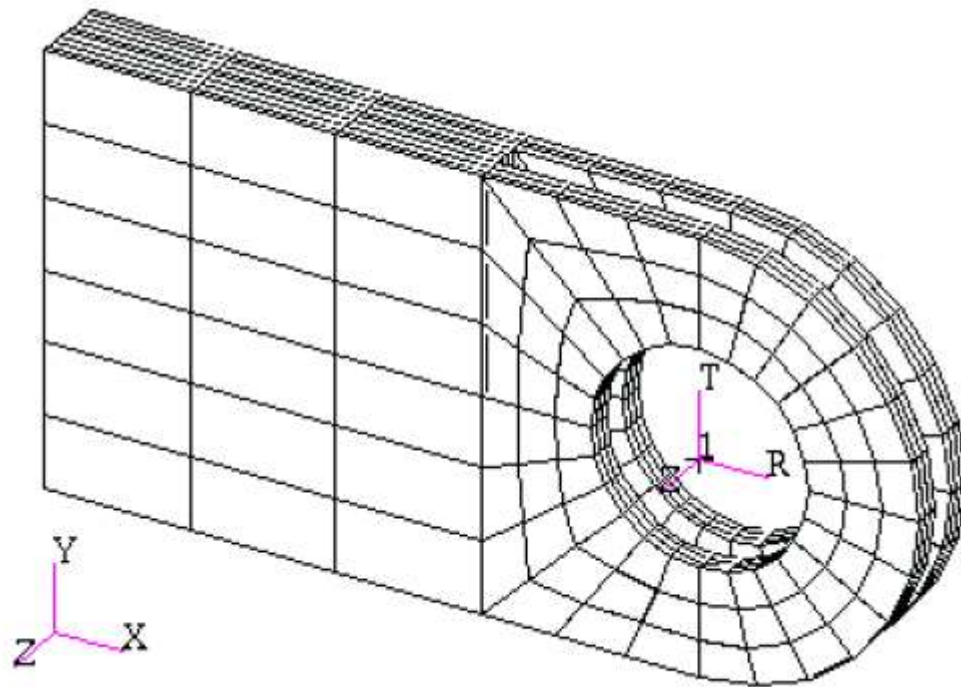


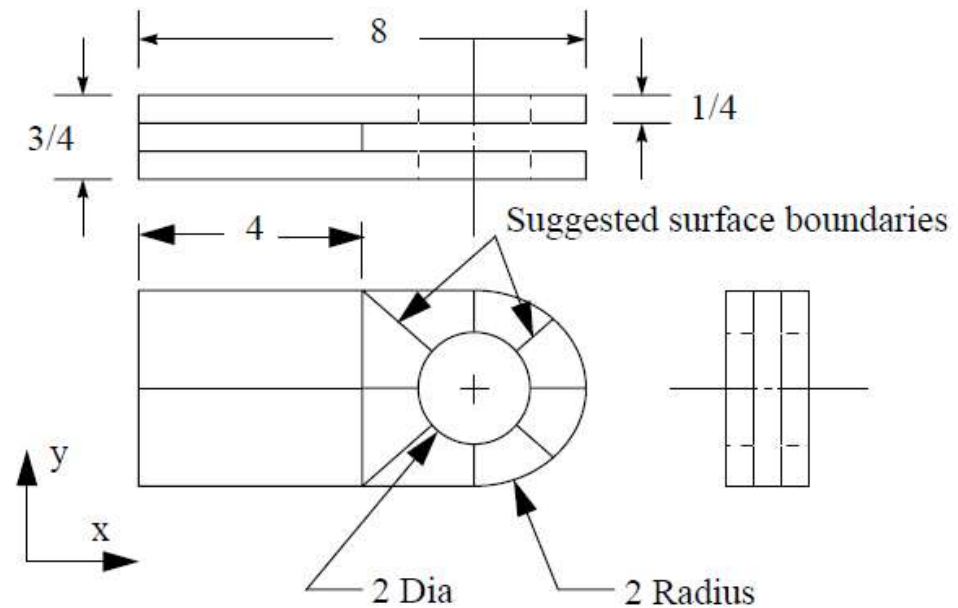
Geometry model of a 3-D Clevis

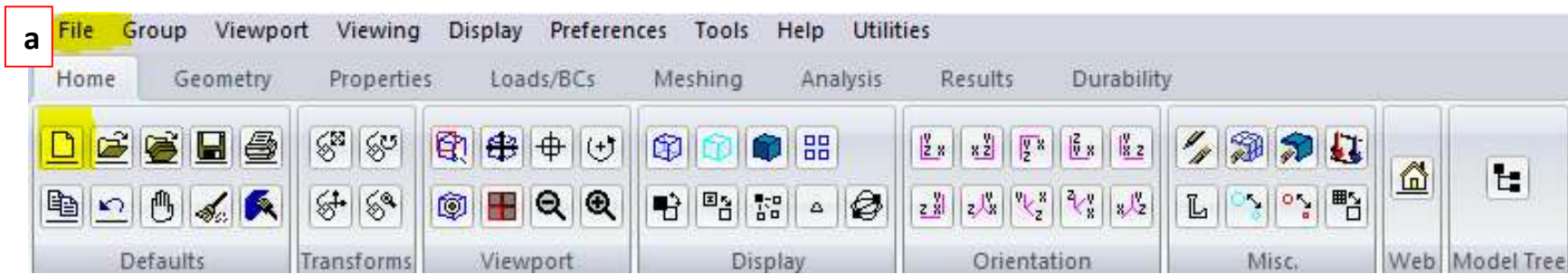


In this exercise you will create an analytic solid model of a clevis by defining MSC/PATRAN points, curves, surfaces, solids, and a user define coordinate system. Throughout this exercise you will become more familiar with the use of the MSC/PATRAN select menu. Shown below is a drawing of the model you will build and suggested steps for its construction.

Suggested Exercise Steps:

- 1) Create a new database and name it **Clevis.db**.
- 2) Create a surface model of the top half of the clevis as shown in the front view on the right side. Place the center of the hole at $[0,0,0]$.
- 3) Create solids that represent the first third of the solid model's total width
- 4) Create the bottom half of your model by mirroring all of the solids about the y-axis mirror plane located at $y=0$.
- 5) Create the remaining solids that represent the last two thirds of your model in the width direction (z-direction).





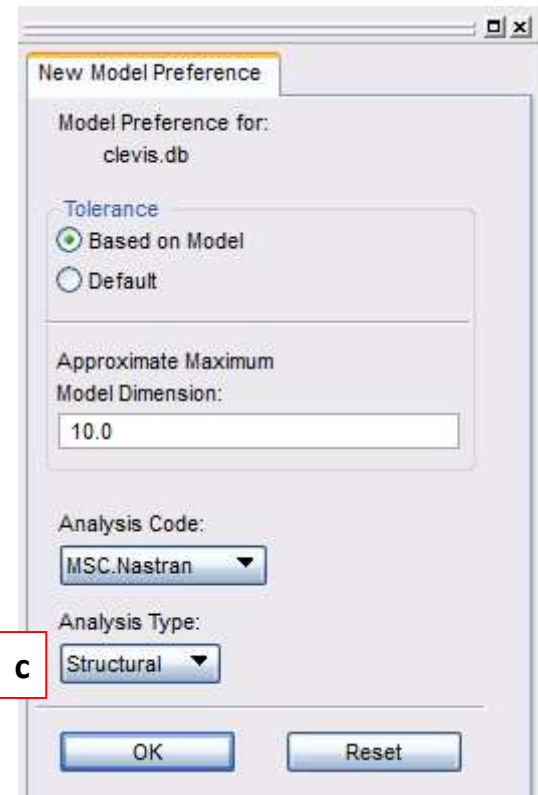
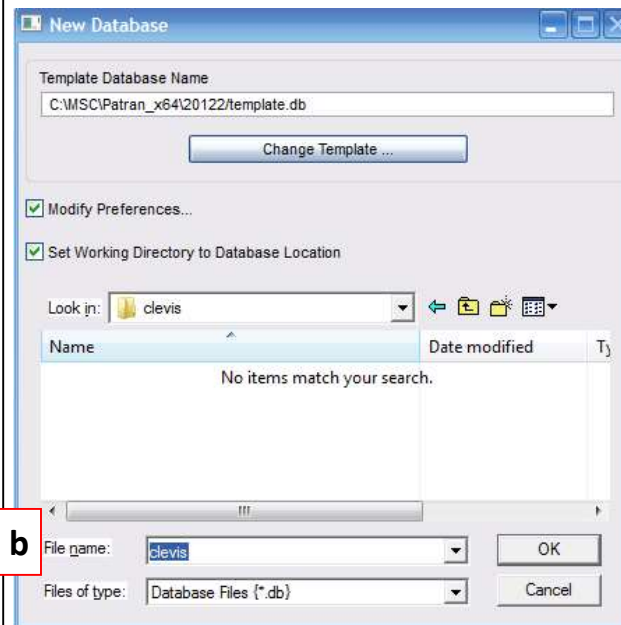
In order to create a new database

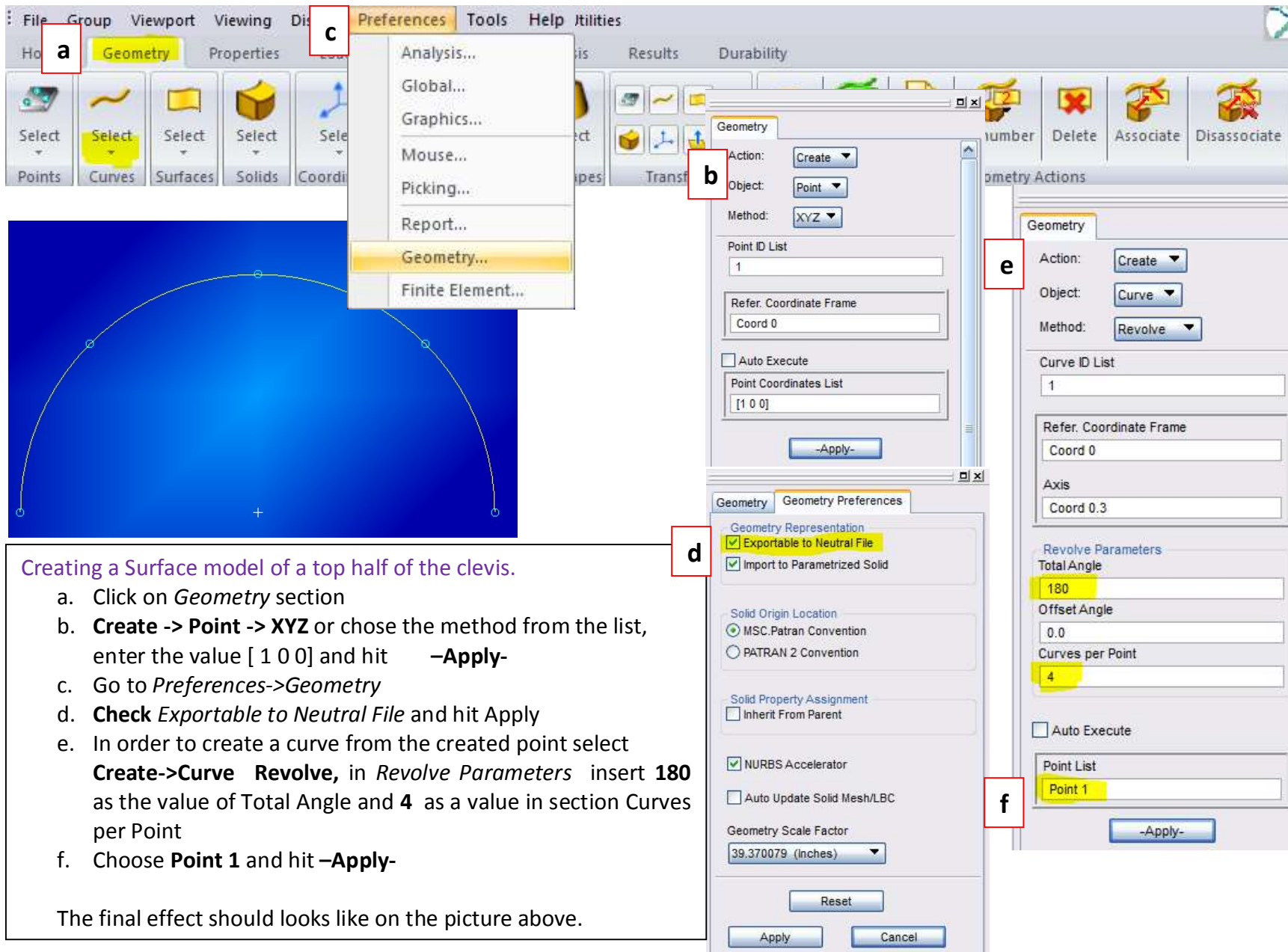
You have to do as follows:

- a. **File / New...** or on a symbol **New** in *Home/Defaults* section
- b. Enter **clevis** as the File name add Click **OK**

/ New Model Preference

- c. Select **Structural** and Click **OK**






Creating a Surface model of a top half of the clevis.

- a. Click on *Geometry* section
- b. **Create -> Point -> XYZ** or chose the method from the list, enter the value [1 0 0] and hit **-Apply-**
- c. Go to *Preferences->Geometry*
- d. **Check *Exportable to Neutral File*** and hit **Apply**
- e. In order to create a curve from the created point select **Create->Curve Revolve**, in *Revolve Parameters* insert **180** as the value of *Total Angle* and **4** as a value in section *Curves per Point*
- f. Choose **Point 1** and hit **-Apply-**

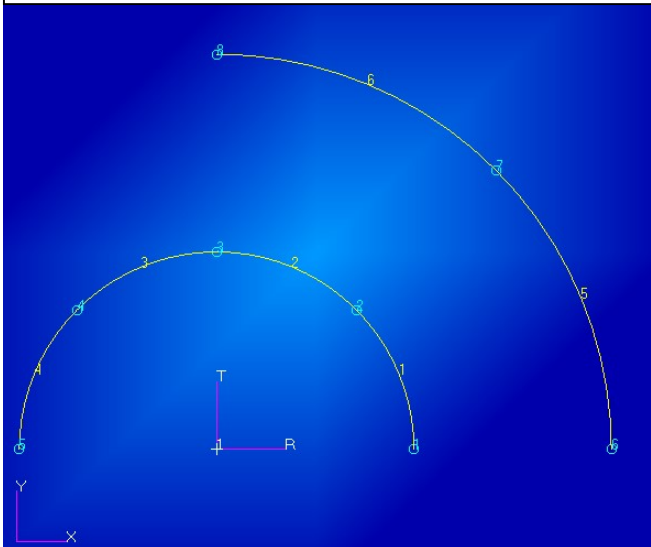
The final effect should looks like on the picture above.

You will now use Curvilinear transformation to create the outer radius of the lug by radially translating the curves that define a quarter of the hole.

To accomplish this you will first need to create a cylindrical coordinate frame located at the center of the hole.

- Create->Coord->3point** chose **Cylindrical** as a Type of Coord and hit **-Appl**
- Transform->curve->Translate** check **Curvilinear in Refer. CF**, click on newly created coord - **Coord 1** as a referenc coordinate frame and **Uncheck** Auto Execute .
- Show labels by pressing  in *Home/Display* section.
- Insert Curve 1 and Curve 2 to the curve list

TIP In order to choose more than one curve HOLD down **L.Shift** while selecting curve



a

Geometry

Action: Create

Object: Coord

Method: 3Point

Coord ID List

2

Type: Cylindrical

Refer. Coordinate Frame

Coord 0

Auto Execute

Origin

[0 0 0]

Point on Axis 3

[0 0 1]

Point on Plane 1-3

[1 0 0]

-Apply-

b

Geometry

Action: Transform

Object: Curve

Method: Translate

Curve ID List

5

Type of Transformation

Cartesian in Refer. CF

Curvilinear in Refer. CF

Refer. Coordinate Frame

Coord 1

Translation Vector

<1 0 0>

Translation Parameters

Repeat Count

1

Delete Original Curves

Auto Execute

Curve List

Curve 1 2

-Apply-

d

You have now created all the curves that you will need to complete your clevis model. Next, you will create the necessary surfaces for the model. You will start by creating a 4x2 (in x Surface that defines part of the upper half of the clevis body



- a. **Create->Surface->XYZ** insert value $\langle -4 \ 2 \ 0 \rangle$ to define Vector, and $[-2 \ 0 \ 0]$ to define point of origin and hit **Apply-**

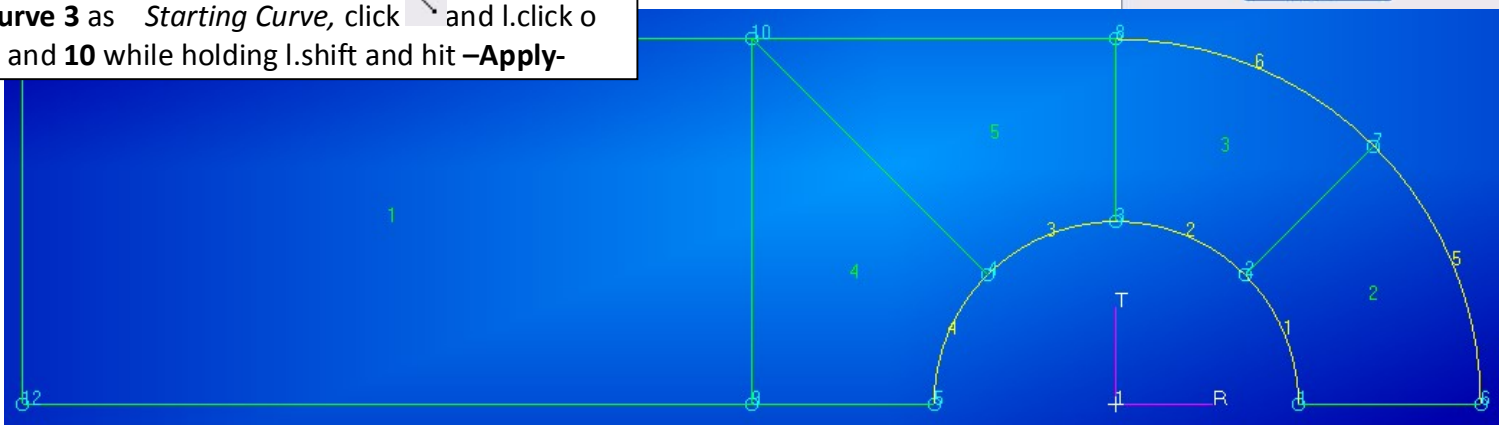
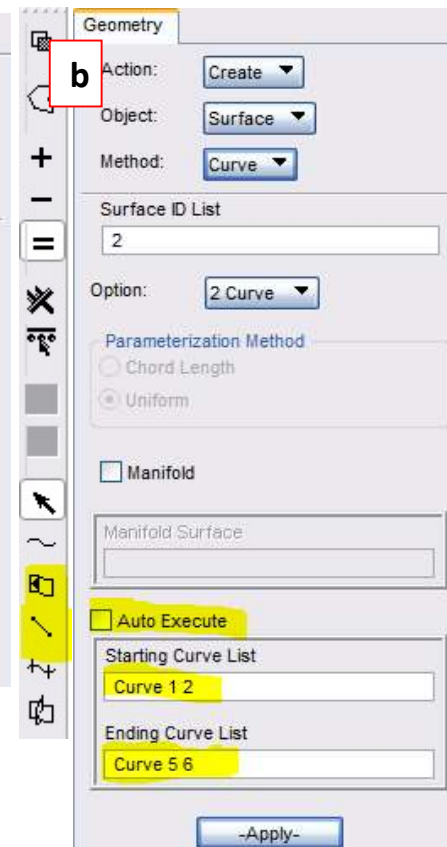
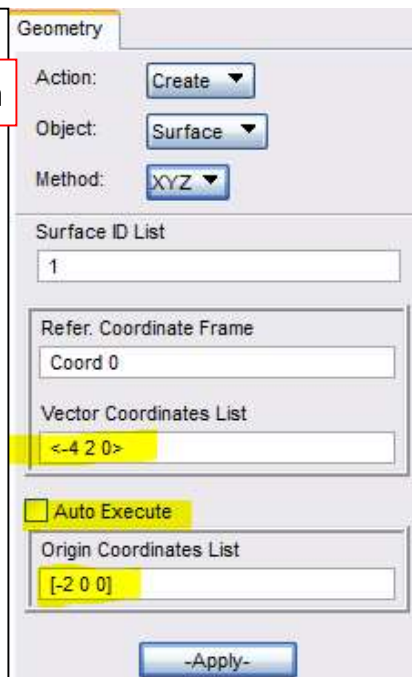
The next series of Surfaces will be created using the *Curve* Method:

- b. **Create->Surface->Cur** , **uncheck Auto Execute** and select **Curve 1** and **Curve 2** in a *Starting Curve List* section and **Curve 5 6** as a *Ending* and hit **Apply-**

Click on  in the *Home/Misc.* section to display the lines.


To create the next surface you will use the Select Menu to help you define an existing curve and surface edge as the boundaries of the new surface.

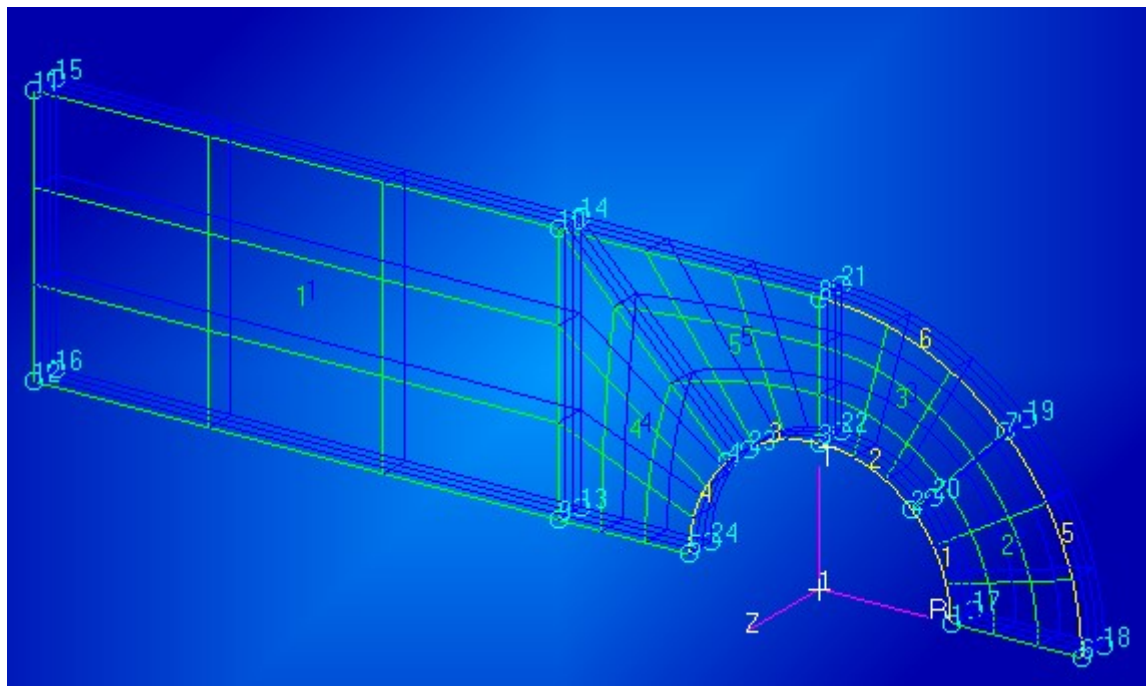
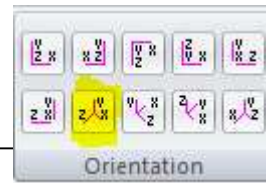
- c. Chose **Curve 4** as *Starting Curve*, click  and I.click on edge 9-10 of a surface 1 and hi **Apply-**
- d. Chose **Curve 3** as *Starting Curve*, click  and I.click o **Points 8** and **10** while holding I.shift and hit **Apply-**



You will now use the Surfaces you have just created as patterns to define solids (3-dimensional entities)

- a. **Create->Solid- Normal** , insert **0.25** as a Thickness, **Uncheck** Auto execute
- b. Chose all surfaces and click **-Apply-**

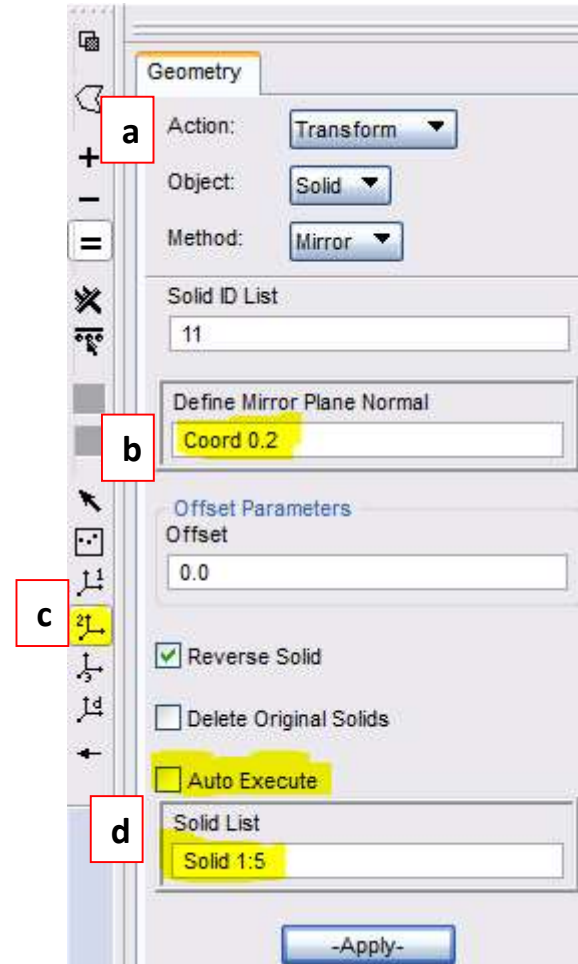
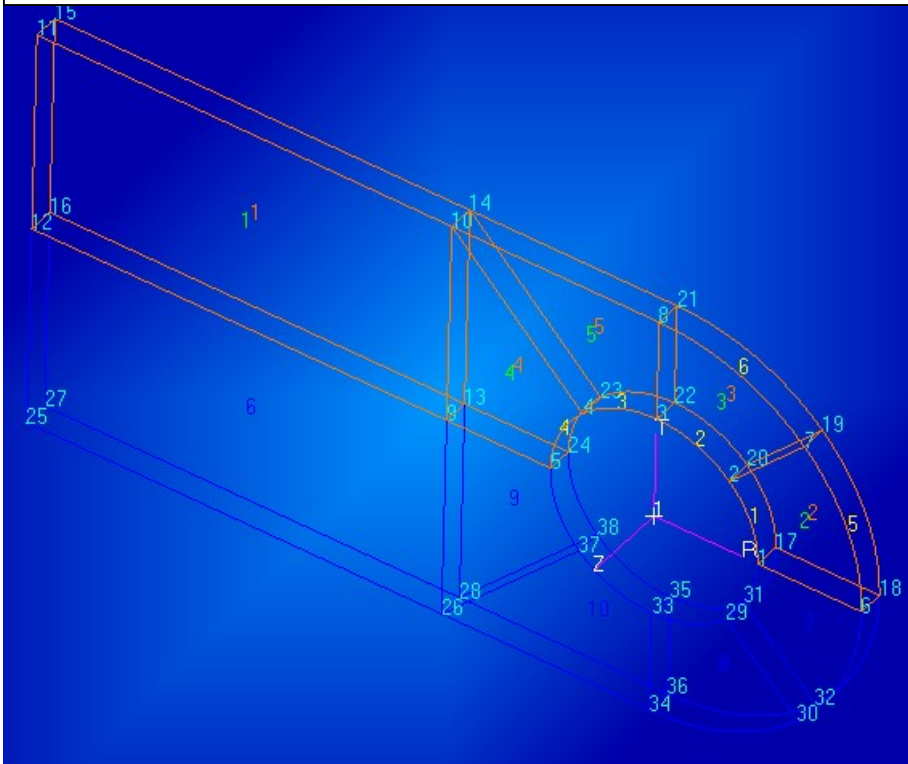
Change the view to **Iso 1** in *Home/Orientation* section and **Fit view** 



Creating the lower part of the clevis

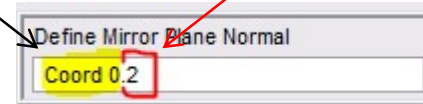
- Transform->Solid->Mirror**
- To *Defin Mirror Plane Normal* click the **Coord 0**
- Click on the **Frame Direction 2** on the *Select Menu* to point out the direction of a vector normal to the minor surface.
- Uncheck **Auto Execute**, select all solids **Solid 1:5** and press **-Apply-**

Click Display Line in Home/Misc. section 




This way of representing indicates the two important things: **which Coord is being used** and **which axis is under consideration**


TIP




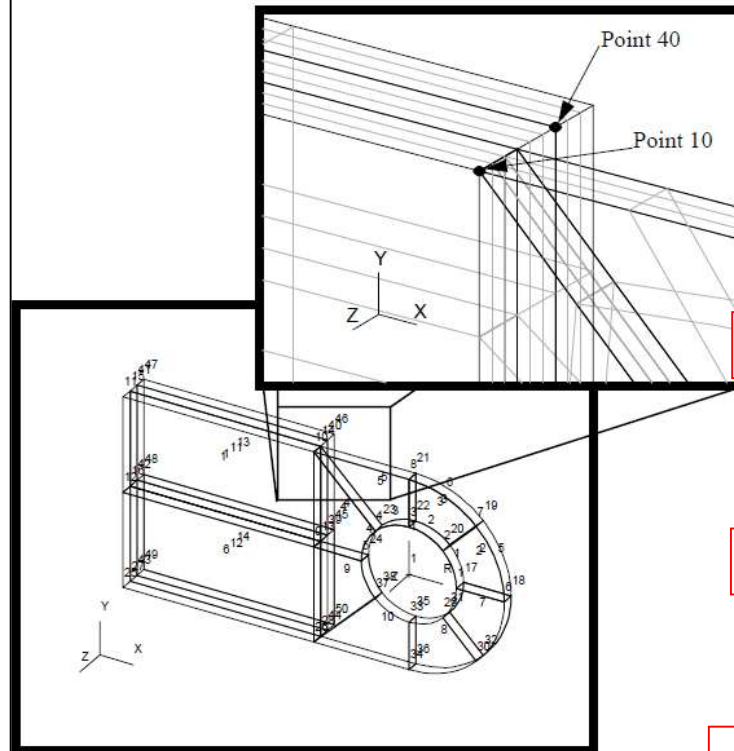
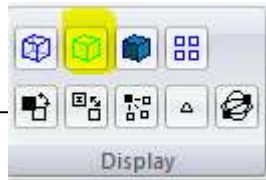
The remaining solids will be created using the translate method.

- Transform->Solid->Translate**
- Enter **<0 0 -0.25** in a *Direction Vector* section
- Repeat Count: **2**
- Uncheck **Auto Execute**, choose **Solid 1** and **6** and press **-Appl**

Your last construction step is to translate copies of the solids that surround the hole to create the final solids. Click in the **Translation Vector databox** and click on  in *Select Menu*

- To define the translation vector, pick **Point 10** then **Point 40** as shown below. Use  **View Corners** from **Viewing** to zoom in. After selecting the points use **Fit View** from **Viewing** to zoom out.
- Repeat Count: **1**
- Select **Solids 2 to 5** and **7 to 10** and hit **-Appl**

To disable the grid View and enable Body press  in display menu



Geometry

Action: **Transform**

Object: **Solid**

Method: **Translate**

Solid ID List
23

Type of Transformation
 Cartesian in Refer. CF
 Curvilinear in Refer. CF

Refer. Coordinate Frame
Coord 0

Translation Vector
 Reverse Direction
 Auto Update Magnitude

Direction Vector
<0 0 -0.25>

Vector Magnitude
0.25

Translation Parameters
Repeat Count
2

Delete Original Solids

Auto Execute

Solid List
Solid 1 6

-Apply-

a

b

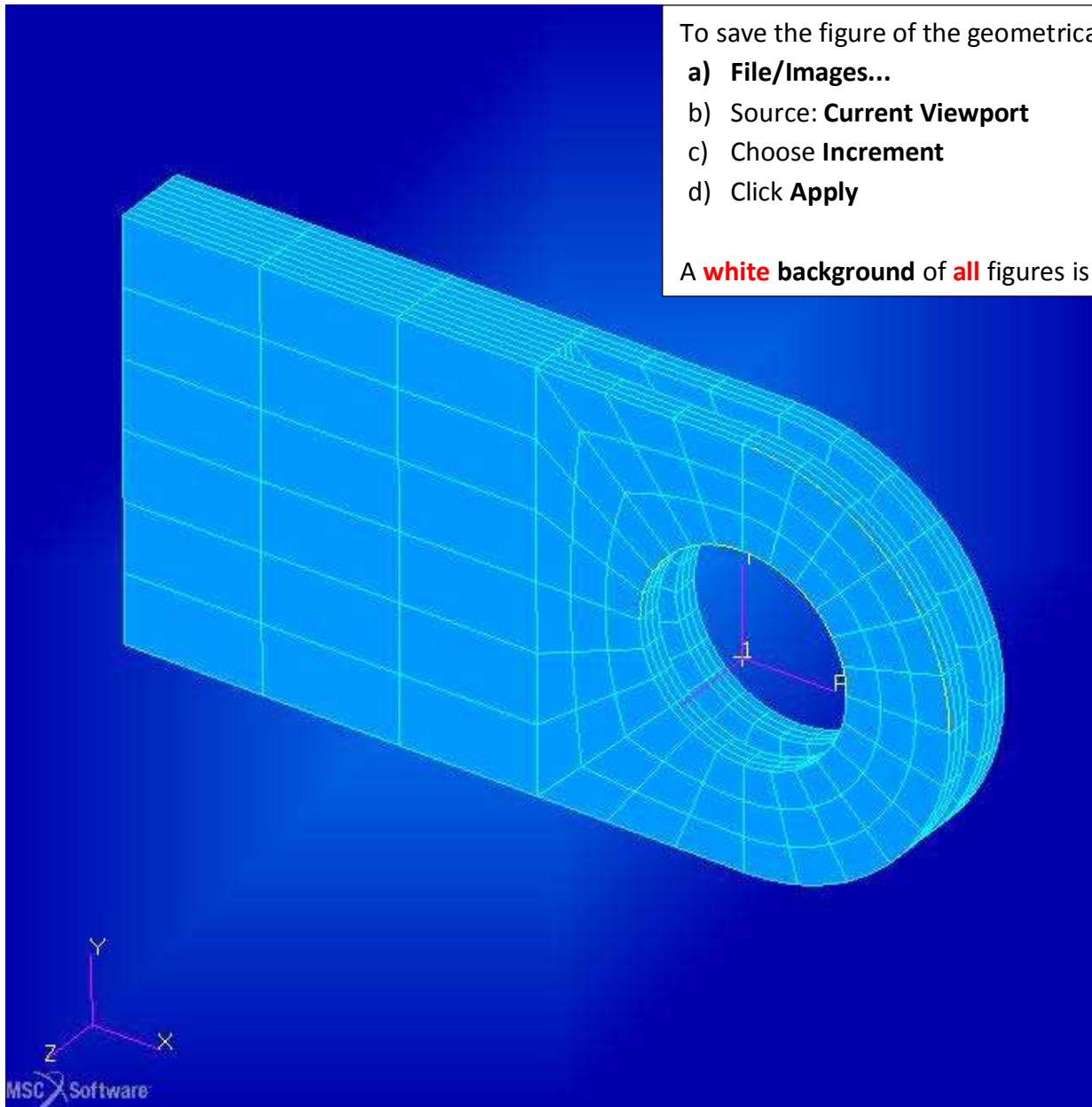
c

d

To save the figure of the geometrical mode

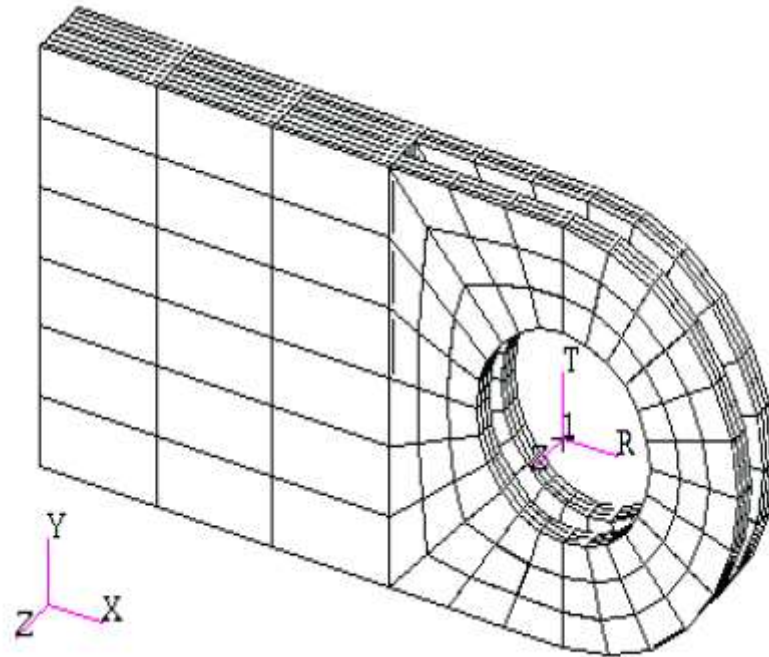
- a) **File/Images...**
- b) Source: **Current Viewport**
- c) Choose **Increment**
- d) Click **Apply**

A **white** background of **all** figures is **obligatory**.



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

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

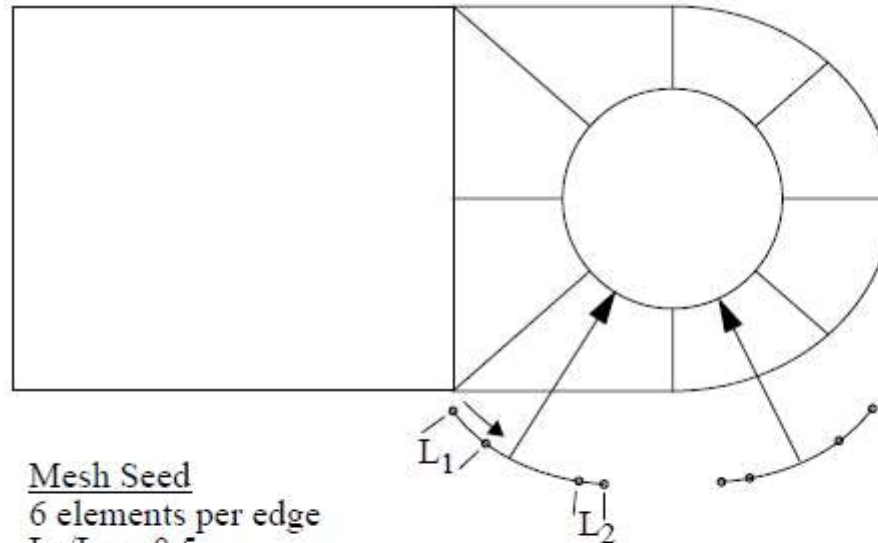


Model Description:

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 refine mesh with a higher mesh density near the bottom of the hole where you will apply a force load in a future exercis

Suggested Exercise Steps:

1. Database opening / creating a new View
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**.
2. Create the mesh seeds needed to increase th density in the area where the distributed load will be applied.
3. Create a finite element mesh using the elemen topology and size listed in the diagram on the right
4. Create an Isotropic material, named **Steel** which uses Linear Elastic Constitutive Model. The Steel's Elastic Modulus and Poisson's Ratio are respectively 30E6 and 0.30.
5. Create a 3-D element property named, Solid_Elements_Steel, for the entire includes the steel material definition



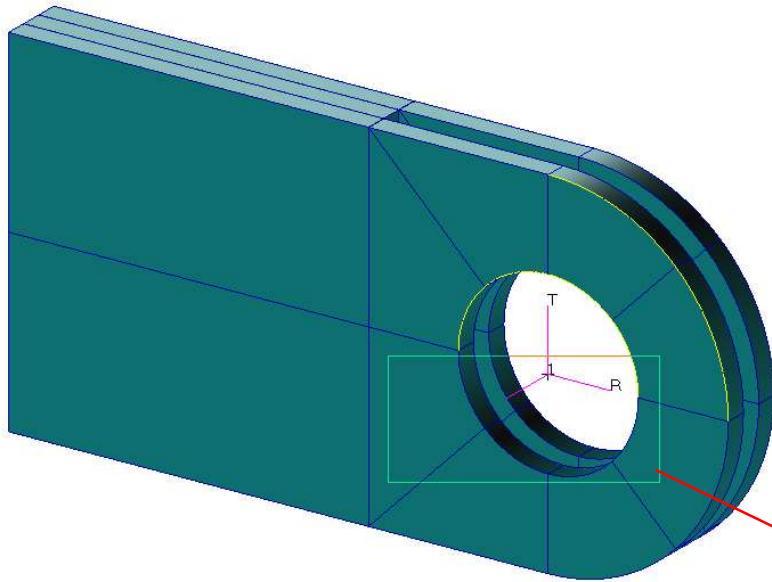
Mesh Seed
6 elements per edge
 $L_2/L_1 = 0.5$

Finite Element Mesh
Global Edge Length = 0.5
HEX8 elements

1. New View

Assuming that you have already opened data **Clevis.db**, use the *Viewing/ Select Corners* option to zoom a model in a specific area.

- a. Go to *Viewing/Named View Option*
- b. Create a new View, name it **zoom_in** and hit *-Apply-*



TIP To simplify the View you can always hide or display lines by pressing *-* only model's boundaries are shown.




a


The "Viewing" menu is open, showing options like Transformations..., Fit View, Select Center, Select Corners, Zoom (% of View), Angles..., View From/To..., Scale Factors..., **Named View Options...**, Clipping/Perspective..., and Arbitrary Clipping... The "Named View Options..." option is highlighted in yellow. A blue arrow points from this option to the "Named View Options" dialog box.

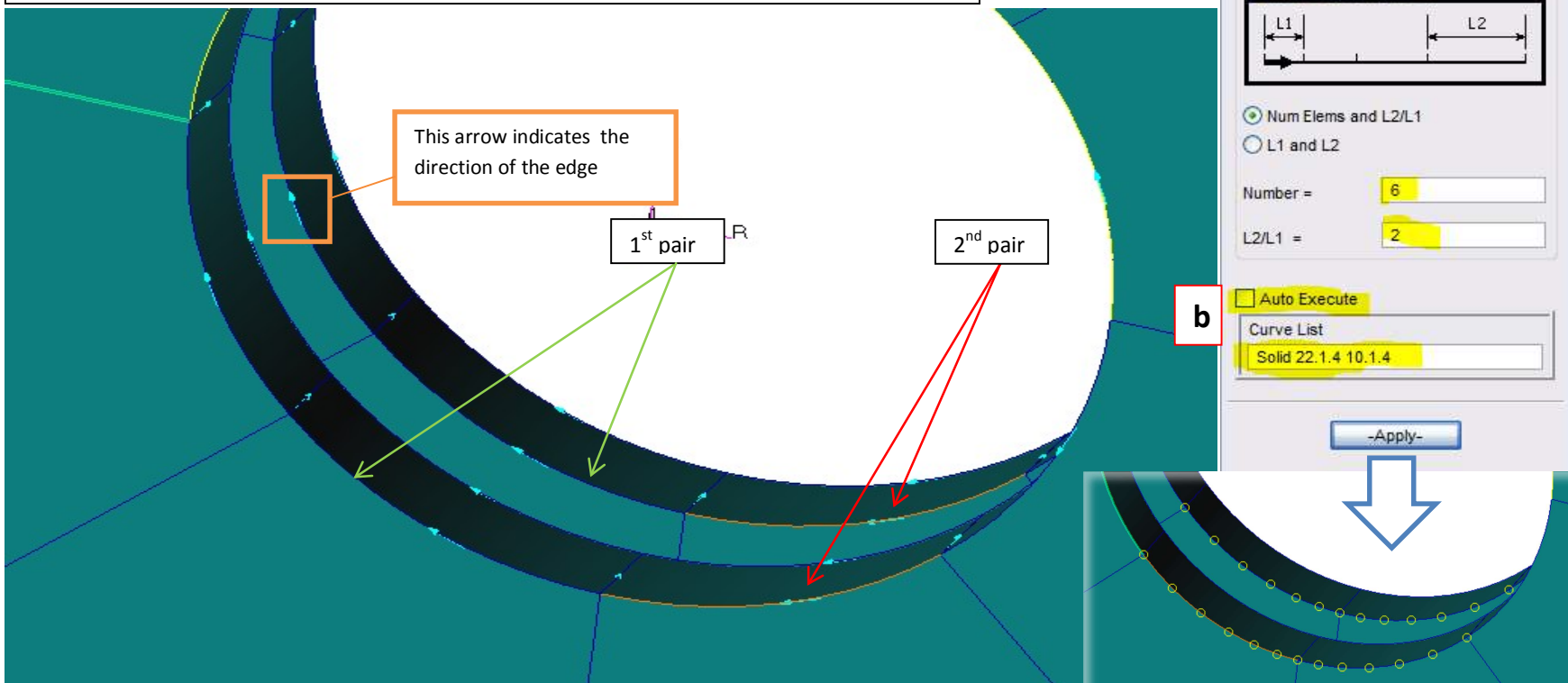
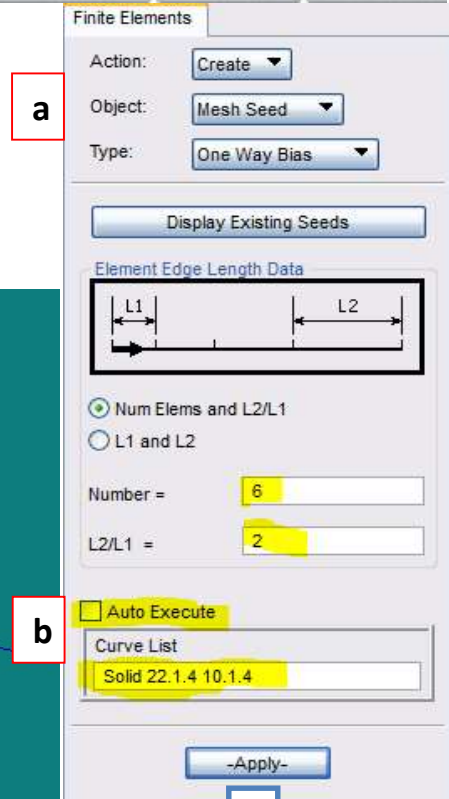
b

The "Named View Options" dialog box is shown. It has a "Current Viewport" field set to "default_viewport". Under "Select Named View", a list of views is shown: bottom_view, default_view, isometric_view, side_view, top_view, and zoom_in. The "Create View..." button is highlighted in yellow. Below it are "Rename View..." and "Delete View..." buttons. A blue arrow points from the "Create View..." button to the "Select Named View" dialog box.The "Select Named View" dialog box is shown. It has a list of views: bottom_view, default_view, isometric_view, side_view, top_view, and zoom_in. The "zoom_in" view is highlighted in yellow. Below the list is a "Create New View" field containing "zoom_in". At the bottom are "-Apply-" and "Cancel" buttons.

2. Creating mesh seeds

- Click on *Meshing* and as follows : **Create->Mesh seed->One Way Bias** or just click on 
- Insert **6** as a *Number* (number of seeds on the curve/edge) a **2** as a *L2/L1* which indicates seeds varying size along an edge. Uncheck *Auto Execute* and holding L.Shift choose the **1st pair of edges** shown in figure and press *-Apply-*. Do the same for the **2nd pair of edges** but invert the *L2/L1* to maintain symmetry of the seeds, thus *Number*: **6**, *L2/L1*: **0.5** and choose **2nd pair of edge** and press *-Apply-*.

Tip In order to select edge click on  in select menu



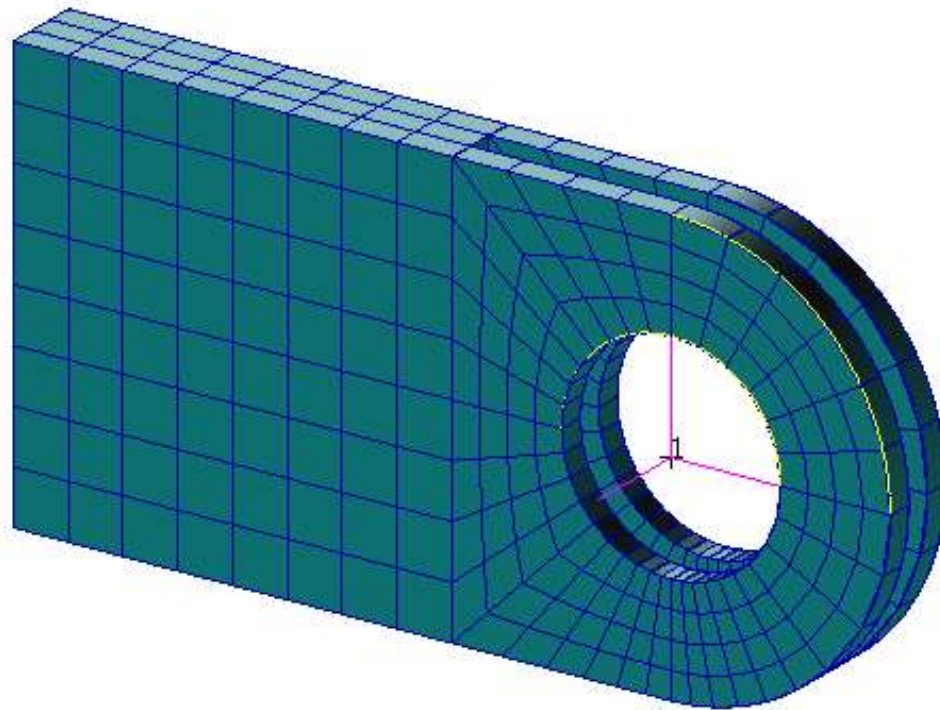
3. Creating Mesh

- Create->Mesh->Solid** or simply click on
- Change element shape to **Hex**
- Select all solid parts
- Uncheck *Automatic calculation* and insert Value **0.5** and hit *-Apply*



Tip

If project is unreadable
You have to hide Labels
and decrease the size of
nodes by pressing in *Home*
section:



Finite Elements

Action: **Create**

Object: **Mesh**

Type: **Solid**

Output ID List

Node: 1361

Element: 457

Elem Shape: **Hex**

Mesher: **IsoMesh**

Topology: **Hex8**

Solid List

Solid 1:22

Global Edge Length

Automatic Calculation

Value: **0.5**

Prop. Name: -None -

Prop. Type: -N/A -

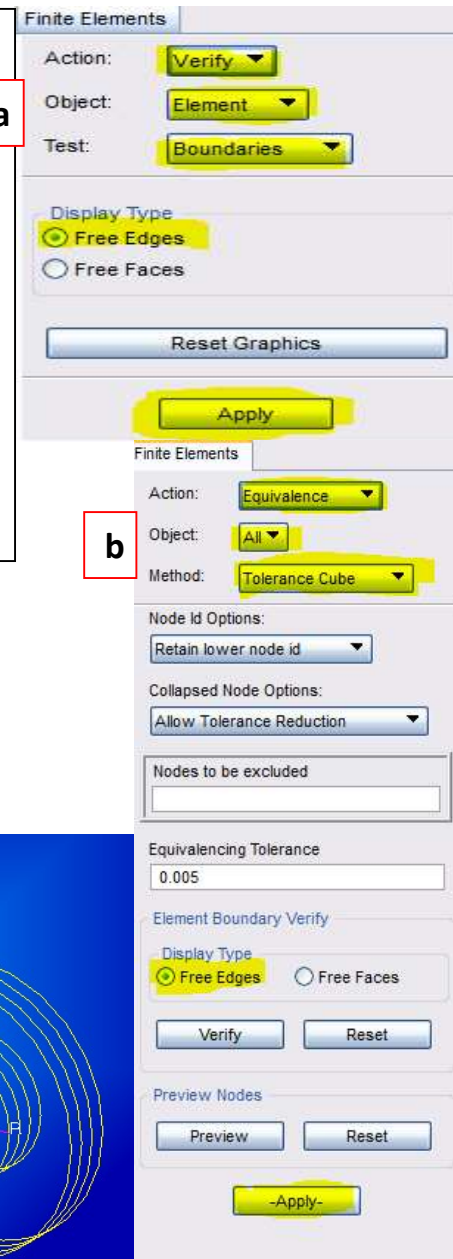
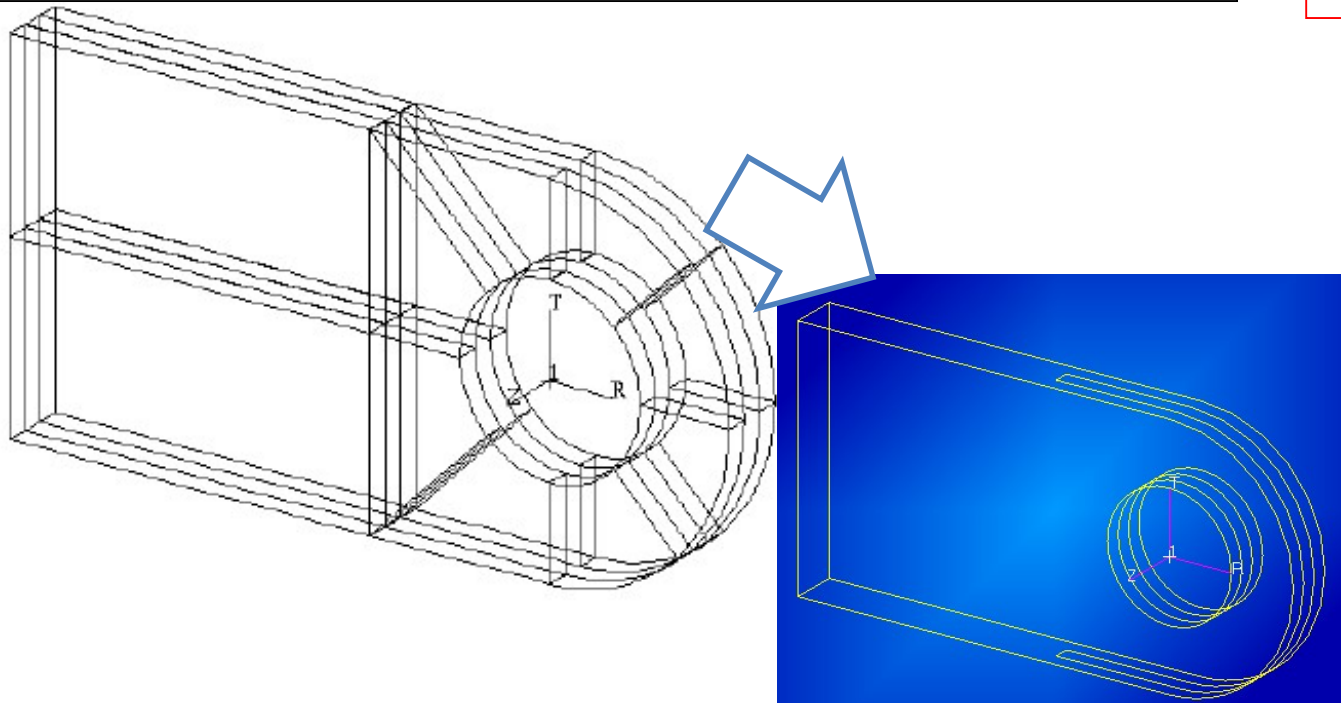
Now that you have created your finite element mesh, it is time to determine whether you need to “equivalence the model”. To do this:

- a. **Verify->Element->Bounda** check the *Free Edges* and hit *-Apply-*

As you can see Your model consist of a group of solids residing next to each other in three dimensiona space. Since you do not 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 locatio

- b. **Equivalence->All->Tolerance Cube** , check the **Free Edges** and *-Appl*

You now have one contiguous model of finite elements. To check whether this is tru : Repeat the step described in point a. (Your model should looks like the one below at blue background)






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

- Click on *Properties / Isotropic*
- Insert **Steel** as the name of material and click *Input Properties*
- Enter Value **30e6** in Elastic Modulus field and **0.3** in Poisson's Ratio and hit *OK*
- Check in *Material Status* if the steel is in *ACTIVE* material zone press *-Apply-* an *-Apply-* in *Materials* window

5. Create a 3D element property named, **Solid_Elements_Steel**, or the entire model which includes the steel material definitio

- a. *Properties Create->3D-> Solid* insert **Solid_Elements_Steel** as a *Property Set Name*
- b. *Input Properties* Click on  and choose sooner create **Steel** and hit *OK*
- c. Click on *Select Application Region* mark whole solid click **Add-> OK** and *-App*

The image displays a sequence of software dialog boxes for creating a 3D element property. The 'Material Properties' dialog shows 'Material Name' as 'm.Steel' and 'Mat Prop Name' with a steel material icon. The 'Select Existing Material' dialog shows 'Steel' selected. The 'Element Properties' dialog shows 'Action' as 'Create', 'Object' as '3D', 'Type' as 'Solid', and 'Property Set Name' as 'Solid_Elements_Steel'. The 'Select Application Region' dialog shows 'Application Region' as 'Solid 1:22'. Red boxes labeled 'a', 'b', and 'c' highlight the 'Action' dropdown, the 'Input Properties' button, and the 'Select Application Region' button respectively.

Loads and Boundary Conditions on a 3-D Clevis

Objectives:

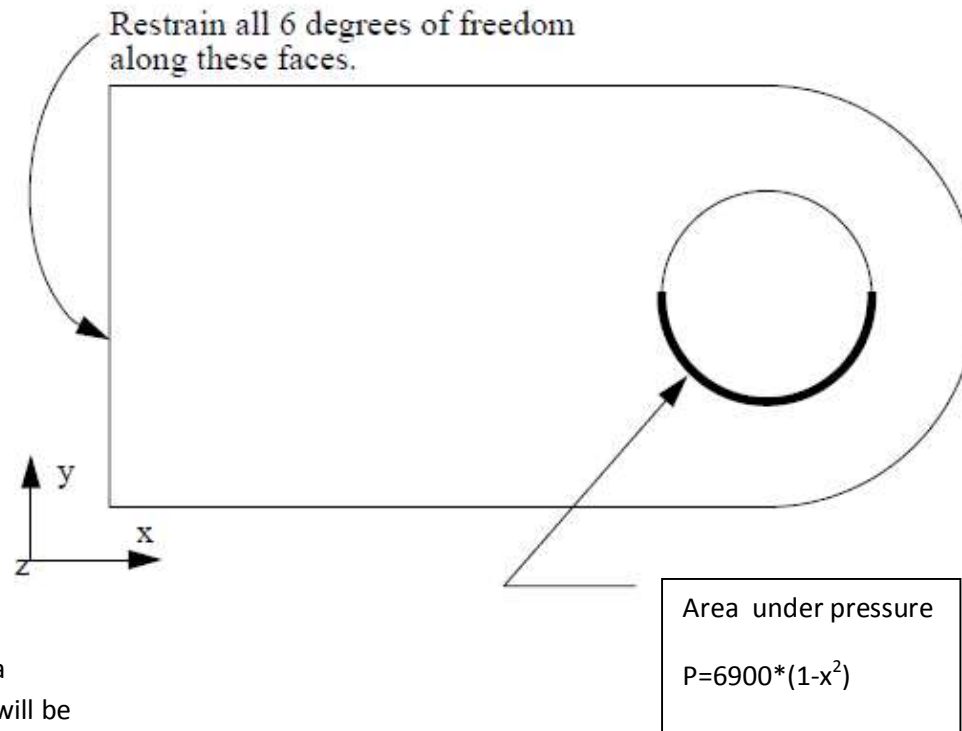
- ❖ Apply constraints to your model.
- ❖ Create and apply a Pressure

Suggested Steps

1. Create a Pressure case
2. Create a nodal displacement boundary condition named **Clamped**
3. Create a Pressure boundary condition

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 under quadratically varying pressure $P = 6900 \cdot (1-x^2)$.



1. Creating a pressure case (ATTENTION: 'X', below, is **NOT TYPED**, it is chosen from the list of Independent Variables!!):

a. Load/BCs // LBC FIELDS->Create spatial function insert Pressure_field as name check Scalar and Real and insert $6900*(1-X**2)$ as a Scalar Function and hit -Apply-

b. Element Variable/ Pressure, Enter Pressure as a name, change target to 3D and click Input Data.

c. Double click on Pressure_field and ok

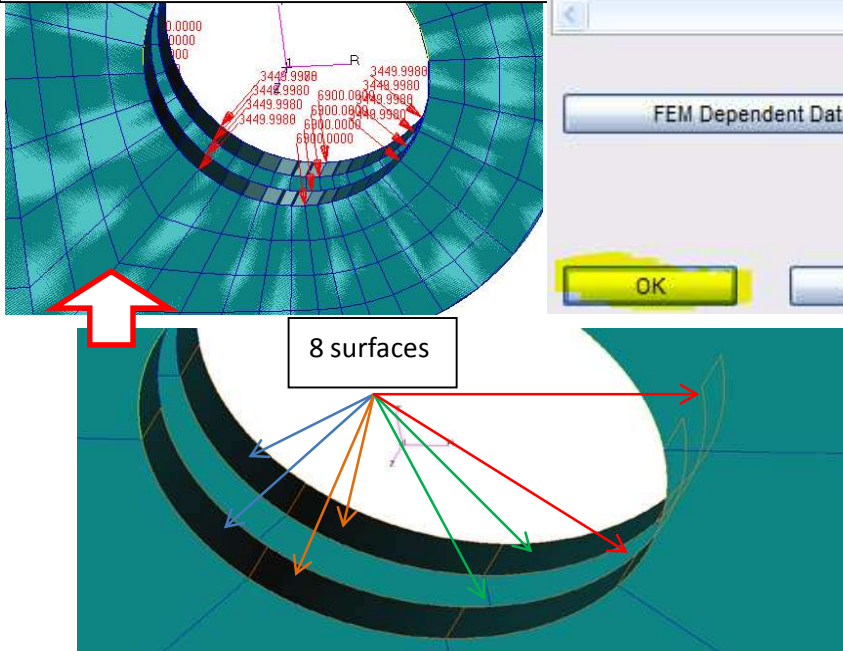
d. Select Application Region -> holding L.Shift select all 8 surfaces than click Add-> OK and -Apply-

The screenshot shows the 'LBC Fields' dialog box with the following settings:

- Action: Create
- Object: Pressure
- Type: Element Variable
- Current Load Case: Default...
- Type: Static
- Existing Sets: Pressure
- New Set Name: Pressure
- Target Element Type: 3D
- Scalar Function: $6900*(1-X**2)$
- Independent Variables: X, Y, Z

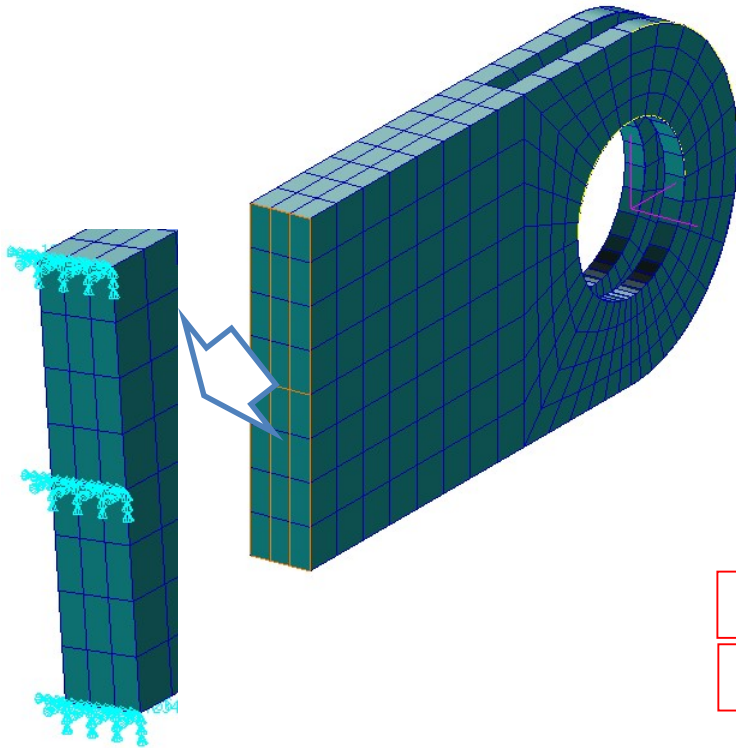
The 'Fields' panel on the right shows:

- Existing Fields: Pressure_field
- Field Name: Pressure_field
- Field Type: Scalar Vector
- Coordinate System Type: Real Parametric
- Coordinate System: Coord 0
- Scalar Function: $6900*(1-X**2)$
- Independent Variables: X, Y, Z



2. Create a Nodal Displacement boundary condition:

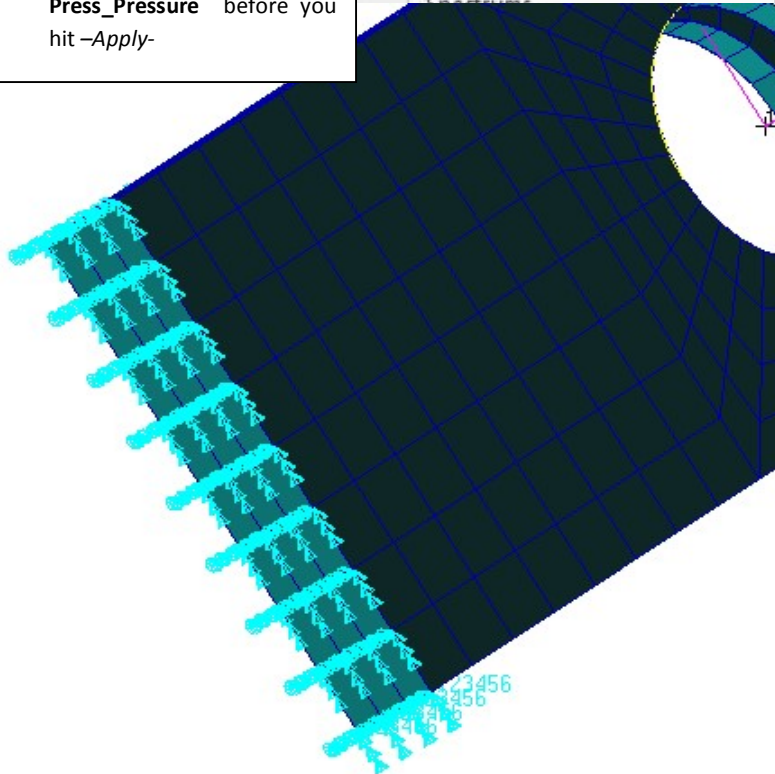
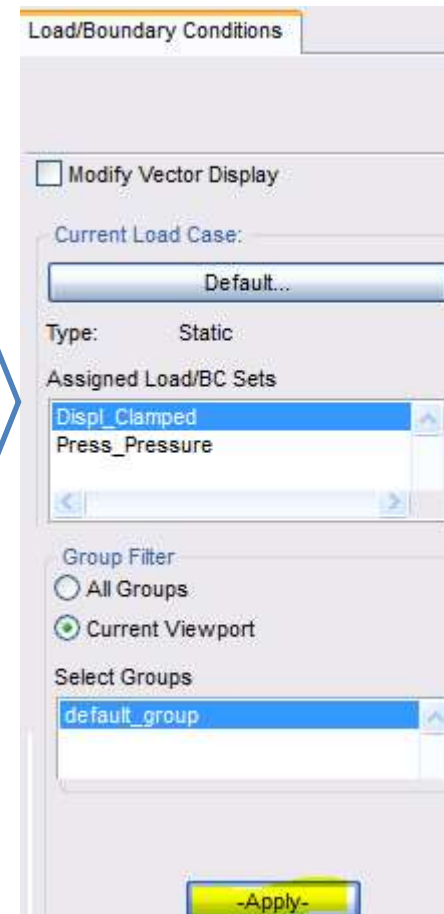
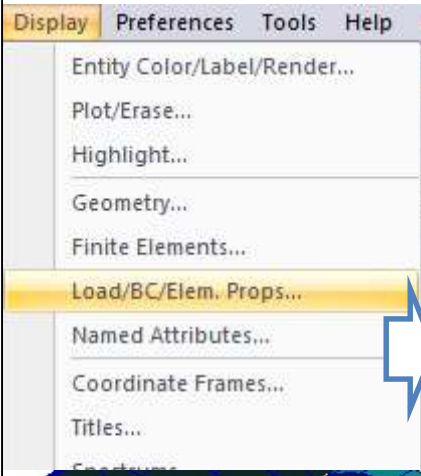
- Load/Boundary Conditions Create-> Displacement ->Nodal set name as Clamped
- Click on Input data and insert <0,0,0> as a Translation and <0,0,0> as Rotations and OK
- Select Application Region-> change the filter to Surface and Add and select surfaces as shown i figure, click on Add, OK and then -Apply-

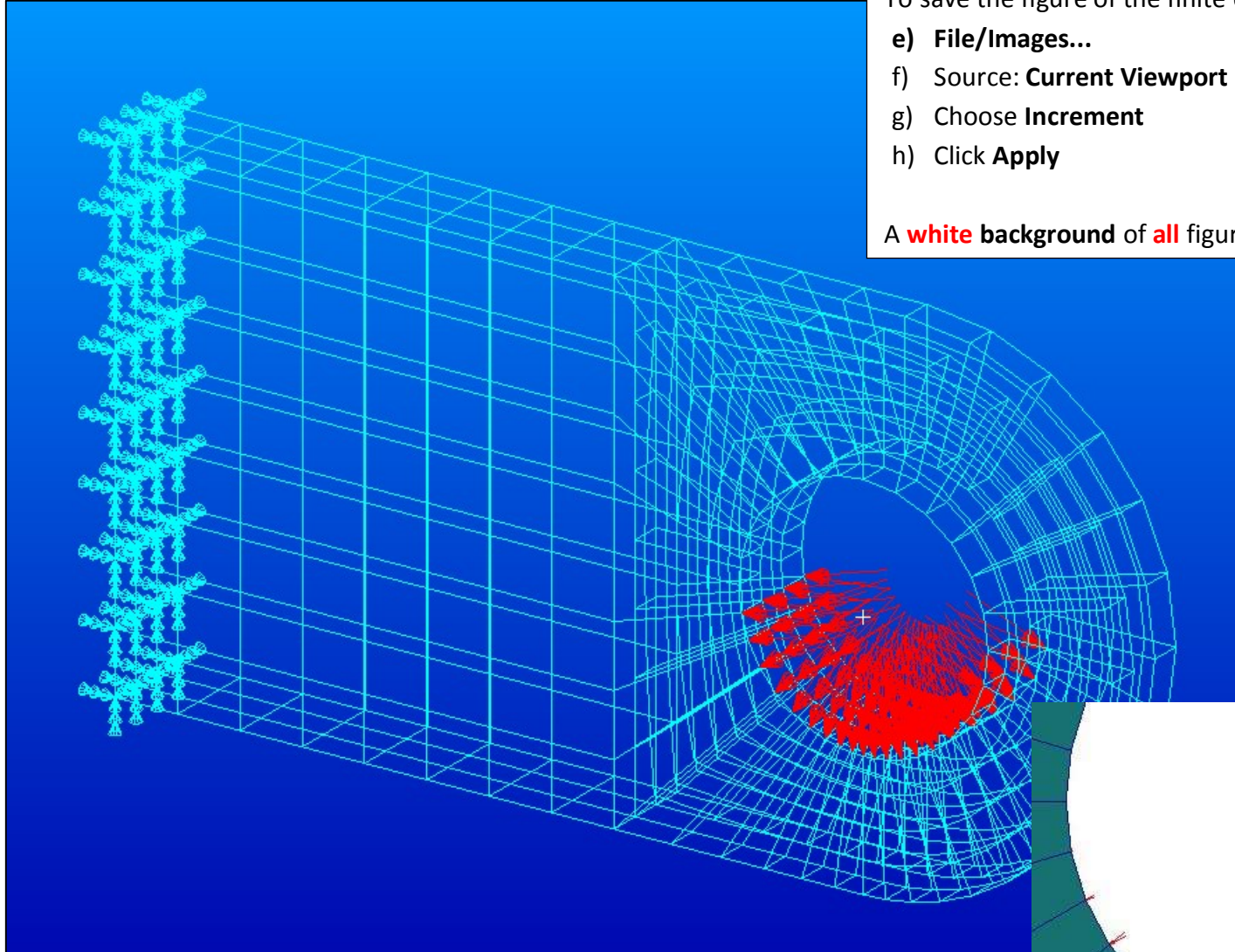


The screenshot displays the software's 'Load/Boundary Conditions' dialog box. The 'Action' is set to 'Create', the 'Object' is 'Displacement', and the 'Type' is 'Nodal'. The 'Option' is 'Standard'. The 'Current Load Case' is 'Default...'. The 'Type' is 'Static'. The 'New Set Name' is 'Clamped'. The 'Input Data...' button is highlighted, and the 'Select Application Region...' button is also highlighted. The 'Input Data' sub-dialog shows 'Translations <T1 T2 T3>' and 'Rotations <R1 R2 R3>' both set to '<0,0,0>'. The 'Select Application Region' sub-dialog shows 'Select: Geometry' and 'Select Geometry Entities' with a list of selected surfaces: 'Solid 13.2 13.2 11.2 1.2 6.4 12.4 14'. The 'Add' button is highlighted. The 'OK' button is highlighted in both sub-dialogs. A toolbar on the left shows a '+' button highlighted in yellow, and a red arrow points from the 'Select Application Region...' button to the 'Add' button in the sub-dialog.

Display both the displacement and force on the finite element model

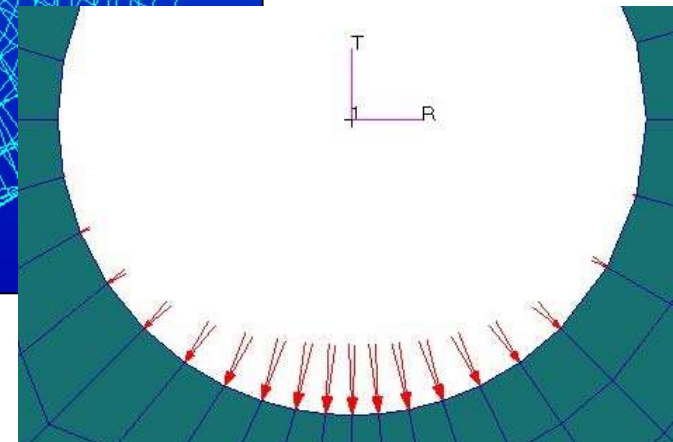
- a) *Display/Load/BC/Elem. Props...* and check **Show on FEM only** and hit *-Apply-*
- b) After that, in *Loads/BCs* tab choose **Plot markers** as a *Action*
- c) Highlight **Displ_Clamped** and hit *-Appl* . If you want to show on the model also a pressure, highlight **Press_Pressure** before you hit *-Apply-*

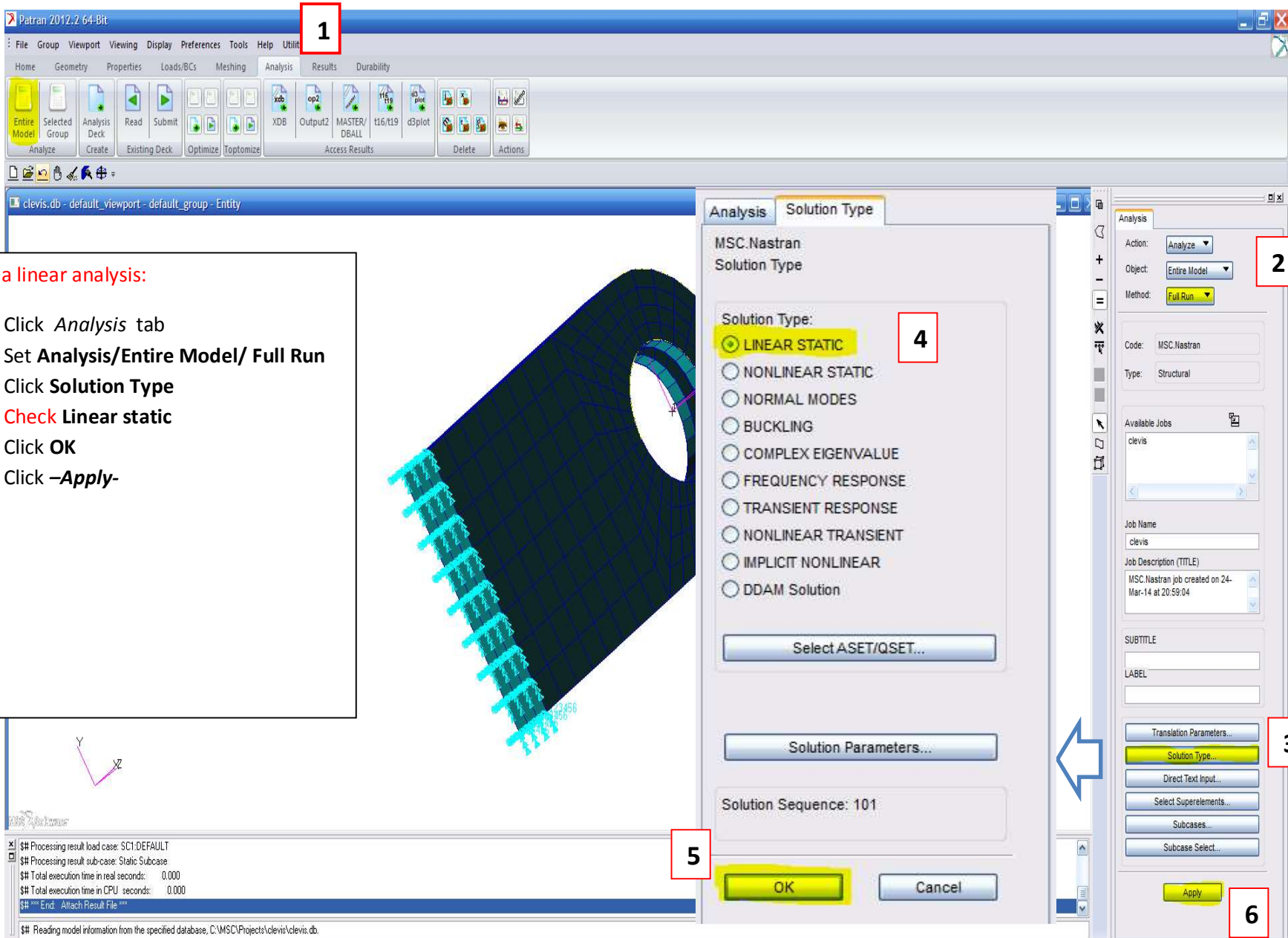




To save the figure of the finite element model:
e) **File/Images...**
f) Source: **Current Viewport**
g) Choose **Increment**
h) Click **Apply**

A **white** background of **all** figures is **obligatory**.





Run a linear analysis:

1. Click *Analysis* tab
2. Set **Analysis/Entire Model/ Full Run**
3. Click **Solution Type**
4. **Check Linear static**
5. Click **OK**
6. Click **Apply**

```
$$$ Processing result load case: SCI.DEFAULT  
$$$ Processing result sub-case: Static Subcase  
$$$ Total execution time in real seconds: 0.000  
$$$ Total execution time in CPU seconds: 0.000  
$$$ End: Attach Result File $$$  
$$$ Reading model information from the specified database, C:\MSC\Projects\clevis\clevis.db
```


Attach the result file, when the analysis job is completed:

1. Click **XDB** HDF5
2. Click **Select Result File** h5 and select Clevis .xdb if necessary
3. Click **Apply**

Bellow there is a fragment of the **fo6** file which contains of a in/output data. This fragment illustrates distribution of the loads. Highlighted value is a value of **resultant fo** working in **Fy** direction

SUBCASE/ DAREA ID	LOAD TYPE	T1	T2	T3	R1	R2	R3
1	FX	-1.011008E+01	----	----	----	3.790687E+00	2.749616E+04
	FY	----	4.583406E+03	----	1.718777E+03	----	-2.750044E+04
	FZ	----	----	8.715680E-05	5.994861E-05	5.229408E-04	----
	MX	----	----	----	0.000000E+00	----	----
	MY	----	----	----	----	0.000000E+00	----
	MZ	----	----	----	----	----	0.000000E+00
	TOTALS	-1.011008E+01	4.583406E+03	8.715680E-05	1.718777E+03	3.791210E+00	-4.273438E+00

MSC.NASTRAN JOB CREATED ON 24-MAR-14 AT 20:59:04

MARCH 24, 2014 MSC.NASTRAN 77 / 6/12 PAGE 63

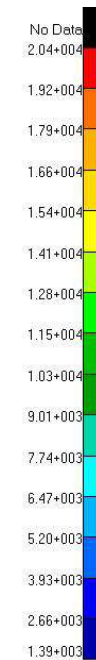
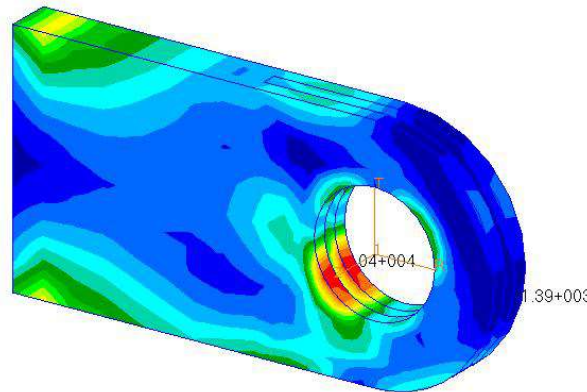
Post Processing of Stress Results

Objectives:

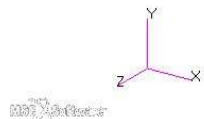
- ❖ To post-process stress results from MSC/NASTRAN
- ❖ To use MSC/PATRAN to create fill and fringe plots to determine if the analyzed part will meet a customer defined criteria or whether the part needs to be redesigned and re-analyzed

Patran 2012.2 64-Bit 25-Mar-14 01:39:31

Fringe: Default, A1:Static Subcase, Stress Tensor, , von Mises, (NON-LAYERED)



default_Fringe3:
Max 2.04+004 @Nd 1246
Min 1.39+003 @Nd 1160



CLEVIS

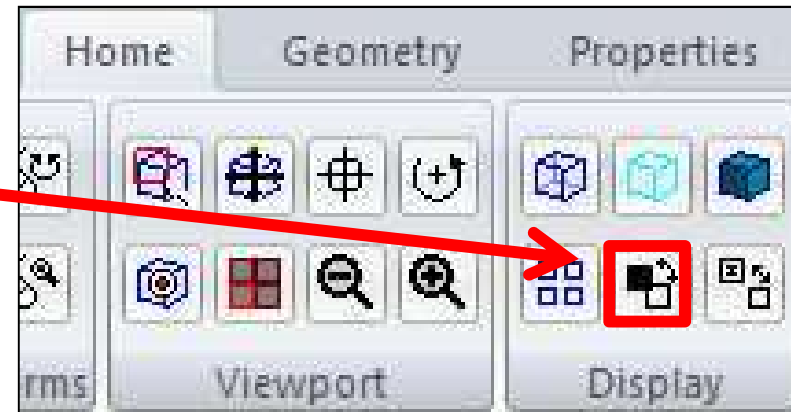
supplement

Create 6 different plots with results:

- 1) Vertical translational displacements in Y direction
- 2) Von Mises stress σ_{equiv}
- 3) Stress in X direction σ_x with averaging, continuous σ_x
- 4) Stress in X direction σ_x without averaging, discontinuous σ_x
- 5) Stress in X direction σ_x with averaging, continuous σ_x for the base of the clevis (2 different views)

Set white background:

Cycle Background



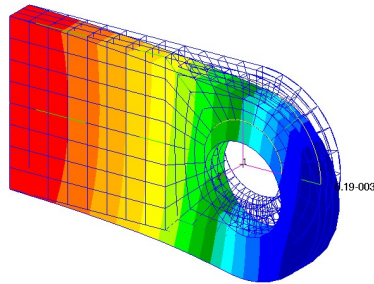
Follow the steps (numbers in red frames). \longrightarrow 1

PLOT no. 1

Vertical translational displacements in Y direction

1 Results -> Create -> Quick Plot

9 File -> Images... -> Apply



Results

Action: Create

Object: Quick Plot

2

Select Result Cases

Default, A1: Static Subcase; MSC.NAS

3 Select Fringe Result

Constraint Forces, Translational
Displacements, Rotational
Displacements, Translational
Principal Stress Direction, Intermed Prir
Principal Stress Direction, Intermed Prir

Quantity: Y Component

4 Select Deformation Result

Constraint Forces, Rotational
Constraint Forces, Translational
Displacements, Rotational
Displacements, Translational

Animate

Apply

Results

Action: Create

Object: Quick Plot

5

6 Show Spectrum
 Show Viewport Legend

Spectrum... Range...

Style: Discrete/Smooth

Shading: None

0.0 1.0 0.0

Element Shrink Factor

Fringe Edges

7 Display: Free Edges

Style:

Width:

Title Editor...

Show Title Lock Title

Show Max/Min Label
 Show Fringe Label

Label Style...

8 Show on Deformed

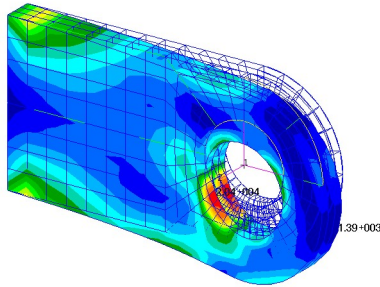
Apply Reset

PLOT no. 2

Von Mises stress σ_{equiv}

1 Results -> Create -> Quick Plot

9 File -> Images... -> Apply



Results

Action: Create

Object: Quick Plot

2

Select Result Cases

Default, A1: Static Subcase; -MSC.NAS

3

Select Fringe Result

Stress Invariants, Mean Pressure
Stress Invariants, Minor Principal
Stress Invariants, Von Mises
Stress Tensor,

Quantity: von Mises

4

Select Deformation Result

Constraint Forces, Rotational
Constraint Forces, Translational
Displacements, Rotational
Displacements, Translational

Animate

Apply

Results

Action: Create

Object: Quick Plot

5

Show Spectrum
 Show Viewport Legend

Spectrum... Range...

Style: Discrete/Smooth 6

Shading: None

0.0 1.0 0.0

Element Shrink Factor

Fringe Edges 7

Display: Free Edges

Style:

Width:

Title Editor...

Show Title Lock Title

Show Max/Min Label
 Show Fringe Label

Label Style...

Show on Deformed

8

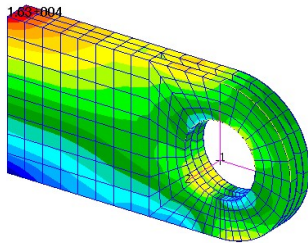
Apply Reset

PLOT no. 3

Stress in X direction σ_x
with averaging, continuous σ_x

1 Results -> Create -> Fringe

9 File -> Images... -> Apply



Results

Action: Create

Object: Fringe

2

3

Select Result Cases

Default, A1:Static Subcase,-MSC.NAS

Select Fringe Result

Stress Invariants, Mean Pressure

Stress Invariants, Minor Principal

Stress Invariants, Von Mises

Stress Tensor,

4

Position...((NON-LAYERED))

Quantity: X Component

Animate

Apply Reset

Results

Action: Create

Object: Fringe

5

6

Show Spectrum

Show Viewport Legend

Spectrum... Range...

Style: Discrete/Smooth

Shading: None

0.0 1.0 0.0

Element Shrink Factor

Fringe Edges

7

Display: Element Edges

Style: [Line Style]

Width: [Width]

Title Editor...

Show Title Lock Title

Show Max/Min Label

Show Fringe Label

Label Style...

Show on Deformed

8

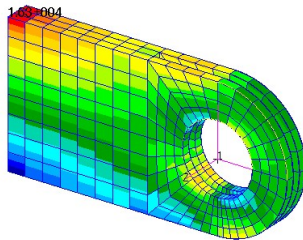
Apply Reset

PLOT no. 4

Stress in X direction σ_x
without averaging, discontinuous σ_x

1 Results -> Create -> Fringe

8 File -> Images... -> Apply



Results

Action: Create

Object: Fringe

2

3

Select Result Cases

Default, A1:Static Subcase,-MSC.NAS

Select Fringe Result

Stress Invariants, Mean Pressure
Stress Invariants, Minor Principal
Stress Invariants, Von Mises
Stress Tensor

Position...((NON-LAYERED))

Quantity: X Component 4

Animate

Apply Reset

Results

Action: Create

Object: Fringe 5

Coordinate Transformation: As Is

Scale Factor: 1.0

Filter Values: None

Averaging Definition: 6

Domain: None

Method: Derive/Average

Extrapolation: Shape Fn.

Use PCL Expression

Define PCL Expression...

Existing Fringe Plots...

Save Fringe Plot As:

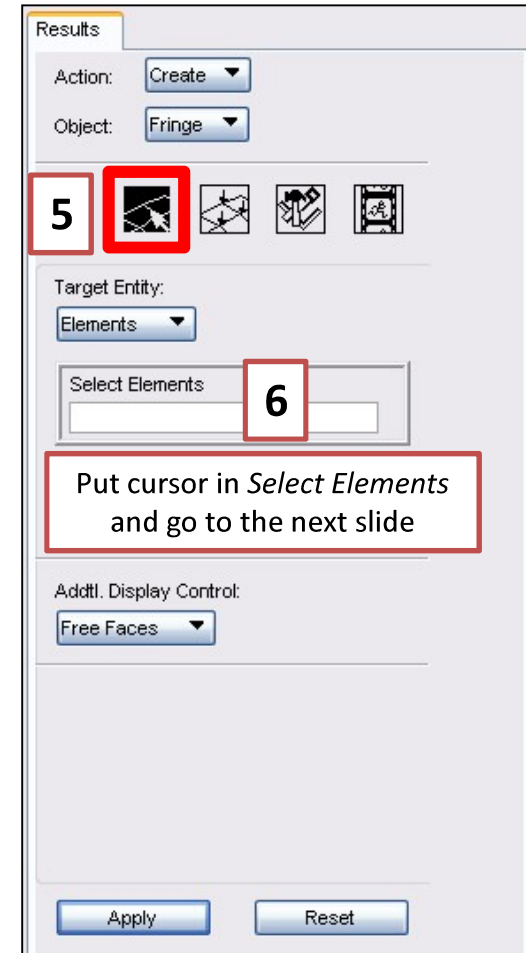
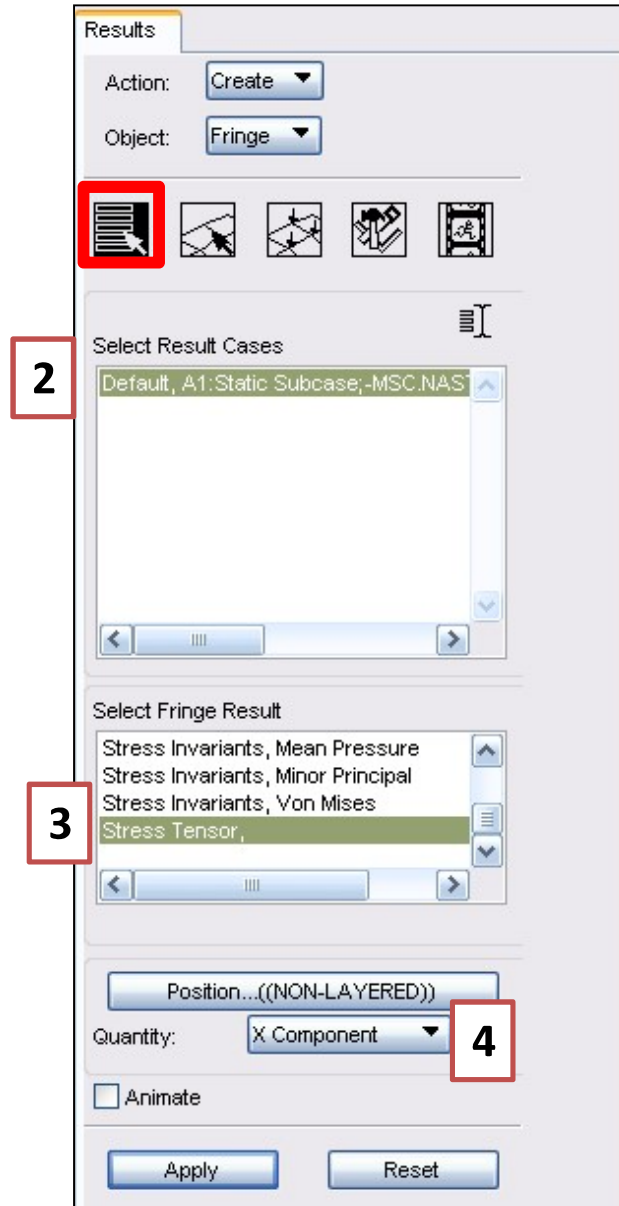
7 Apply Reset

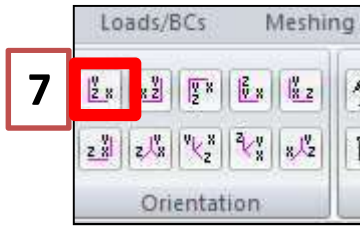
All Entities change to None

PLOTS: no. 5 and no. 6

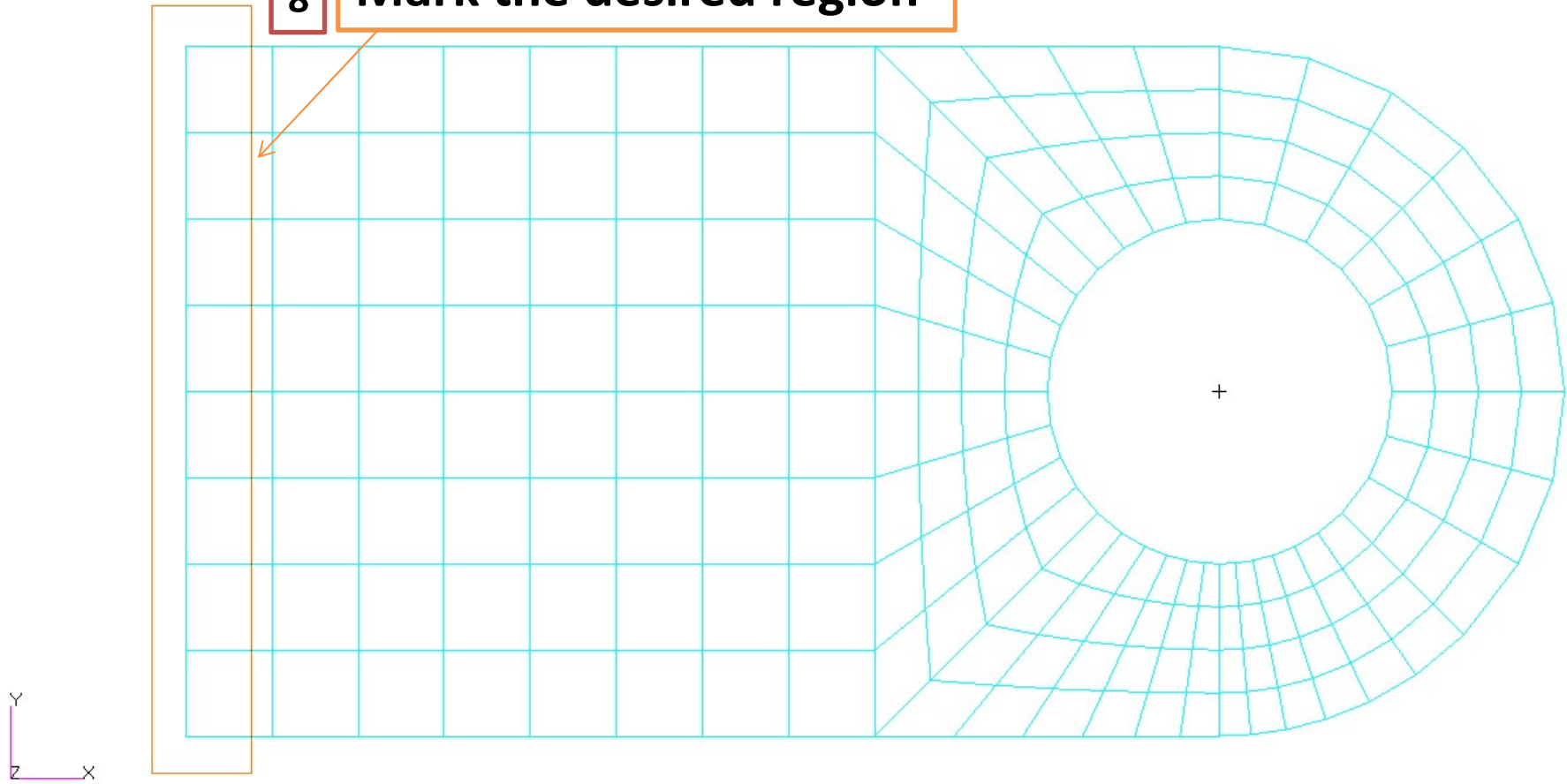
Stress in X direction σ_x
with averaging, continuous σ_x for the base
of the clevis (2 different views)

1 Results -> Create -> Fringe



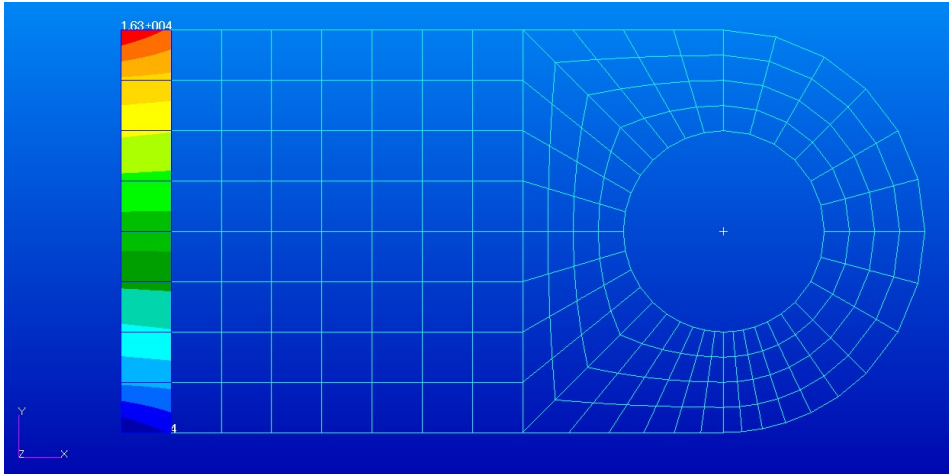


8 Mark the desired region

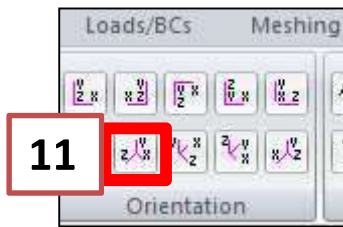


9 Click **Apply**

10 File -> Images... -> Apply



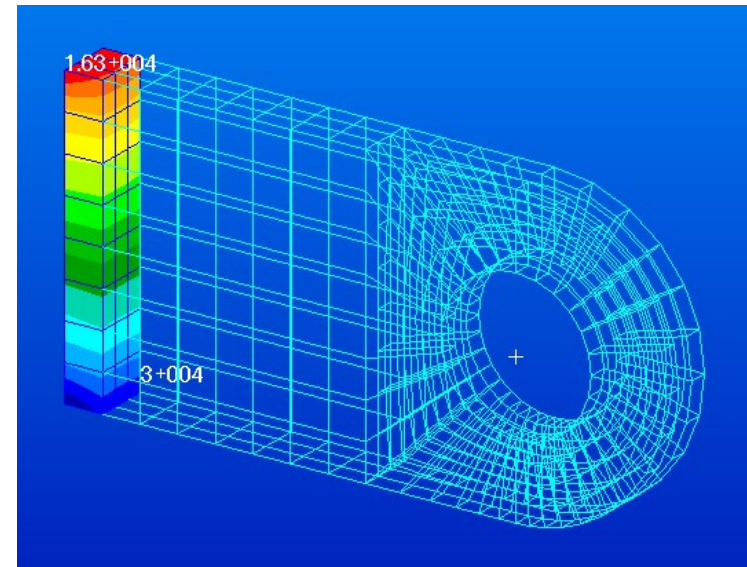
What is the distribution of the σ_x stress at the base of the clevis along the vertical direction?



11

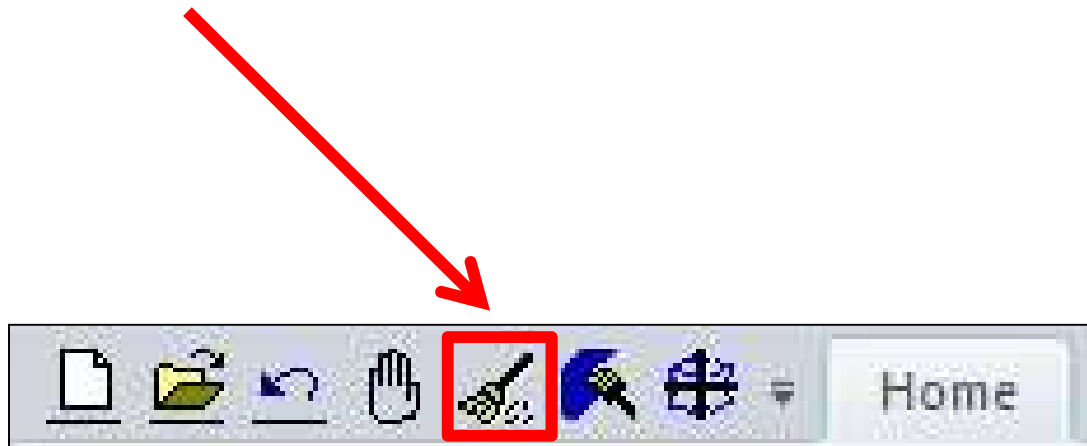


12 File -> Images... -> Apply



**Check the value of the displacement in the direction Y
of the node located on the lower surface of the hole
at the distance 6 [in]:**

Reset Graphics



Check the value of the displacement in the direction Y of the node located on the lower surface of the hole at the distance 6 [in]:

Results -> Create -> Cursor -> Vector

Results

Action: Create

Object: Cursor

Method: Vector

Select Result Cases

1 Default, A1:Static Subcase,-MSC.NAS

Select Cursor Result

2 Displacements, Translational

Position...((NON-LAYERED))

Target Entity: Nodes

3 Apply Reset

Cursor Data

Summary

Cursor Name: default_cursor

Patran 2011

Analysis Code: MSC.Nastran

Load Case: Default, A1:Static Subcase


Select Nodes

Entity ID	XX	YY	ZZ
-----------	----	----	----

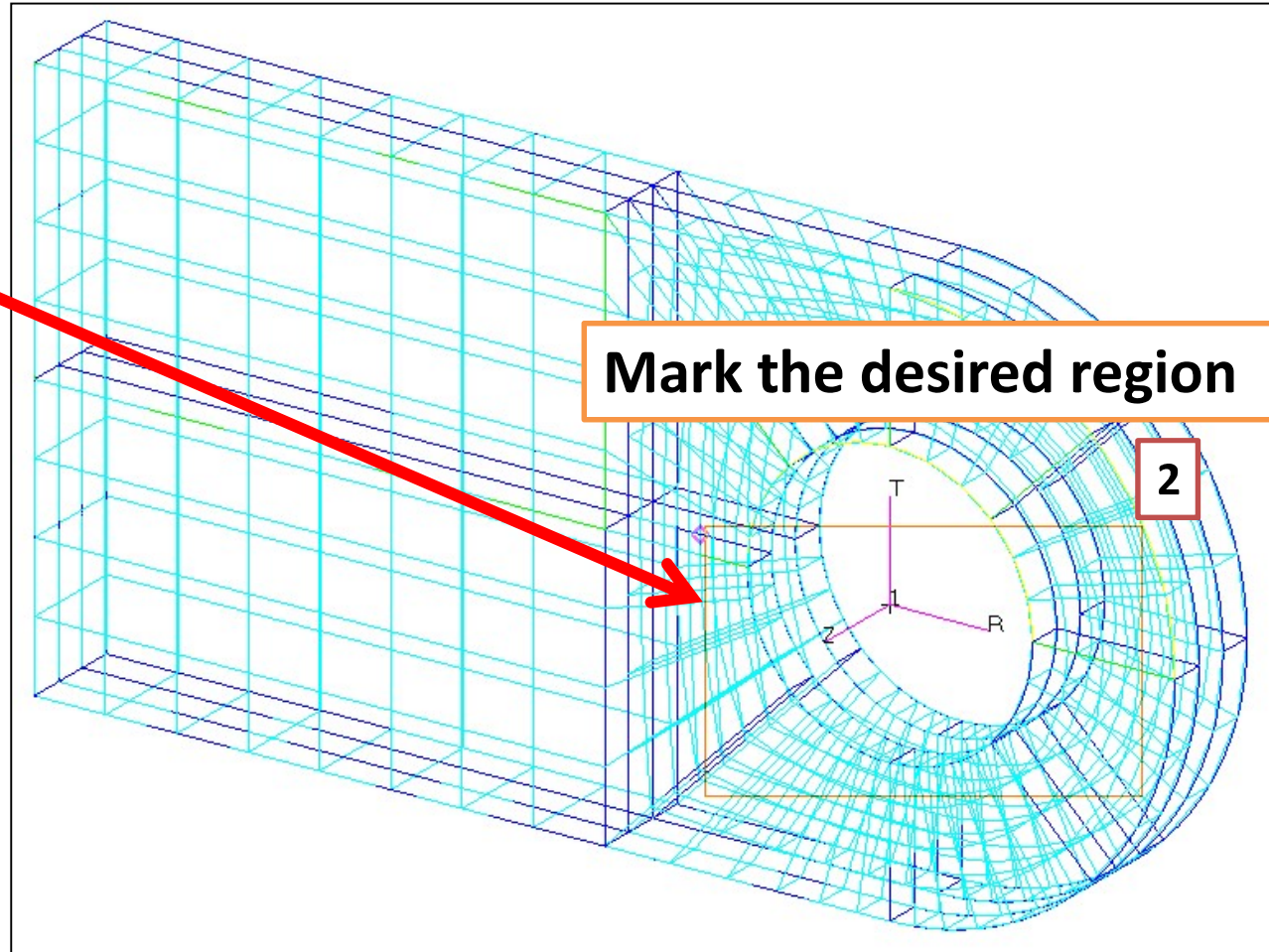
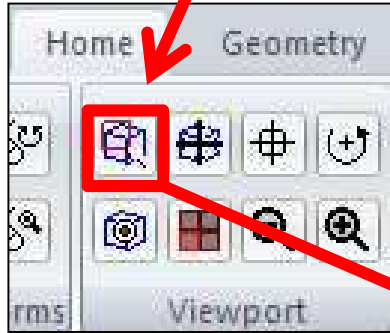
Write Report

Report Setup...

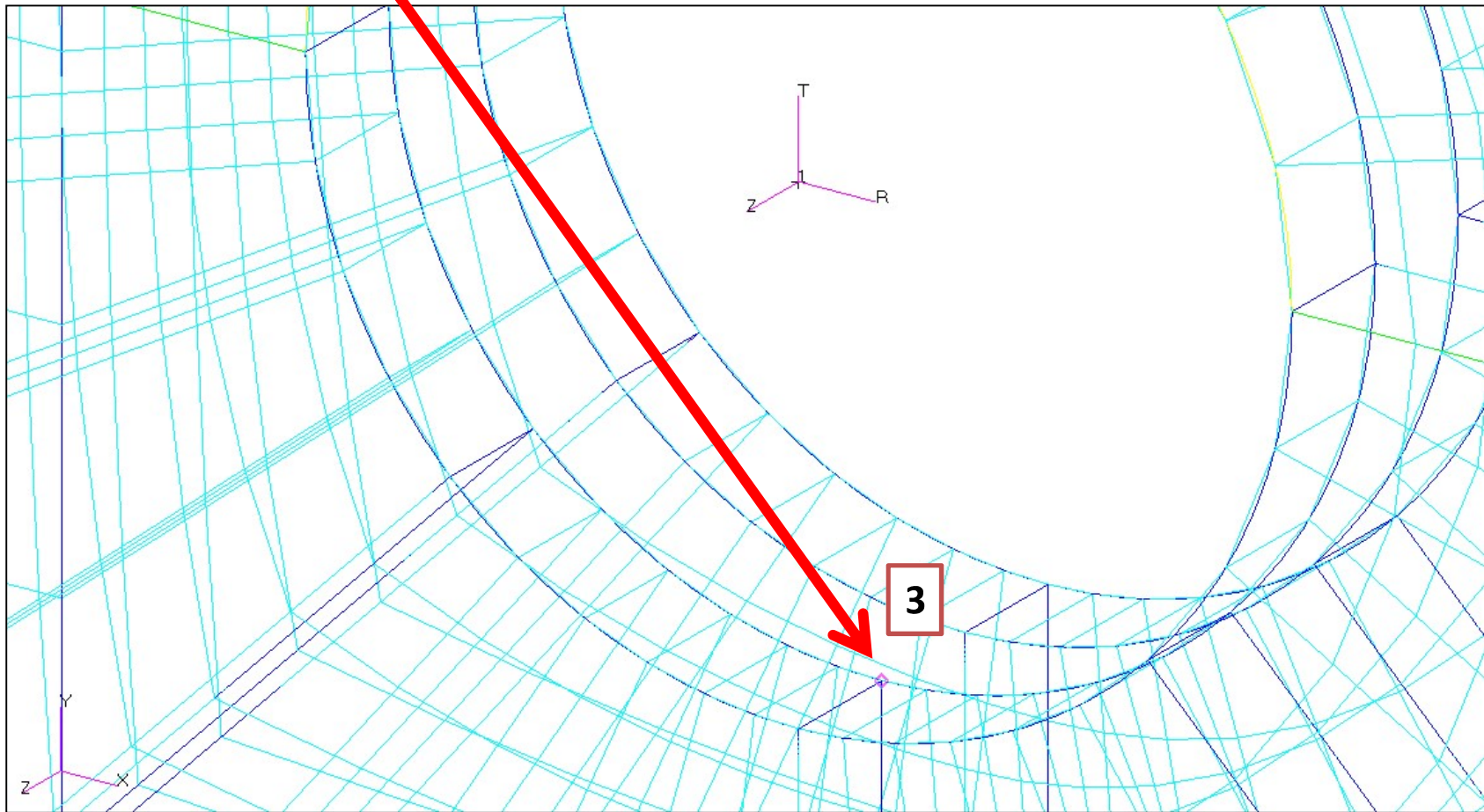
Reset Cancel

 **this window will appear**

1 View Corners

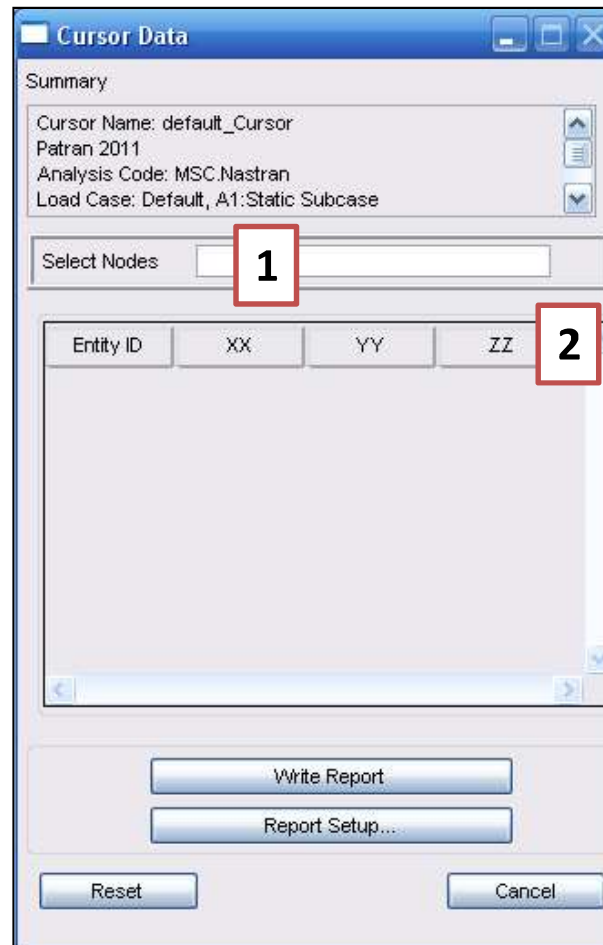


Select the desired node



After selection of the desired node you will see:

- 1 Node ID
- 2 its 3 components of displacement (XX, YY, ZZ)
- 3 Read value of YY

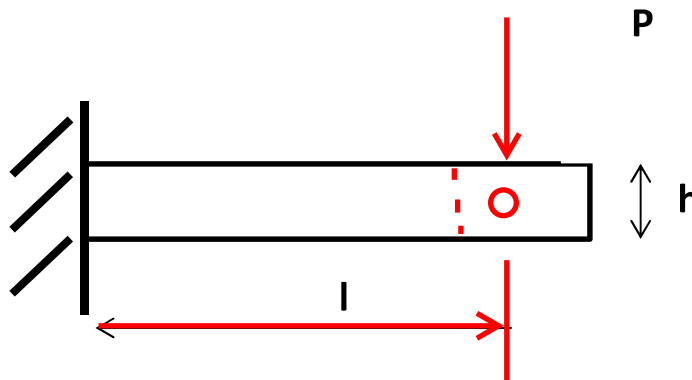


BEAM

Compare the obtained results from the FE analysis (*value of YY, previous slide*) to the **deflection** of the simple model of the beam.

The beam is fixed at one end and loaded by **the same value of force** as for the clevis.

The **material properties** for clevis and beam are **the same**.



BEAM

1. Calculate the deflection of the beam ($f_{beam} = \dots$).

Data:

$l = \dots$ [in] length

$b = \dots$ [in] width

$h = \dots$ [in] height

$E = \dots$ [psi]

$I_y = \dots$ [m⁴]

$P = \dots$ [lbf] resultant load in Y direction (read from the file *clevis.f06*)

2. Calculate the relative error.

3. Draw conclusions.

Report should also contain:

a) Figures:

- 1) Geometrical model (1 figure)
- 2) FE model with load and boundary conditions (1 figure)
- 3) 6 plots with the results:
 - Vertical translational displacements in Y direction
 - Von Mises stress σ_{equiv}
 - Stress in X direction σ_x with averaging, continuous σ_x
 - Stress in X direction σ_x without averaging, discontinuous σ_x
 - Stress in X direction σ_x with averaging, continuous σ_x for the base of the clevis (2 different views)

Total number of figures = 1 + 1 + 6 = **8**

A **white background** of **all** figures is **obligatory**.

A **date** on the plots with the results is **obligatory**.

b) Comparison between **the obtained results from the FE analysis** (*value of YY*) and the **deflection** of the **simple model of the beam**

- the value of the displacement in the direction Y of the node located on the lower surface of the hole at the distance 6 [in]
- formula for the deflection of the beam ($f_{beam} = \dots$)
- data and calculations with proper units
- relative error calculations

c) **Conclusions**