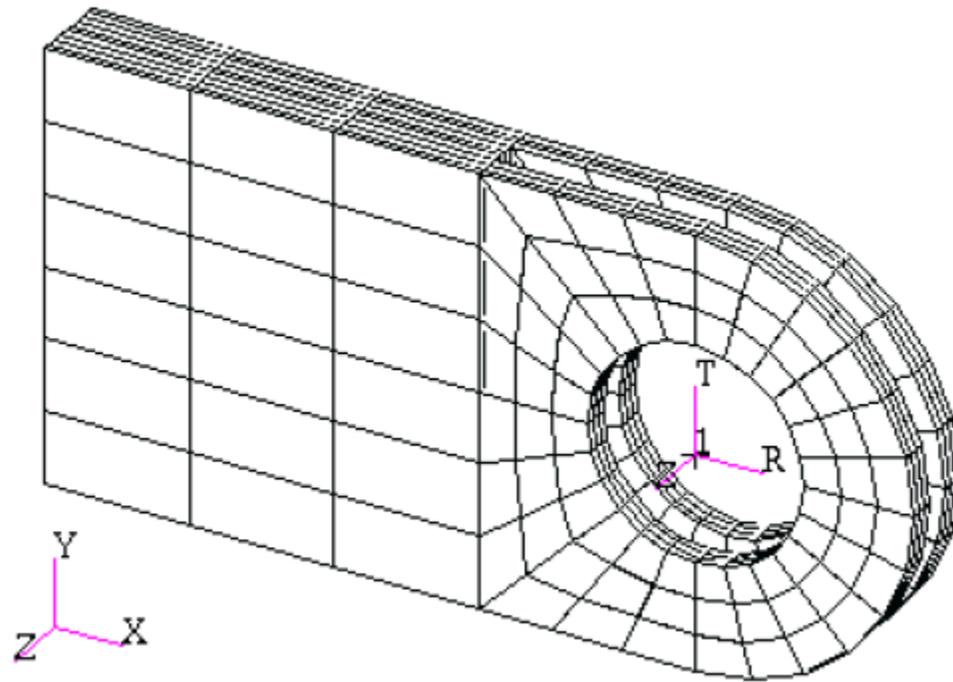


# Geometry model of a 3-D Clevis

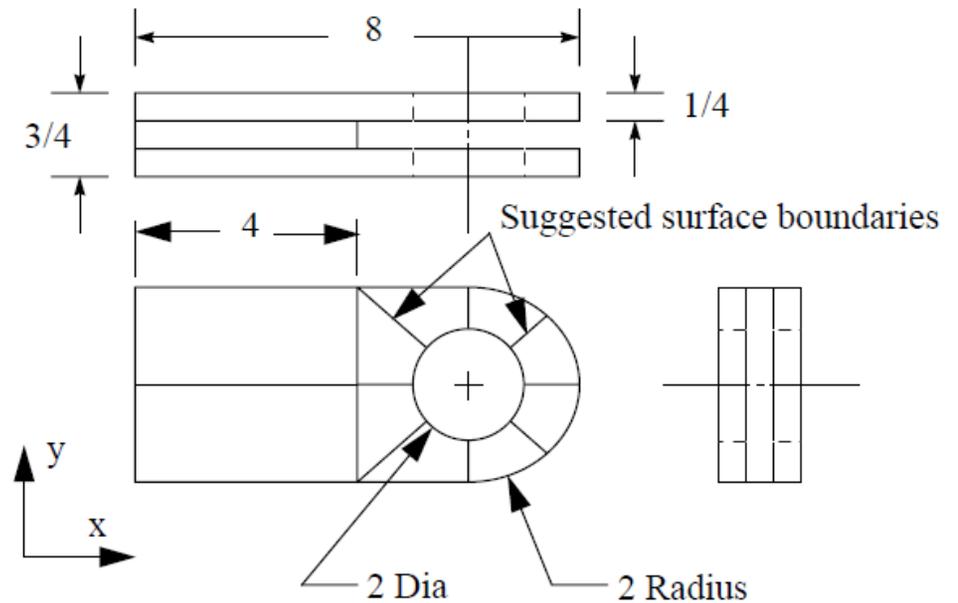
---

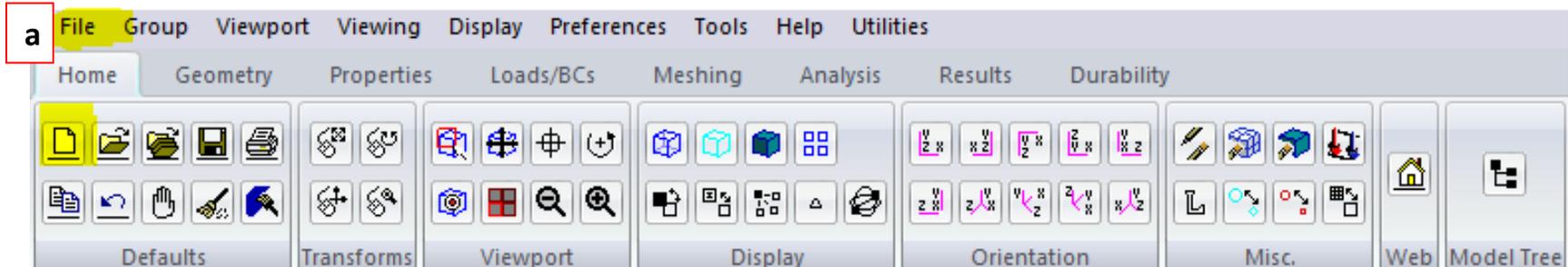


In this exercise you will create an analytic solid model of a clevis by defining MSC/PATRAN points, curves, surfaces, solids, and a user define coordinate system. Throughout this exercise you will become more familiar with the use of the MSC/PATRAN select menu. Shown below is a drawing of the model you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



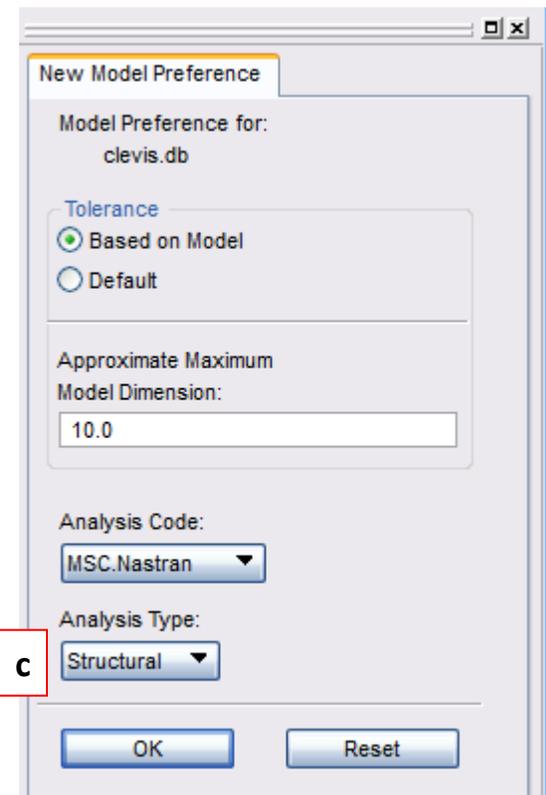
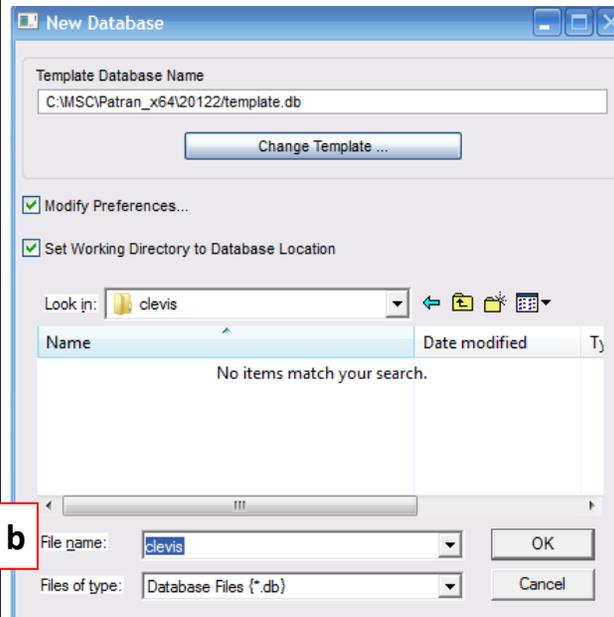


In order to create a new database  
You have to do as follows:

- 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.

- Click on *Geometry* section
- Create -> Point -> XYZ** or chose the method from the list, enter the value [ 1 0 0 ] and hit **-Apply-**
- Go to *Preferences->Geometry*
- Check Exportable to Neutral File** and hit Apply
- 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
- Choose **Point 1** and hit **-Apply-**

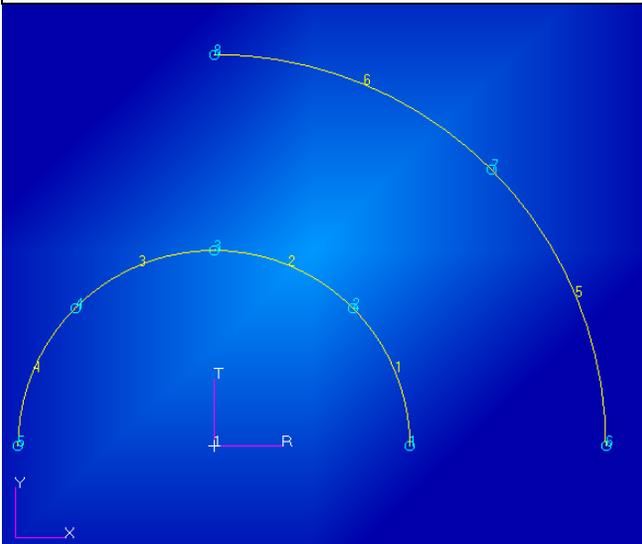
The final effect should looks like on the picture above.

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 **-Apply-**
- Transform->curve->Translate** check **Curvilinear in Refer. CF**, click on newly created coord - **Coord 1** as a reference 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 curves.



**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 in) Surface that defines part of the upper half of the clevis body.

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

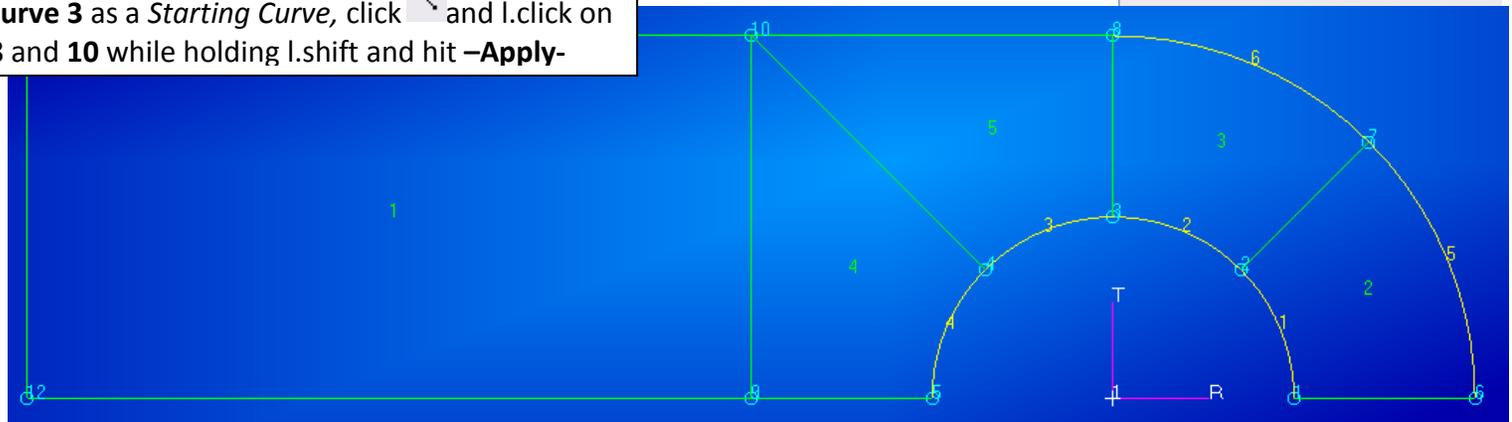
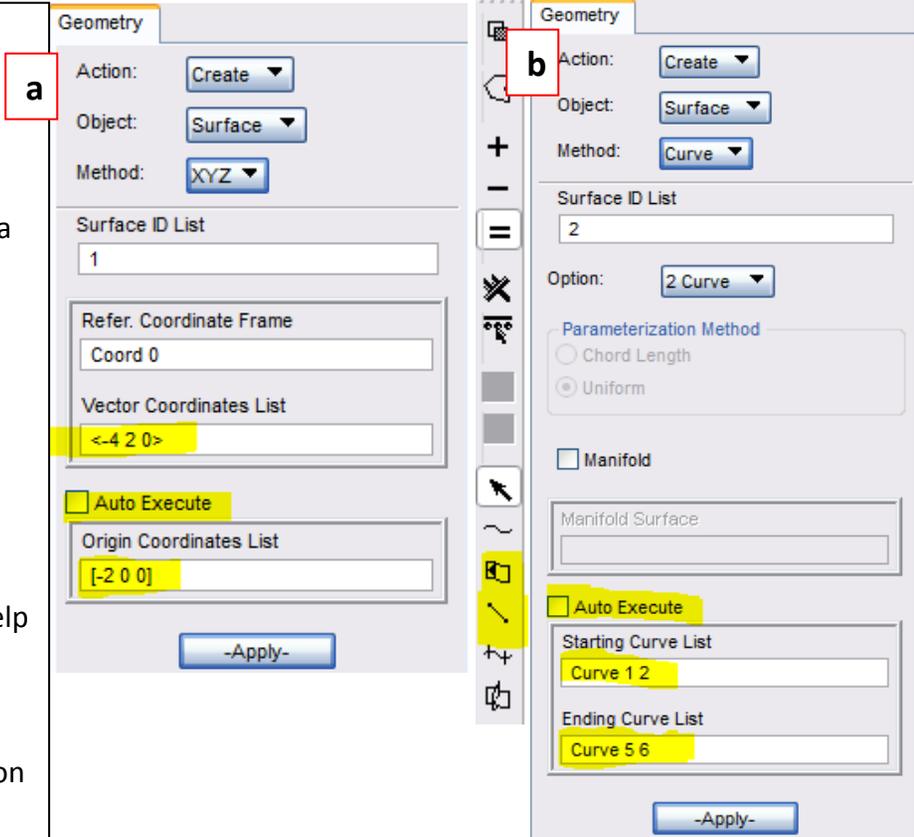
The next series of Surfaces will be created using the *Curve* Method:

- b. **Create->Surface->Curve**, **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.

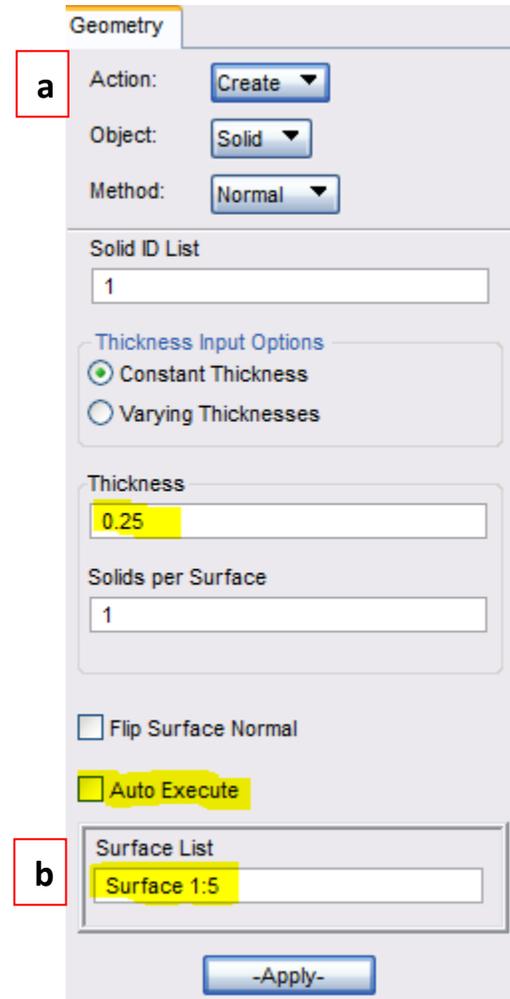
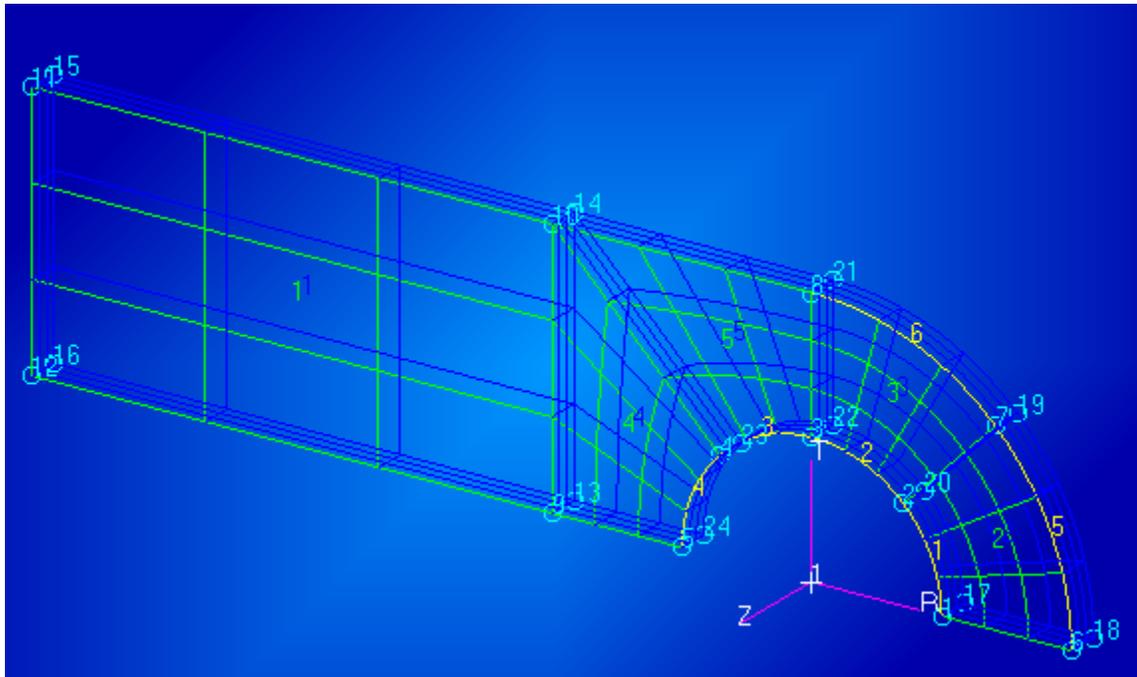
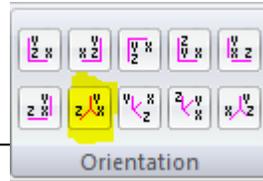
- c. Chose **Curve 4** as a *Starting Curve*, click  and I.click on edge 9-10 of a surface 1 and hit **-Apply-**
- d. Chose **Curve 3** as a *Starting Curve*, click  and I.click on **Points 8** and **10** while holding I.shift and hit **-Apply-**



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-**

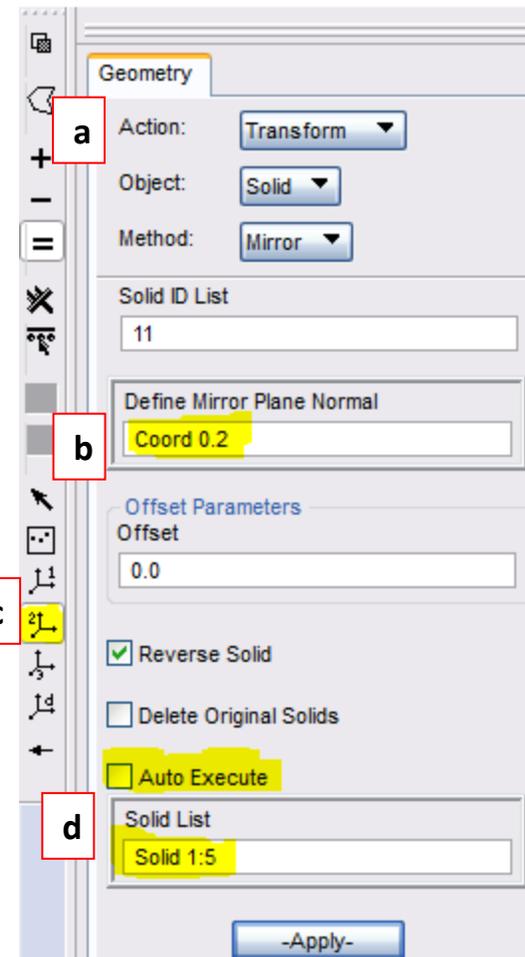
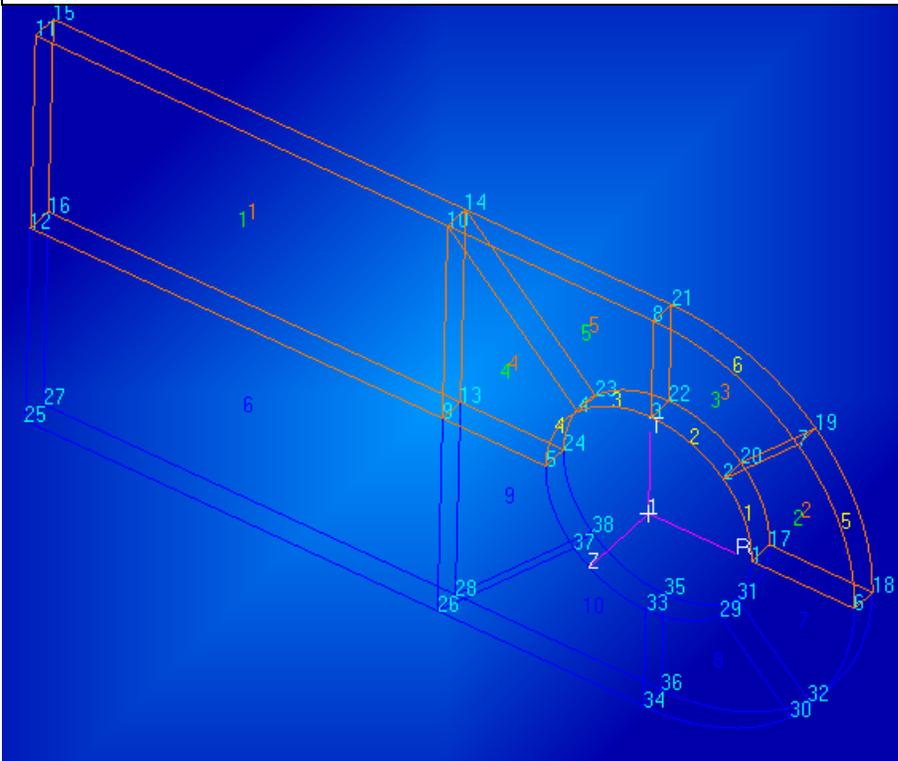
Change the view to **Iso 1** in *Home/Orientation* section and **Fit view** 



### Creating the lower part of the clevis

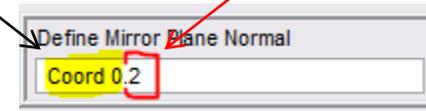
- Transform->Solid->Mirror**
- To *Define Mirror Plane Normal* click the **Coord 0**
- Click on the **Frame Direction 2** on the *Select Menu* to point out the direction of a vector normal to the minor surface.
- Uncheck **Auto Execute**, select all solids **Solid 1:5** and press **-Apply-**

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



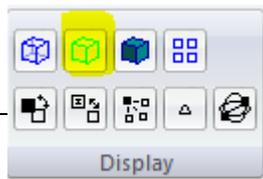
The remaining solids will be created using the translate method.

- 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 **-Apply**

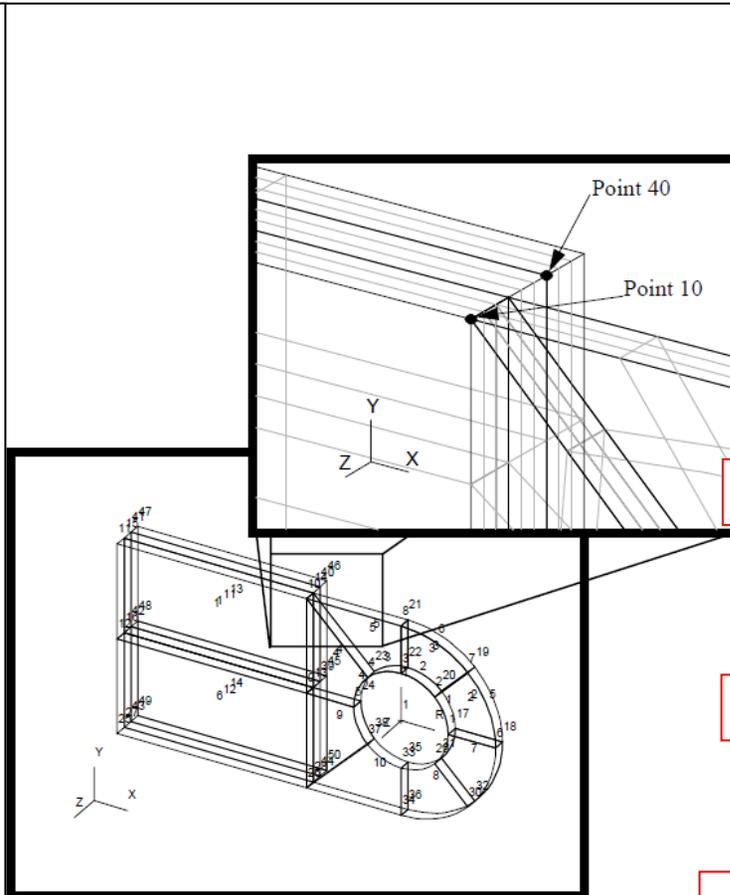
Your last construction step is to translate copies of the solids that surround the hole to create the final solids. Click in the **Translation Vector databox** and click on  in *Select Menu*

- 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** **Solid 2:5 7:10** and hit **-Apply-**

To disable the grid View and enable Body in display 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

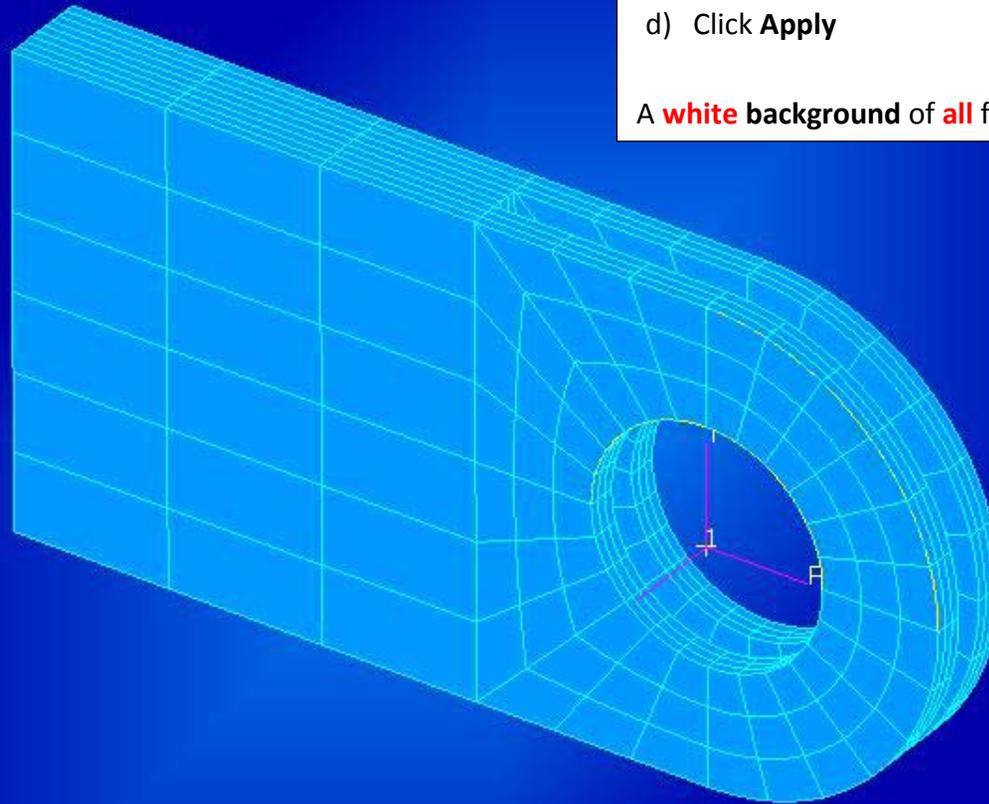
c

d

To save the figure of the geometrical model:

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

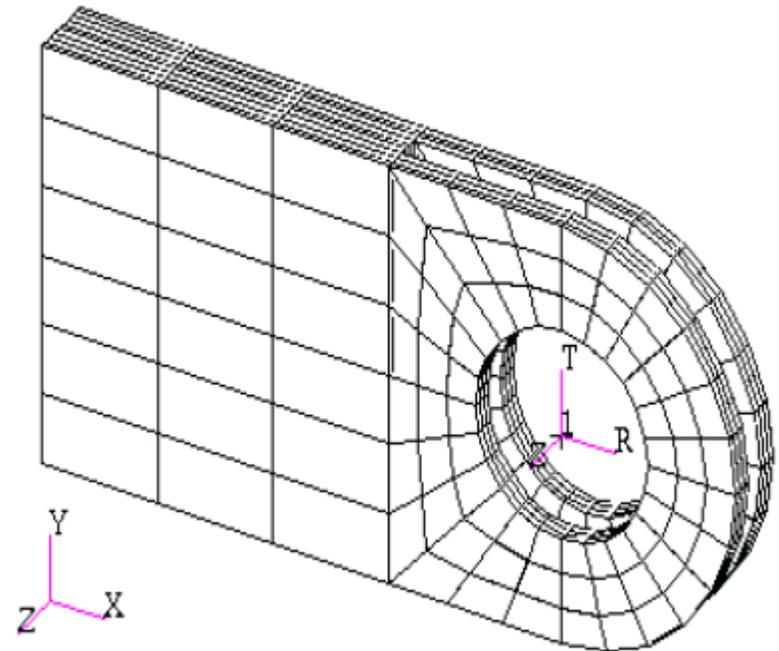
A **white** background of **all** figures is **obligatory**.



# 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.

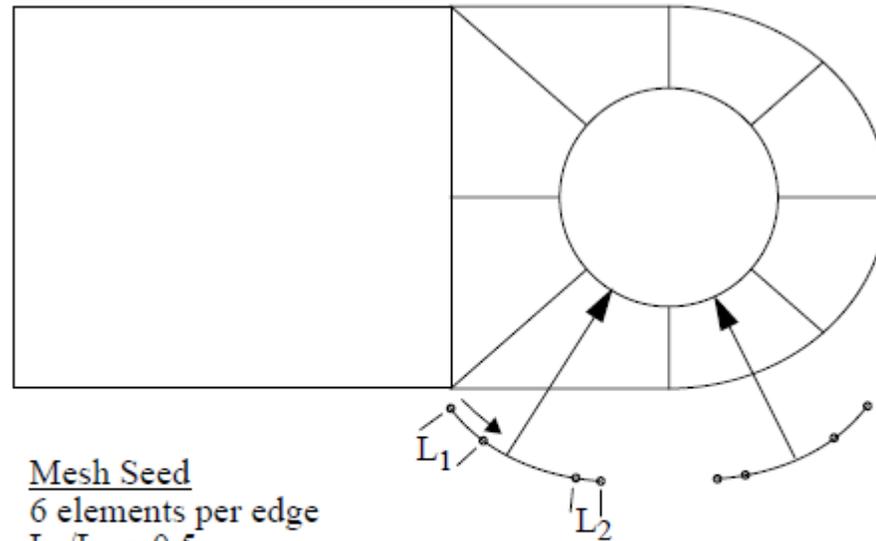


## Model Description:

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 exercise.

## 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 the mesh density in the area where the distributed load will be applied.
3. Create a finite element mesh using the element topology and size listed in the diagram on the right.
4. 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.
5. Create a 3-D element property named, Solid\_Elements\_Steel, for the entire includes the steel material definition



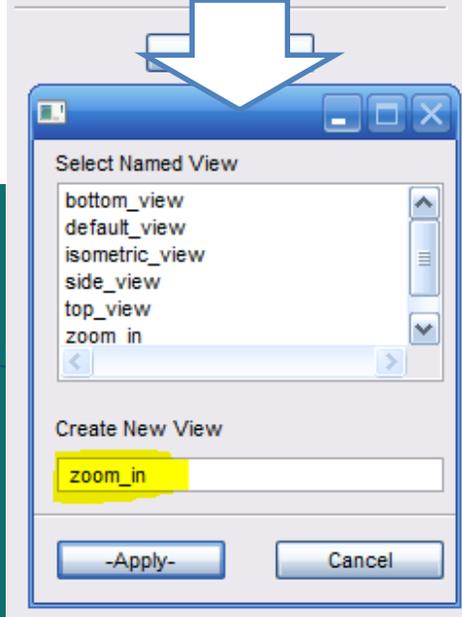
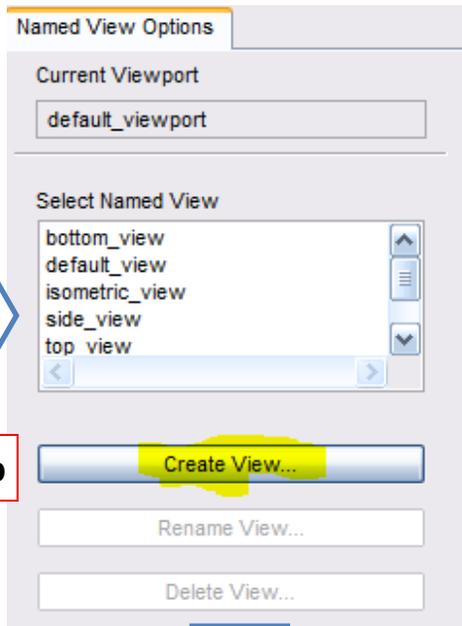
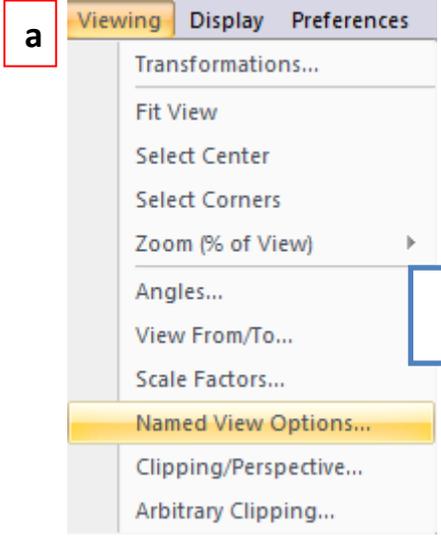
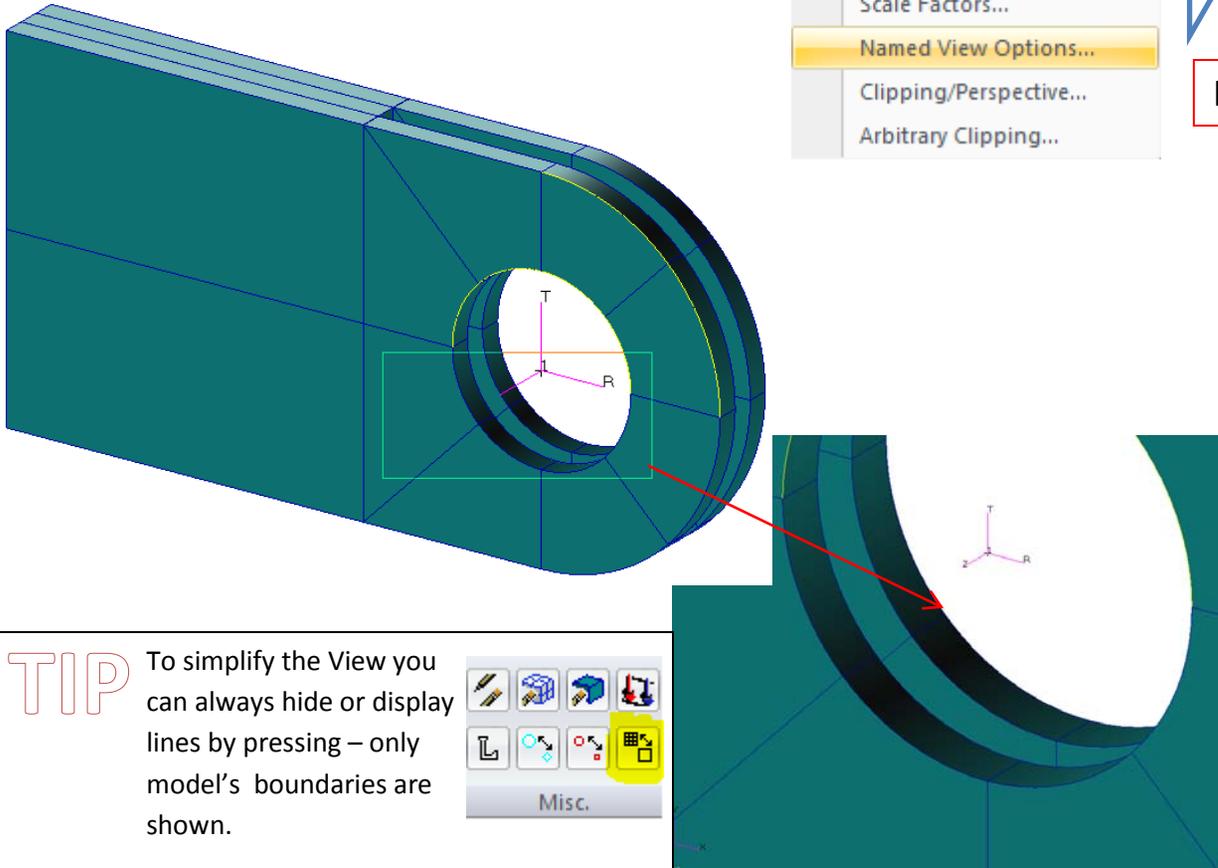
Mesh Seed  
6 elements per edge  
 $L_2/L_1 = 0.5$

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-*



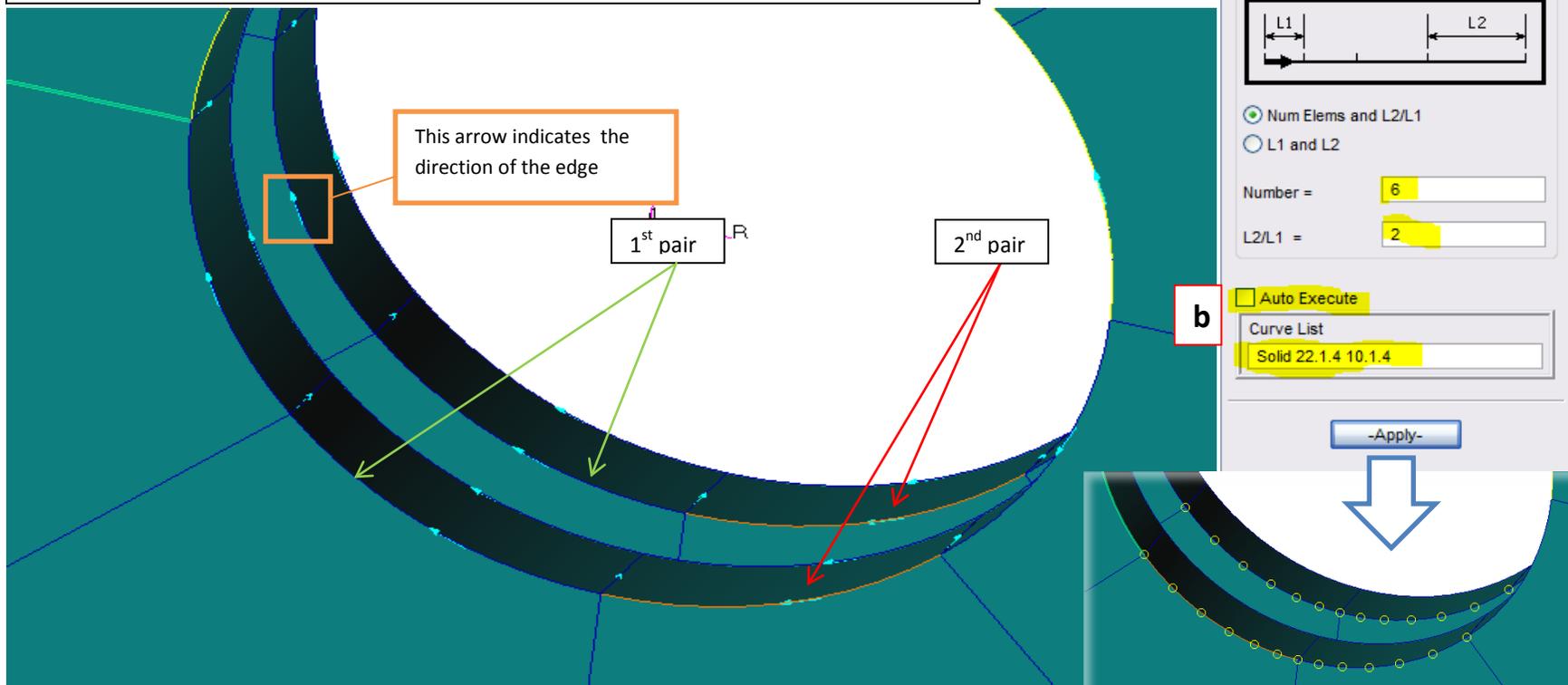
**TIP** To simplify the View you can always hide or display lines by pressing – only model's boundaries are shown.



## 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) and **2** as a *L2/L1* which indicates seeds varying size along an edge. Uncheck *Auto Execute* and holding L.Shift choose the **1<sup>st</sup> pair of edges** shown in figure and press *-Apply-*. Do the same for the **2<sup>nd</sup> pair of edges** but invert the *L2/L1* to maintain symmetry of the seeds, thus Number: **6**, L2/L1: **0.5** and choose **2<sup>nd</sup> pair of edges** and press *-Apply-*

**Tip** In order to select edge click on  in select menu



The image shows a software interface for creating mesh seeds. The main window displays a curved part with two pairs of edges highlighted in green and red, labeled "1<sup>st</sup> pair" and "2<sup>nd</sup> pair". A callout box points to an arrow on the green edge, stating "This arrow indicates the direction of the edge". The "Finite Elements" panel on the right shows the "Create" action, "Mesh Seed" object, and "One Way Bias" type. The "Element Edge Length Data" section shows "Num Elems and L2/L1" selected, with "Number" set to 6 and "L2/L1" set to 2. The "Auto Execute" checkbox is unchecked, and the "Curve List" contains "Solid 22.1.4 10.1.4". The "-Apply-" button is highlighted with a large blue arrow pointing to the resulting meshed part in the bottom right inset.

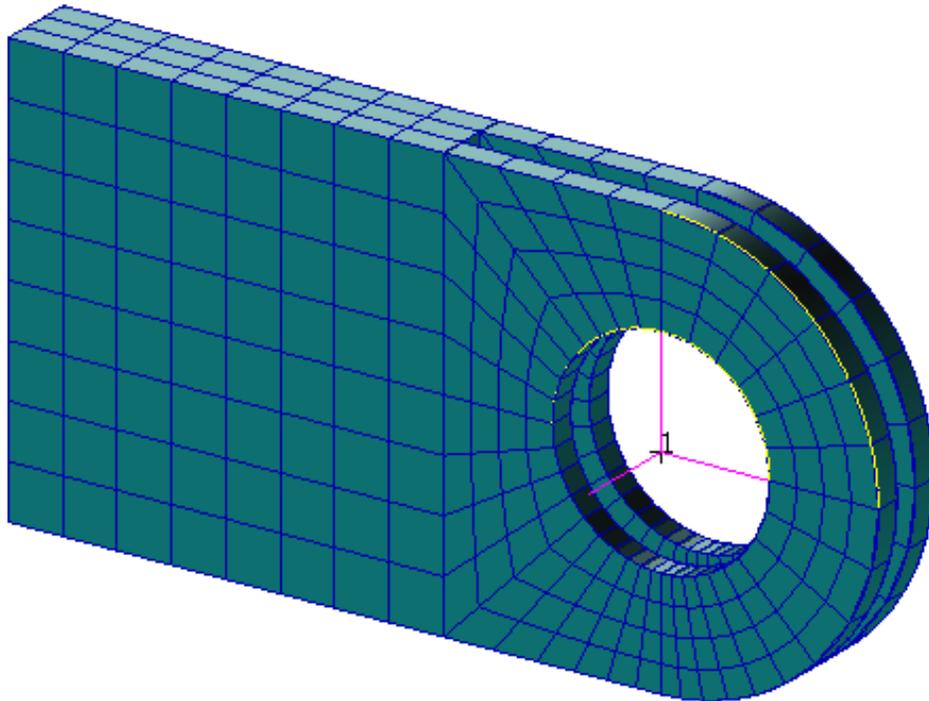
### 3. Creating Mesh

- 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

IsoMesh Parameters...

Node Coordinate Frames...

Solid List

Solid 1:22

Global Edge Length

Automatic Calculation

Value: 0.5

Prop. Name: - None -

Prop. Type: - N/A -

Select Existing Prop...

Create New Property...

-Apply-

a

b

c

d

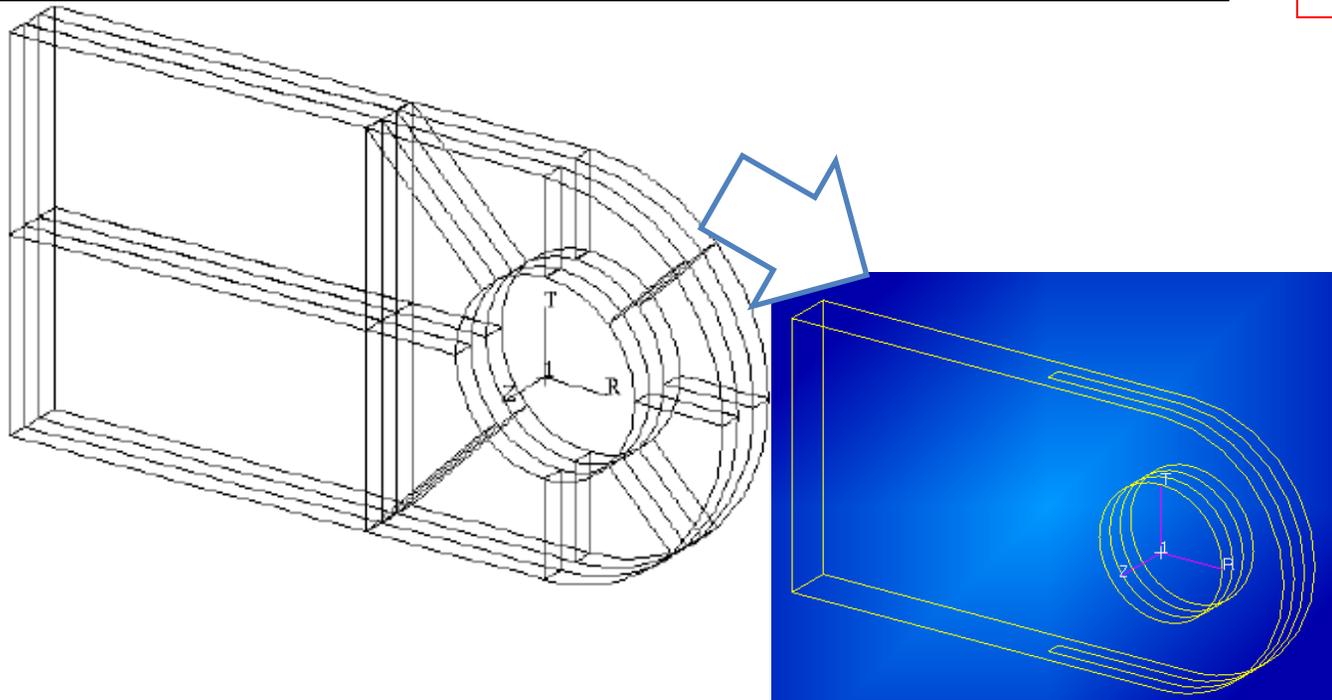
Now that you have created your finite element mesh, it is time to determine whether you need to “equivalence the model”. To do this:

- a. **Verify->Element->Boundaries**, 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 dimensional 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 location.

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

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



Finite Elements

Action: **Verify**

Object: **Element**

Test: **Boundaries**

Display Type

**Free Edges**

Free Faces

Reset Graphics

**Apply**

Finite Elements

Action: **Equivalence**

Object: **All**

Method: **Tolerance Cube**

Node Id Options:

Retain lower node id

Collapsed Node Options:

Allow Tolerance Reduction

Nodes to be excluded

Equivalencing Tolerance

0.005

Element Boundary Verify

Display Type

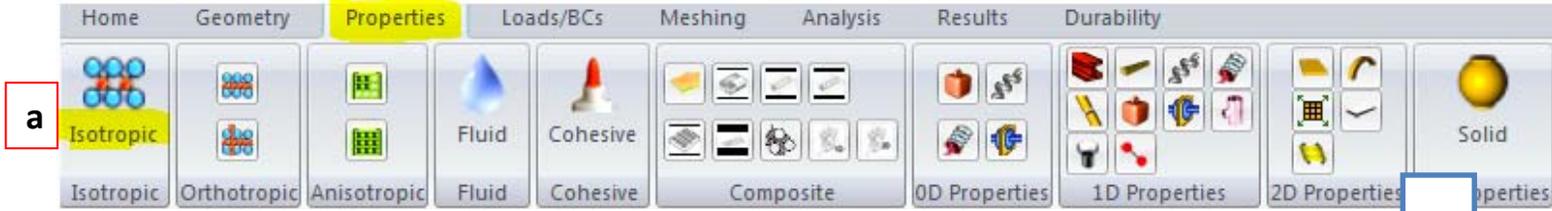
**Free Edges**  Free Faces

Verify Reset

Preview Nodes

Preview Reset

**-Apply-**



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-* and *-Apply-* in *Materials* window

Constitutive Model: Linear Elastic

Property Name	Value
Elastic Modulus =	30e6
Poisson Ratio =	0.3
Shear Modulus =	
Density =	
Thermal Expan. Coeff =	
Structural Damping Coeff =	
Reference Temperature =	

Materials

Action: Create  
Object: Isotropic  
Method: Manual Input

Existing Materials

- Steel

Material Name: Steel

Description: Date: 23-Mar-14 Time: 22:29:04

Input Properties ...  
Change Material Status ...  
Apply

Constitutive Model Status

Active Constitutive Models:

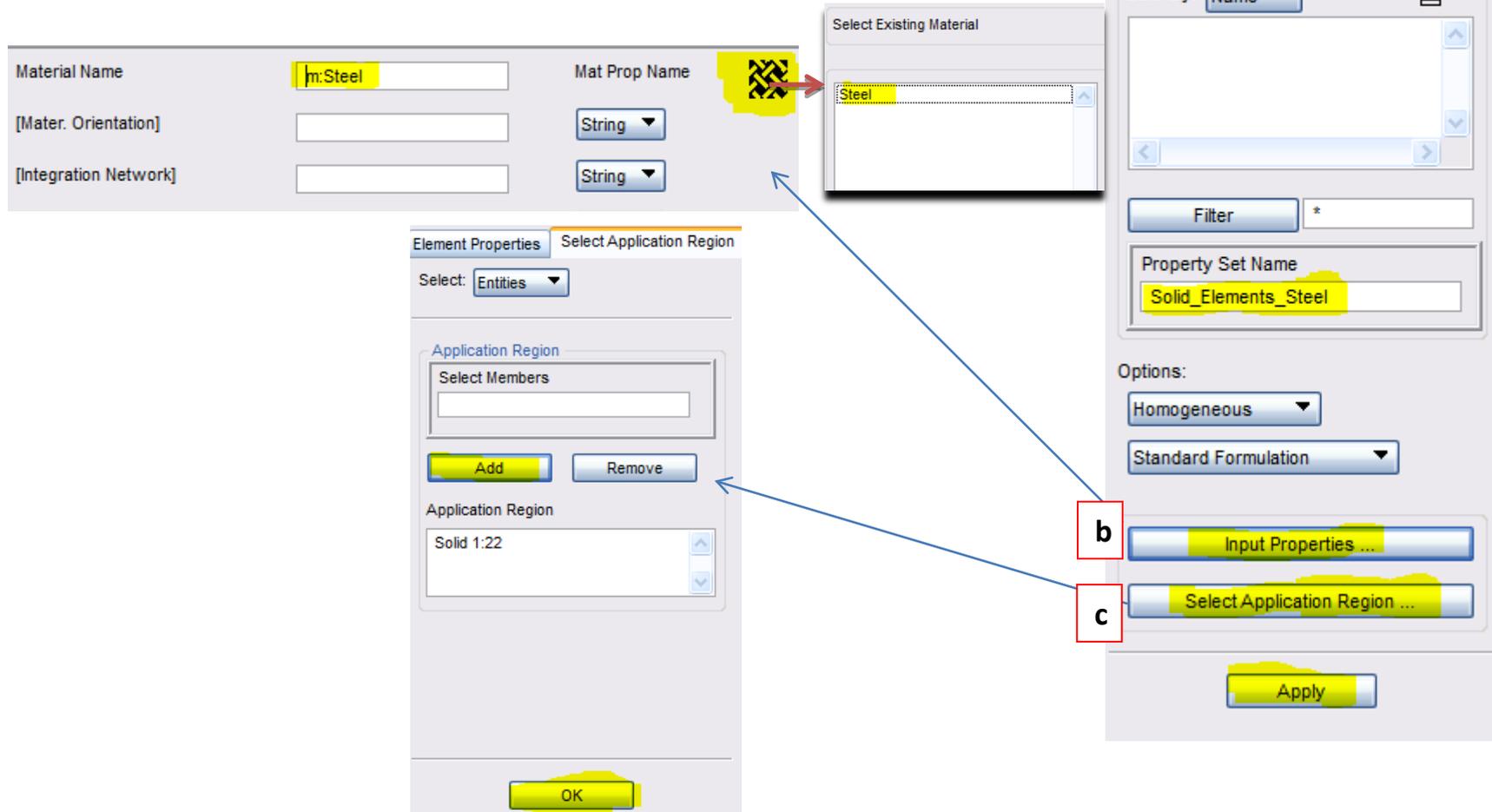
- Linear Elastic

Inactive Constitutive Models:

-Apply- Cancel

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

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



The image displays a software interface for creating a 3-D element property. The main window is titled "Element Properties" and has several sections:

- Action:** Create (dropdown)
- Object:** 3D (dropdown)
- Type:** Solid (dropdown)
- Sets By:** Name (dropdown)
- Property Set Name:** Solid\_Elements\_Steel (text field)
- Options:** Homogeneous (dropdown), Standard Formulation (dropdown)
- Buttons:** Input Properties ..., Select Application Region ..., Apply

Below the main window, there are two smaller dialog boxes:

- Select Existing Material:** Shows a list with "Steel" selected.
- Select Application Region:** Shows "Solid 1:22" in the "Application Region" list. The "Add" button is highlighted.

Annotations include:

- a:** A red box around the "Create" dropdown in the main window.
- b:** A red box around the "Input Properties ..." button in the main window.
- c:** A red box around the "Select Application Region ..." button in the main window.

# Loads and Boundary Conditions on a 3-D Clevis

## Objectives:

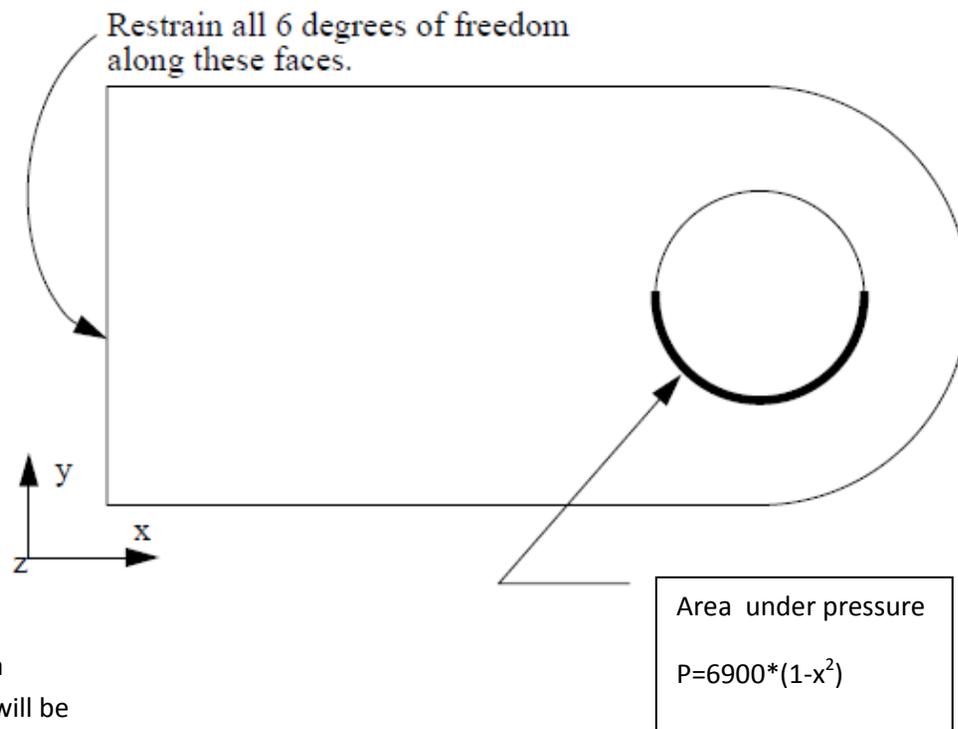
- ❖ Apply constraints to your model.
- ❖ Create and apply a Pressure

## 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  $P = 6900 \cdot (1-x^2)$ .



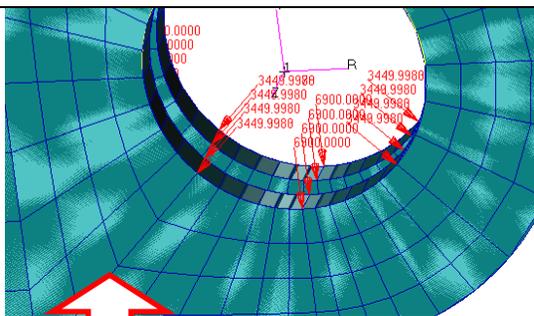
1. Creating a pressure case (ATTENTION: 'X, below, is **NOT** TYPED, it is chosen from the list of Independent Variables!!):

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

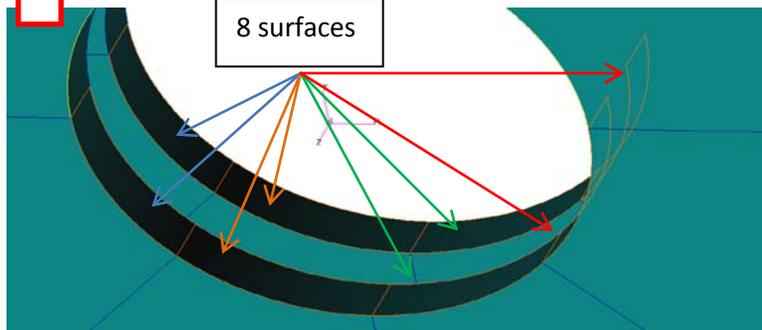
b. **Element Variable/ Pressure**, Enter Pressure as a name, change target to **3D** and click **Input Data**.

c. Double click on **Pressure\_field** and **ok**

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



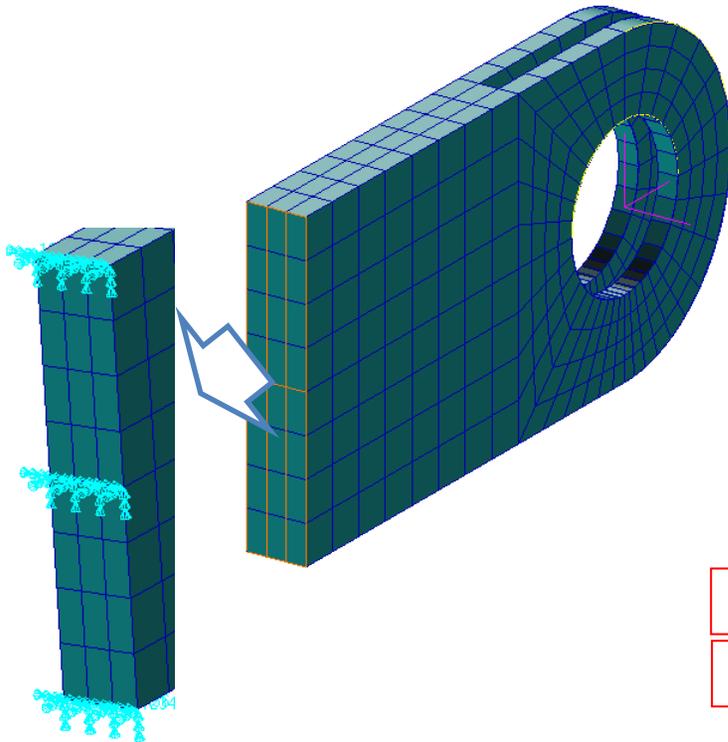
8 surfaces



The screenshot shows the software interface for creating a pressure case. The 'Load/BCs' panel is active, showing the 'Input Data' tab. The 'Pressure' field is defined with the name 'f.Pressure\_field'. The 'Spatial Fields' list contains 'Pressure\_field'. The 'FEM Dependent Data...' button is highlighted. The 'OK' and 'Reset' buttons are also visible. The 'Fields' panel is also visible, showing the 'Create' action, 'Spatial' object, and 'PCL Function' method. The 'Field Name' is 'Pressure\_field', the 'Field Type' is 'Scalar', and the 'Coordinate System Type' is 'Real'. The 'Scalar Function' is  $6900*(1-X**2)$ . The 'Independent Variables' list contains 'X', 'Y', and 'Z'. The 'Input Data...' and 'Select Application Region...' buttons are highlighted. The '-Apply-' button is also visible.

## 2. Create a Nodal Displacement boundary condition:

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



The screenshot shows the software interface for creating a boundary condition. The main window is titled "Load/Boundary Conditions" and has a tab labeled "Input Data". 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 "Existing Sets" list is empty. The "New Set Name" is "Clamped".

The "Input Data..." button is highlighted in yellow, and a green arrow points from it to the "Input Data" tab in the "Load/Boundary Conditions" dialog. The "Select Application Region..." button is highlighted in yellow, and a red arrow points from it to the "Select Application Region" dialog.

The "Load/Boundary Conditions" dialog has the following fields:

- Load/BC Set Scale Factor: 1.
- Translations <T1 T2 T3>:  $\langle 0,0,0 \rangle$
- Rotations <R1 R2 R3>:  $\langle 0,0,0 \rangle$
- Trans Phase <Tp1 Tp2 Tp3>:  $\langle \quad \quad \quad \rangle$
- Rotation Phase <Rp1 Rp2 Rp3>:  $\langle \quad \quad \quad \rangle$
- Spatial Fields:  $\langle \quad \quad \quad \rangle$
- FEM Dependent Data...:  $\langle \quad \quad \quad \rangle$
- Analysis Coordinate Frame: Coord 0
- Buttons: OK, Reset

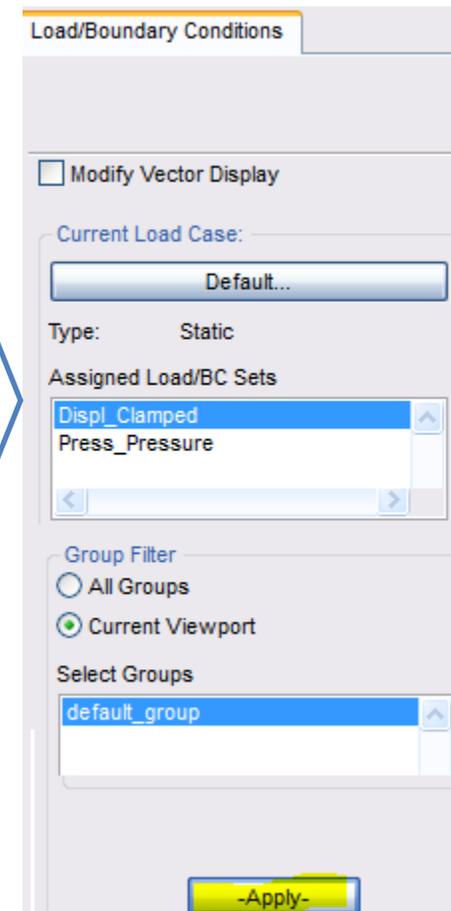
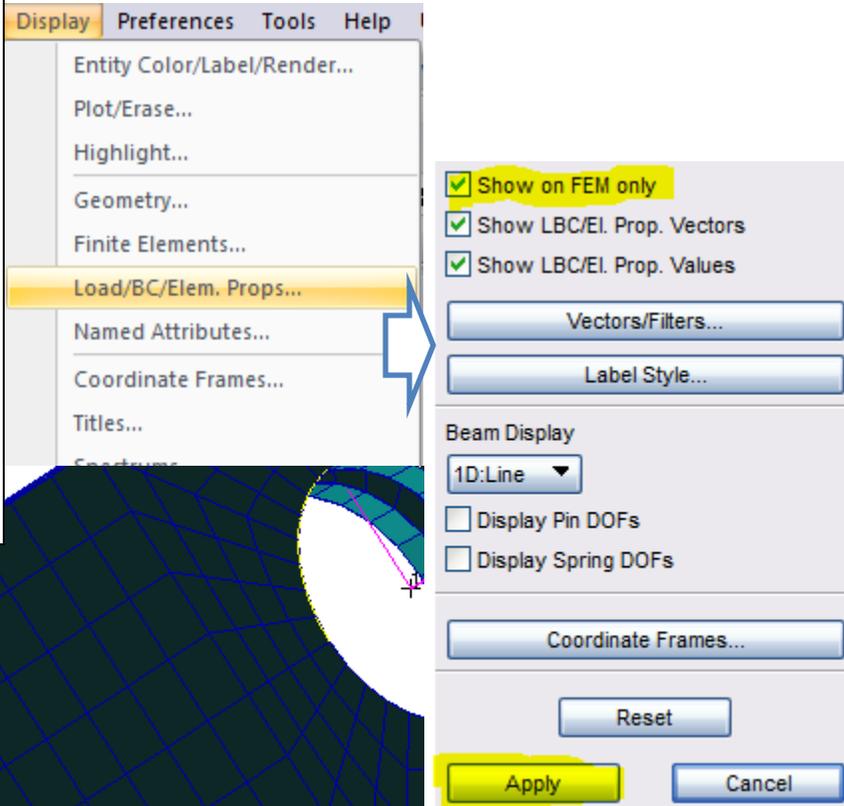
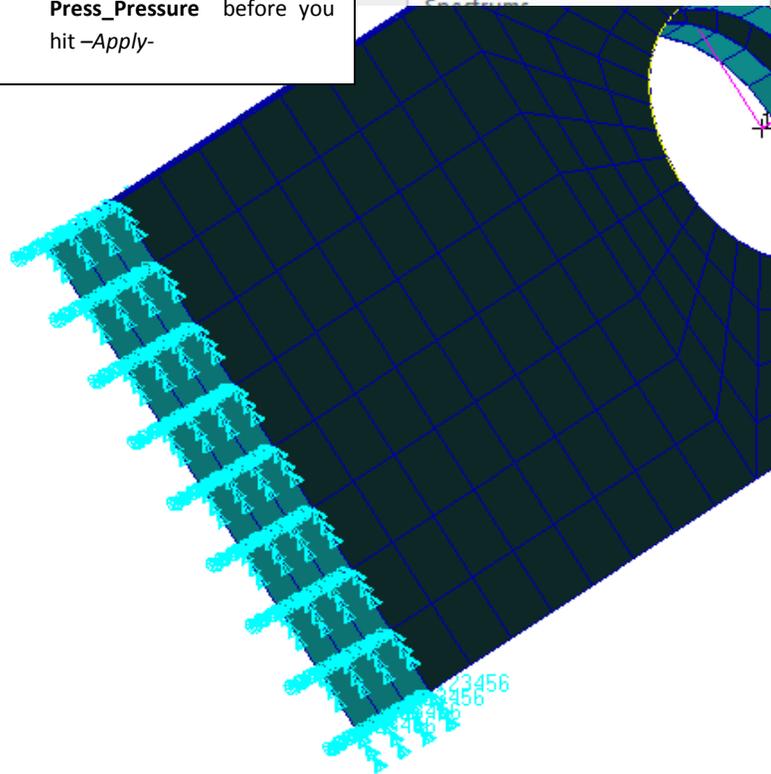
The "Select Application Region" dialog has the following fields:

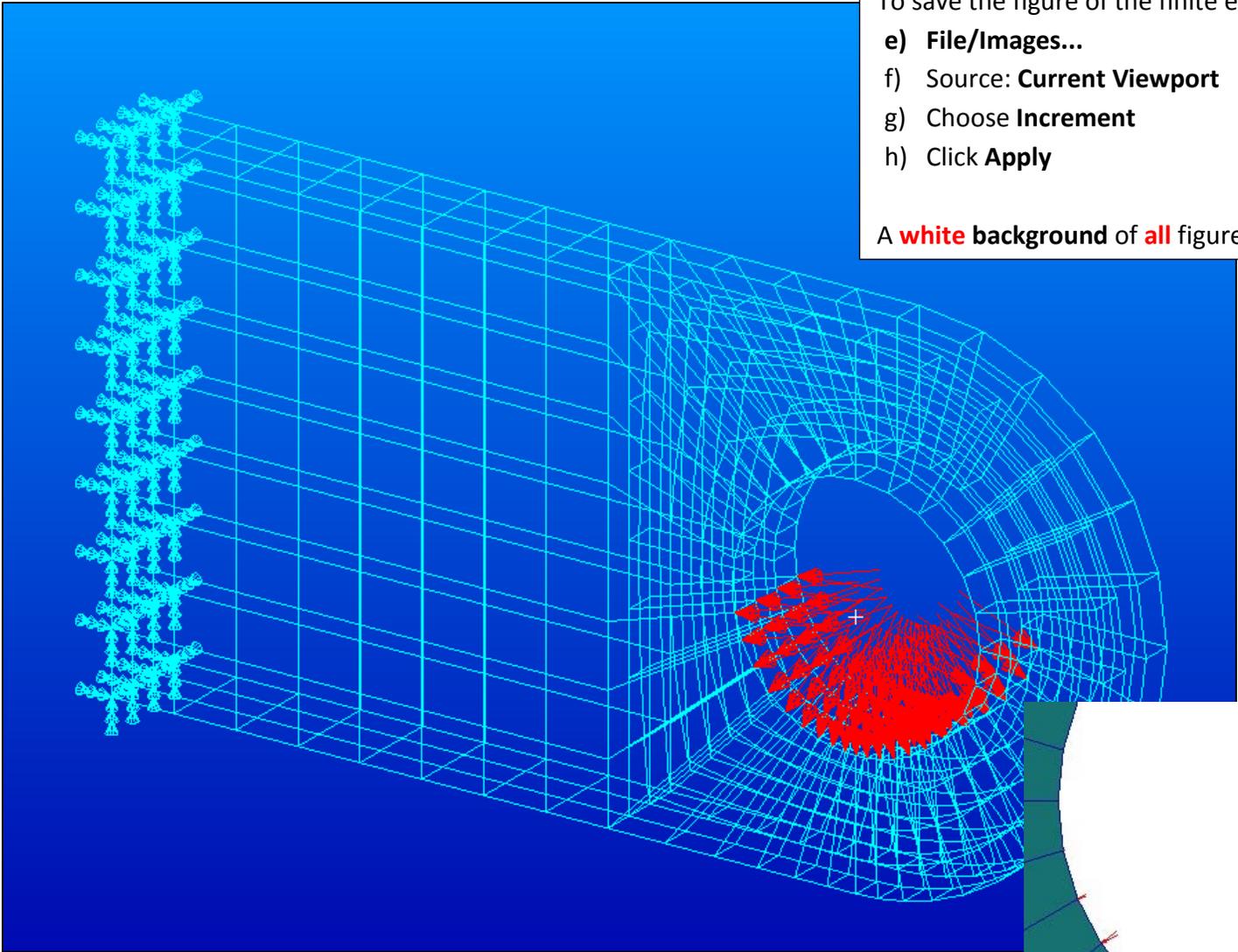
- Select: Geometry
- Auto Select...:  $\langle \quad \quad \quad \rangle$
- Application Region: Select Geometry Entities
- Select Geometry Entities: Solid 13.2 13.2 11.2 1.2 6.4 12.4 14
- Buttons: Add, Remove
- Application Region: Solid 1.2 11:13:2.2 6.4 12:14:2.4
- Buttons: OK

The interface also shows a toolbar with various icons, including a plus sign, a minus sign, a selection tool, and a copy icon.

**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 an *Action*
- c) Highlight **Displ\_Clamped** and *-Apply-*. If you want to show on the model also a pressure, highlight also **Press\_Pressure** before you hit *-Apply-*

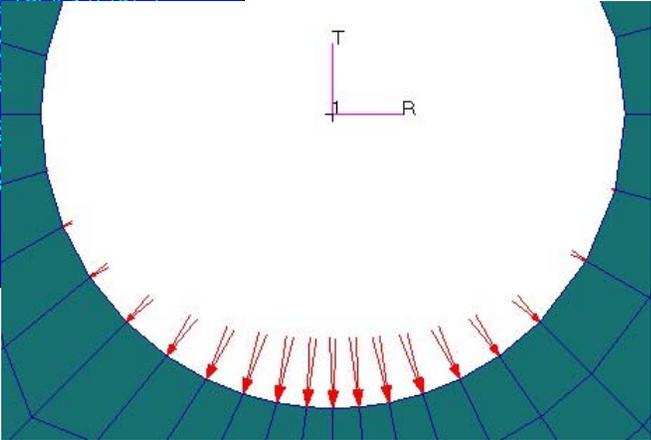




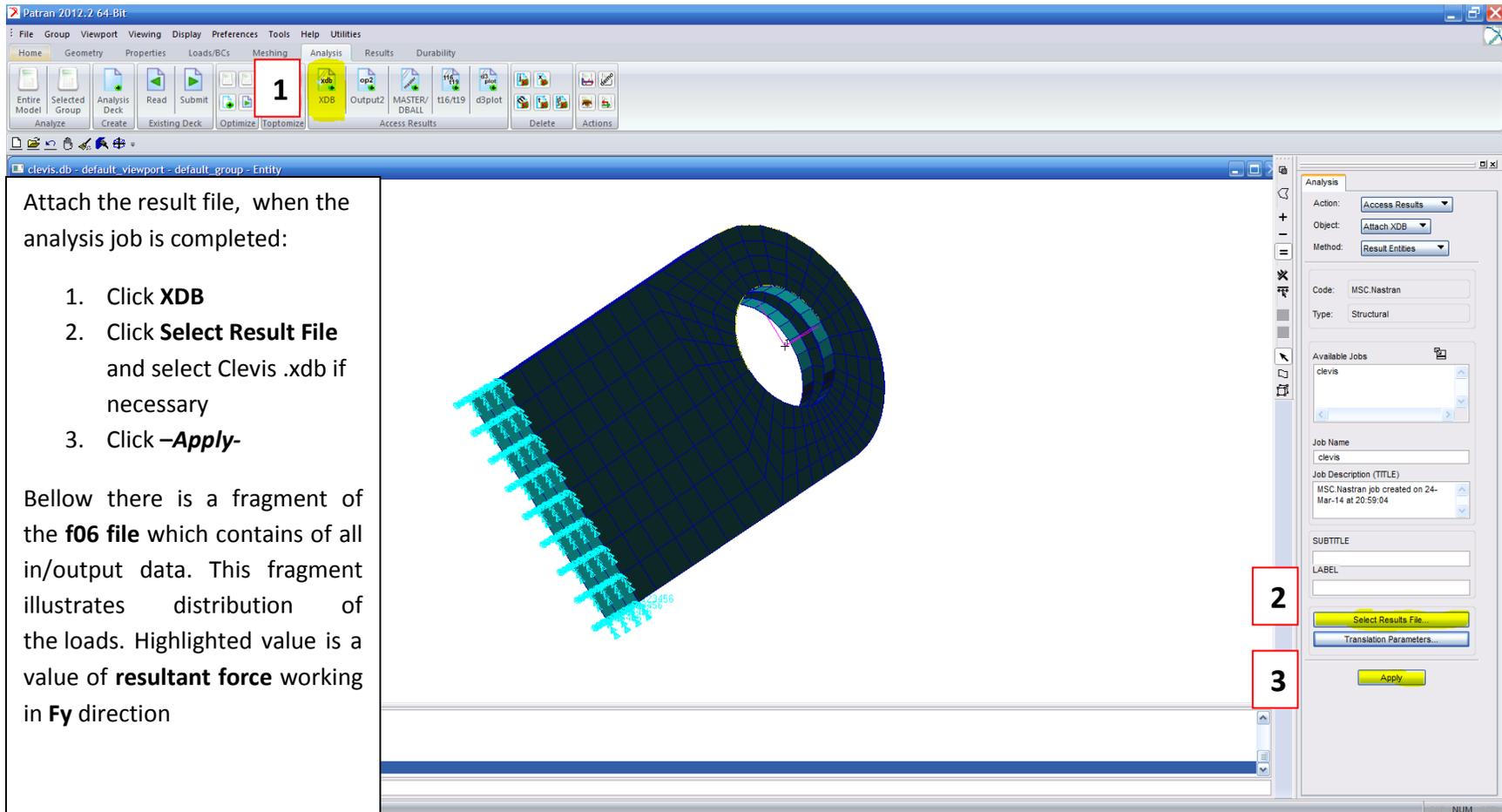
To save the figure of the finite element model:

- e) **File/Images...**
- f) Source: **Current Viewport**
- g) Choose **Increment**
- h) Click **Apply**

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







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 7/ 6/12 PAGE 63							

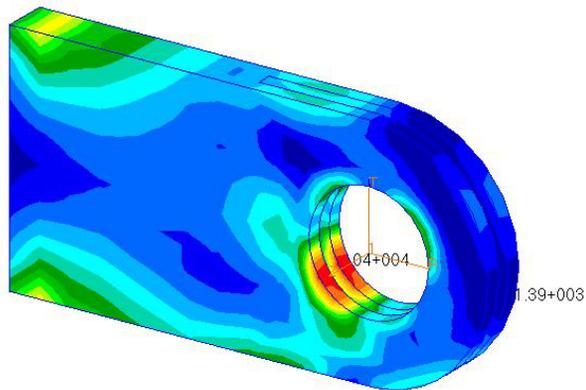
# Post Processing of Stress Results

## 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)



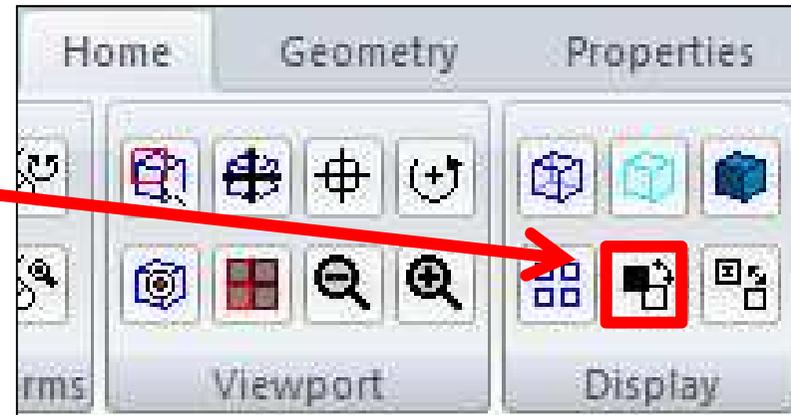
default\_Fringe3 :  
Max 2.04+004 @Nd 1246  
Min 1.39+003 @Nd 1160

## Create 6 different plots with results:

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

Set white background:

**Cycle Background**



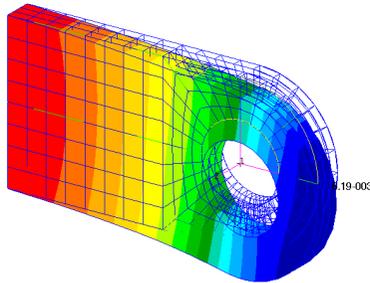
Follow the steps (numbers in **red frames**). **→** **1**

# 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

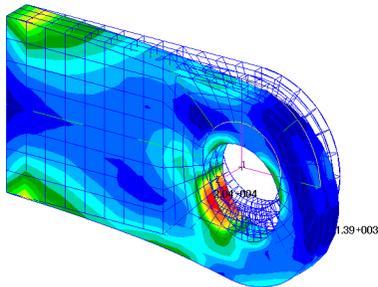
Apply Reset

# PLOT no. 2

Von Mises stress  $\sigma_{equiv}$

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  6 

Show Spectrum  
 Show Viewport Legend

Spectrum... Range...

Style: Discrete/Smooth

Shading: None

0.0 1.0 0.0

Element Shrink Factor

Fringe Edges 

Display: Free Edges

Style: 

Width: 

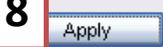
Title Editor...

Show Title  Lock Title

Show Max/Min Label  
 Show Fringe Label

Label Style...

Show on Deformed

8  

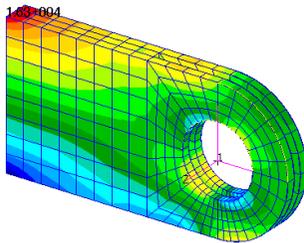
4

### PLOT no. 3

Stress in X direction  $\sigma_x$   
with averaging, continuous  $\sigma_x$

**1** Results -> Create -> Fringe

**9** File -> Images... -> Apply



Results

Action: Create

Object: Fringe

**2** [Icon: Fringe]

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, **4**

Position...((NON-LAYERED))

Quantity: X Component

Animate

Apply Reset

Results

Action: Create

Object: Fringe

**5** [Icon: Fringe]

Show Spectrum

Show Viewport Legend

Spectrum... Range...

Style: Discrete/Smooth **6**

Shading: None

0.0 1.0 0.0

Element Shrink Factor

Fringe Edges

Display: Element Edges **7**

Style: [Dropdown]

Width: [Dropdown]

Title Editor...

Show Title  Lock Title

Show Max/Min Label

Show Fringe Label

Label Style...

Show on Deformed

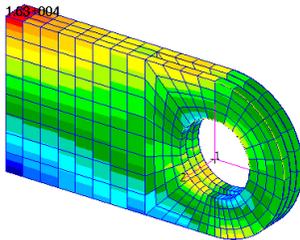
**8** Apply Reset

# PLOT no. 4

Stress in X direction  $\sigma_x$   
without averaging, discontinuous  $\sigma_x$

1 Results -> Create -> Fringe

8 File -> Images... -> Apply



Results

Action: Create

Object: Fringe

2 [Icon: Fringe]

Select Result Cases

Default, A1:Static Subcase,-MSC.NAS

Select Fringe Result

Stress Invariants, Mean Pressure

Stress Invariants, Minor Principal

Stress Invariants, Von Mises

3 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:

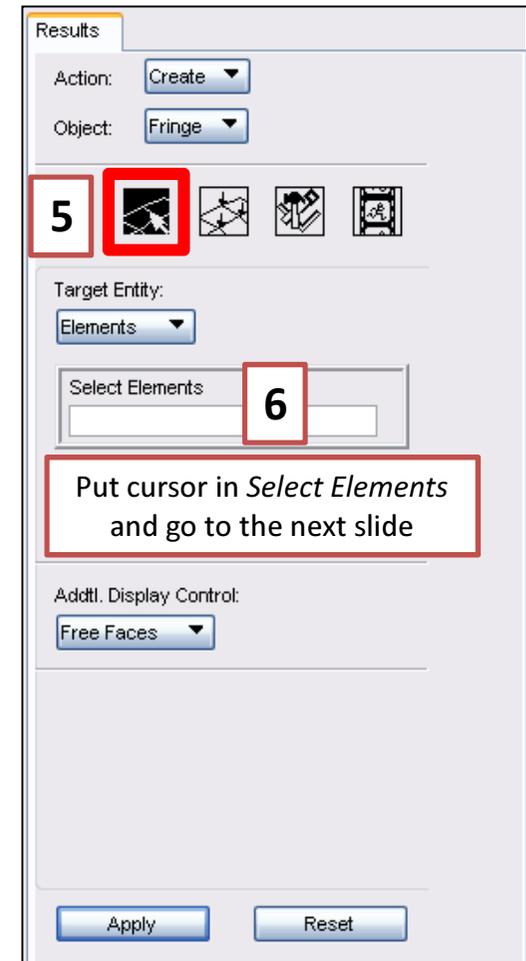
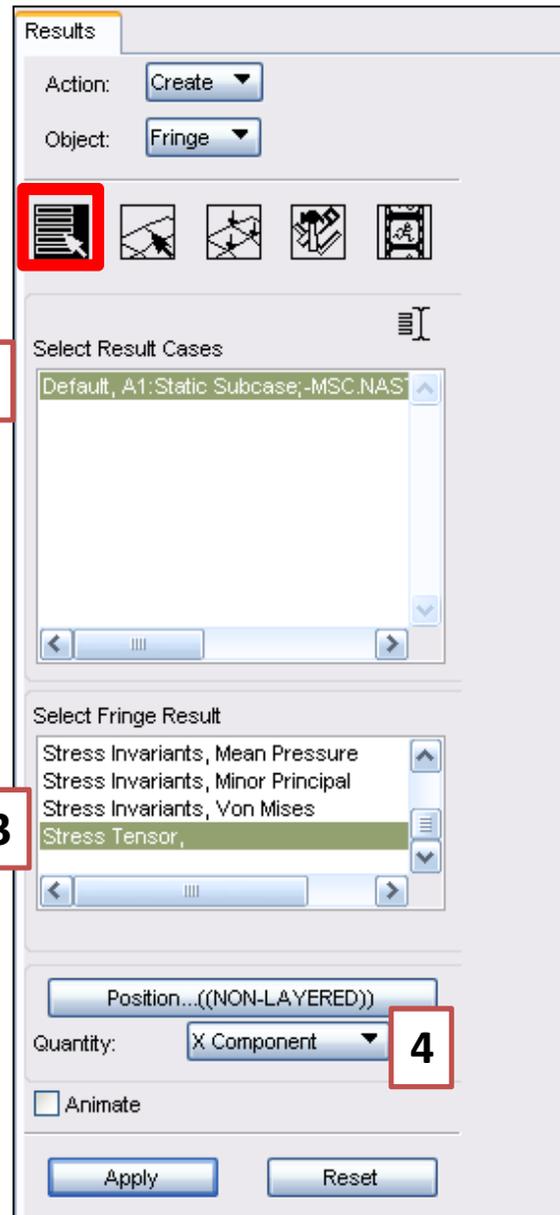
7 Apply Reset

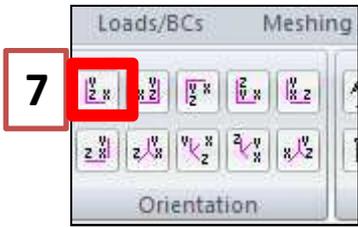
All Entities change to None

## PLOTS: no. 5 and no. 6

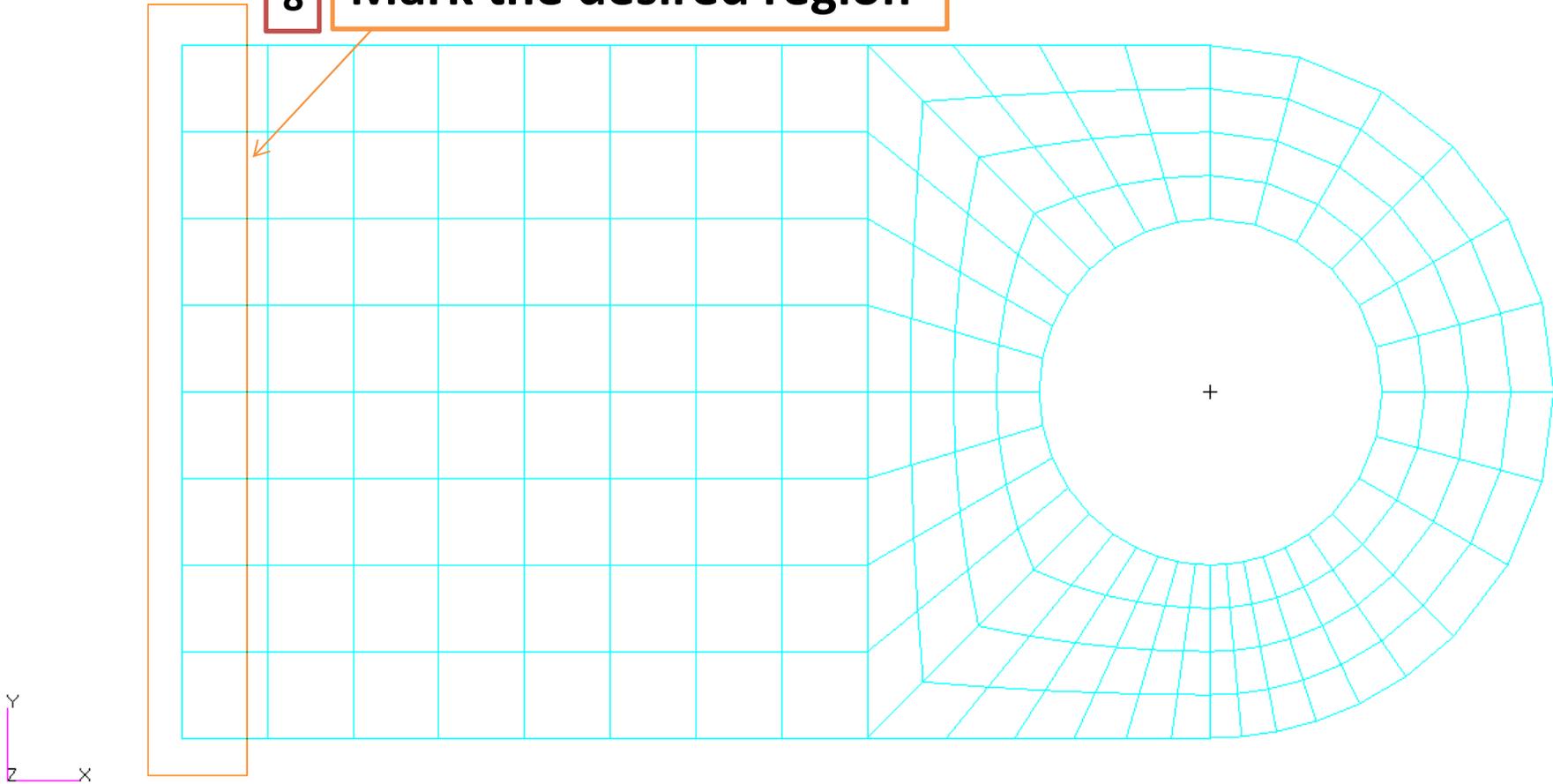
Stress in X direction  $\sigma_x$   
with averaging, continuous  $\sigma_x$  for the base  
of the clevis (2 different views)

**1** Results -> Create -> Fringe



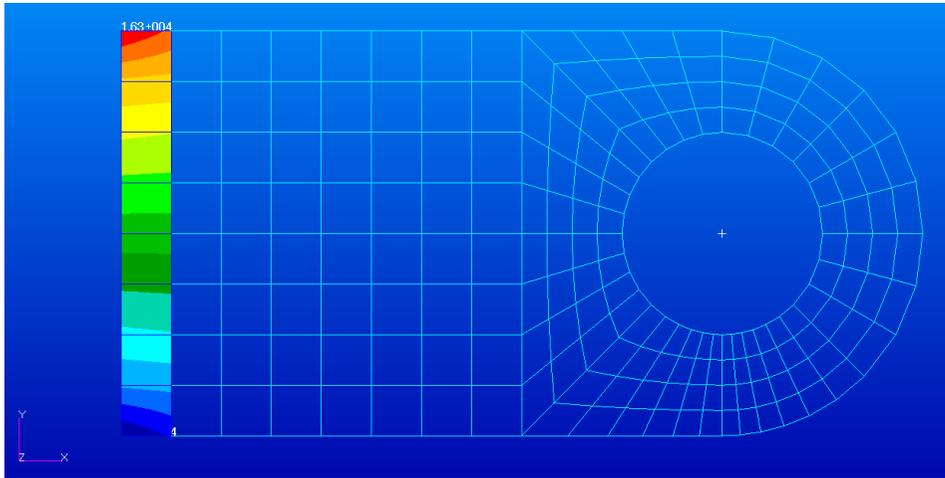


8 Mark the desired region

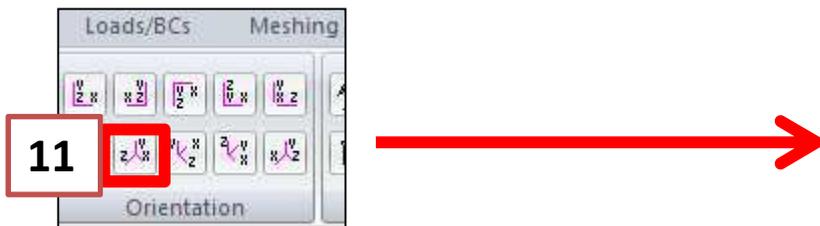


9 Click **Apply**

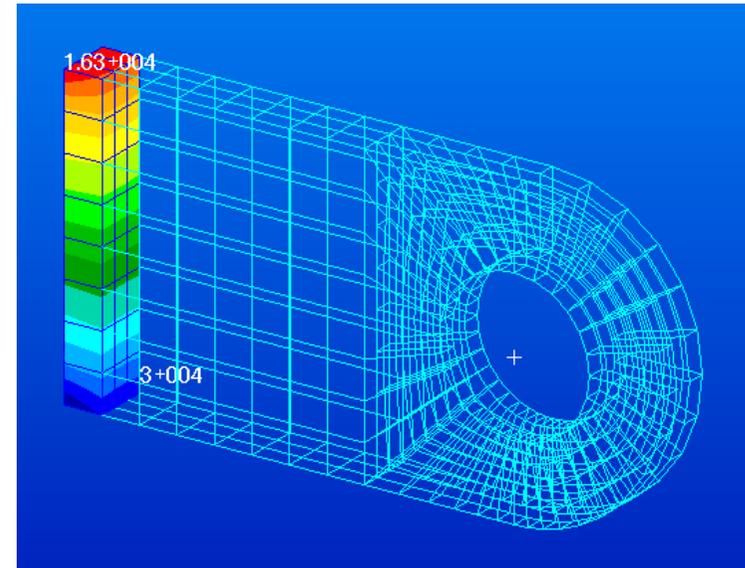
10 File -> Images... -> Apply



What is the distribution of the  $\sigma_x$  stress at the base of the clevis along the vertical direction?

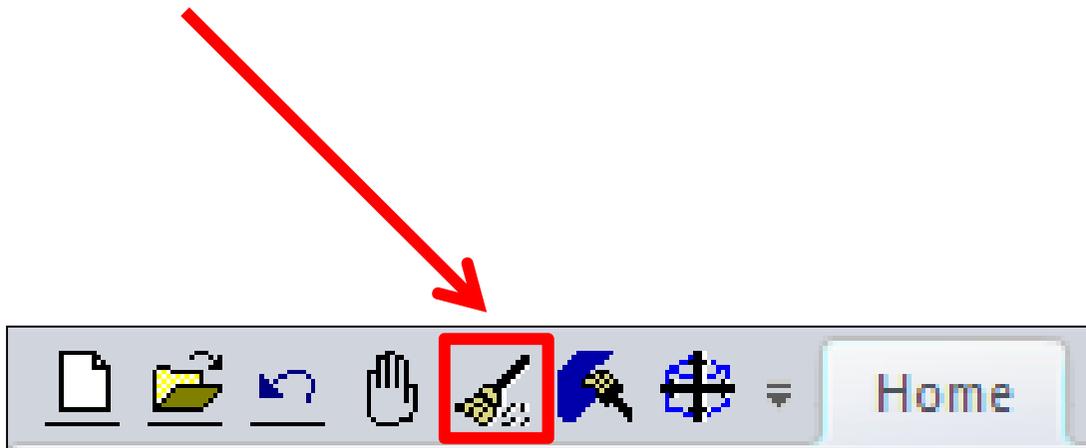


12 File -> Images... -> Apply



**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
-----------	----	----	----

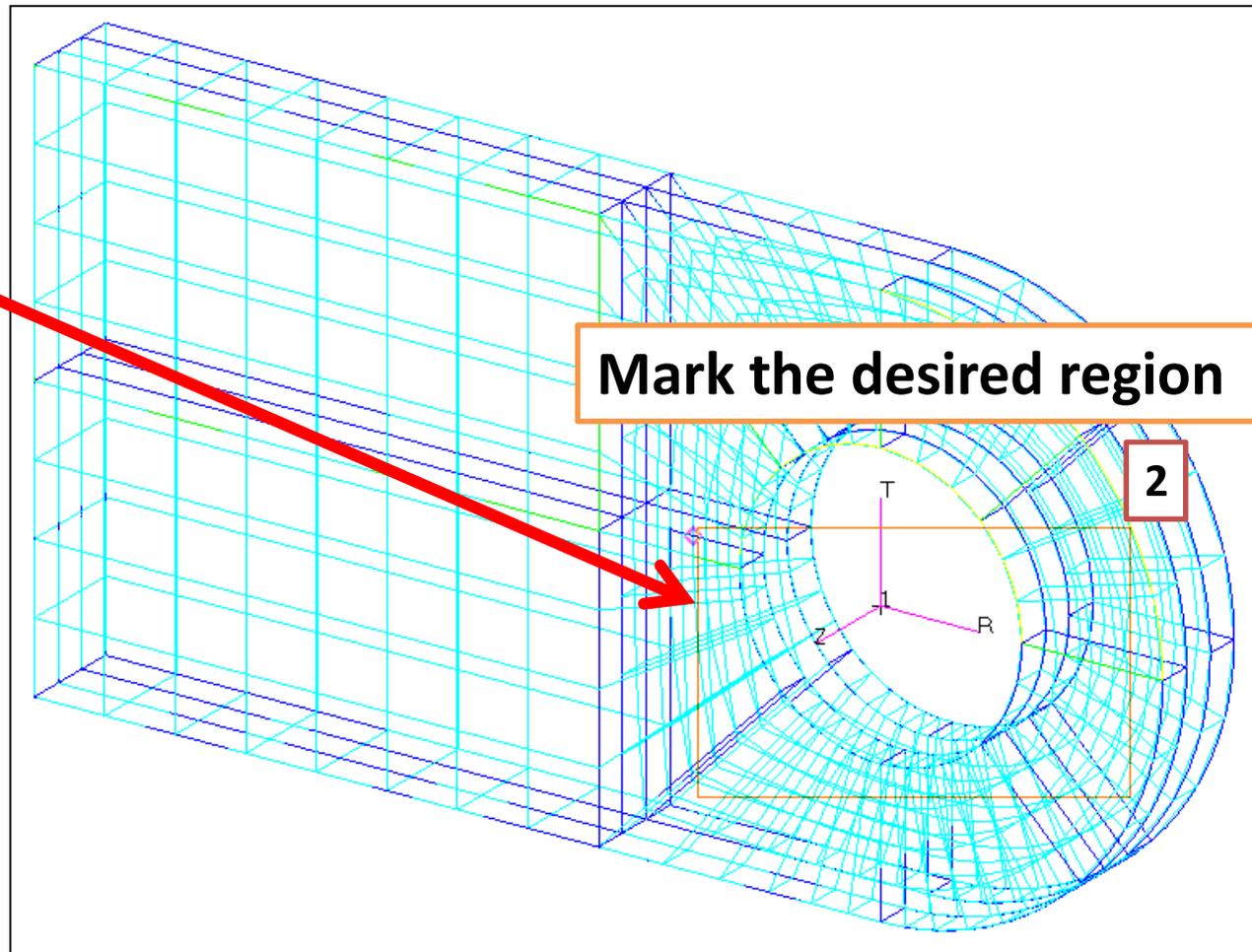
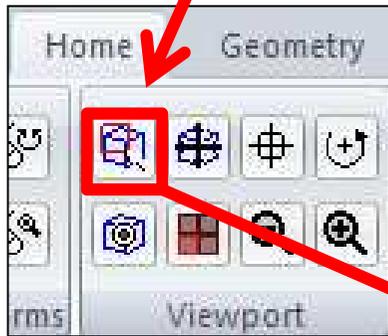
Write Report

Report Setup...

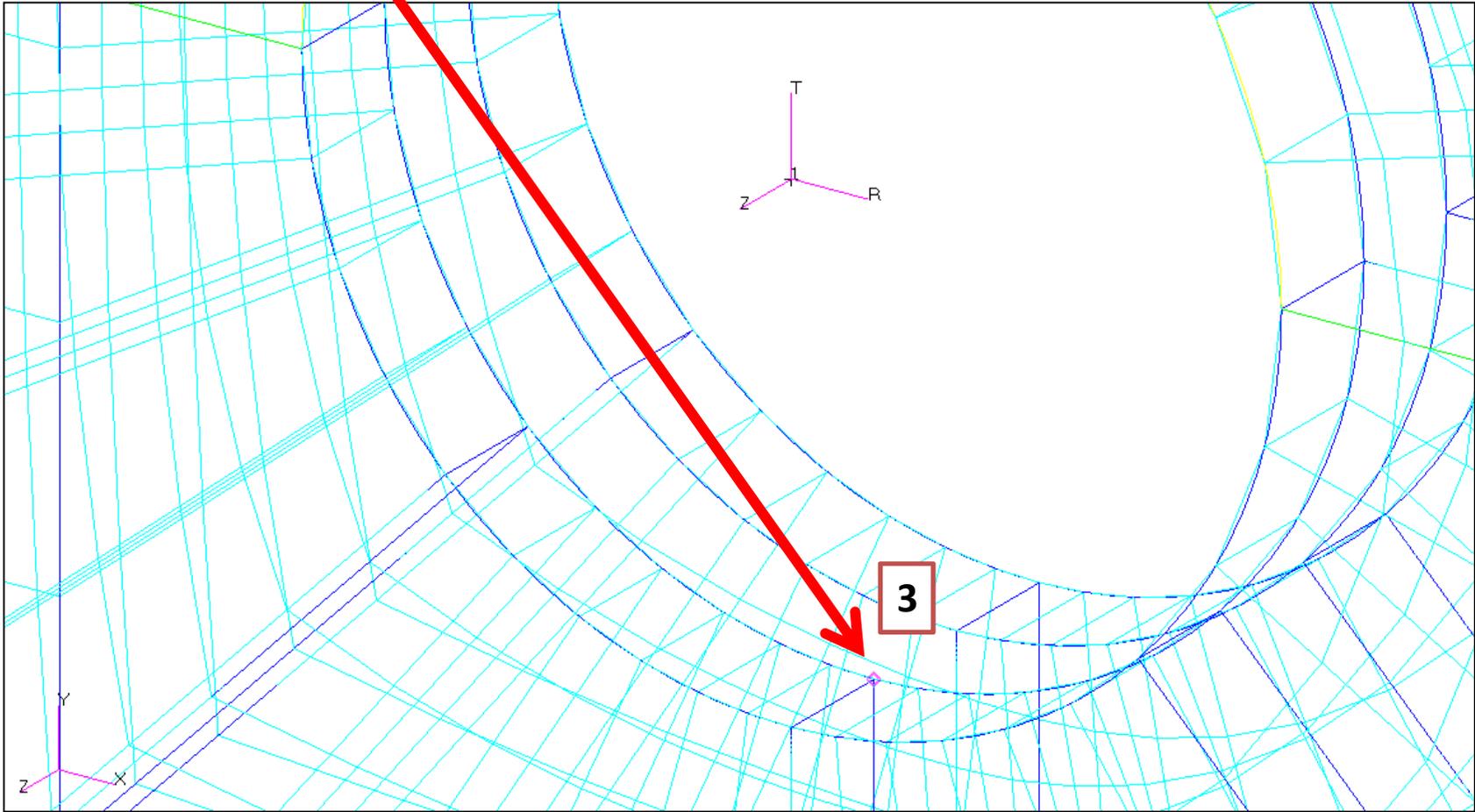
Reset Cancel

 **this window will appear**

# 1 View Corners

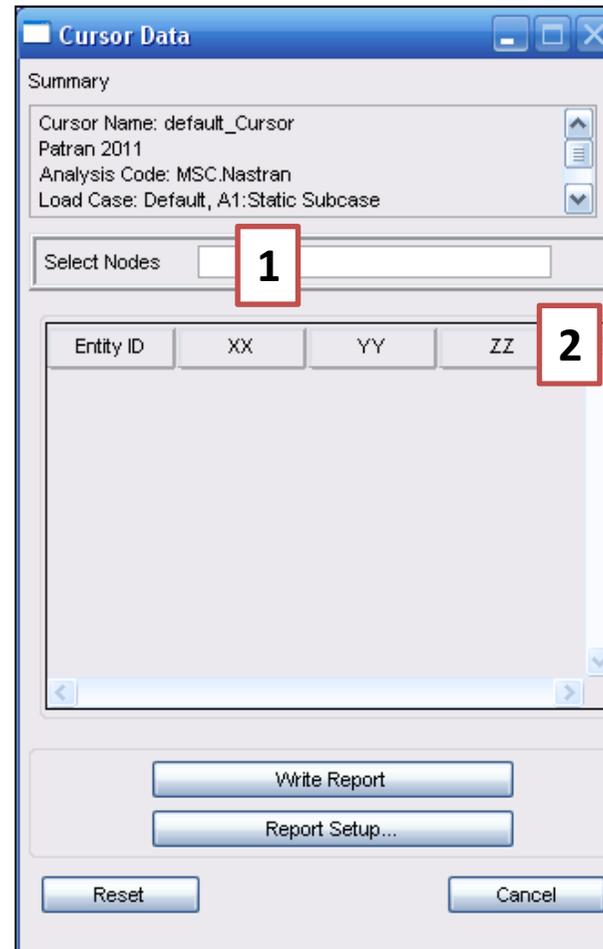


Select the desired node



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

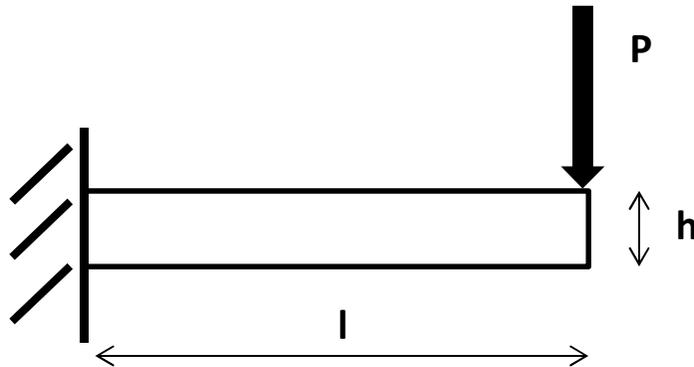


# BEAM

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]

$I_y = \dots$  [m<sup>4</sup>]

$P = \dots$  [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  $\sigma_{equiv}$
  - Stress in X direction  $\sigma_x$  with averaging, continuous  $\sigma_x$
  - Stress in X direction  $\sigma_x$  without averaging, discontinuous  $\sigma_x$
  - Stress in X direction  $\sigma_x$  with averaging, continuous  $\sigma_x$  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**