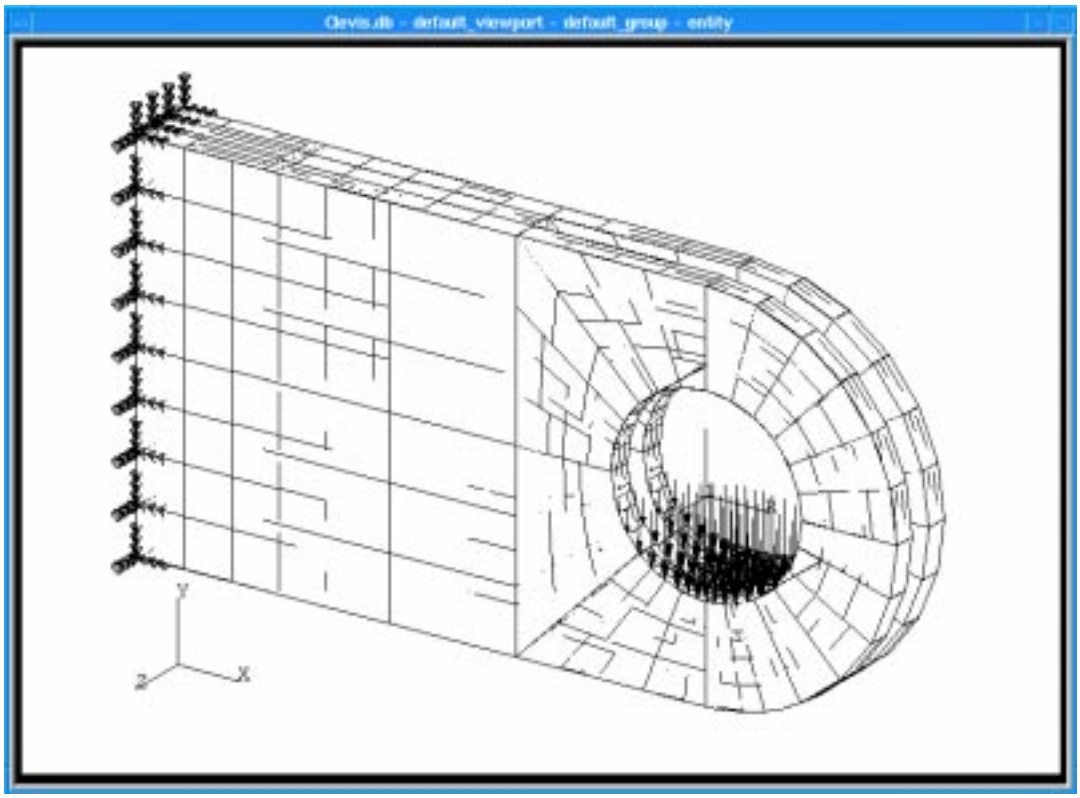
LESSON 11

Loads and Boundary Conditions on a 3-D Clevis



Objectives:

- Apply constraints to your model.
- Create and apply a **Field** to describe a spatially varying load.

Model Description:

In this exercise you will create a loading condition and a constraint set for the clevis model. The base of the lug will be clamped. The hole will be loaded downward with a quadratically varying load $F_y = -100(1-x^2)$ generated from a vector field.

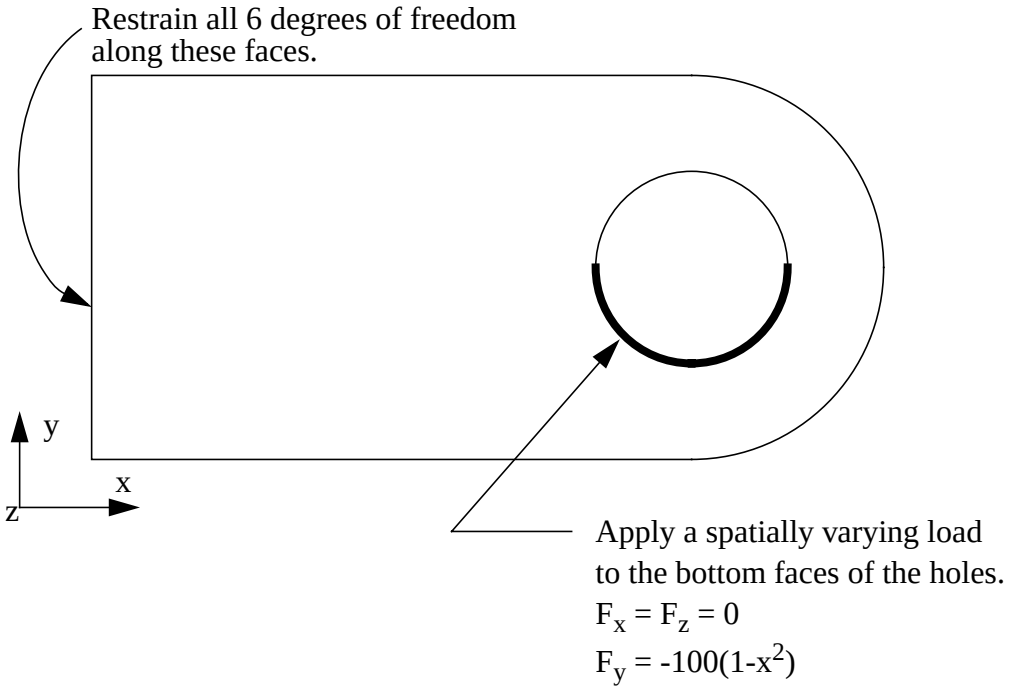


Figure 10-1

Suggested Exercise Steps:

- Open the database, **`Clevis.db`**.
- Create a spatially varying vector field named **Quadratic_load**, using the vector components described in the figure above.
- Create a nodal displacement boundary condition named **Clamped**, which restrains all degrees of freedom. Apply it to the geometry faces shown in the figure above.
- Create a force boundary condition named **Vertical_load**, which uses the **Quadratic_load** field. Apply it to the solid faces along the bottom half of the holes.
- Display both the displacement and force on the finite element model.

Exercise Procedure:

1. Open the database, **`Clevis.db`**.

File/Open Database...

Existing Database Name **Clevis.db**

OK

Fields to Define a Spatially Varying Load

2. Create a spatially varying vector field named **Quadratic_load**, using the vector components described in Figure 10-1.

Fields

Action: **Create**

Object: **Spatial**

Method: **PCL Function**

Field Name **quadratic_load**

Field Type **Vector**

To define the *Vector Function ('X,'Y,'Z)*, click in the *Second Component* databox, and type the equation for the load as shown below. Remember to precede the independent variable, capital X, with a single quote.

-100*(1-'X2)**

Second Component **-100*(1-'X**2)**

Apply

Applying Constraints

3. Create a nodal displacement boundary condition named **clamped**, which restrains all degrees of freedom. Apply it to the geometry faces shown in Figure 10-1.

Loads/BCs

Action: **Create**

Object: **Displacement**

Type: **Nodal**

New Set Name **clamped**

Input Data...

Translations <T1 T2 T3>	**<0, 0, 0>**
Rotations <R1 R2 R3>	**<0, 0, 0>**

OK

Select Application Region...

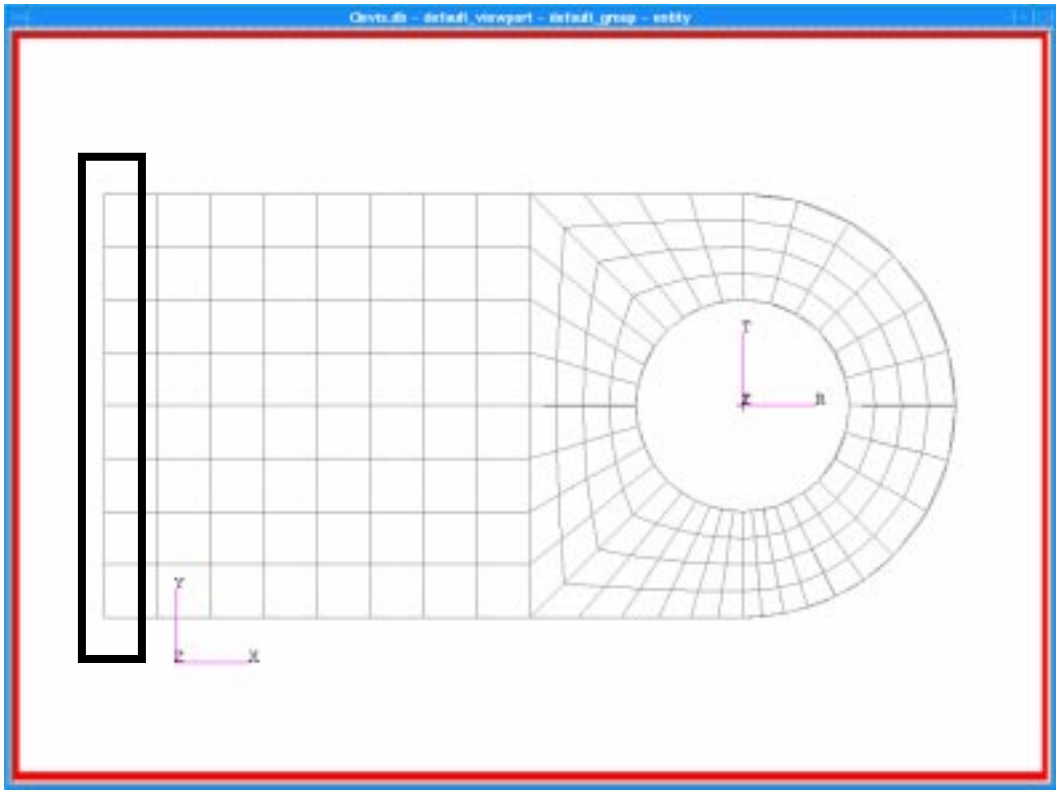
Geometry Filter	**Geometry**
Select Geometry Entities	Select the left side of the model as shown below

To prepare for the application of loads and boundary conditions, you need to orient the model to facilitate cursor picking. Click on the **Front View** icon from the *Main Form,* then select the **Surface** icon from the *Select Menu.*

Front View

Surface

Use the rectangle selection technique (click and drag) to choose the application region for the constraint. Make sure that your *Picking Preference* is set to **Enclose Entire Entity**.



Add

OK

Apply

Vectors showing the constraints will be displayed at the display lines of the solid. If display lines are set to zero, the vectors will appear at the corners of the solid. To remove the vectors click on the Reset Graphics icon in the toolbar.

To redisplay vectors:

Action:	**Plot Markers**
Assigned Load/BC Sets	**Displ-clamped**
Select Groups	**default_group**

Apply

Vectors indicating constraints in translations and rotations will reappear at the solid's display lines**.**

Applying a Spatially Varying Load

4. Create an applied load named **Vertical_load**, which uses the **Quadratic_load** field. Apply it to the solid faces along the bottom half of the holes.

Action:	**Create**
Object:	**Force**
Type:	**Nodal**
New Set Name	**vertical_load**

Input Data...

Force <F1 F2 F3>	**f:quadratic_load**

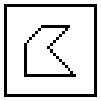
In the *Force <F1 F2 F3>* databox, move the cursor down to the *Spatial Fields* listbox, and select **quadratic_load**.

OK

Select Application Region...

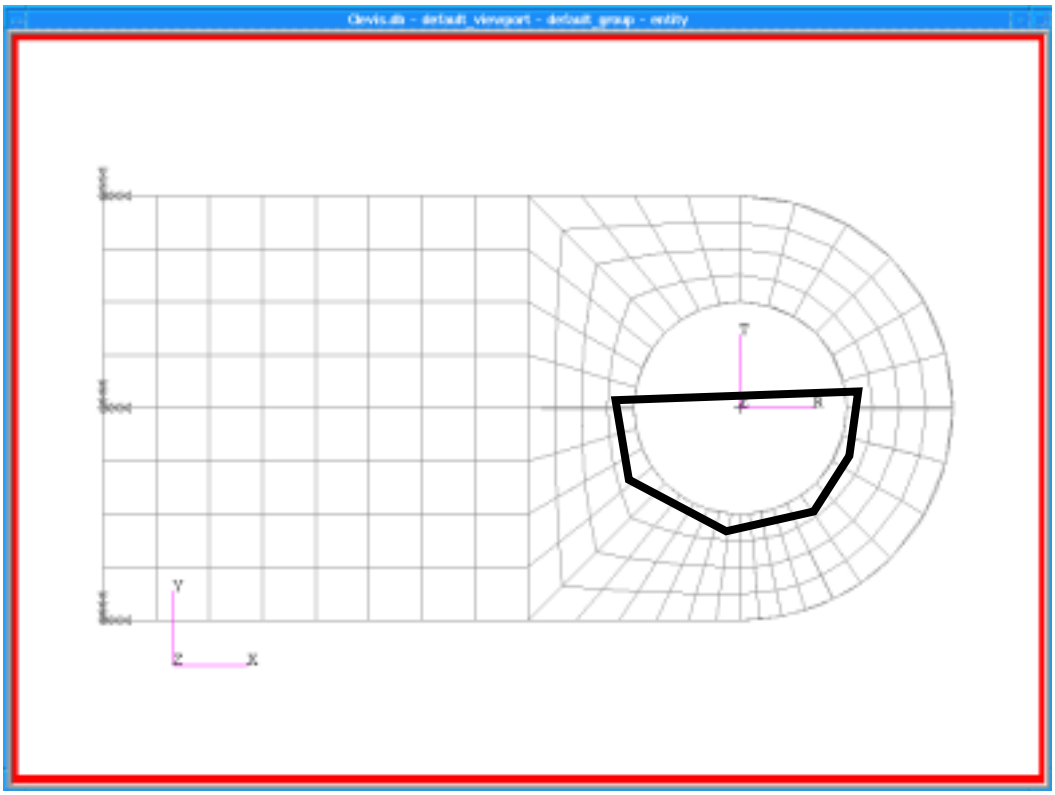
Geometry Filter **Geometry**

Be sure that the **Surface** icon is still highlighted in the *Select Menu.* Click in the *Select Geometry Entities* databox, and select the surfaces for the load. Use the polygon pick method to select the solid faces that bound the bottom half of the holes, as shown in the figure below



Polygon Picking

You can select this icon or hold down **control** while dragging the cursor.

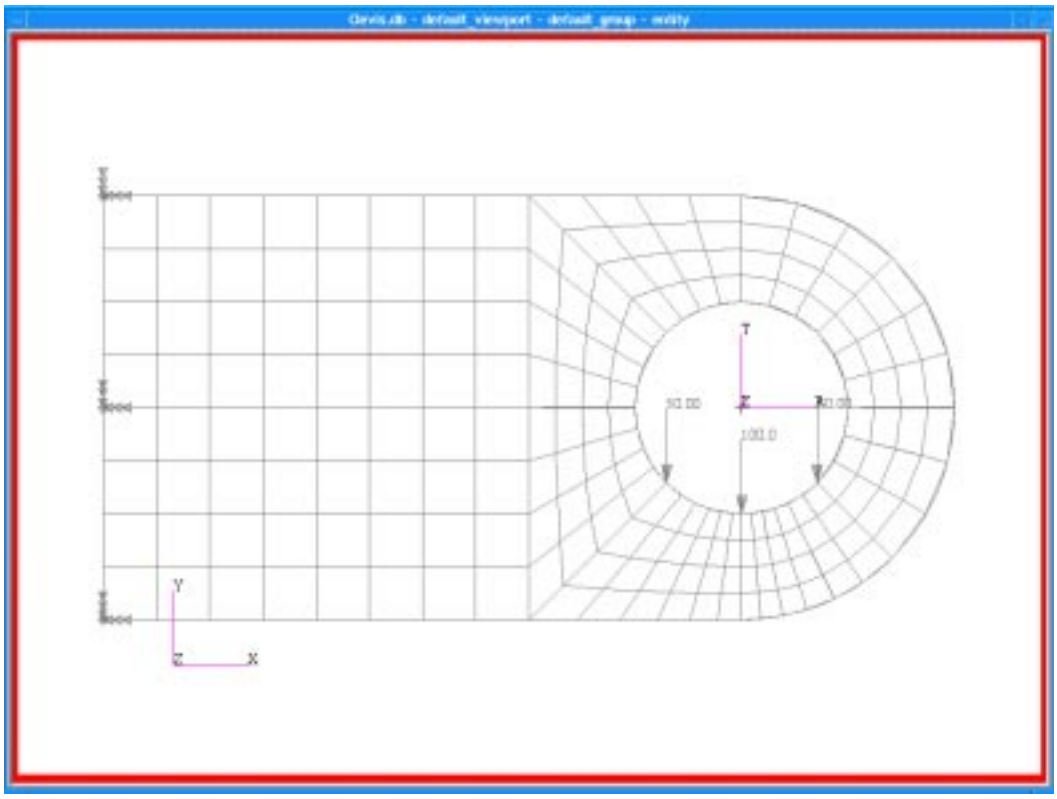


Add

OK

Apply

Vector markers indicating the applied load will appear as shown below. It's ok if your vectors from the clamped end are pointing in the opposite direction.



Display on FEM

5. Display both the displacement and force on the finite element model.

Display/Load/BC/Elem. Props...

Show on FEM Only

Apply

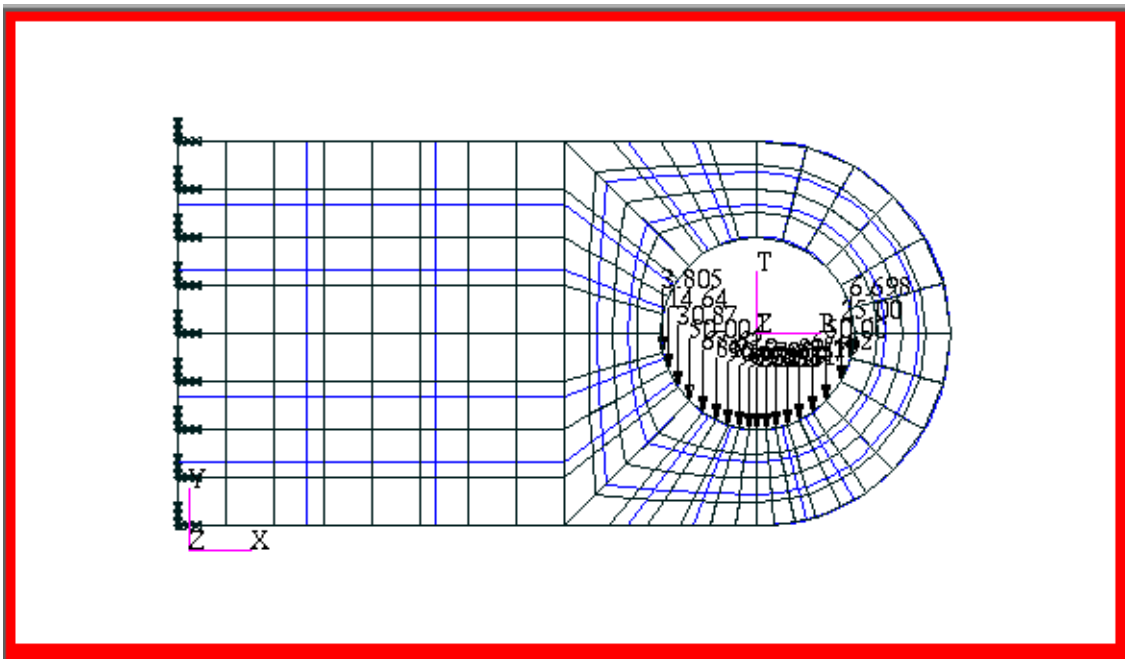
Cancel

In the *Load/Boundary Conditions* form, change the *Action* to **Plot Markers**.

Action:	**Plot Markers**
Assigned Load/BC Sets	**Displ_clamped** **Force_vertical_load**
Select Groups	**default_group**

Apply

The loads and constraints should now be displayed at the nodal locations as shown below (orientation of the constraint arrows may vary).



To turn off the values, use

Labels on Vectors

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

Vectors/Filters...

☐ **Show LBC/El. Prop. Values**

Apply

To see the vectors scaled to the values, go to

◆ **Scaled - Model Relative**

Apply

Cancel

File/Quit