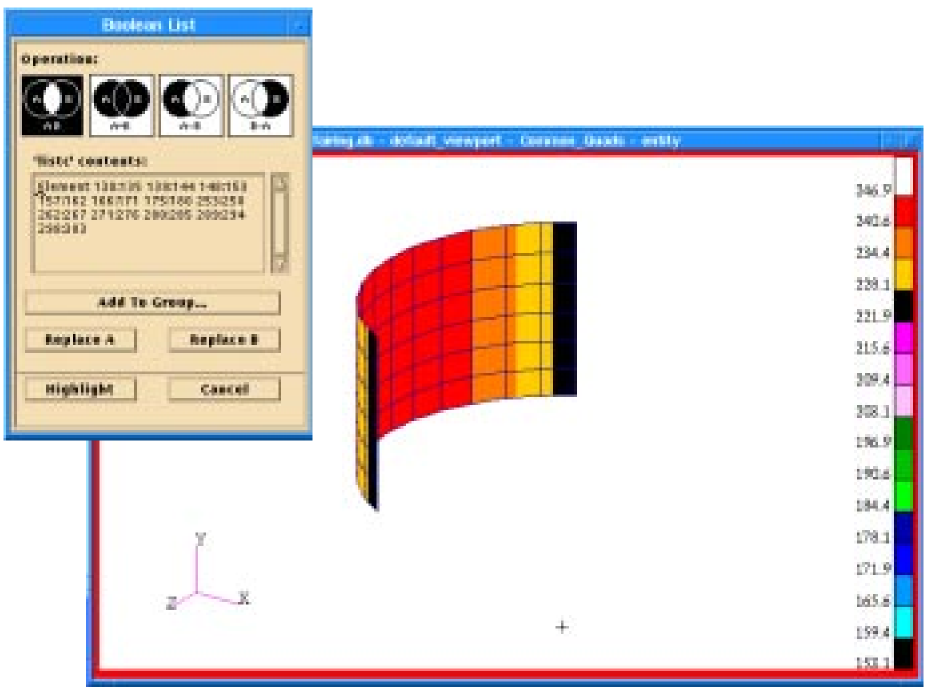
WORKSHOP 14

Using Groups and Lists



Objectives:

- Build a finite element model that includes element properties and boundary conditions.
- Use lists to identify parts of the model with specified attributes.
- Explore the Group Display mode.

Model Description:

In this exercise you will import or construct a portion of a fairing. Shown below is a drawing of the assembled structure and its dimensions. Use curves and surfaces to define the fairing geometry. The finite element model will consist of 2-dimensional elements with 1-dimensional elements applied at various edges of the geometry. The 1-dimensional elements will represent stiffeners for the structure.

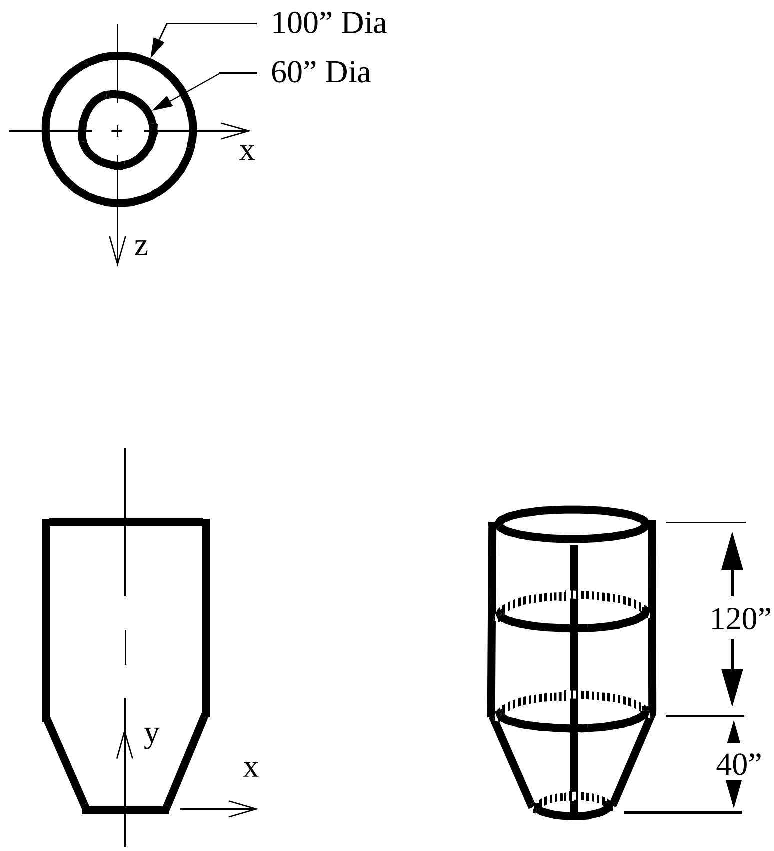


Figure 14-1

Analysis Code	P3/FEA	
Element Types	Bar2 (horizontal fairing edges)	
	Quad4 (fairing surface)	
Material Name	Alum_1	Alum_2
Modulus of Elasticity, E (psi)	1.05E7	1.18E7
Poisson's Ratio, ν	0.33	0.33
Density, ρ (lb/in3)	2.6E–4	2.4E–4
Model Thickness	1.5 - Y/160.	
Model Temperature Distribution	200.-(150./160.)X	

Table 14-1

Suggested Exercise Steps:

- Create a new database and name it **fairing.db**. Select **Default** for the *Tolerance* and **MSC/NASTRAN** for the *Analysis Code.*
- Either import the Geometry and Finite Element model from the neutral file **`fairing.out`** or create the model using Figure 14-1.
- Create the points and curves that represent the outline of the fairing.

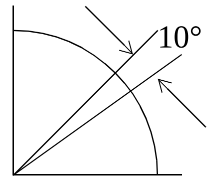
  Point 1 (XYZ method): [30,0,0];

  curve 1 (XYZ method): vector length=<0,120,0>; origin=[50,40,0]

  curve 2 (point method): between points 1 and 2.

- Sweep Curves 1 & 2 through 360^{o} angles about the center-line of the fairing in 4 steps using the Surface Revolve method.

a) Seed the circumference of the fairing at the upper edge with 9 nodes per quarter of the circumference.



b) Create non-uniform seed distributions along the vertical edge of the fairing represented by Curve 1 &2..

Curve 1, L2 = 10, L1 = 7

Curve 2, L2 = 7, L1 = 4

c) Create the mesh for the surface using Quad4 elements.

d) Create Bar2 elements along the circumference representing the edges of the upper cylinder of the lower cone.

- Create a group containing only the finite element model. Name the group FEM. Post only that group to the viewport.
- Create the materials for the fairing. Materials **alum_1** and **alum_2** will be applied to the top (cylindrical) and bottom (tapered) portions of the fairing respectively. Use Table 14-1 to define the Material Properties.
- Define fields that represent the varying **thickness** and **temperature** distribution. Use Table 14-1 to define the fields.
- Create the element properties which include the material definitions and the varying thickness. Use the names **prop_1** and **prop_2** for the element property names.
- Define the model's varying temperature distribution. Use the name **temp** for the temperature set name.
- Use Lists and Groups to display the Quad elements that have the following attributes:

  Material:alum_1 (MATRL.1)

  Thickness:> 0.98

  Temperature: > 230.0

Create a new group named **Common_Quads** and add these elements to that group. Plot the temperature contours on these elements. Reset Graphics.

- Post only the group named FEM and change the render style to hidden line (the bars will disappear).
- Create a group containing only the bar elements. Name the group **BARS**.
- Change to group display mode and modify the FEM and BARS render style as follows:

Group	Render Style	Shade Color	Entity Labels
FEM	**Hidden Line**	**Cyan**	**Off**
BARS	**Wireframe**	**Yellow**	**Off**

- Change the render style for the group BARS to **Wireframe/Accurate**.

Exercise Procedure:

1. Create a new database and name it **fairing.db**. Select **Default** for the *Tolerance* and **MSC/NASTRAN** for the *Analysis Code*.

File/New...

New Database Name | **fairing**

OK

New Model Preference

Tolerance | ◆ **Default**

Analysis Code: | **MSC/NASTRAN**

OK

2. Either import the Geometry and Finite Element model from the neutral file **`fairing.out`** or create the model using Figure 14-1.

If you are going to import the Geometry and Finite Element model of the fairing, perform the following import procedure, then skip to step 11. If you are going to build the fairing model, skip to step 3.

File/Import...

Object:	**Model**
Source:	**Neutral**
Import File	**fairing.out**

Apply

Respond **Yes** when asked to continue on the *Import Summary* form.

To see what was just imported, go to **Group/Modify** and look at the *Member List*. Both geometry (points, curves and surfaces) and finite elements (nodes and elements) have been imported into the default_group. Click on **OK** to close the form. To see what kinds of elements were imported, select the **Finite Elements** radio button, then **Show/Element/Attributes**, highlight all the elements and hit **Apply**. Scroll down through the spreadsheet to see that both Quad4 and Bar2 elements are in the model.

Now create a group containing only the finite element model.

Group/Create...

New Group Name	**FEM**
Group Contents	**Add All FEM**

Apply

Go to Step 11.

Create Model Geometry

3. Create the points and curves that represent the outline of the fairing.

   Point 1 [30,0,0];
   Curve 1: vector length=<0,120,0>; origin=[50,40,0]

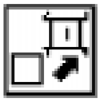
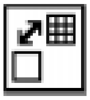
   Curve 2: between points 1 and 2.

Geometry

Action:	**Create**
Object:	**Point**
Method:	**XYZ**
Point Coordinate List	**[30, 0, 0]**

Apply

Turn on labels and display lines using the **Show Labels** and **Display Lines** icons,

Now you will create curves that represent the profile of the fairing. They will be swept to create the fairing's surface.

Action:	**Create**
Object:	**Curve**
Method:	**XYZ**
Vector Coordinate List	**<0, 120, 0>**
Origin Coordinate List	**[50, 40, 0]**

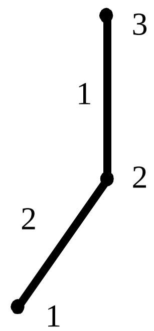
Apply

Next change the *Method* option menu to **Point**.

Action:	**Create**
Object:	**Curve**
Method:	**Point**
Starting Point List	**Point 1**
Ending Point List	**Point 2**

Apply

Your model should appear as follows:



4. Create the fairing from an assembly of quarter circular surfaces defined by revolving curves 1 and 2 about the fairing's vertical center line.

Action:	**Create**
Object:	**Surface**
Method:	**Revolve**
Surface Type	**PATRAN 2 Convention**
Axis	**Coord 0.2**
Total Angle	**360**
Surface per Curve	**4**
Curve List	**Curve 1, 2**

Apply

Change the view *Angle* to **30 0 0**

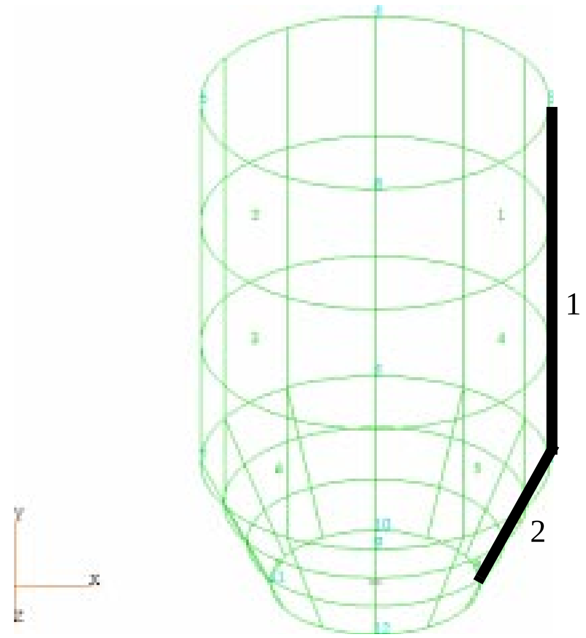
Viewing/Angles...

Angles	**30, 0, 0**

Apply

Cancel

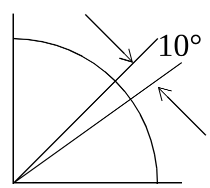
Your model should appear as follows:



Create Mesh Seeds

5. Create a finite element mesh that has the following attributes:

Along the circumferential edges create 4 node Quad elements every 10°



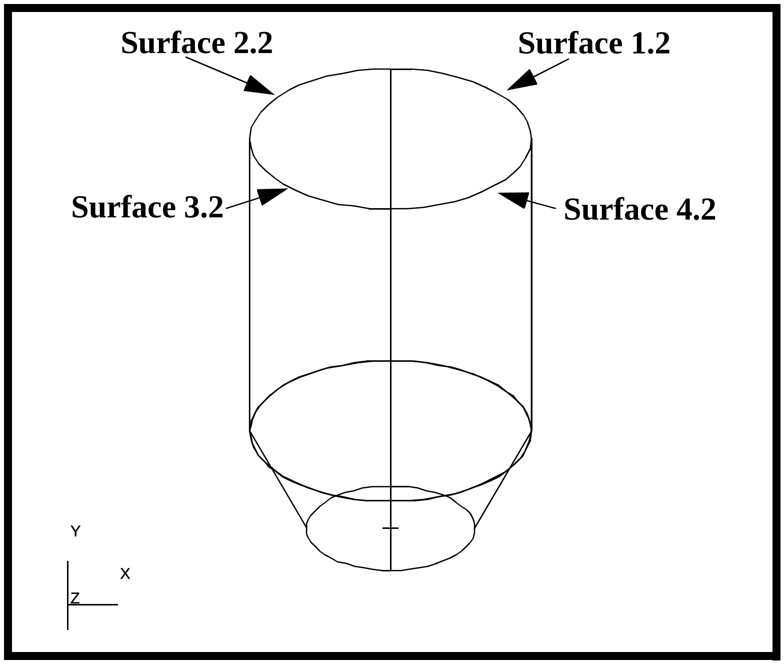
◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh Seed**
Type:	**Uniform**

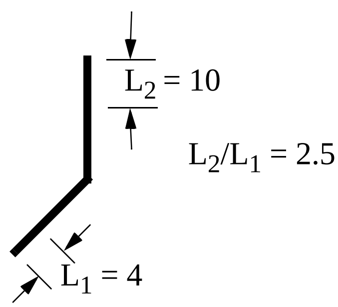
◆ **Number of Elements**

Number =	**9**
Curve List	Select the Upper Circumferential Edges of Surfaces 1 through 4. See the figure below

Apply



In the vertical direction (y-direction), define a smoothly transitioning mesh density, the elements along the top of the cylinder are 2.5 times as large as those along the bottom edge (tapered end) of the fairing.



Action:	**Create**
Object:	**Mesh Seed**
Type:	**One Way Bias**

◆ **L1 and L2**

L1 =	**7**
L2 =	**10**
Curve List	**Curve 1**

Apply

Action:	**Create**
Object:	**Mesh Seed**
Type:	**One Way Bias**

◆ **L1 and L2**

L1 =	**4**
L2 =	**7**
Curve List	**Curve 2**

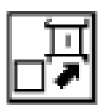
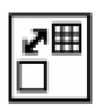
Apply

Now that the seed has been created you will mesh the model.

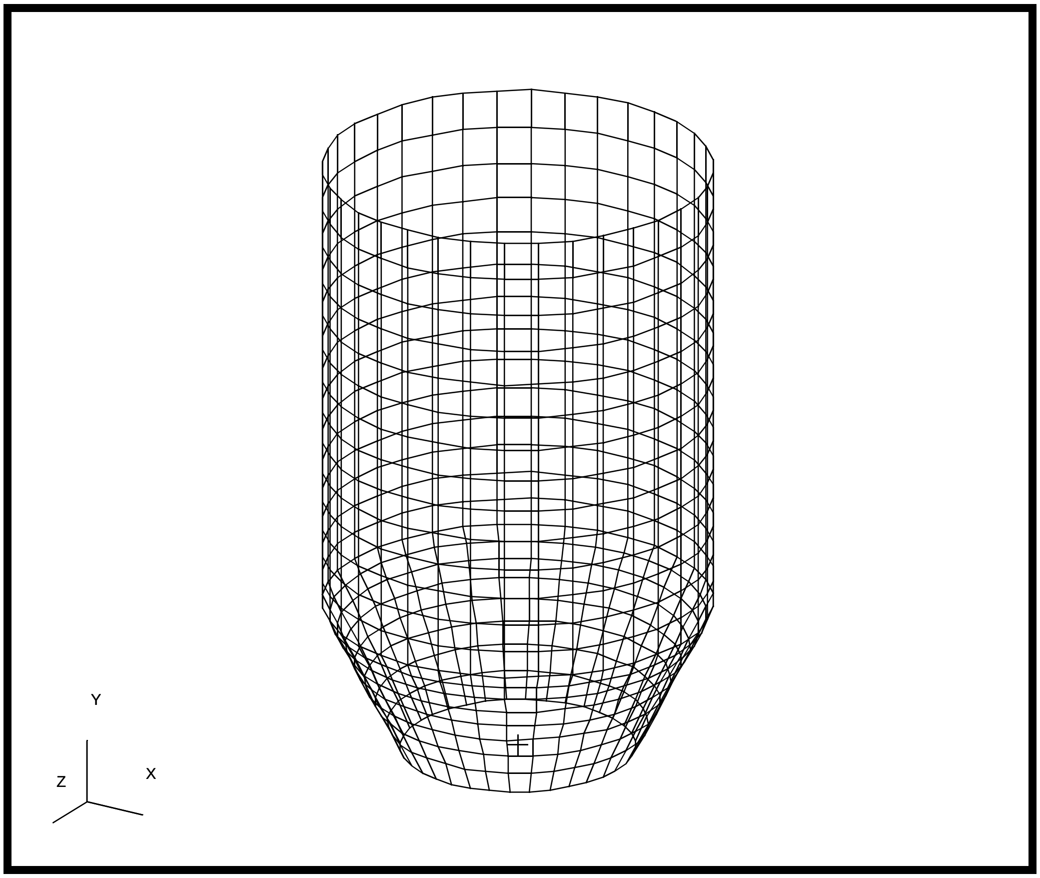
Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Element Topology	**Quad 4**
Surface List	Select All Surfaces

Apply

Turn of display lines and labels by using the appropriate icons.

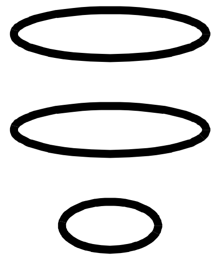
Your model should appear as follows:



Mesh the horizontal (circumferential) edges of each surface with two-noded bar elements.

Action:	**Create**
Object:	**Mesh**
Type:	**Curve**
Element Topology	**Bar 2**
Curve List	

Select the surface edges shown below. A hint on selecting the appropriate edges. Set the view to the default, then use click and drag picking technique.



Also you may want to Erase all FEM in **Display/Plot/Erase...** to make the selection easier.When you are done remember to replot the FEM.

Apply

Equivalence the Finite Elements to reduce the number of elements by eliminating duplicate nodes.

Action:	**Equivalence**
Object:	**All**
Type:	**Tolerance Cube**

Apply

Create Groups

6. Create a group containing only the finite element model. Name the group FEM. Post only that group to the viewport

Group/Create...

New Group Name	**FEM**

Unpost All Other Groups

Group Contents	**Add All FEM**

Apply

Cancel

Create Material Properties

7. Create the materials for the fairing. Materials **alum_1** and **alum_2** will be applied to the top (cylindrical) and bottom (tapered) portions of the fairing respectively. Use Table 14-1 to define the Material Properties.

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name	**alum_1**
Input Properties...	
Constitutive Model:	**Linear Elastic**
Elastic Modulus	**1.05E7**
Poisson's Ratio	**0.33**
Density	**2.6E-4**
Apply	

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name	**alum_2**
Input Properties...	
Constitutive Model:	**Linear Elastic**
Elastic Modulus	**1.18E7**
Poisson's Ratio	**0.33**
Density	**2.4E-4**
Apply	

Create Fields

8. Define fields that represent the varying thickness and temperature distribution. Use Table 14-1 to define the fields.

◆ **Fields**

Action:	**Create**
Object:	**Spatial**
Method:	**PCL Function**
Field Name	**thickness**
Scalar Function	**1.5-'Y/160**

Apply

Action:	**Create**
Object:	**Spatial**
Method:	**PCL Function**
Field Name	**temperature**
Scalar Function	**200.-(150./160.)*'X**

Apply

Create Element Properties

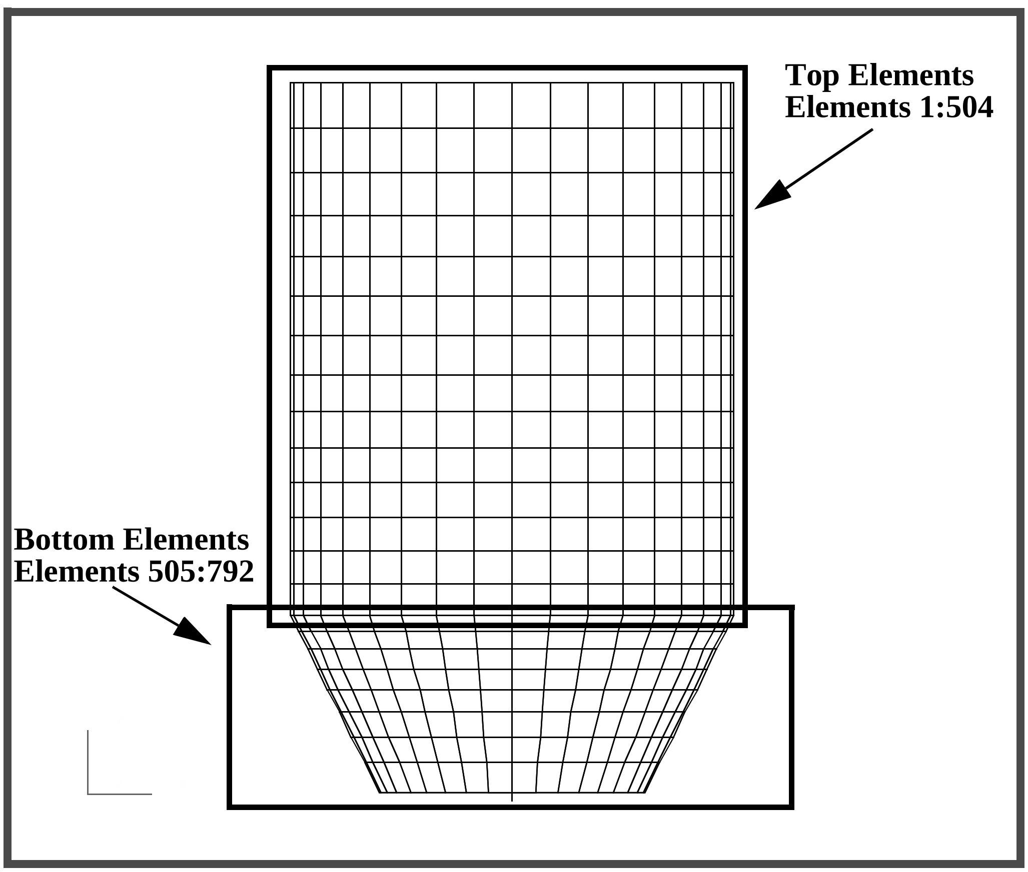
9. Create the element properties which include the material definitions and the varying thickness. Use the names **prop_1** and **prop_2** for the element property names.

Click on the **Properties** radio button in the *Main Form*. Using the information on Table 14-1 create element properties **prop_1** and **prop_2** for the top (cylindrical) and bottom (tapered) portions of the fairing respectively. Apply the element properties to the Quad elements. Use the **thickness** field you defined earlier to represent the varying shell thickness and materials **alum_1** and **alum_2** for the top and bottom portions of the model respectively.

◆ **Properties**

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

Options:	**Homogeneous**
	Standard Formulation
Input Properties...	
Material Name	**m:alum_1**
Thickness	**f:thickness**
OK	
Select Members	Select the Top Elements of the Model. See figure below.
Add	
Apply	



Action:	**Create**

Dimension:	**2D**
Type:	**Shell**
Property Set Name	**prop_2**
Options:	**Homogeneous**
	Standard Formulation
Input Properties...	
Material Name	**m:alum_2**
Thickness	**f:thickness**
OK	
Select Members	Select the Bottom Elements of the Model. See figure on previous page.
Add	
Apply	

Create Temperature Boundary Conditions

10. Define the model's varying temperature distribution. Use the name **temp** for the temperature set name.

◆ **Load/BCs**

Action:	**Create**
Object:	**Temperature**
Type:	**Nodal**
New Set Name	**temp**
Input Data...	
Temperature	**f:temperature**
OK	
Select Application Region...	
Geometry Filter	◆ **FEM**
Select Nodes	Select All Nodes
Add	
OK	

Apply

Turn off the temperature labels

Display/Load/BC/El. Props...

Loads/BCs ❑ **Temperature**

Apply

Cancel

11. Use Lists and Groups to filter then group the quad elements that have the following attributes:

    Material:alum_1 (MATRL.1 if you imported the model)

    Thickness:> 0.98

    Temperature: > 230.0

Add to *List A* the elements which have the alum_1 (MATRL.1) material as one of their attributes.

Create Lists

Tools/List/Create...

Model:	**FEM**
Object:	**Element**
Method:	**Attribute**
Attribute	**Material**
Existing Materials	**alum_1**
Target List	◆ **A**

Apply

Next, you will define *List B* to include only the Quad elements that have a ***thickness greater than 0.98***.

◆ **Properties**

Action:	**Show**
Existing Properties	**Thickness**

Display Method	**Scalar Plot**
Select Groups	**FEM**

Apply

Tools/List/Create...

Model:	**FEM**
Object:	**Element**
Method:	**Attribute**
Attribute	**Fringe Value**
Fringe Tools:	**default_Fringe**
F >	**0.98**
Target List	**B**

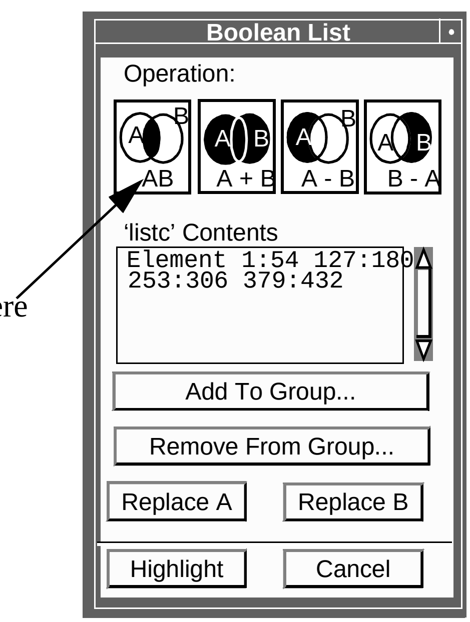
Apply

Intersect Lists

Next, you will intersect Lists A and B and replace the contents of List A with the elements found in the intersection.

Tools/List/Boolean...

On the form that appears click on the intersect icon. The form should appear as follows:



Click here

To transfer the contents of List C to List A, click on the **Replace A** button in the *Boolean List* form.

List A currently satisfies the first two of our three conditions: Quad elements associated with material alum_1 *(MATRL.1)* and having thickness > 0.98.

Now you will perform a final classification of the elements. You will isolate those elements that satisfy the third condition of applied temperature load > 230.0.

◆ **Load/BCs**

Action:	**Plot Contours**
Object:	**Temperature**

If you have imported the model from the neutral file, you need to switch the current load case to Load_Case.1 to be able to select the temperature boundary condition.

Existing Sets	**temp (TEMPN.1.1)**
Select Data Variable	**Temperature**
Select Groups	**FEM**

Apply

Tools/List/Create...

Model:	**FEM**
Object:	**Element**
Method:	**Attribute**
Attribute	**Fringe Value**
Fringe Tools:	**default_Fringe**
F [>]	**230.0**
Target List	◆ **B**

Click on the **Clear** button in the *List B* form.

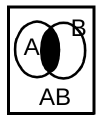
Apply

Add List to Group

In the last portion of this step, you will intersect Lists A and B again to create List C. This will provide you with a list of elements that satisfy all 3 of the conditions. You will then put the contents of List C into the **common_quads** group.

Tools/List/Boolean...

Click on the *intersect* icon.



Add To Group...

Group Name **common_quads**

Apply

Cancel

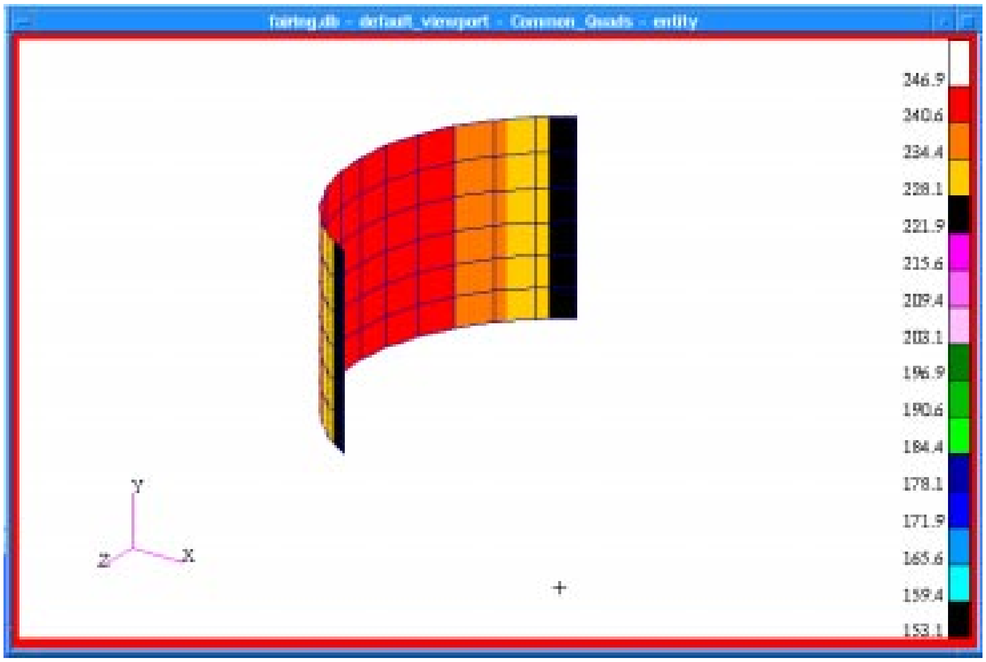
Finally click on **Group** in the *Main Form.*

Group/Post...

Select Groups to Post **common_quads**

Apply

In the *Load/Boundary Conditions* form rerender the temperature contours and Your model should appear as follows:



On the *Load/Boundary Conditions* form, click on the **Reset Graphics** button.

Group Display Method

12. Create two groups by properties containing **prop_1** and **prop_2** respectively. In this step, you will be introduced to Group display mode concept. You will practice how to change the display attributes of a group of entities that represents a collection of different entity types (i.e. quad and bar elements). A major usage of this feature is demonstrated through displaying the same set of entities placed in two different groups in different render styles.

Group/Create...

New Group Name	**prop1_group**
Group Contents:	**Add Entity Selection**

Apply

Now to add the contents to the group you must create a list.

Tools/List/Create...

Model:	**FEM**

Object: **Element**

Method: **Attribute**

Attribute **Property Set**

Existing Property Sets **prop_1**

Apply

Next on the **List A** form select:

Add To Group...

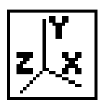
Group Name **prop1_group**

Apply

Cancel

Repeat this process. Label the next group **prop2_group** and select **prop_2** from the *Existing Property Set*. Be sure to clear **List A** before you select **Apply** on the *List Create* form.

Change the view to **Isometric View 1**.



Now render each group with different render styles.

Display/Entity Color/Label/Render...

Entity Coloring and Labeling ◆ **Group**

Target Group **prop1_group**

Render Style **Hidden Line**

Apply

Now that MSC.Patran is in group display mode, you can modify each group's display properties individually.

Target Group **prop2_group**

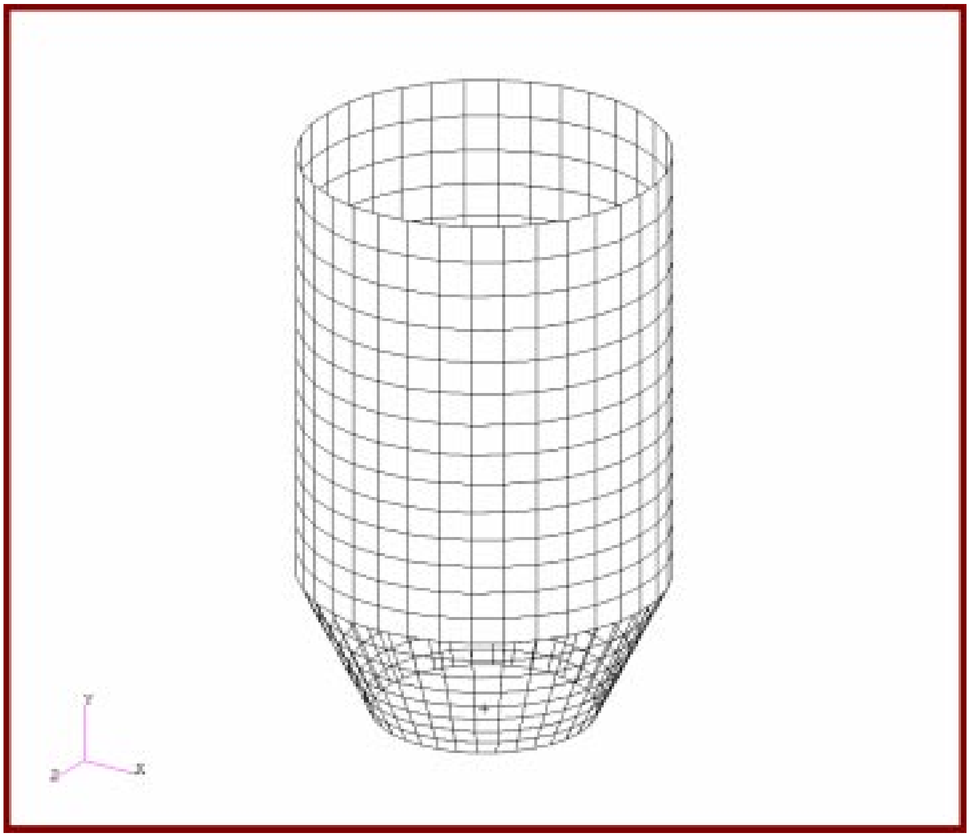
Render Style **Wireframe**

Shade Color: **Yellow**

Apply

Display each group separately using **Group/Post...** Note how the same set of entities can be displayed in different render styles. This feature proves to be extremely useful in the results post-processing. An example would be to display different results on the same set of finite elements, such as stress and temperature.

This figure shows both groups posted at once.



File/Quit...