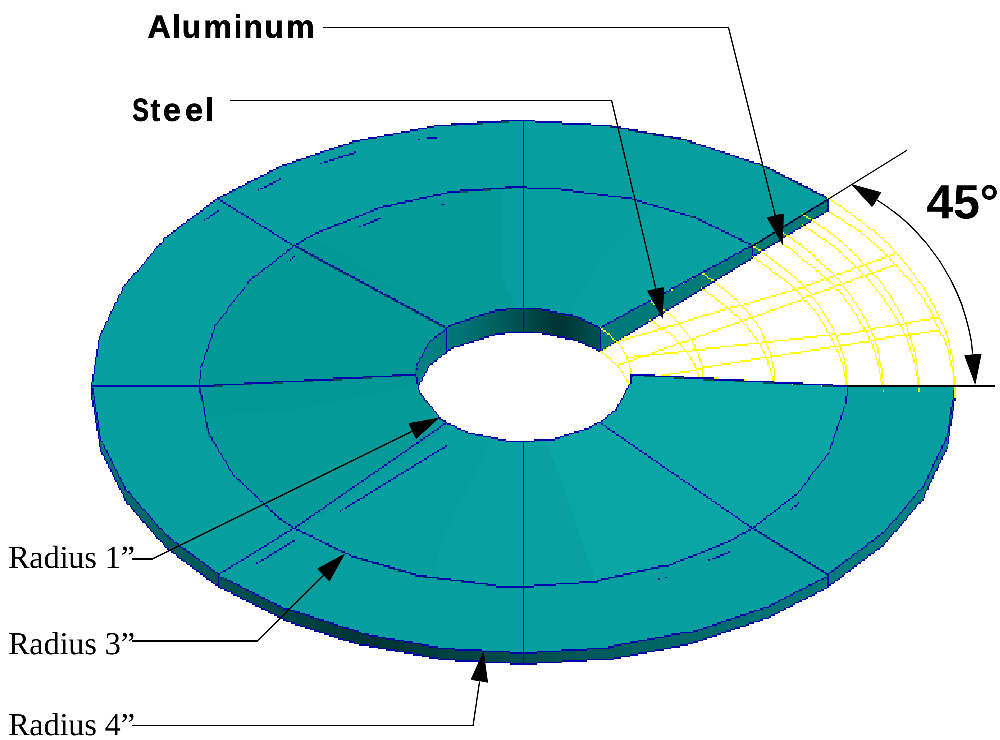
# WORKSHOP 12

# *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.

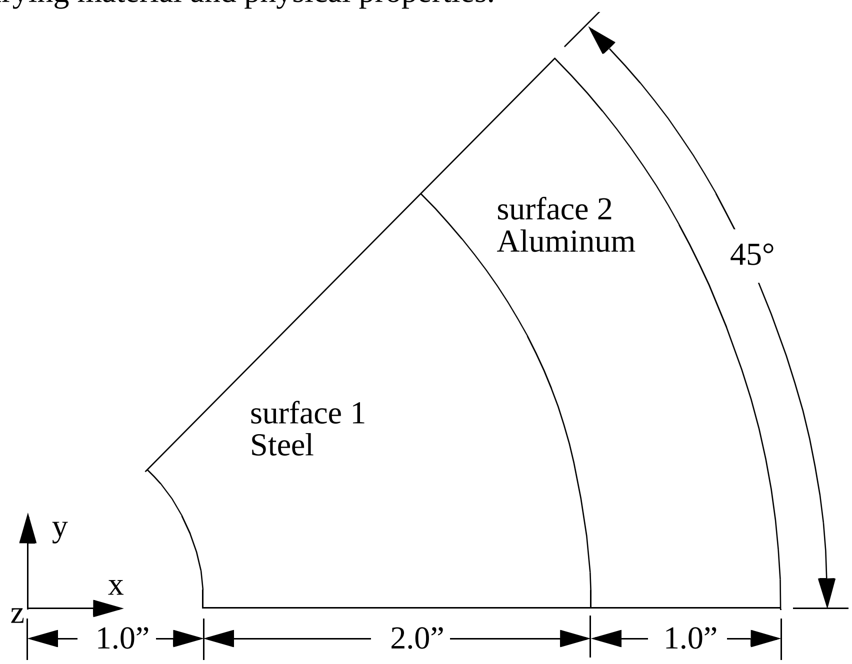



**Figure 12-1**

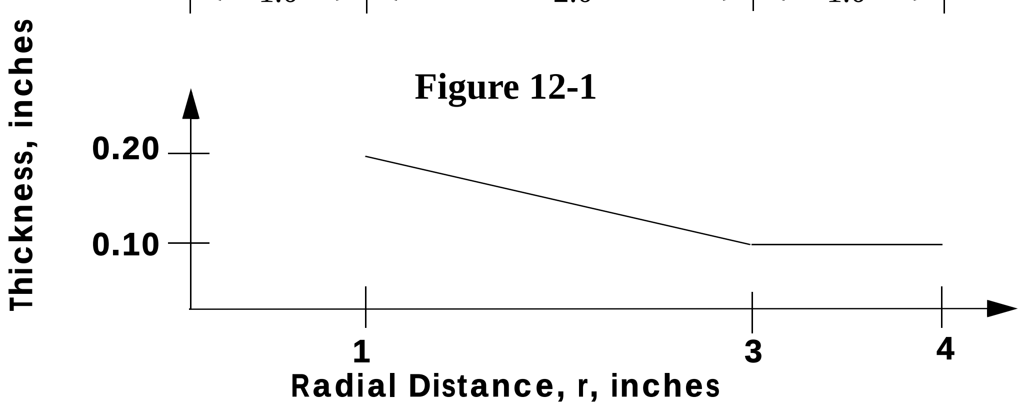



**Table 12-1**

| | | |
|---|---|---|
| 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 Figure12-1.
- Create the finite element mesh using the information listed in Table 12-1.
- 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 12-1.
- 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...**

| | |
|---|---|
| *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**

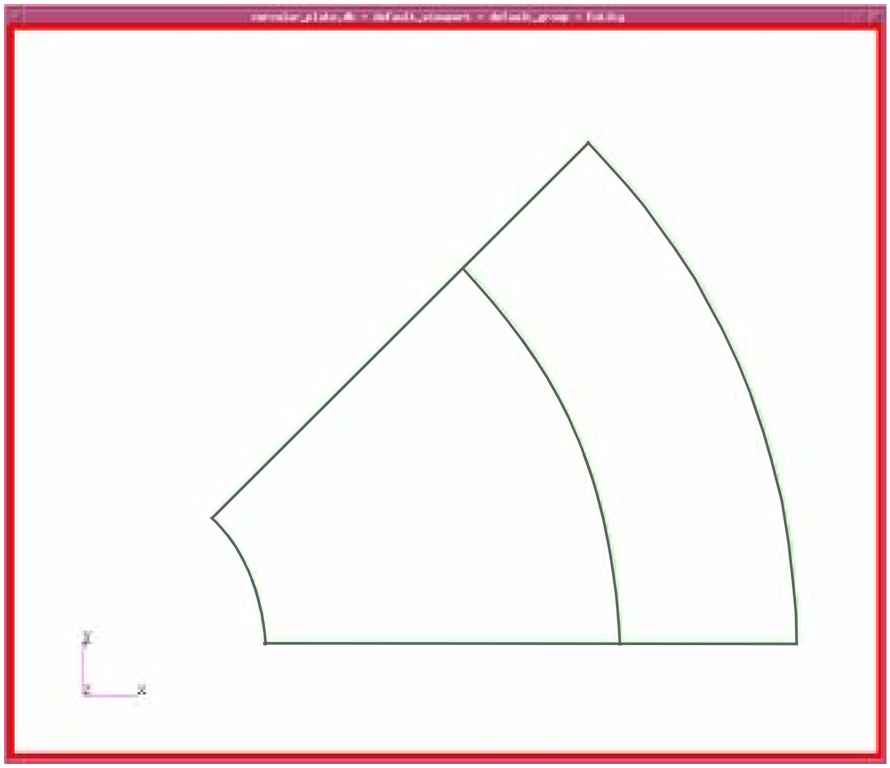
| | |
|---|---|
| *Tolerance* | ◆ **Default** |
| *Analysis Code:* | **MSC/NASTRAN** |
| *Analysis Type* | **Structural** |
| **OK** | |

3. Create the geometry that represents the 45° slice of the circular plate shown in Figure 12-1.

**Create the Circular Plate model**

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 12-1 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

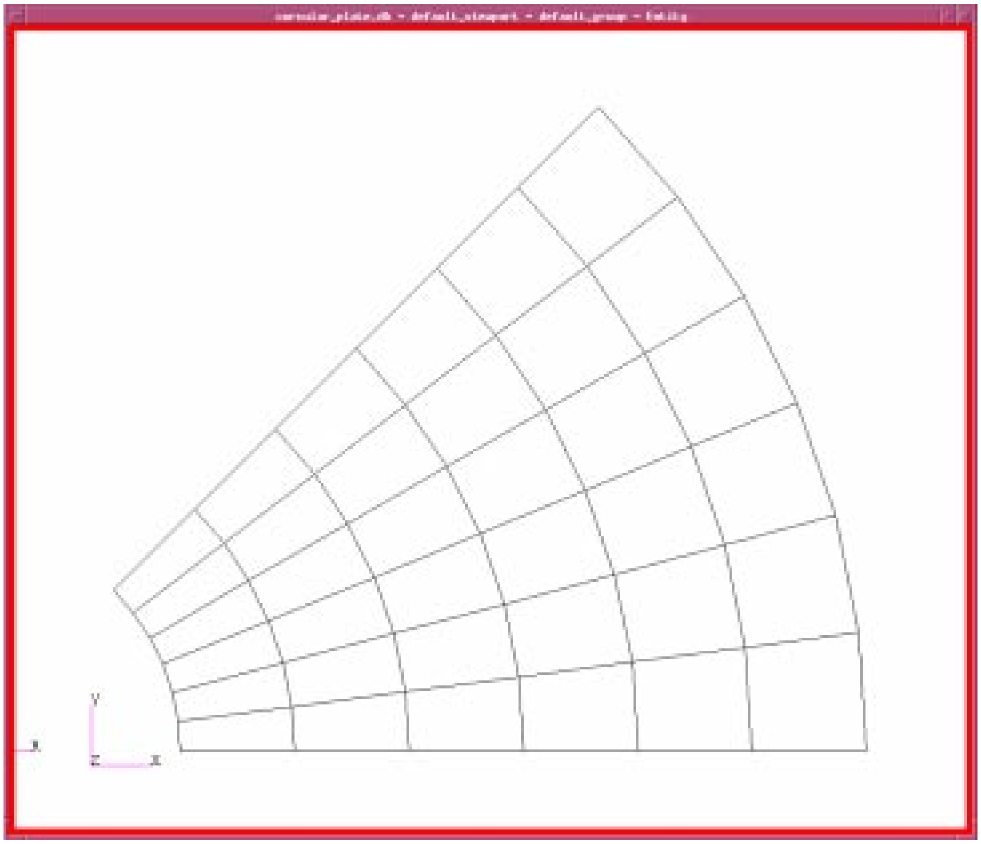
4. Create the finite element mesh using the information listed in Table 12-1.

### ◆ 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.



**Create a Cylindrical Coordinate Frame**

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.

◆ **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**

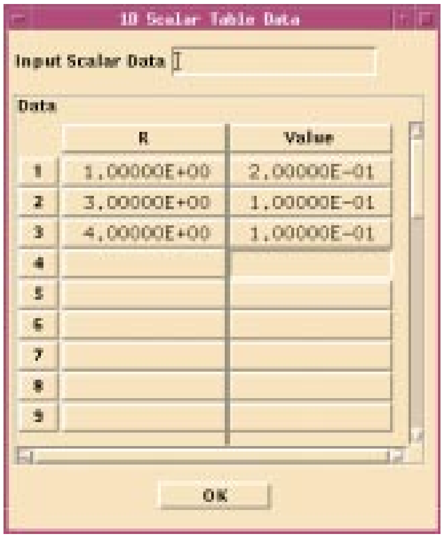
| | |
|---|---|
| *Action:* | **Create** |
| *Object:* | **Spatial** |
| *Method:* | **Tabular Input** |
| *Field Name* | **thickness_spatial** |
| *Coordinate System* | **Coord 1** |
| *Active Independent Variable* | **R** |

**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**

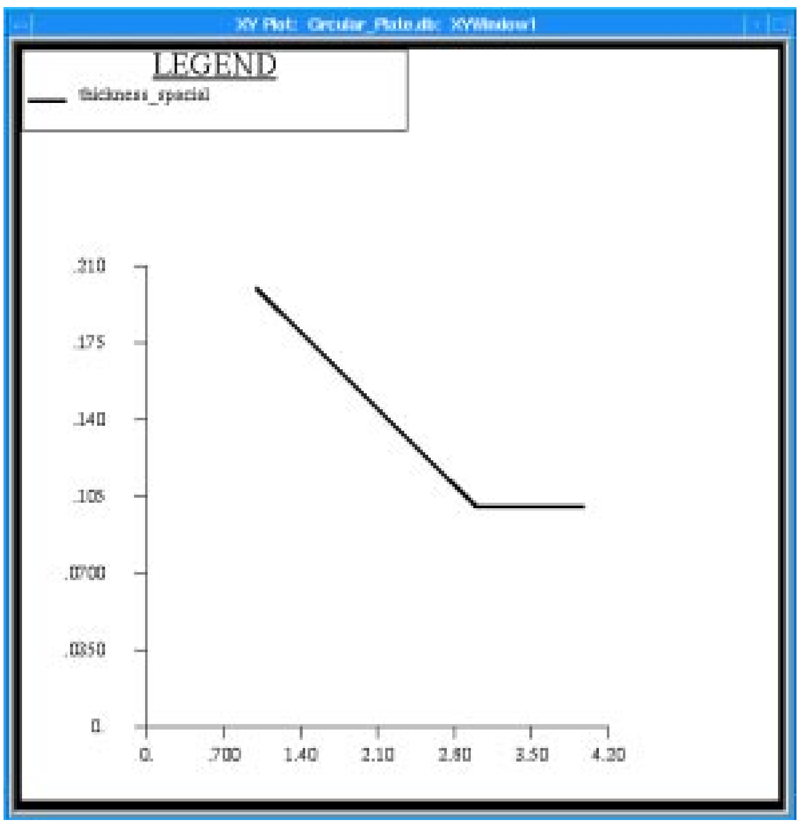
**Specify Range...**

**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 1212-1.

◆ **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.

**Specify the Physical Properties**

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.

◆ **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**.

**Create an Element Fill Plot**

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.

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...**

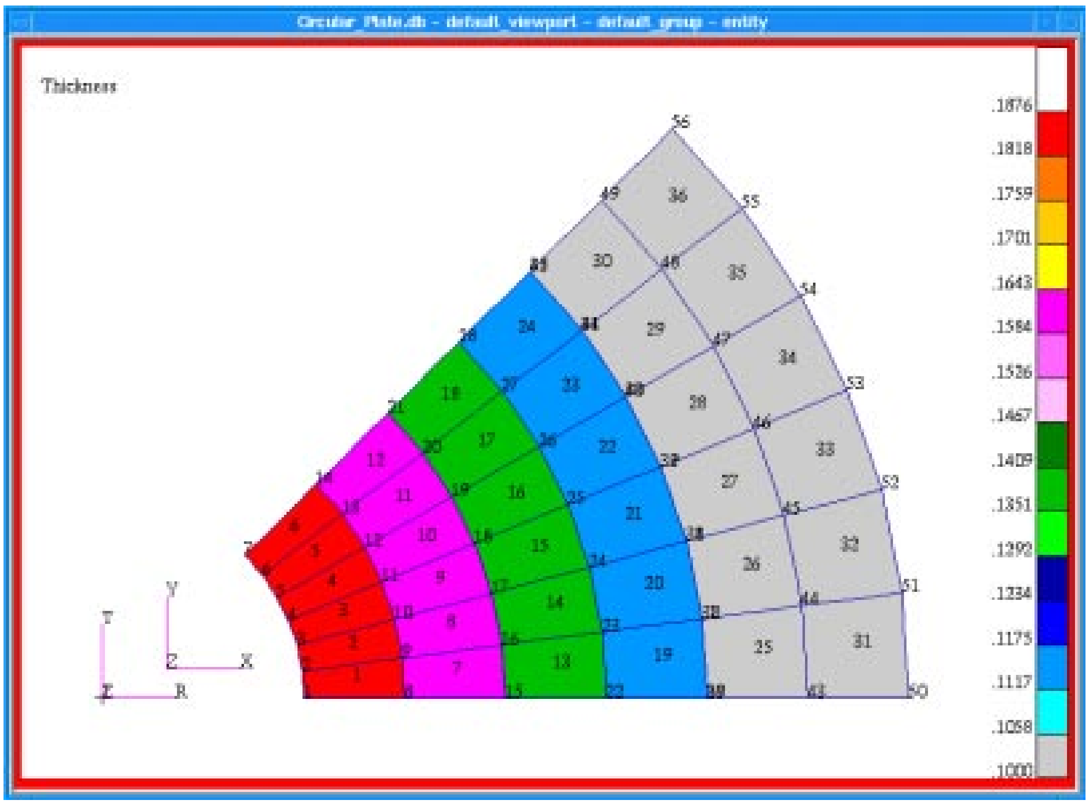
**Fit Results**

**Calculate**

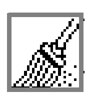
**Apply**

**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...**