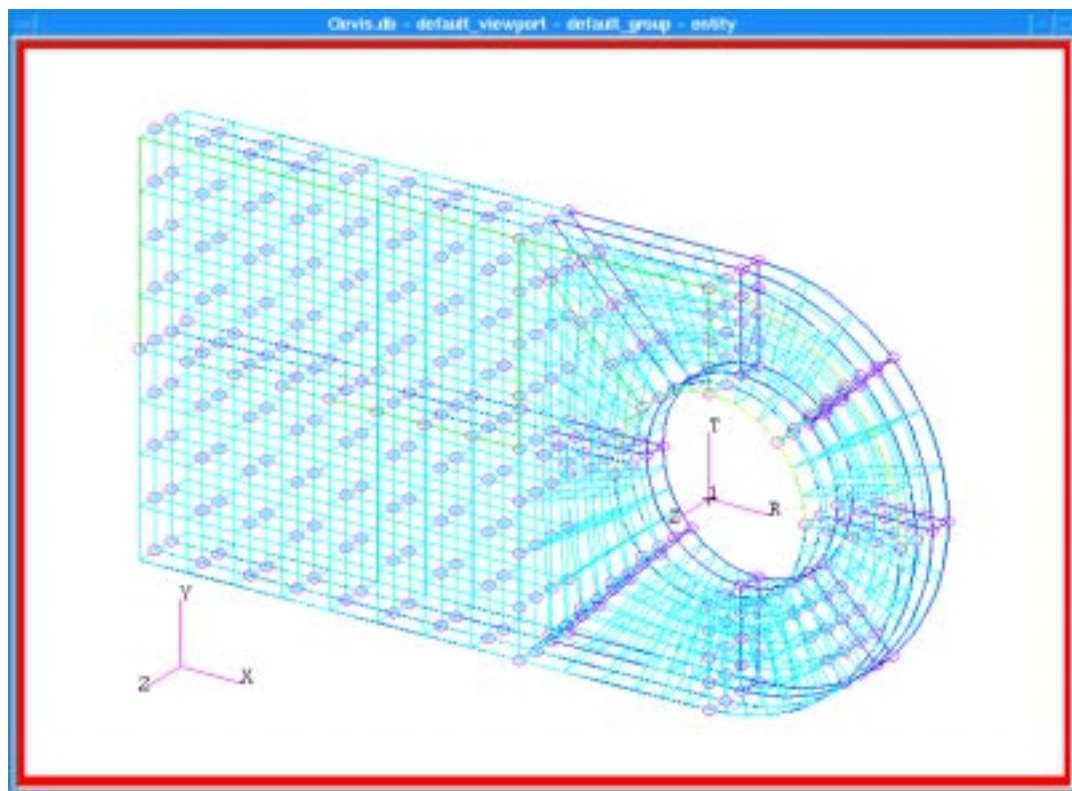


## WORKSHOP 9

# *Verification and Property Assignment*



### **Objectives:**

- Prepare the model for analysis by eliminating duplicate nodes and verifying element attributes.
- Apply material and element properties.



## Model Description:

This exercise continues to prepare the clevis model for analysis. You will remove duplicate nodes, check the hex elements aspect ratio, and optimize the finite element model. You will also input material and element properties for the model.

## Suggested Exercise Steps:

- Start MSC.Patran and open your old file, **clevis.db**.
- Check the model for internal free edges which indicate where cracks exist in your model.
- Equivalence the Clevis model and then verify the element boundaries.
- Verify the Hex element's aspect ratio using a threshold value of 2.0.
- Select the analysis code P3/ADVANCEDFEA.
- 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 your old file, **Clevis.db**.

### File/Open...

Existing Database Name

Clevis.db

OK

2. Check the model for internal free edges which indicate where cracks exist in your model.

### ◆ Finite Elements

Action:

Verify

Object:

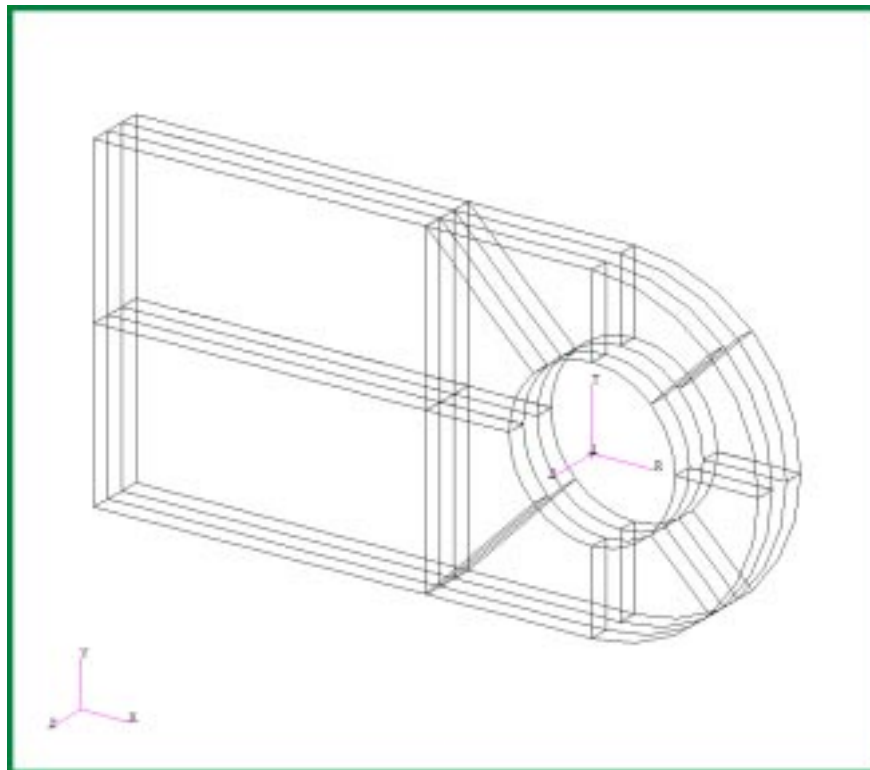
Element

Test:

Boundaries

Apply

MSC.Patran will render your model's free edges as yellow lines. Your model should look like the one shown below.



Verify  
Element  
Boundaries

Notice that prior to equivalencing, all geometric boundaries appear as free edges (cracks) in your model. **MSC.Patran** defines free edges as an edge that is shared by only one element. Click on the Reset Graphics button on the form to re-render your model in its original render style.

**Reset Graphics**

3. Equivalence the Clevis model and then verify the element boundaries.

**Equivalence**

By equivalencing the model, all duplicate nodes will be removed. Hence, the finite element model represents the object as a single solid piece.

*Action:*

**Equivalence**

*Object:*

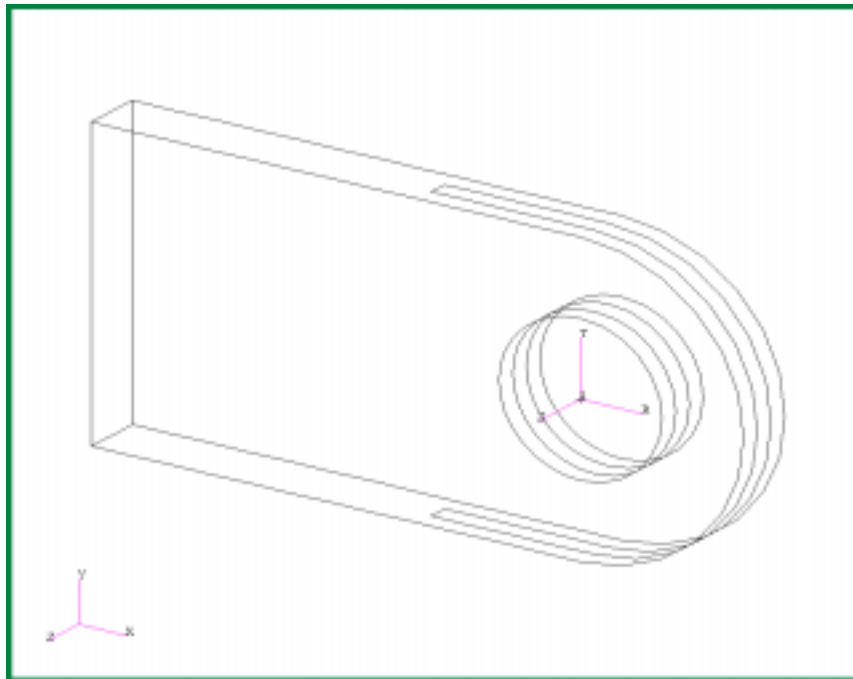
**All**

*Method:*

**Tolerance Cube**

**Apply**

As the equivalencing process proceeds, the locations which have been modified will be identified by circles. Verify the Element Boundaries again. Now you should only see the exterior edges of the clevis, as shown in the figure below.



---

## Aspect Ratio Verification

4. Verify the Hex element's aspect ratio using a threshold value of 2.0.

*Action:*

**Verify**

*Object:*

**Hex**

*Method:*

**Aspect**

Next, set the *Aspect Ratio* slide bar value to approximately **2.0**.

**Apply**

This will color code the Hex elements based on their Aspect Ratio values. Any element with an Aspect Ratio greater than or equal to 2.0 will be colored red according to the default spectrum.

Remember to **Reset Graphics** before performing the next step.

Set the analysis code to MSC/ADVANCED\_FEA.

**Preferences/Analysis...**

*Analysis Code*

**MSC/ADVANCED\_FEA**

**OK**

---

## Selecting an Analysis Code

5. Create an Isotropic material, named **Steel**, which uses a Linear Elastic Constitutive Model. The material's Elastic Modulus and Poisson's Ratio are 30E6 and 0.30, respectively.

◆ **Materials**

<i>Action:</i>	<b>Create</b>
<i>Object:</i>	<b>Isotropic</b>
<i>Method:</i>	<b>Manual Input</b>
<i>Material Name</i>	<b>steel</b>
<b>Input Properties...</b>	
<i>Constitutive Model</i>	<b>Elastic</b>
<i>Elastic Modulus</i>	<b>30E6</b>
<i>Poisson's Ratio</i>	<b>0.3</b>
<b>Apply</b>	

You will know the model has been created when the Current Constitutive Model list is updated.

**Cancel**

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

**Create Element Properties**

◆ **Properties**

<i>Action:</i>	<b>Create</b>
<i>Dimension:</i>	<b>3D</b>
<i>Type:</i>	<b>Solid</b>
<i>Property Set Name</i>	<b>solid_elements_steel</b>
<b>Input Properties...</b>	

In the *Input Properties* form click in the *Material Name* data box. The material properties available for selection will appear in the Material Property Sets list.

Select the proper material from the list. The selected material name will appear with the prefix “m:” in the *Material Name* data box.

Property Name	Value	Value Type
Material Name	m:steel	Mat Prop Name
[Orientation System]		CID
[Orientation Axis]		Real Scalar
[Orientation Angle]		Integer

Material Property Sets

- Steel

OK

**OK**

*Select Members*

Select All Geometry

**Add**

**Apply**

**File/Quit...**