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

Objectives:

- Apply a non-uniform mesh seed near a critical section 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.

Exercise Procedure:

1. Open up the database named clevis.db.

Type p3 in your xterm. The Main Window and Command Window will appear.

File/Open ...

Database List: clevis.db

OK

2. Create a named view of the lower half of the clevis hole.
First, zoom in on the lower half of the hole using the following toolbar icon:

![View Corners](image)

**Figure 2.1 - Region to Zoom in on**

Since this is a region where both the mesh seeds and load will be applied for this model, it only seems fitting that we create a named view of this region to use when we need it.

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

<table>
<thead>
<tr>
<th>Create View ...</th>
</tr>
</thead>
<tbody>
<tr>
<td>Create New view:</td>
</tr>
<tr>
<td>Apply</td>
</tr>
<tr>
<td>Close</td>
</tr>
</tbody>
</table>

3. Lay a biased mesh seed across the bottom half of the hole.

**Finite Elements**

<table>
<thead>
<tr>
<th>Action:</th>
</tr>
</thead>
<tbody>
<tr>
<td>Create</td>
</tr>
</tbody>
</table>

---

2-4  PATRAN 322 Exercise Workbook
Object: Mesh Seed  
Method: One Way Bias  
◆ Num Elems and L2/L1  
Number = 6  
L2/L1 = 2  

Figure 2.2 - First Set of Edges to Place Mesh Seeds on  

select edges in Figure 2.2
\[
\frac{L_2}{L_1} = -2
\]

**Figure 2.3 - Second Set of Mesh-Seeded Edges**

---

**Curve List:** select edges in Figure 2.3

Zoom out to view the entire model using the following toolbar icon:

![Fit View](image-url)

4. Mesh the entire solid, and equivalence the nodes.

**Action:** Create

**Object:** Mesh

**Method:** Solid

**Global Edge Length:** .5

**Mesher:** ♦ IsoMesh

**Solid List:** select all solids
The meshed model in Figure 2.4 should appear:

**Figure 2.4 - Meshed Lug Model**

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**

<table>
<thead>
<tr>
<th>Action:</th>
<th>Create</th>
</tr>
</thead>
<tbody>
<tr>
<td>Object:</td>
<td>Isotropic</td>
</tr>
</tbody>
</table>
6. Create a 3D element property called `steel_solid_elements`, which includes the defined material `steel`.

**Properties**

- **Action:** Create
- **Dimension:** 3D
- **Type:** Solid
- **Property Set Name:** `steel_solid_elements`

You have now created a finite element mesh for the clevis model, including material and element property definitions. Close the database.

File/Close

This ends the exercise.