LESSON 15

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.
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 15-1
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 15-1.

- 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° angles about the center-line of the fairing in 4 steps using the **Surface Revolve** method.

---

**Table 15-1**

<table>
<thead>
<tr>
<th>Analysis Code</th>
<th>P3/FEA</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element Types</td>
<td>Bar2 (horizontal fairing edges) Quad4 (fairing surface)</td>
</tr>
<tr>
<td>Material Name</td>
<td>Alum_1 Alum_2</td>
</tr>
<tr>
<td>Modulus of Elasticity, E (psi)</td>
<td>1.05E7 1.18E7</td>
</tr>
<tr>
<td>Poisson’s Ratio, ν</td>
<td>0.33 0.33</td>
</tr>
<tr>
<td>Density, ρ (lb/in3)</td>
<td>2.6E−4 2.4E−4</td>
</tr>
<tr>
<td>Model Thickness</td>
<td>1.5 - Y/160.</td>
</tr>
<tr>
<td>Model Temperature Distribution</td>
<td>200.- (150./160.)X</td>
</tr>
</tbody>
</table>
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, \( L_2 = 10, L_1 = 7 \)

Curve 2, \( L_2 = 7, L_1 = 4 \)

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 15-1 to define the Material Properties.

- Define fields that represent the varying thickness and temperature distribution. Use Table 15-1 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 Temperature 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:

<table>
<thead>
<tr>
<th>Group</th>
<th>Render Style</th>
<th>Shade Color</th>
<th>Entity Labels</th>
</tr>
</thead>
<tbody>
<tr>
<td>FEM</td>
<td>Hidden Line</td>
<td>Cyan</td>
<td>Off</td>
</tr>
<tr>
<td>BARS</td>
<td>Wireframe</td>
<td>Yellow</td>
<td>Off</td>
</tr>
</tbody>
</table>

- 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 Database...**
   
   **New Database Name**
   
   ![New Database Name Input]
   
   **OK**

   **New Model Preference**
   
   **Tolerance**
   
   ![Default Tolerance]

   **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 15-1.
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.

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

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

![Diagram showing mesh creation with 4 node Quad elements]

**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

[Diagram showing surfaces and mesh]
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.

\[
\frac{L_2}{L_1} = 2.5
\]

**Action:**
- Create

**Object:**
- Mesh Seed

**Type:**
- One Way Bias

**L1 and L2**
- \( L_1 = 7 \)
- \( L_2 = 10 \)

**Curve List**
- Curve 1

**Apply**

**Action:**
- Create

**Object:**
- Mesh Seed

**Type:**
- One Way Bias

**L1 and L2**
- \( L_1 = 4 \)
- \( L_2 = 7 \)

**Curve List**
- Curve 2

**Apply**

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

**Action:**
- Create

**Object:**
- Mesh
Your model should appear as follows:

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

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

Add All FEM
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 15-1 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**

---

8. Define fields that represent the varying thickness and temperature distribution. Use Table 15-1 to define the fields.

**Fields**
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 15-1 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**

<table>
<thead>
<tr>
<th>Action:</th>
<th>Create</th>
</tr>
</thead>
<tbody>
<tr>
<td>Object:</td>
<td>Spatial</td>
</tr>
<tr>
<td>Method:</td>
<td>PCL Function</td>
</tr>
<tr>
<td>Field Name</td>
<td>thickness</td>
</tr>
<tr>
<td>Scalar Function</td>
<td>1.5-’Y/160</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Action:</th>
<th>Create</th>
</tr>
</thead>
<tbody>
<tr>
<td>Object:</td>
<td>Spatial</td>
</tr>
<tr>
<td>Method:</td>
<td>PCL Function</td>
</tr>
<tr>
<td>Field Name</td>
<td>temperature</td>
</tr>
<tr>
<td>Scalar Function</td>
<td>200-.(150./160.)*’X</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Create Element Properties</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Dimension: 2D</td>
<td></td>
</tr>
<tr>
<td>Type: Shell</td>
<td></td>
</tr>
<tr>
<td>Property Set Name prop_1</td>
<td></td>
</tr>
<tr>
<td>Options: Homogeneous</td>
<td></td>
</tr>
<tr>
<td>Standard Formulation</td>
<td></td>
</tr>
</tbody>
</table>
Create Element Properties

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
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*  

*OK*

*Select Application Region...*  

*Geometry Filter*  

*FEM*  

*Select Nodes*  

*Add*

*OK*

*Apply*
Turn off the temperature labels

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

<table>
<thead>
<tr>
<th>Loads/BCs</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Temperature</td>
</tr>
<tr>
<td><strong>Apply</strong></td>
<td></td>
</tr>
<tr>
<td><strong>Cancel</strong></td>
<td></td>
</tr>
</tbody>
</table>

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.

**Tools/List/Create...**

<table>
<thead>
<tr>
<th>Model:</th>
<th>FEM</th>
</tr>
</thead>
<tbody>
<tr>
<td>Object:</td>
<td>Element</td>
</tr>
<tr>
<td>Method:</td>
<td>Attribute</td>
</tr>
<tr>
<td>Attribute</td>
<td>Material</td>
</tr>
<tr>
<td>Existing Materials</td>
<td>alum_1</td>
</tr>
<tr>
<td><strong>Target List</strong></td>
<td>A</td>
</tr>
<tr>
<td><strong>Apply</strong></td>
<td></td>
</tr>
</tbody>
</table>

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

**Properties**

<table>
<thead>
<tr>
<th>Action:</th>
<th>Show</th>
</tr>
</thead>
<tbody>
<tr>
<td>Existing Properties</td>
<td>Thickness</td>
</tr>
<tr>
<td>Display Method</td>
<td>Scalar Plot</td>
</tr>
<tr>
<td>Select Groups</td>
<td>FEM</td>
</tr>
<tr>
<td><strong>Apply</strong></td>
<td></td>
</tr>
</tbody>
</table>
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:
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

\[ F \geq 230.0 \]

*Target List*  
B

Click on the Clear button in the List B form.

**Apply**
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.

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
LESSON 15  Using Groups and Lists

Method:

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.