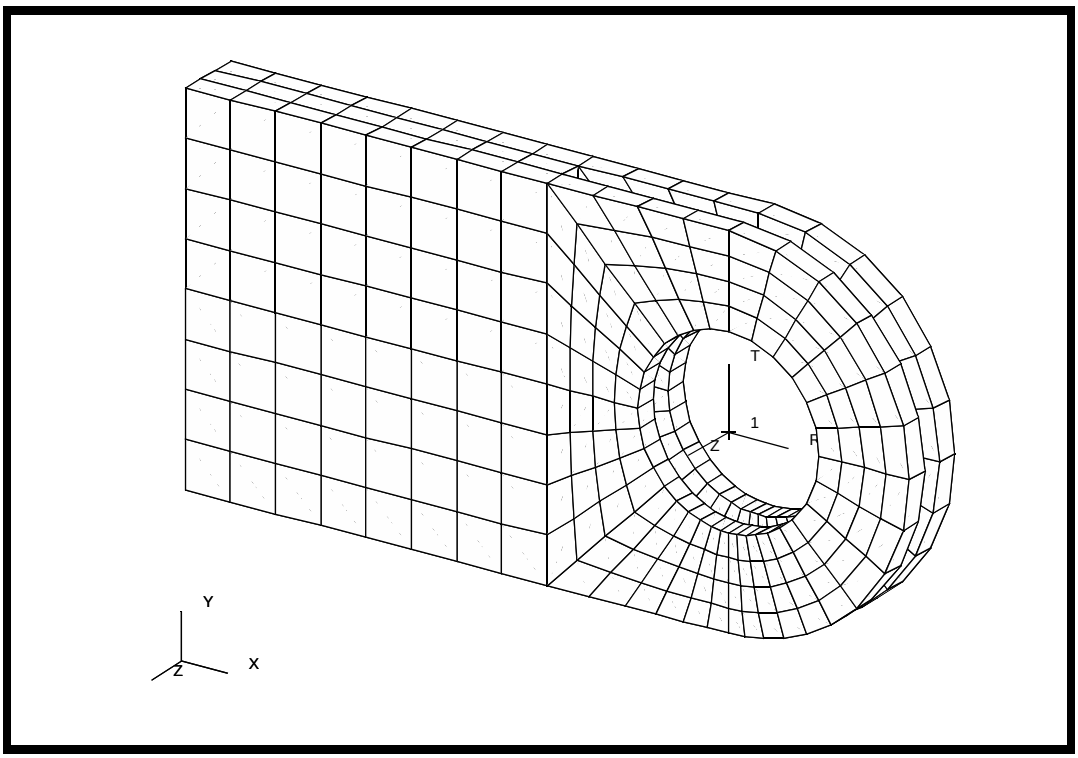
WORKSHOP 2

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

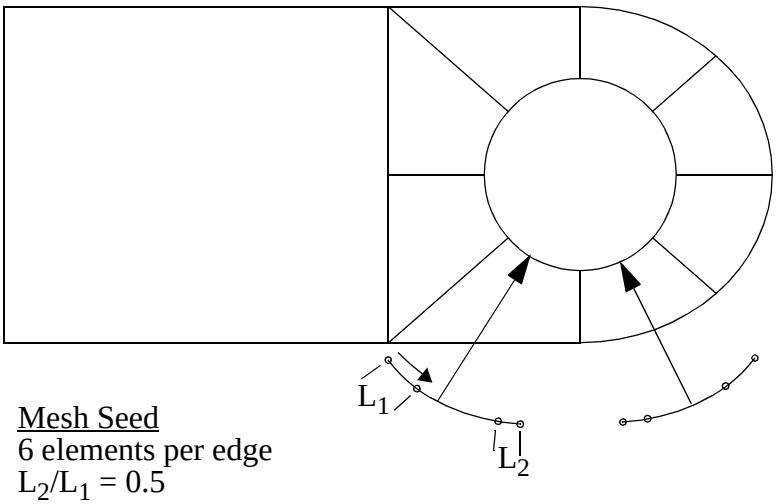


Objectives:

- 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.



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

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**

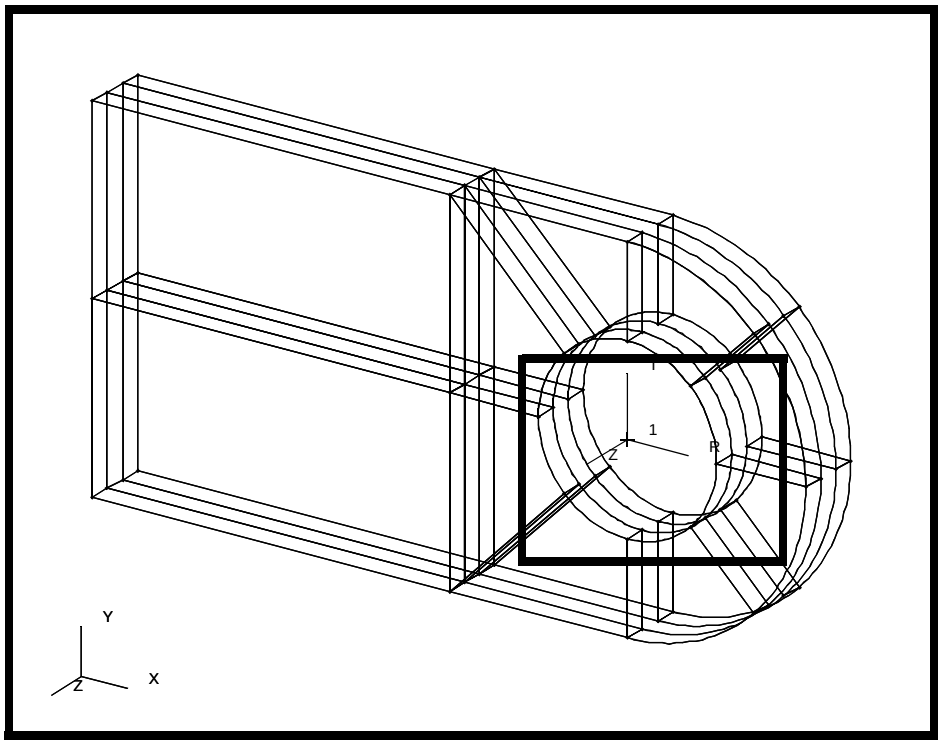
OK

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

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...

Create View ...

Create New view: **my_view**

Apply

Close

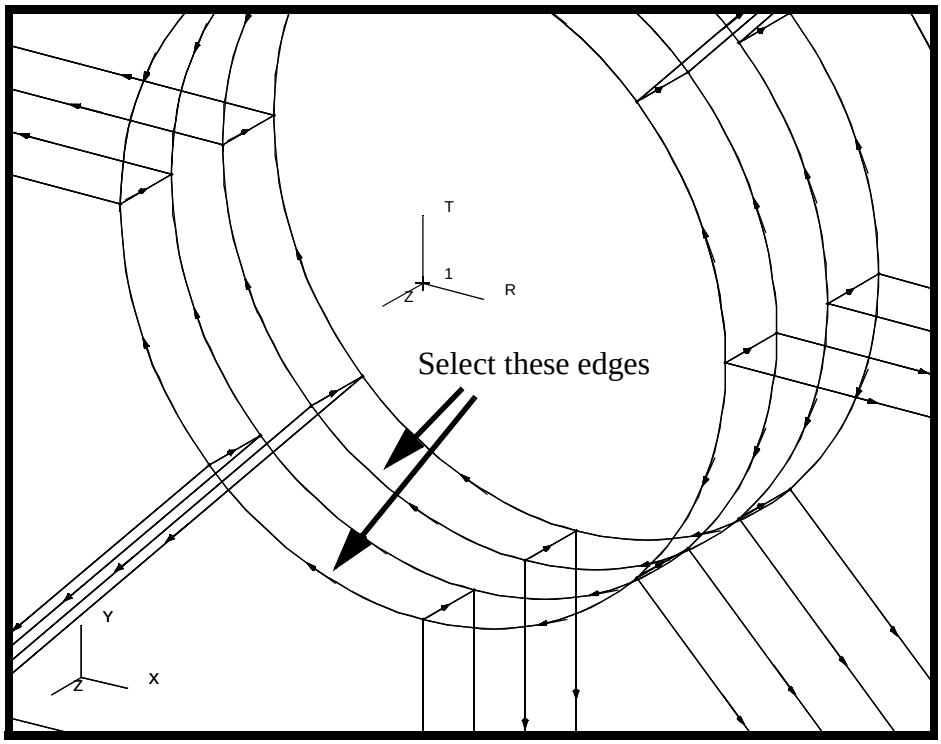
3. Lay a biased mesh seed across the bottom half of the hole.

◆ **Finite Elements**

Action: **Create**

Object:	**Mesh Seed**
Method:	**One Way Bias**
	◆ **Num Elems and L2/L1**
Number =	**6**
L2/L1 =	**2**

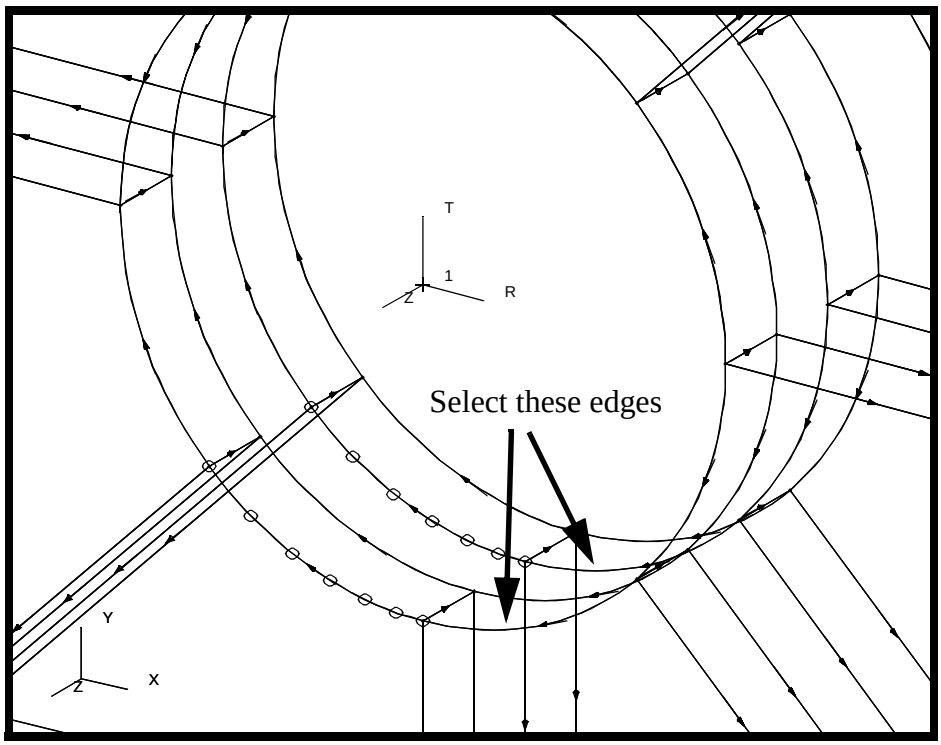
Figure 2.2 - First Set of Edges to Place Mesh Seeds on



Curve List:	select edges in Figure 2.2

L2/L1 =	**-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

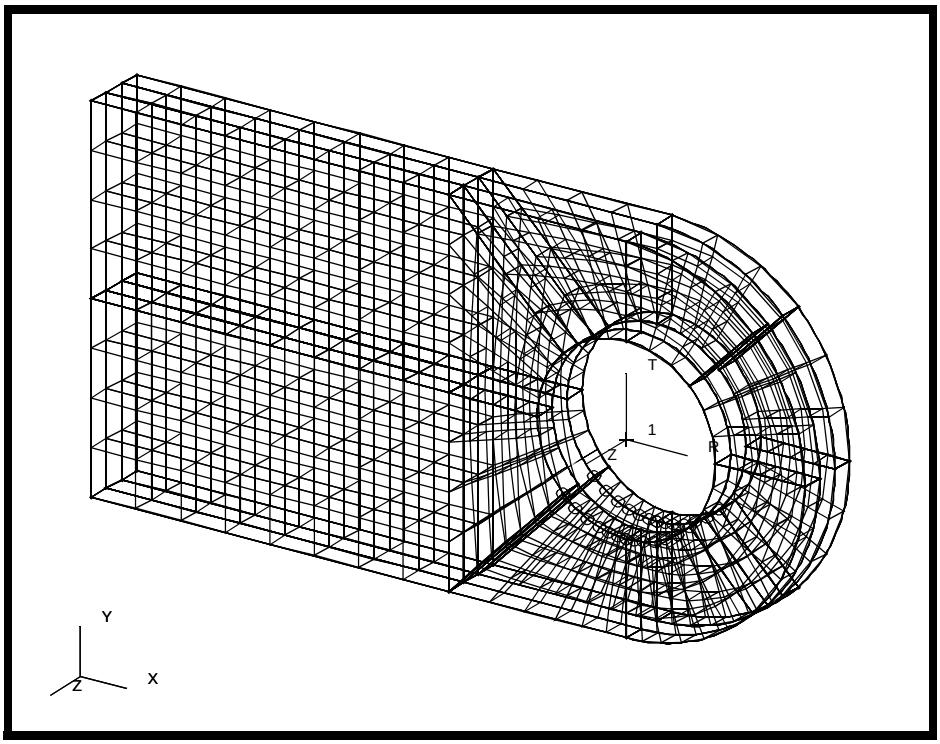
4. Mesh the entire solid, and equivalence the nodes.

Action:	**Create**
Object:	**Mesh**
Method:	**Solid**
Global Edge Length:	**.5**
Mesher:	◆ **IsoMesh**
Solid List:	select all solids

Apply

The meshed model in Figure 2.4 should appear:

Figure 2.4 - Meshed Lug Model



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

Apply

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**

Action:	**Create**
Object:	**Isotropic**

Method: **Manual Input**

Material Name: **steel**

Input Properties ...

Elastic Modulus: **30E6**

Poisson Ratio: **.3**

Apply

Cancel

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**

Input Properties ...

Material Name: **steel**

OK

Select Members: select all solids

Add

Apply

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.