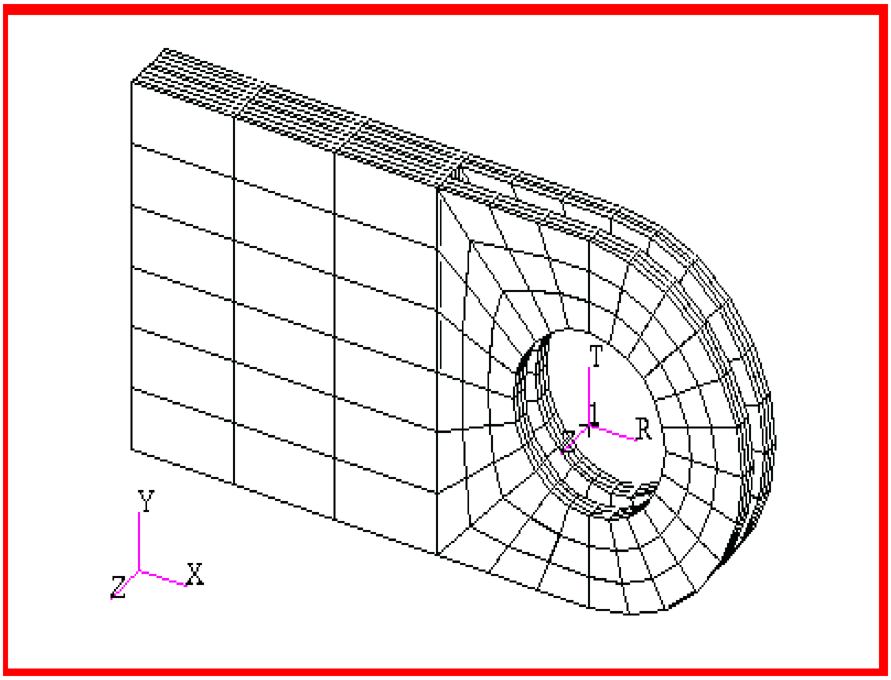
# LESSON 3

# *Finite Element Model of a 3-D Clevis and Property Assignment*



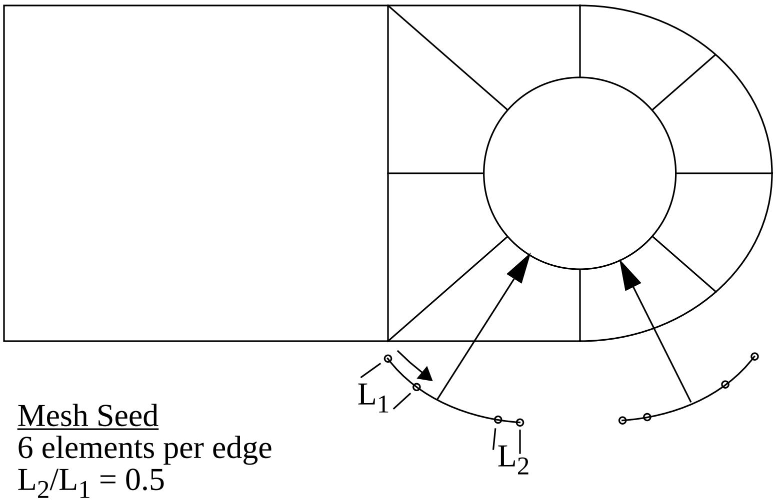

## Objectives:

- Apply a nonuniform mesh seed near a critical location of the model.
- Apply a global mesh to the seeded model.
- Apply material and element properties.

# Model Description:

In this exercise you will define a finite element mesh for the Clevis model you developed earlier. You will use mesh seeding to create a refined mesh with a higher mesh density near the bottom of the hole where you will apply a force load in a future exercise

.




Mesh Seed
6 elements per edge
$L_2/L_1 = 0.5$

Finite Element Mesh
Global Edge Length = 0.5
HEX8 elements

**Figure 3-1**

## Suggested Exercise Steps:

- Start MSC/PATRAN and open the database **`Clevis.db`**.
- Using an isometric view of your model, zoom in on the lower half of the clevis hole. Save this view as a named view. Use the name **zoom_in**.
- To further simplify the rendering of your clevis model you will now turn off the display lines so only the model's boundaries are shown.
- Create the mesh seeds needed to increase the mesh density in the area where the distributed load will be applied.
- Create a finite element mesh using the element topology and size listed in the diagram above.
- Create an Isotropic material, named **Steel**, which uses a Linear Elastic Constitutive Model. The Steel's Elastic Modulus and Poisson's Ratio are respectively 30E6 and 0.30.
- Create a 3-D element property named, **Solid_Elements_Steel**, for the entire model which includes the steel material definition.

## Exercise Procedure:

1. Start MSC/PATRAN and open the database **`Clevis.db`**.

**File/Open...**

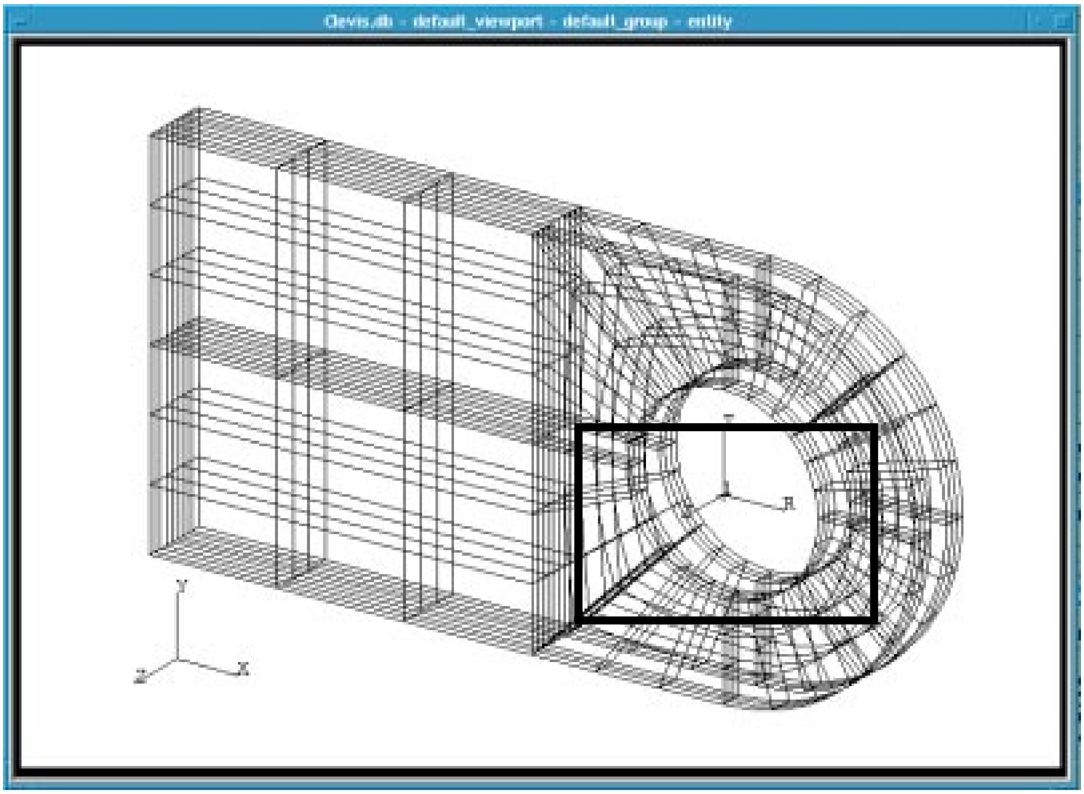
*Existing Database Name:* **clevis.db**

**OK**

1a. Use the **Viewing/Select Corners** option to zoom in on a specific area of the clevis model.

**Viewing/ Select Corners**

When the cursor changes to a plus sign (+) select the lower half of the front clevis hole, as shown in the figure below. Click on a corner of the desired view rectangle (remember, click and hold down the left mouse button) and drag the mouse cursor to the position of the diagonally opposite corner of the view rectangle. Releasing the left mouse button gives the new view.




Since you will need this view in a future exercise, save it by creating a named view of the model's current orientation.

**Viewing/ Named View Options...**

**Create View...**

*Create New View:* **my_view**

**Apply**

**Close**

2. To simplify the rendering of your clevis model, turn off the display lines so only the solid boundaries are shown..

**Display/Geometry...**

*Number of Display Lines:* **0**

**Apply**

**Cancel**

3. Create the mesh seeds needed to increase the mesh density in the area where the distributed load will be applied.

◆ **Finite Elements**

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

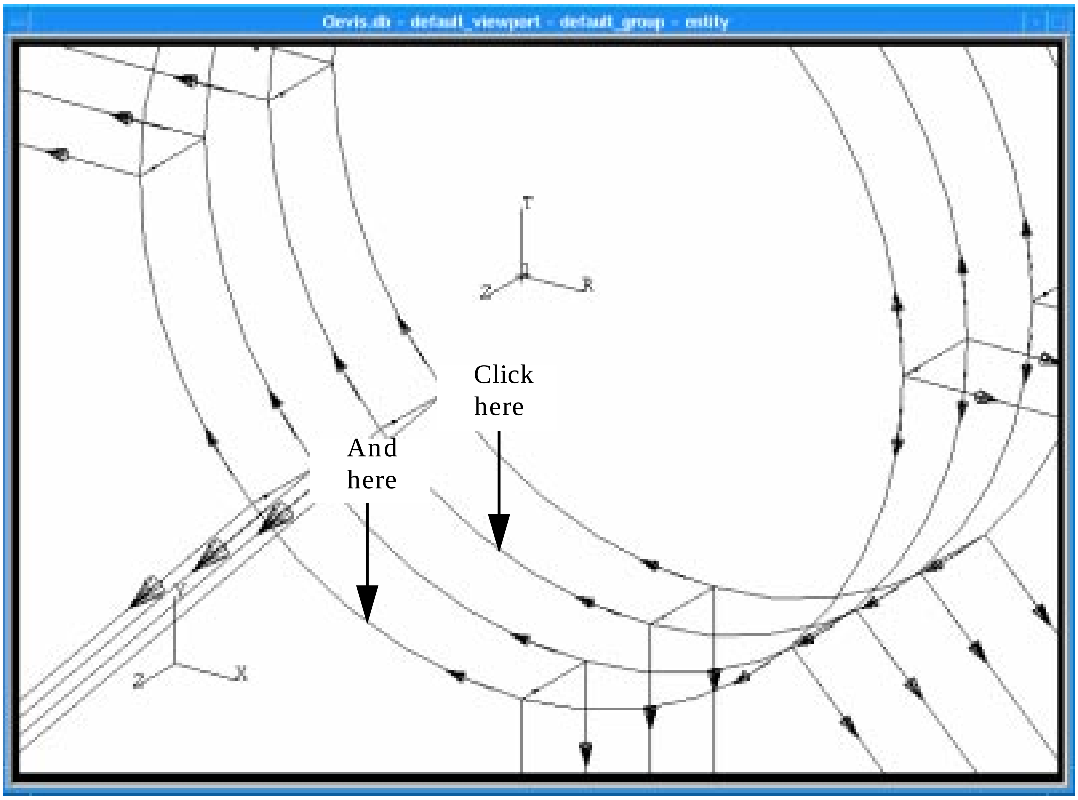
■ **Num and Elems and L2/L1**

Clicking on the **Num Elems** and **L2/L1** switch allows you to specify the number of elements and their varying size along an edge of an entity. The symbol **L2/L1** represents the ratio of the length of the last element to that of the first element along the edge. The directionality of the edges is shown by the cyan arrows that appear on the model when you enter this form.

| | |
|---|---|
| *Number:* | **6** |
| *L2/L1:* | **2** |

❑ **Auto Execute**

In the **Curve List** box, click on the both curves as shown in the figure below (Use the shift+mouse click button to include both curves)..

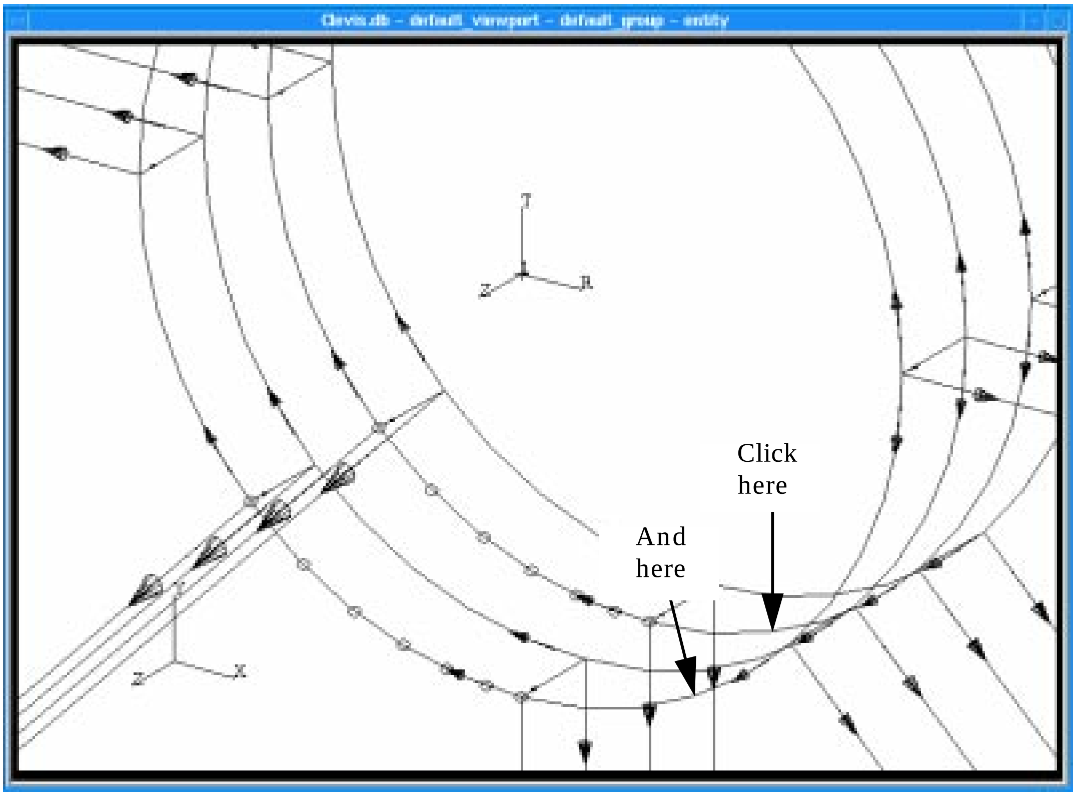



**Apply**

Next, you will select the adjacent edges shown in the following figure. Before you select the edges notice that the directionality of these edges is the same as that of the edges just selected. To obtain a symmetric mesh seed about the lowest point of the hole you must invert the *L2/L1* ratio, by changing its current value to **0.5** (or **-2**). .

| | |
|---|---|
| *Number:* | **6** |
| *L2/L1:* | **0.5** |

In the **Curve List** box, click on the both curves as shown in the figure..




**Apply**

Before creating the model's finite element mesh, zoom out so you can see the entire model.

**Viewing/Fit View**

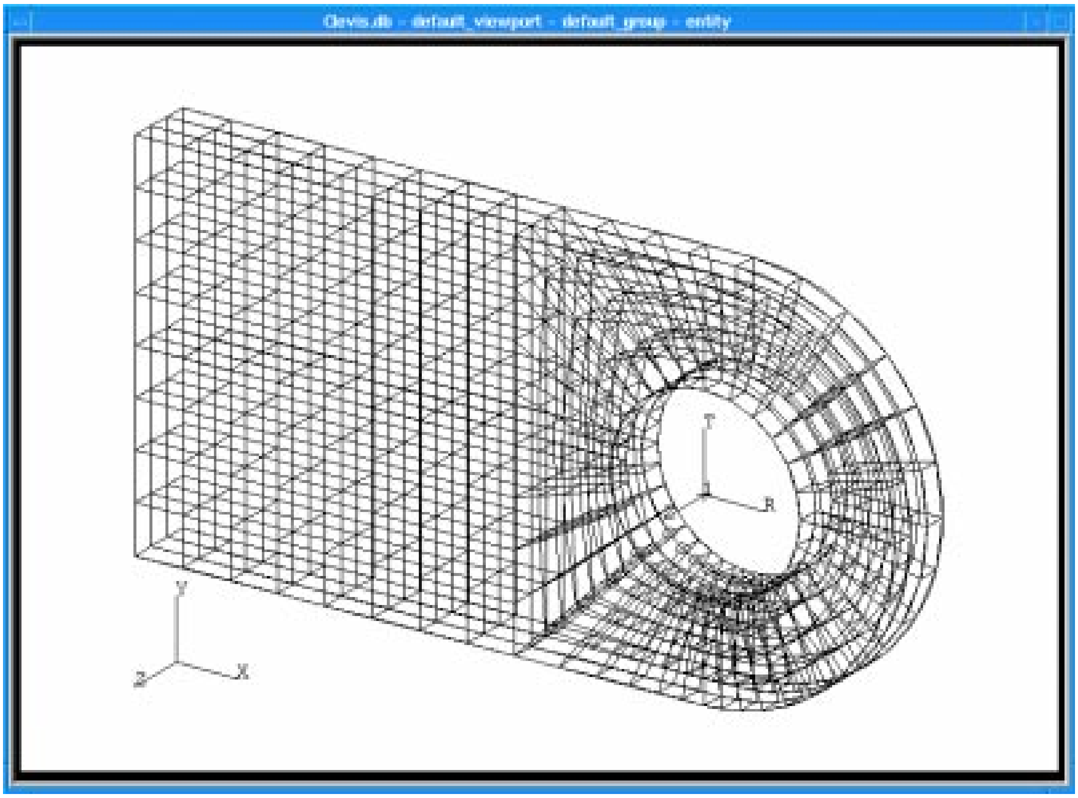
4. Create a finite element mesh using the element topology and size listed below.

◆ **Finite Elements**

| | |
|---|---|
| *Action:* | **Create** |
| *Object:* | **Mesh** |
| *Type:* | **Solid** |
| *Global Edge Length:* | **0.5** |
| *Solid List:* | **Solid 1:22** |

**Apply**

Your clevis model should look like the one shown below.




Now that you have created your finite element mesh, it is time to determine whether you need to "equivalence the model". To do this:

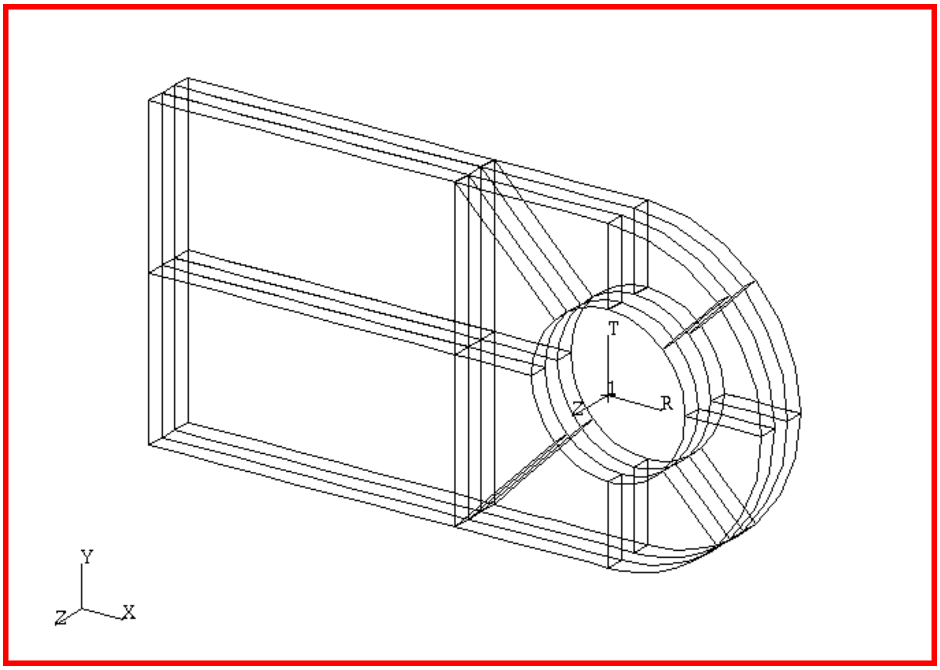
◆ **Finite Elements**

| | |
|---|---|
| *Action:* | **Verify** |
| *Object:* | **Element** |
| *Type:* | **Boundaries** |

◆ **Free Edges**

**Apply**

As shown in the figure below, your model consists of a group of solids residing next to each other in three dimensional space.




Since you don't want your model to be in pieces, you must equivalence your model. Equivalencing results in all the nodes coexisting in the same location, to be reduced to the node with the lowest ID number in that location.

◆ **Finite Elements**

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

**Apply**

You now have one contiguous model of finite elements. To check to that this is true,

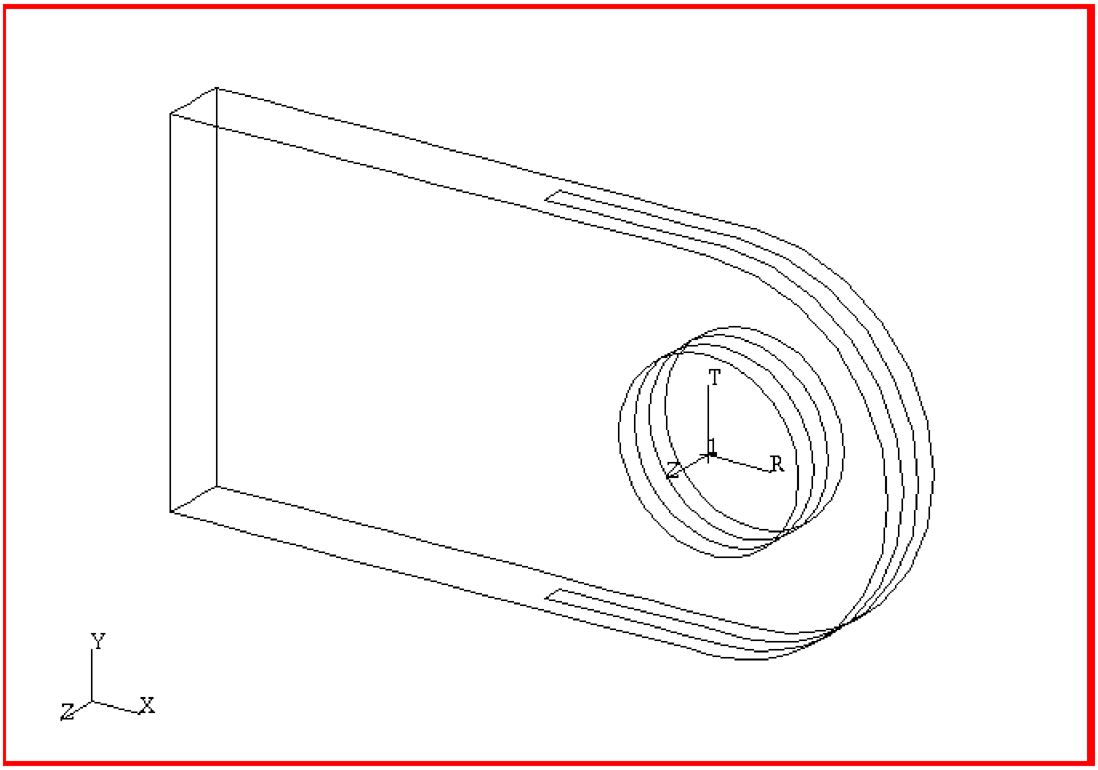
◆ **Finite Elements**

| | |
|---|---|
| *Action:* | **Verify** |
| *Object:* | **Element** |
| *Type:* | **Boundaries** |

◆ **Free Edges**

**Apply**

You should now have a contiguous model as shown in the figure below.




5. Create an Isotropic material, named **Steel**, which uses a Linear Elastic Constitutive Model.

◆ **Materials**

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

**Input Properties...**

| | |
|---|---|
| *Constitutive Model:* | **Linear Elastic** |
| *Elastic Modulus =* | **30e6** |
| *Poisson Ratio =* | **0.3** |

**Apply**

The *Current Constitutive Models* form should appear as below:

**Linear Elastic - [,,,,] - [Active]**

**Cancel**

6. Create a 3-D element property named, **Solid_Elements_Steel**, for the entire model which includes the steel material definition.

◆ **Properties**

| | |
|---|---|
| *Action:* | **Create** |
| *Object:* | **3D** |
| *Type:* | **Solid** |
| *Property Set Name:* | **Solid_Elements_Steel** |
| **Input Properties...** | |
| *Material Name:* | **m:Steel** |
| **OK** | |
| *Select Members:* | **Solid 1:22** |
| **Add** | |
| **Apply** | |

Quit MSC/PATRAN when you are finished with this exercise.