active Independent Variable	**R**

Input Data...

Enter the following three sets of points:

R=1.0,	Value=0.20;
R=3.0,	Value=0.10;
R=4.0,	Value=0.10.

To do this, click on the cell you wish to edit, the cursor will appear in the *Input Scalar* databox. Enter the data, and press <Return>. Your table should look like this.



OK

Apply

At this point, you should verify the created field by using MSC/PATRAN's XY plot feature.

Verify the Created Field

Action: **Show**

Select Field to Show **thickness_spatial**

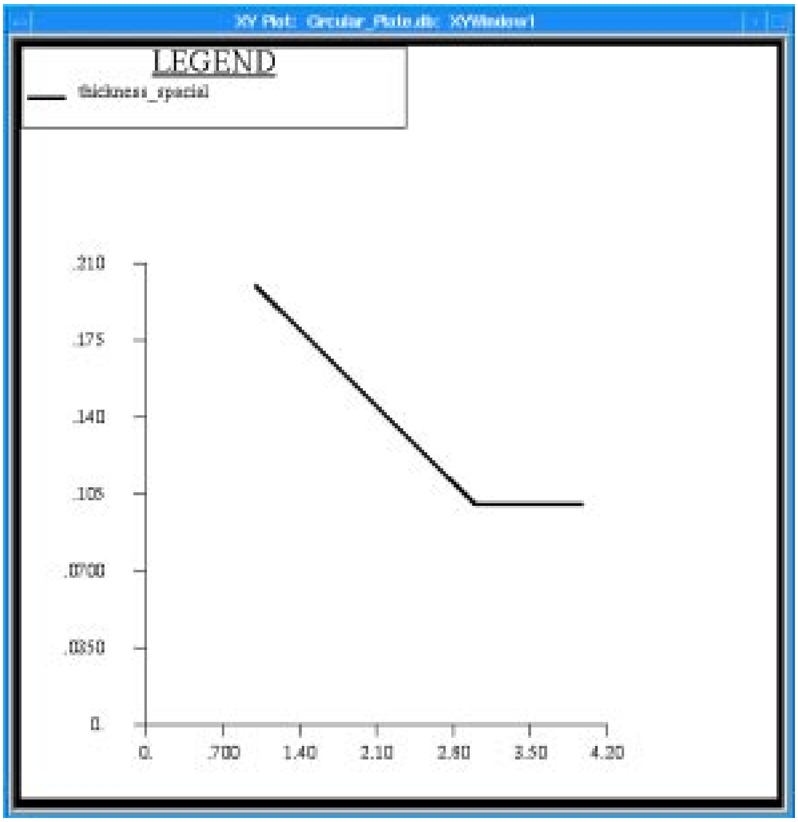
Specify Range...

Use Existing Points

OK

Apply

Your plot should appear like the one shown below. Later you will learn how to change the titles, colors, line styles, tick marks, and other attributes of the graph.



Unpost the XY Plot Window

To unpost and delete the *XY Plot* window first click on the **Unpost Current XYWindow** button.

◆ **XY Plot**

Action:	**Delete**
Object:	**XY Window**
Existing XY Windows	**XY Result Window**

Apply

Click on **Yes** when asked if you are sure you want to delete the XY result window.

Specify the Material Constants for Aluminum and Steel

7. Create the isotropic steel and aluminum material properties using the material constants shown in Table 13-1.

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name	**steel**

Input Properties...

Elastic Modulus	**30E6**
Poisson's Ratio	**0.3**
Density	**0.0007324**

Apply

Cancel

Repeat the process for **aluminum.**

Verify the Material Constants

8. Inspect the constitutive (stiffness) matrices, C_{ijkl}, of each material type.

To verify the material constants you have entered, select **Show** from the *Action* option menu on the *Materials* form.

Action:	**Show**
Material Name	**steel**

Show Properties...

Show Material Stiffness...

To view the component in any cell of the matrix, simply click on that cell. For example, click on the upper left cell.

9. Create the model's element properties assigning the material type and element thickness to the correct region of the model. Use the names **prop_1** and **prop_2** for your element property definitions.

Specify the Physical Properties

◆ **Properties**

Action:	**Create**
Dimension:	**2D**
Type:	**Shell**
Property Set Name	**prop_1**

Input Properties...

Material Name	**m:steel**
Thickness	**f:thickness_spatial**

OK

Select Application Region	**Surface 1**

Add

Apply

The same process must be repeated to specify the **aluminum** material property for **Surface 2**.

10. Verify that the spatial variation of the element thickness has been assigned correctly to your model by rendering a scalar plot of the thickness.

Create an Element Fill Plot

In this final step you will create an element fill plot of the specified thickness of the plate elements.

Action:	**Show**
Existing Properties	**Thickness**
Display Method	**Scalar Plot**

Group Filter **Default_group**

Apply

You may need to reset the range to span the actual property range.

Display/Ranges...

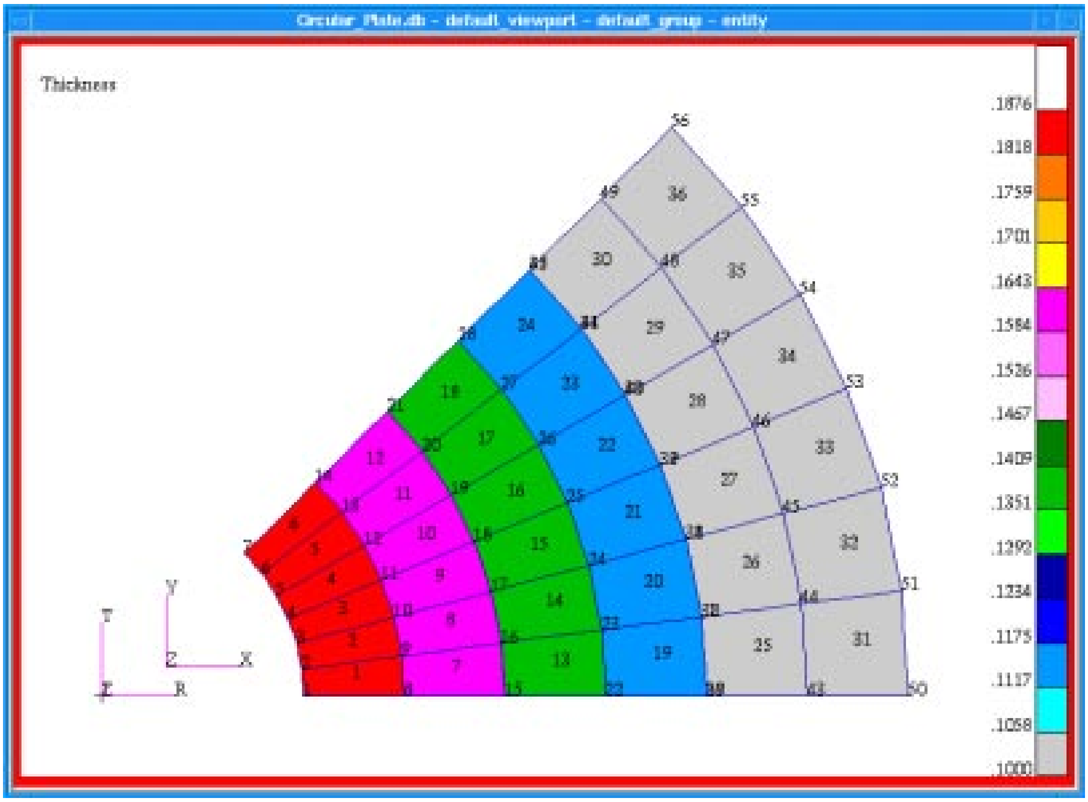
Fit Results

Calculate

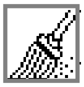
Apply

Cancel

Your Viewport will appear as follows.



The viewport may now be reset by clicking on the broom icon in the main window.



File/Quit