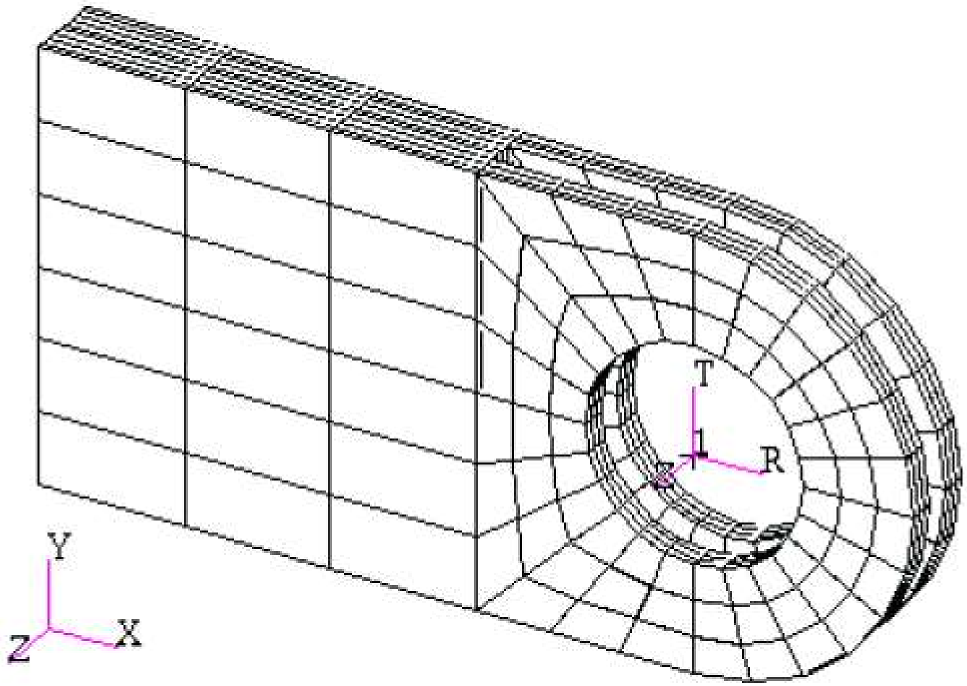
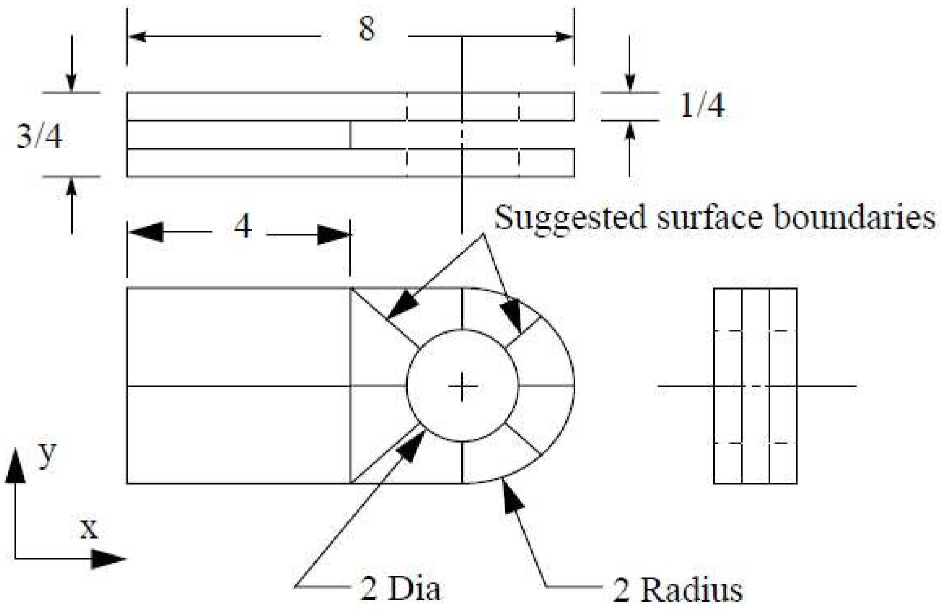
Geometry model of a 3-D Clevis

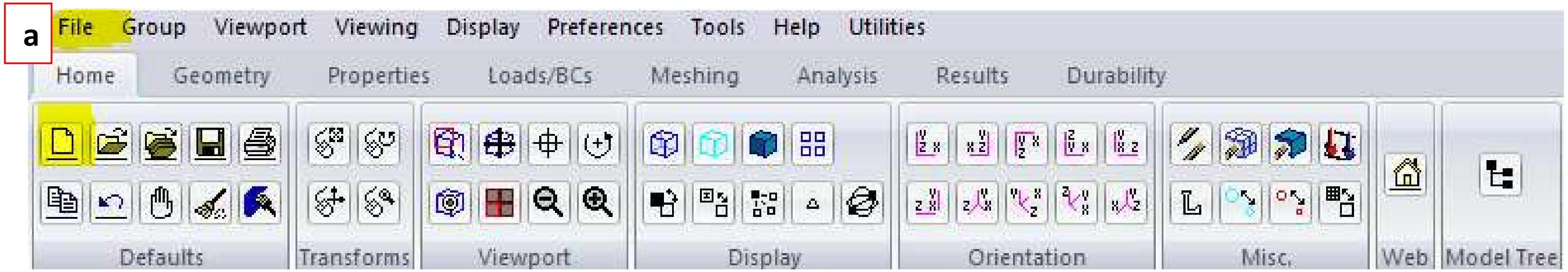


In this exercise you will create an analytic solid model of a clevis by defining MSC⁄PATRAN points, curves, surfaces, solids, and a user define coordinate system. Throughout this exercise you will become more familiar with the use of the MSC⁄PATRAN select menu. Shown below is a drawing of the model you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



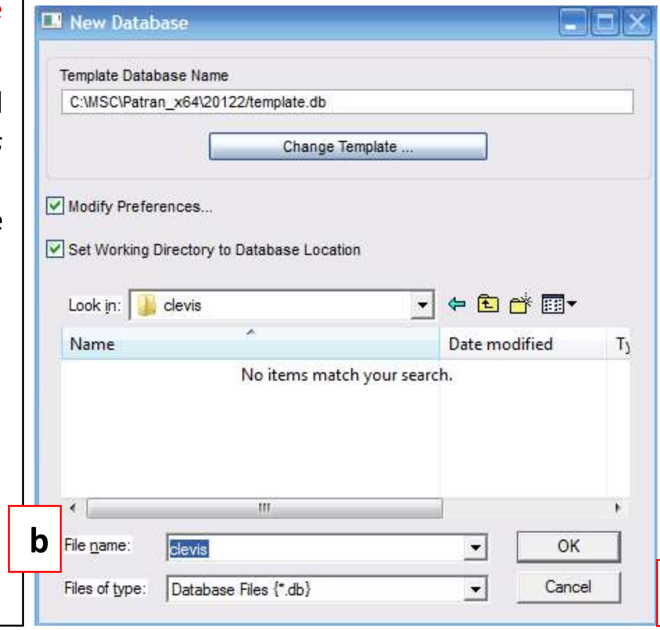


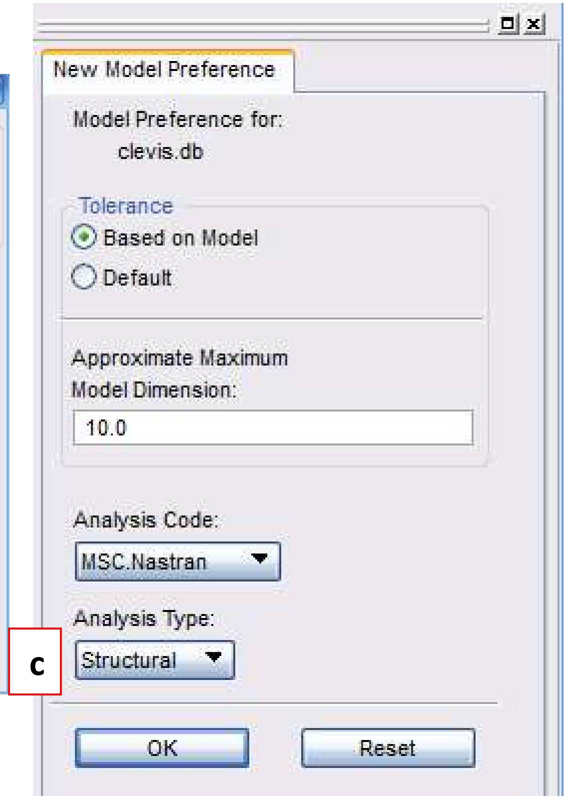
In order to create a new database
You have to do as follows:

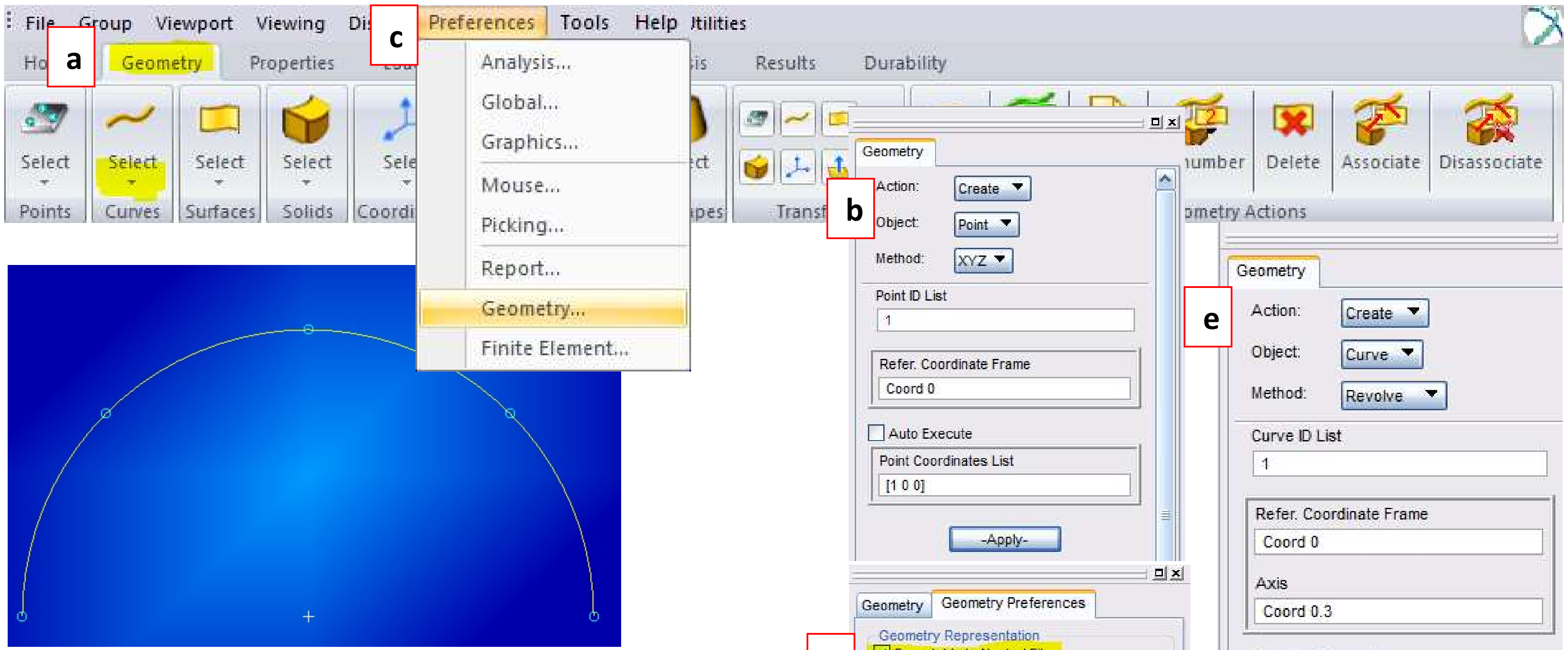
a. **File / New...** or on a symbol **New** in *Home/Defaults* section
b. Enter **clevis** as the File name add Click **OK**

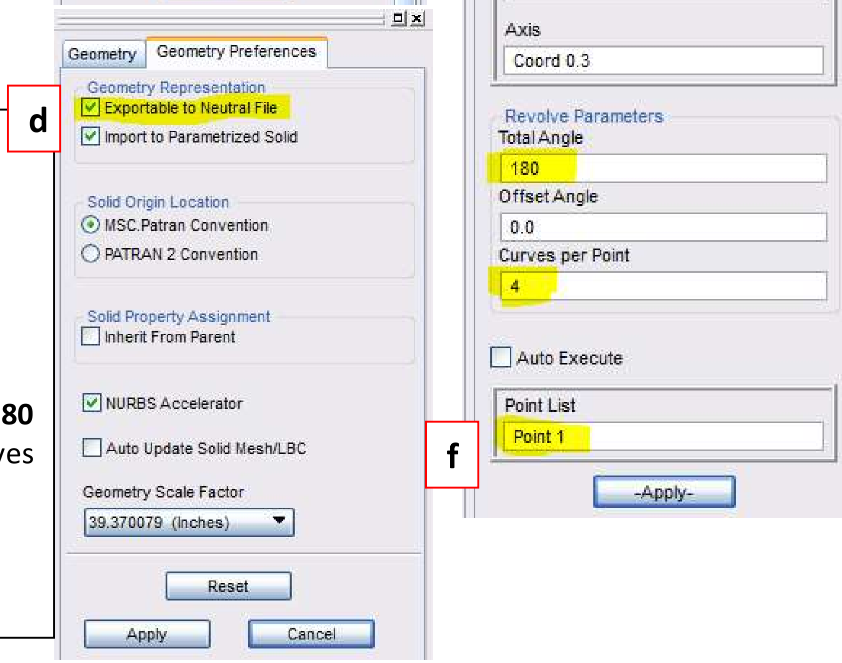
/ New Model Preference

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-**

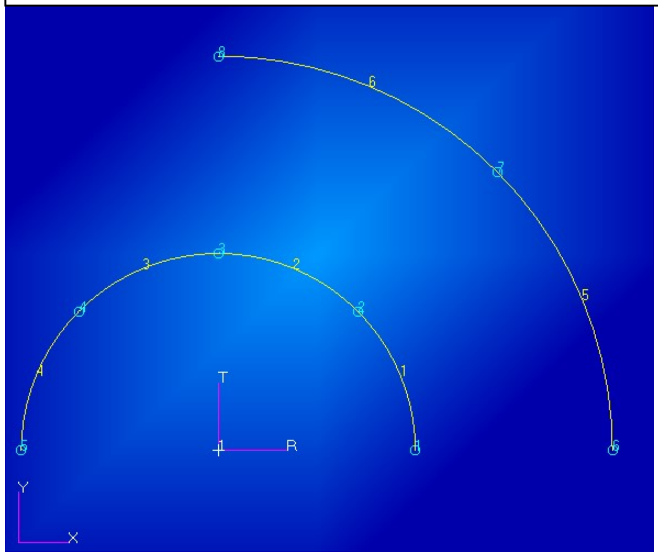
The final effect should looks like on the picture above.

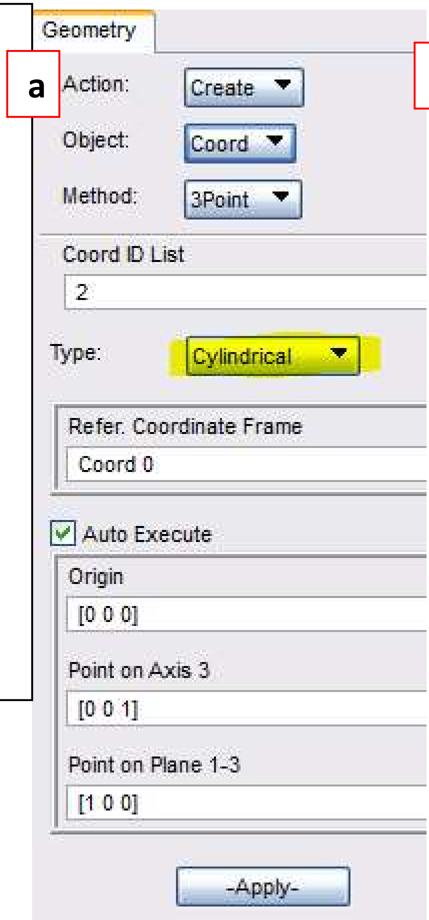
You will now use Curvilinear transformation to create the outer radius of the lug by radially translating the curves that define a quarter of the hole.

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

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





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 **<-4 2 0>** 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:

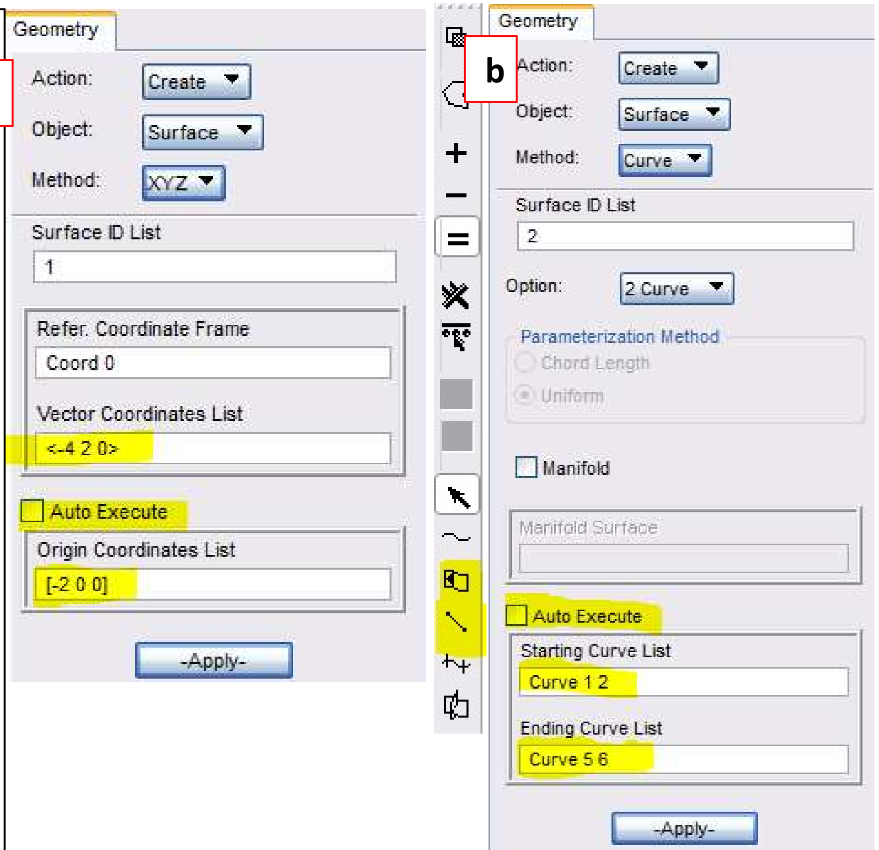
b. ***Create->Surface->Cur***   , ***uncheck*** *Auto Execute* and select **Curve 1** and **Curve 2** in a *Starting Curve List section* and **Curve 5 6** as a *Ending* and hit **–Apply-**

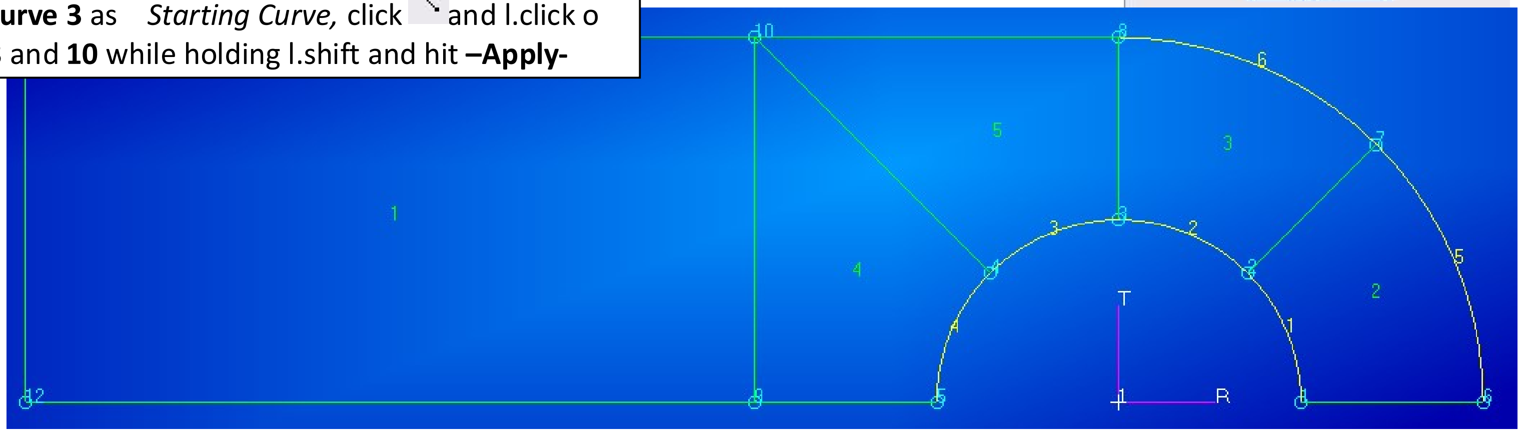
Click on in the *Home/Misc.* section to display the lines.

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 l.click on edge 9-10 of a surface 1 and hi **–Apply-**

d. Chose **Curve 3** as *Starting Curve,* click and l.click o **Points 8** and **10** while holding l.shift and hit **–Apply-**

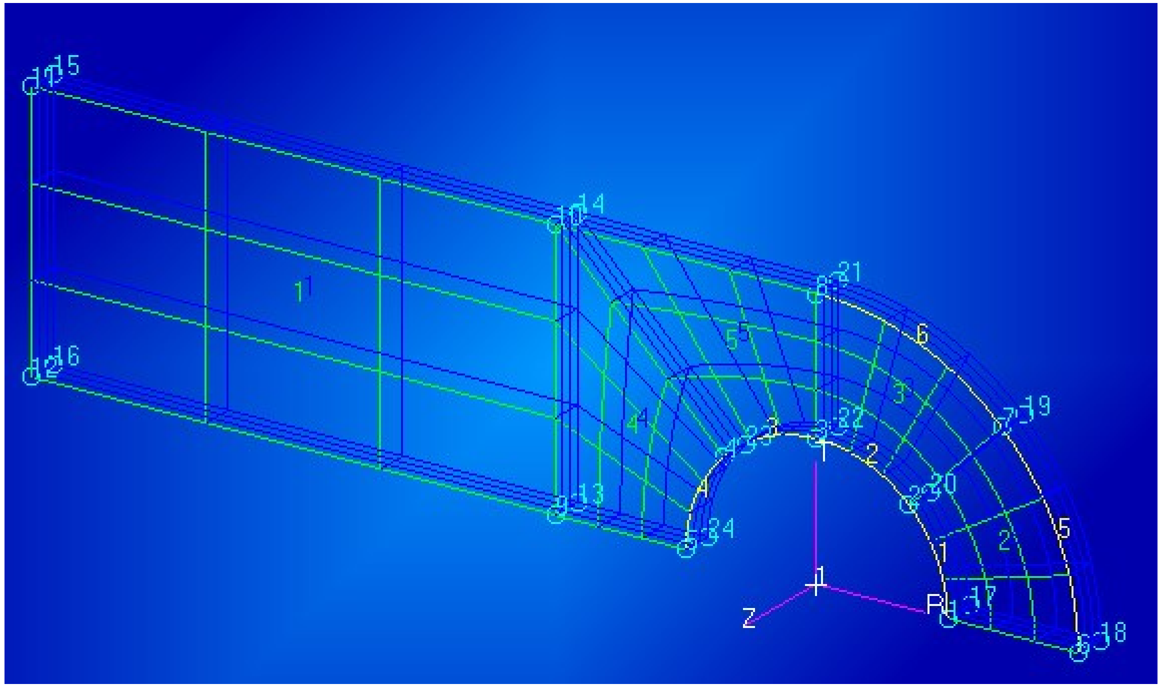




You will now use the Surfaces you have just created as patterns to define solids (3-dimensional entities)

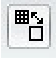
a. **Create->Solid- Normal ,** insert **0.25** as a Thickness, **Uncheck** Auto execute
b. Chose all surfaces and click **–Apply-**

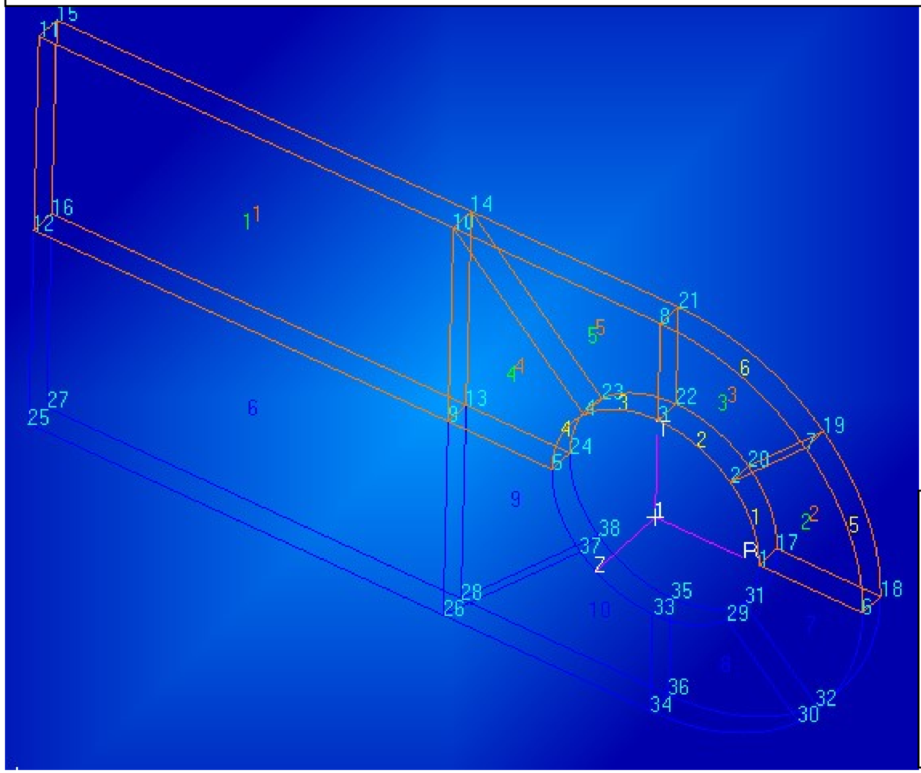
Change the view to **Iso 1** in *Home/Orientation* section and **Fit view** 

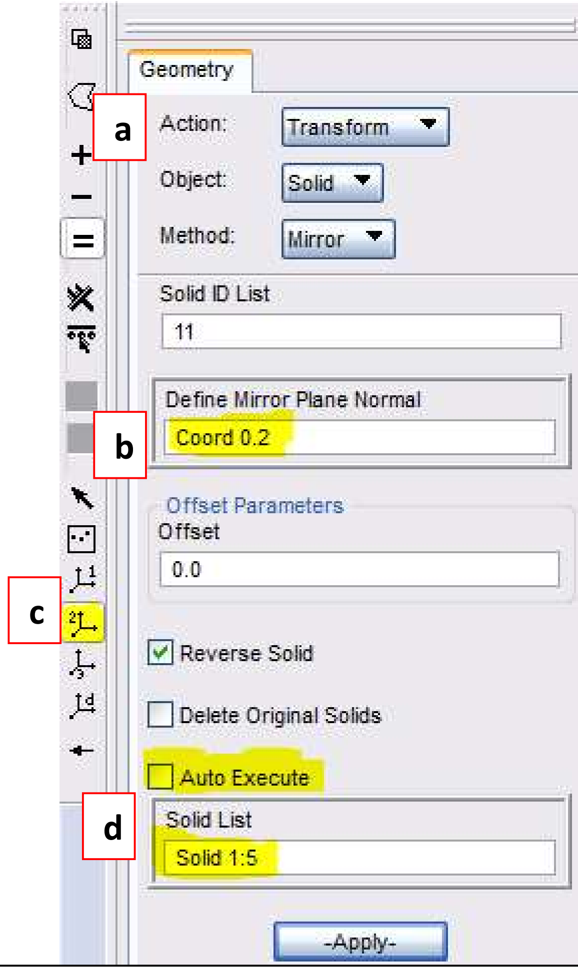


Creating the lower part of the clevis

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

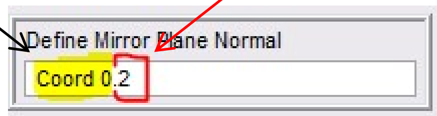
Click Display Line in Home/Misc. section 





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

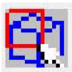
TIP



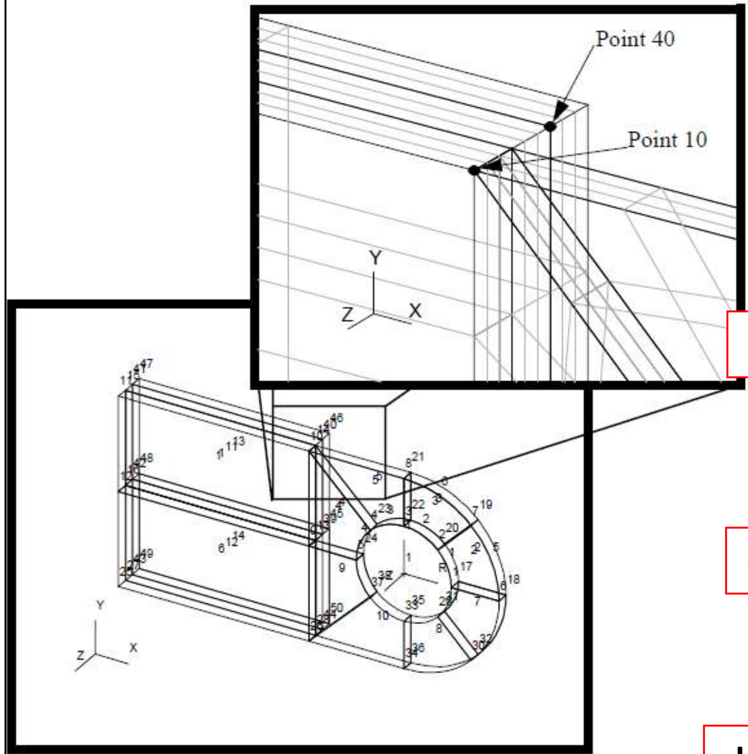
The remaining solids will be created using the translate method.

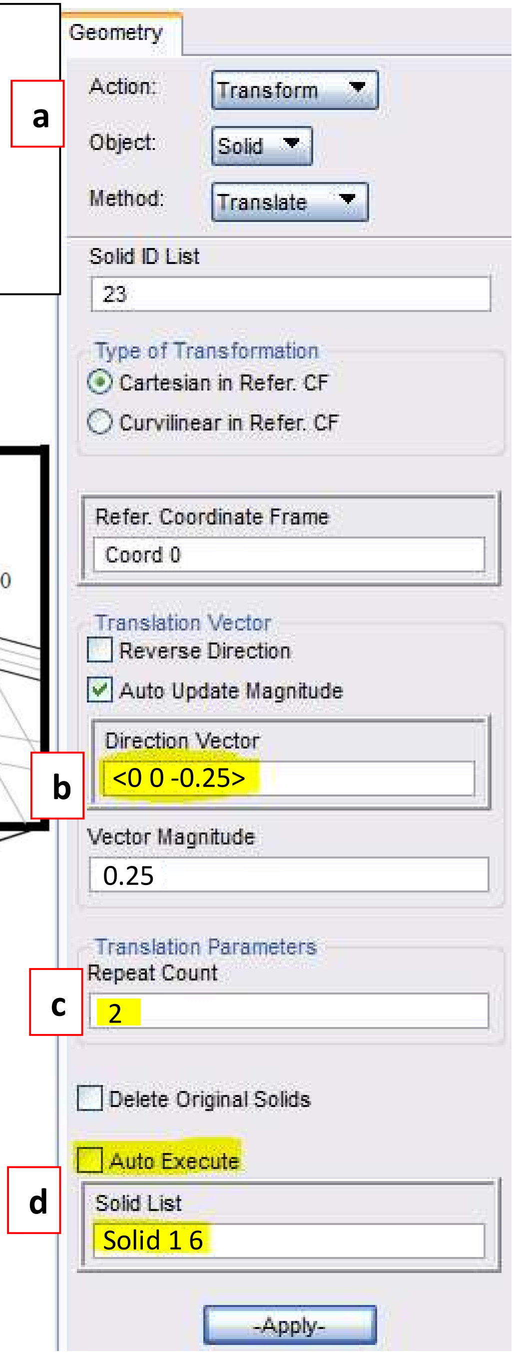
a. **Transform->Solid->Translate**
b. Enter **<0 0 -0.25** in a *Direction Vector* section
c. Repeat Count: **2**
d. Uncheck **Auto Execute**, choose **Solid 1** and **6** and press **–Appl**

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*

d. 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.
e. Repeat Count: **1**
f. Select **Solids** 2 to 5 and 7 to 10 **Solid 2:5 7:10** and hit **–Appl**

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

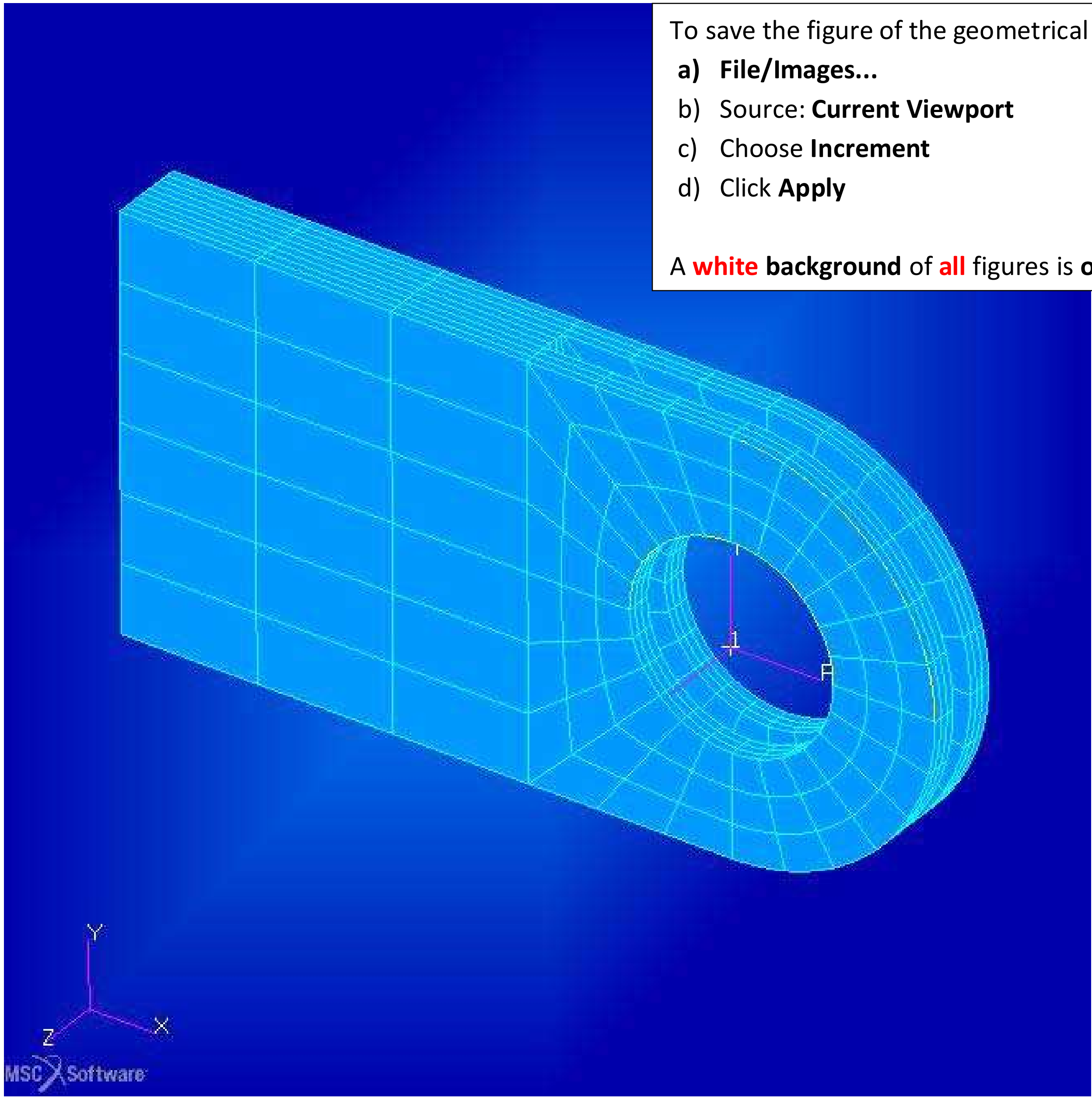




To save the figure of the geometrical mode

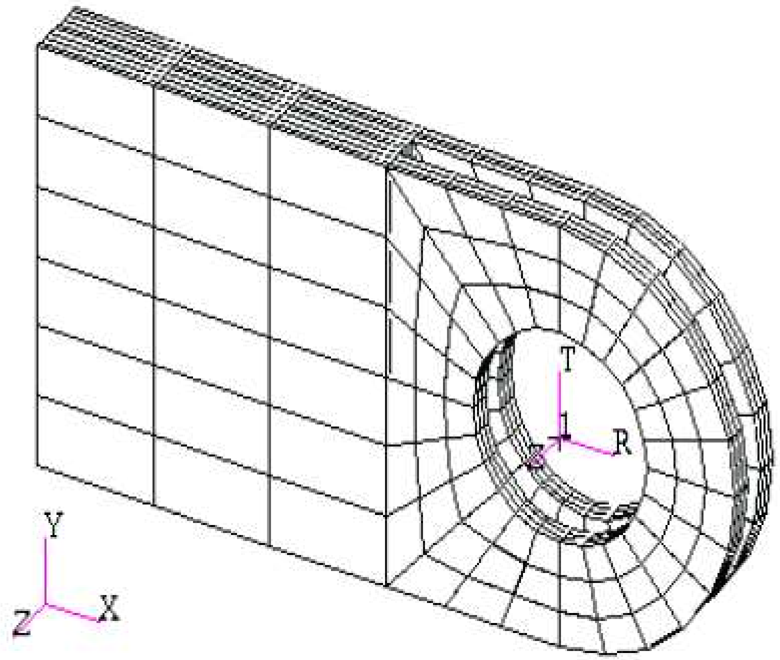
a) **File/Images...**
b) Source: **Current Viewport**
c) Choose **Increment**
d) Click **Apply**

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



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

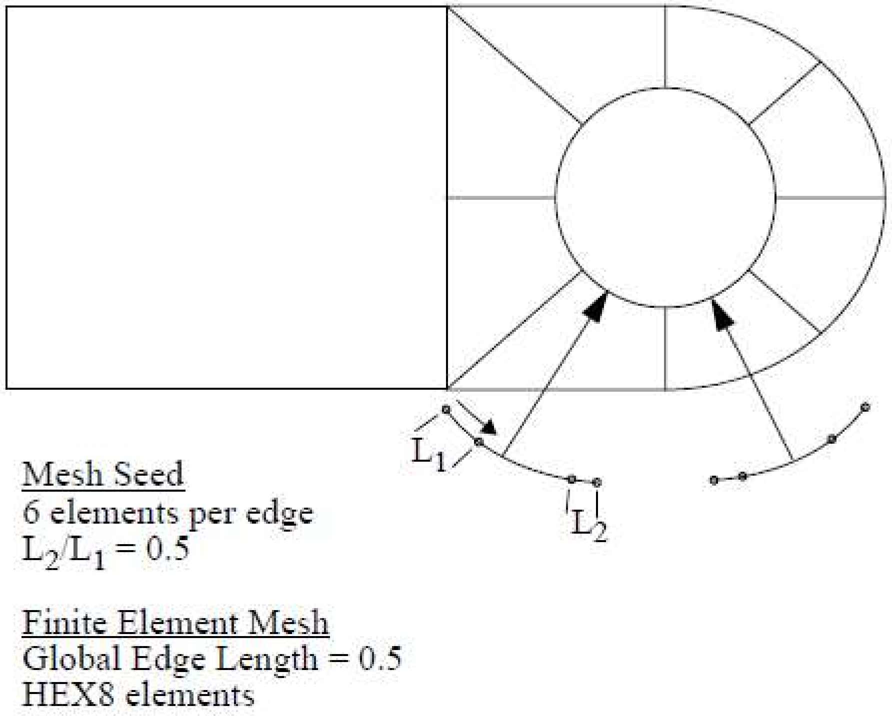
- Apply a non-uniform mesh seed near a critical location of the model.
- Apply a global mesh to the seeded model.
- Apply material and element properties.



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.

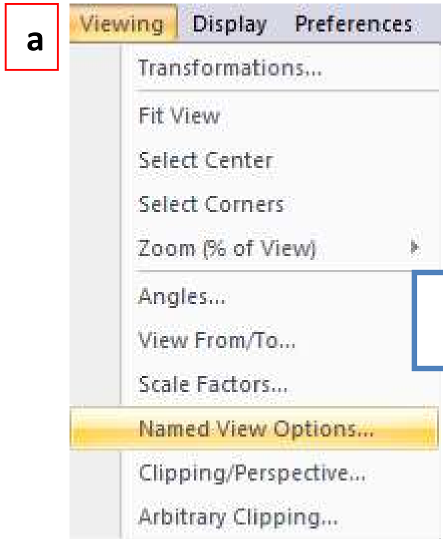
5. Create a 3-D element property named, Solid_Elements_Steel, for the entire includes the steel material definition

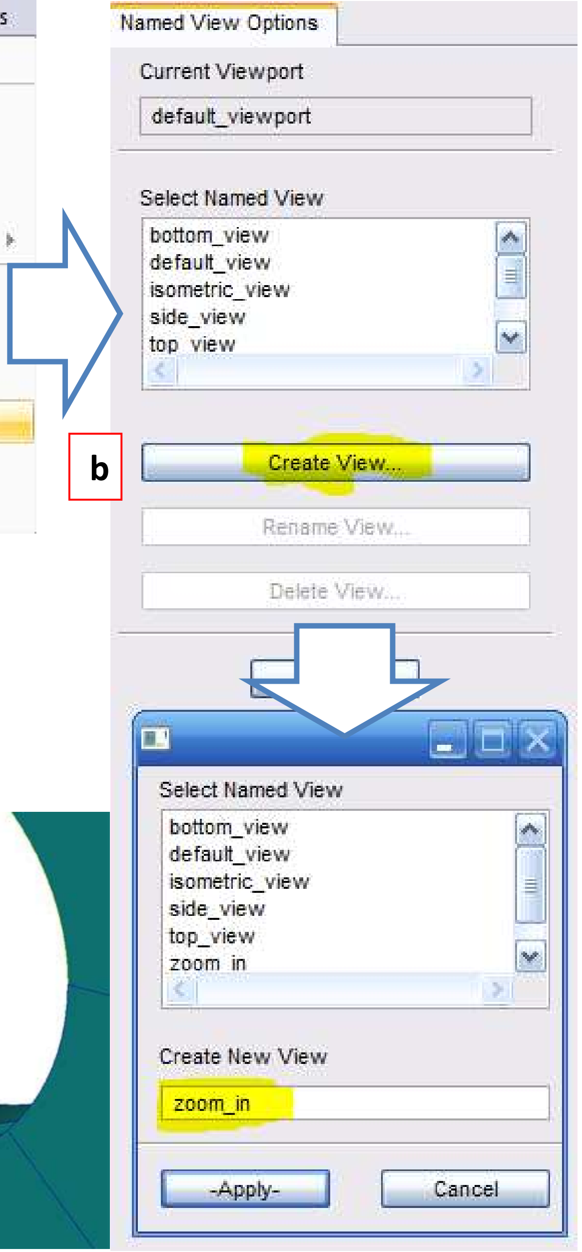
1. New View

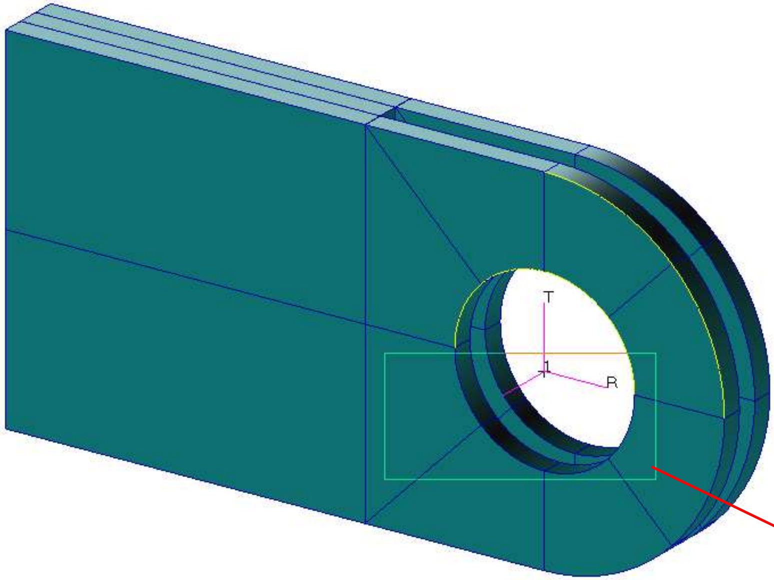
Assuming that you have already opened datae **Clevis.db** , use the *Viewing/ Select Corners* option to zoom a model in a specific area.

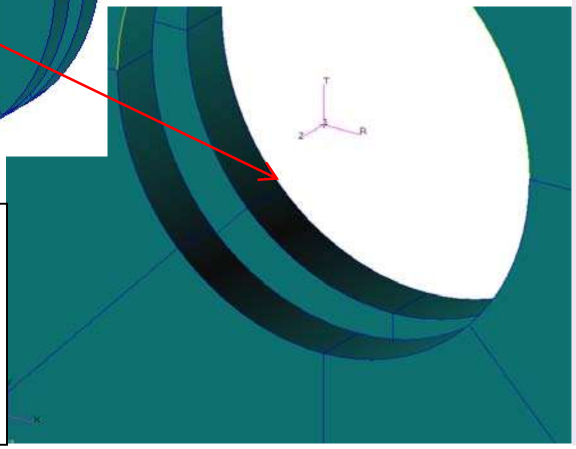
a. Go to *Viewing/Named View Option*

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.

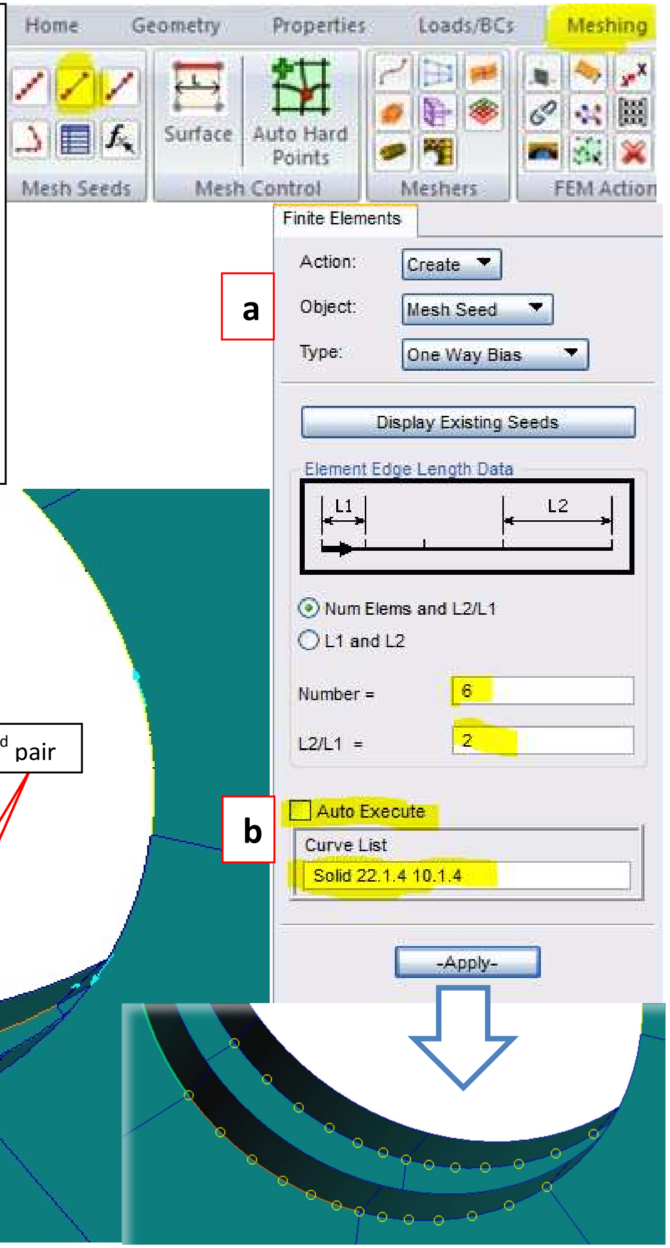
2. Creating mesh seeds

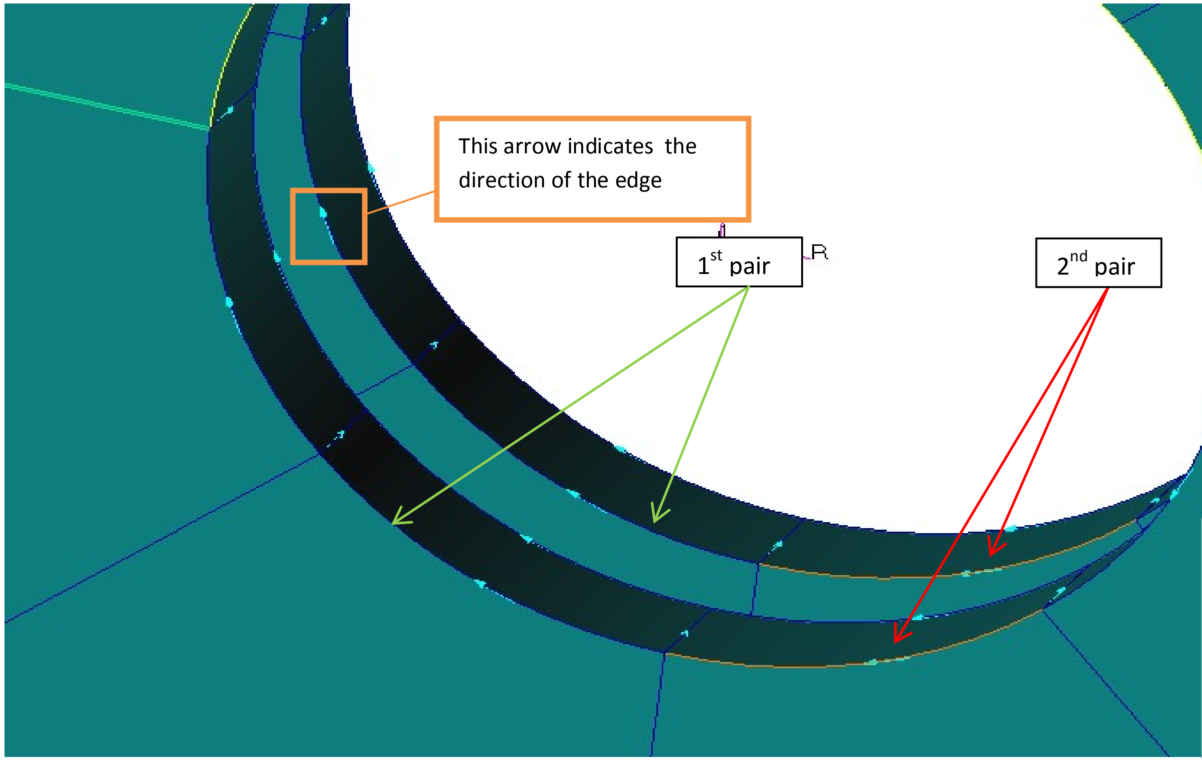
a. Click on *Meshing* and as fallows : **Create->Mesh seed->One Way Bias** or just click on

b. 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 th **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 edg** and press *–Apply-*

Tip In order to select edge click on in select menu



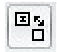


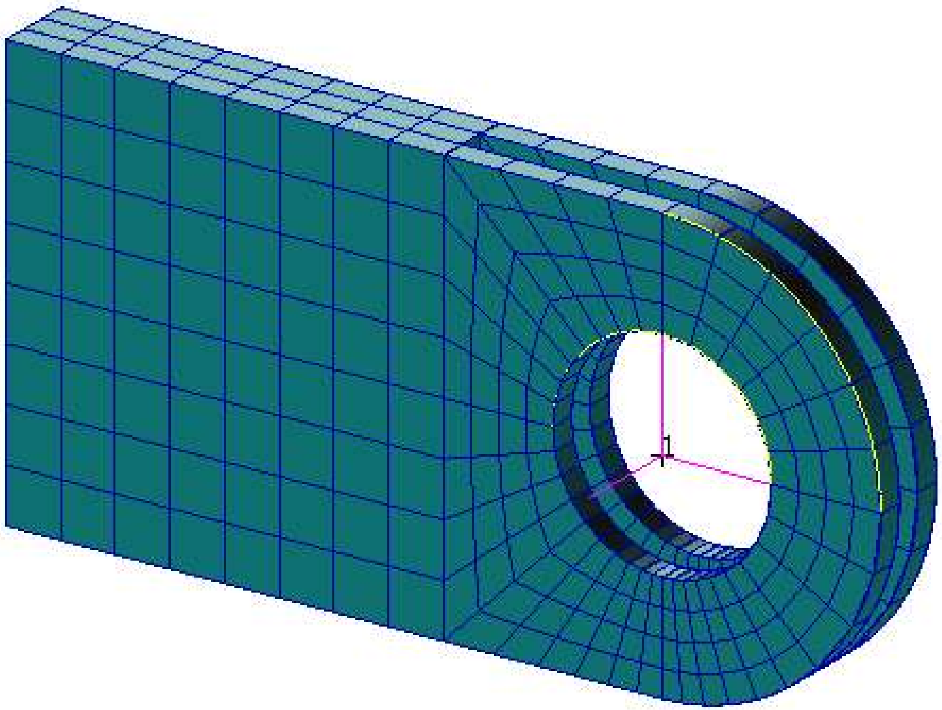
3. Creating Mesh

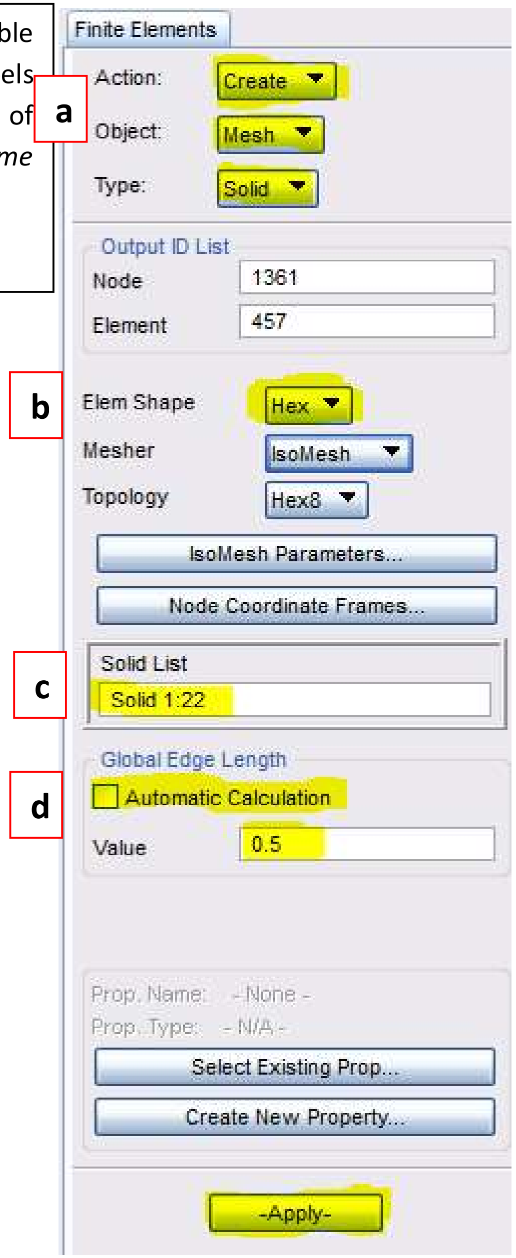
a. **Create->Mesh->Solid** or simply click on
b. Change element shape to **Hex**
c. Select all solid parts
d. Uncheck *Automatic calculation* and insert Value **0.5** and hit *–Apply*

Tip

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







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

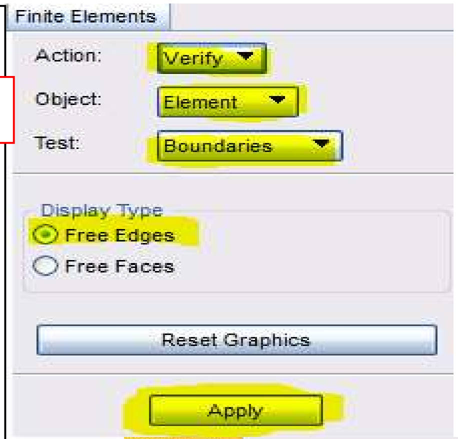
a. **Verify->Element->Bounda** check the *Free* Edges and hit *–Apply-*

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

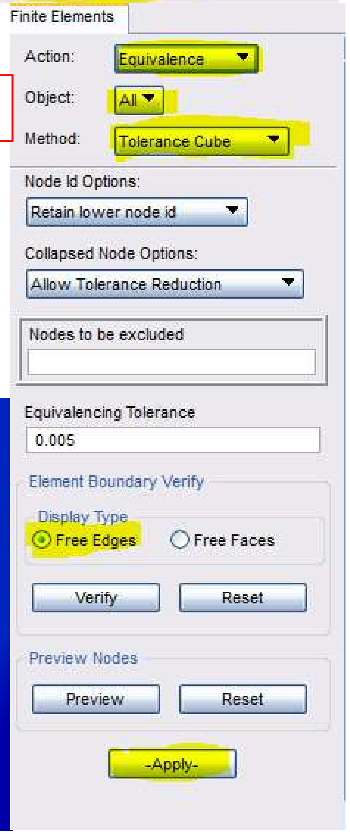
b. **Equivalence->All->Tolerance Cube ,** check the **Free Edges** and *–Appl*

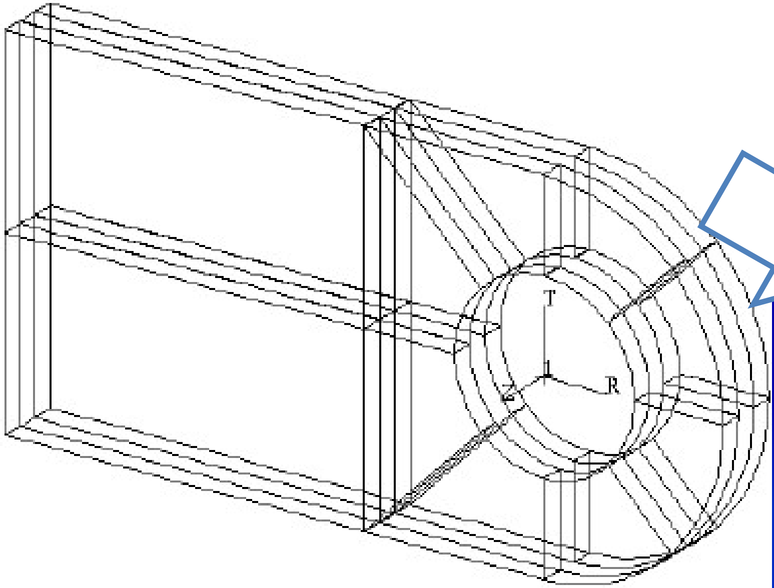
You now have one contiguous model of finite elements. To check whether this is tru : Repeat the step described in point **a.** (Your model should looks like the one below at blue background)

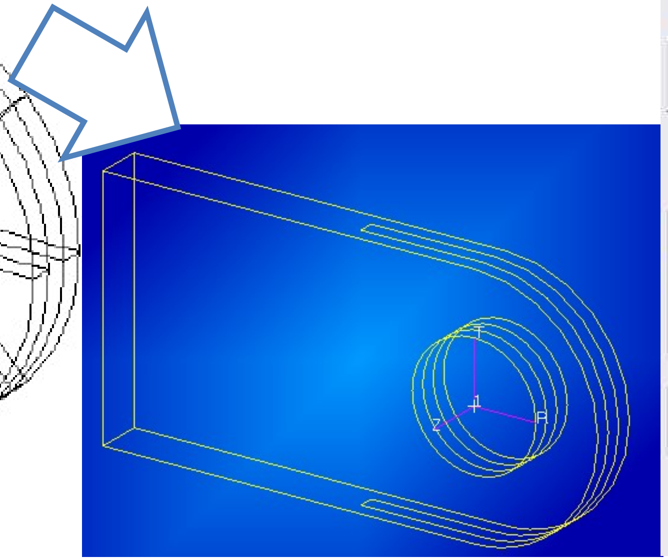
a



b

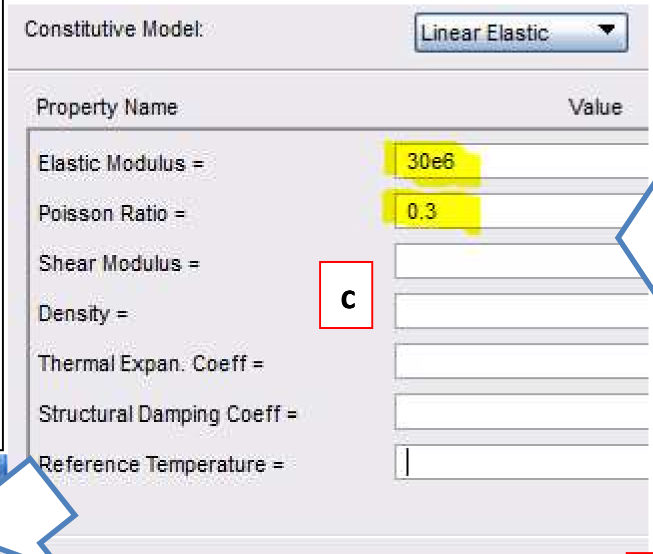


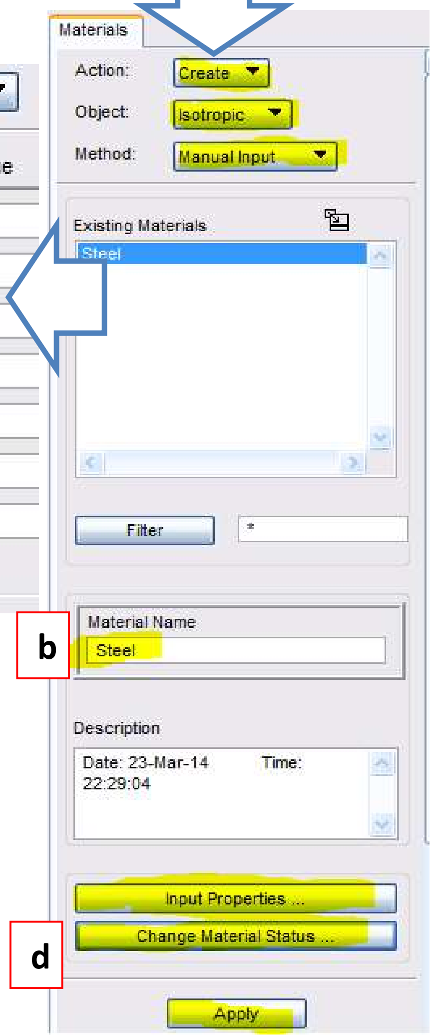


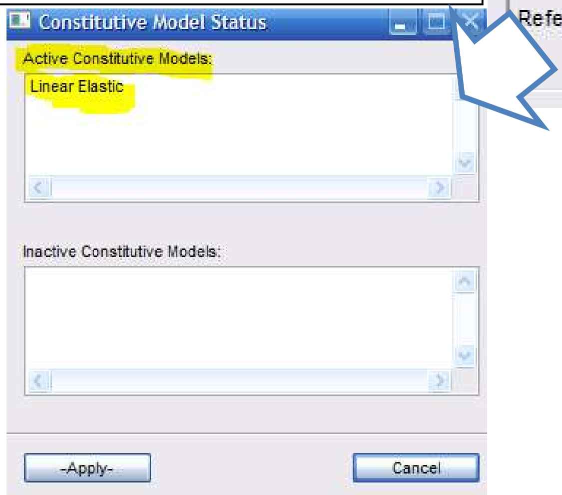


4. Create an Isotropic material, named **Steel**, which uses a Linear Elastic Constitutive Model.

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

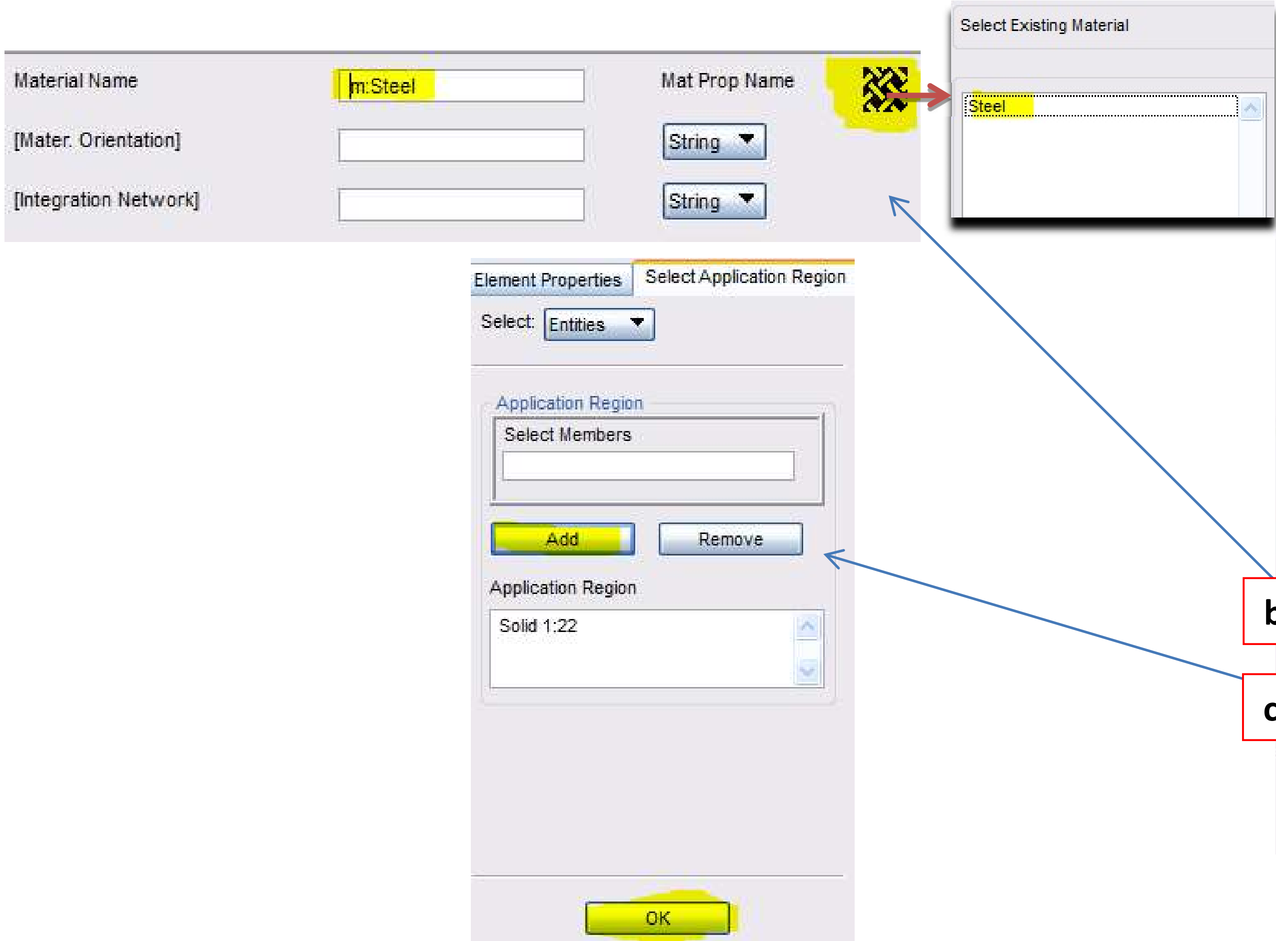


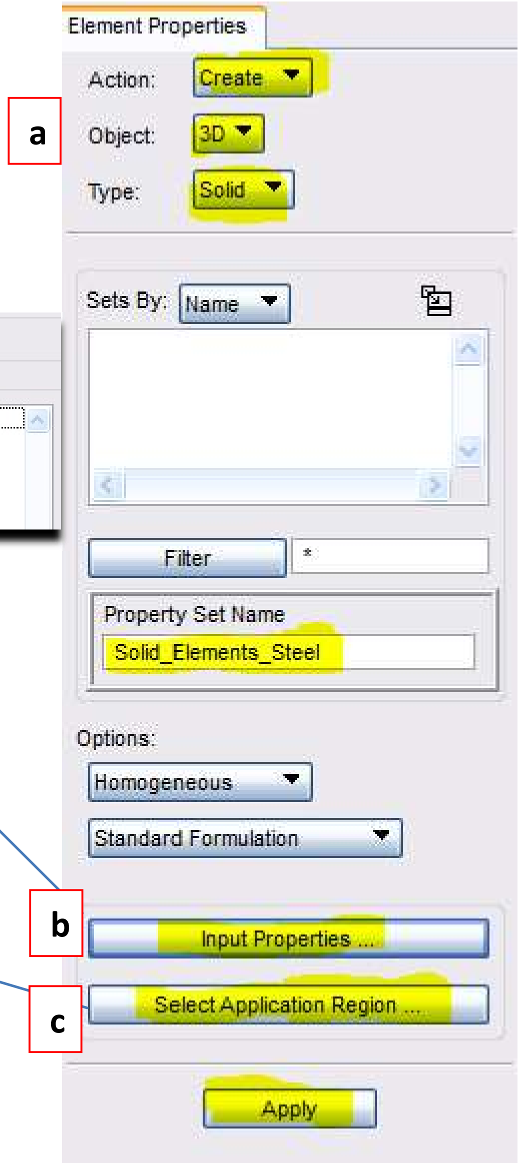




5. Create a D element property named, **Solid_Elements_Steel,** or the entire model which includes the steel material definitio

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*





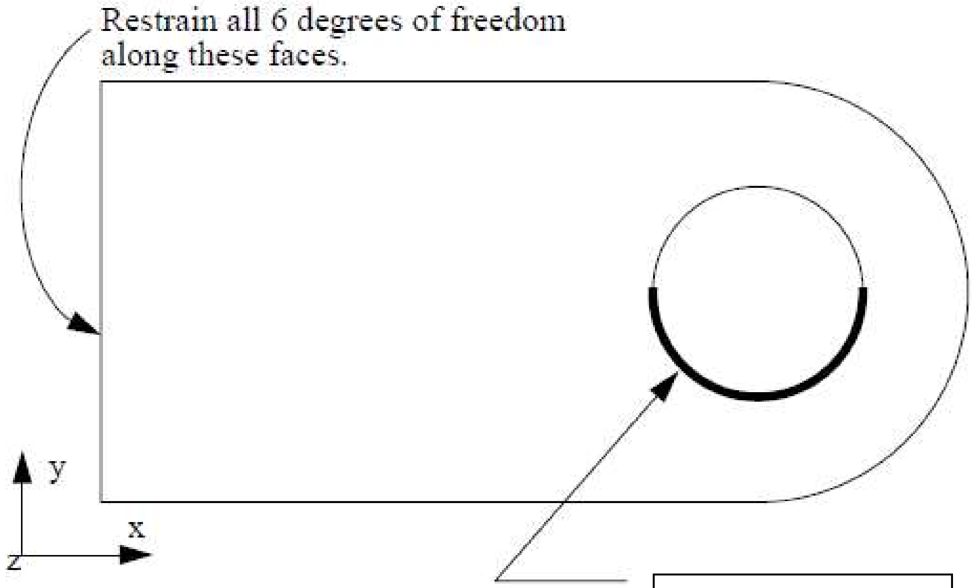
Loads and Boundary Conditions on a 3-D Clevis

Objectives:

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

Suggested Ste

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



Area under pressure

P=6900*(1-x^2)

Model Description:

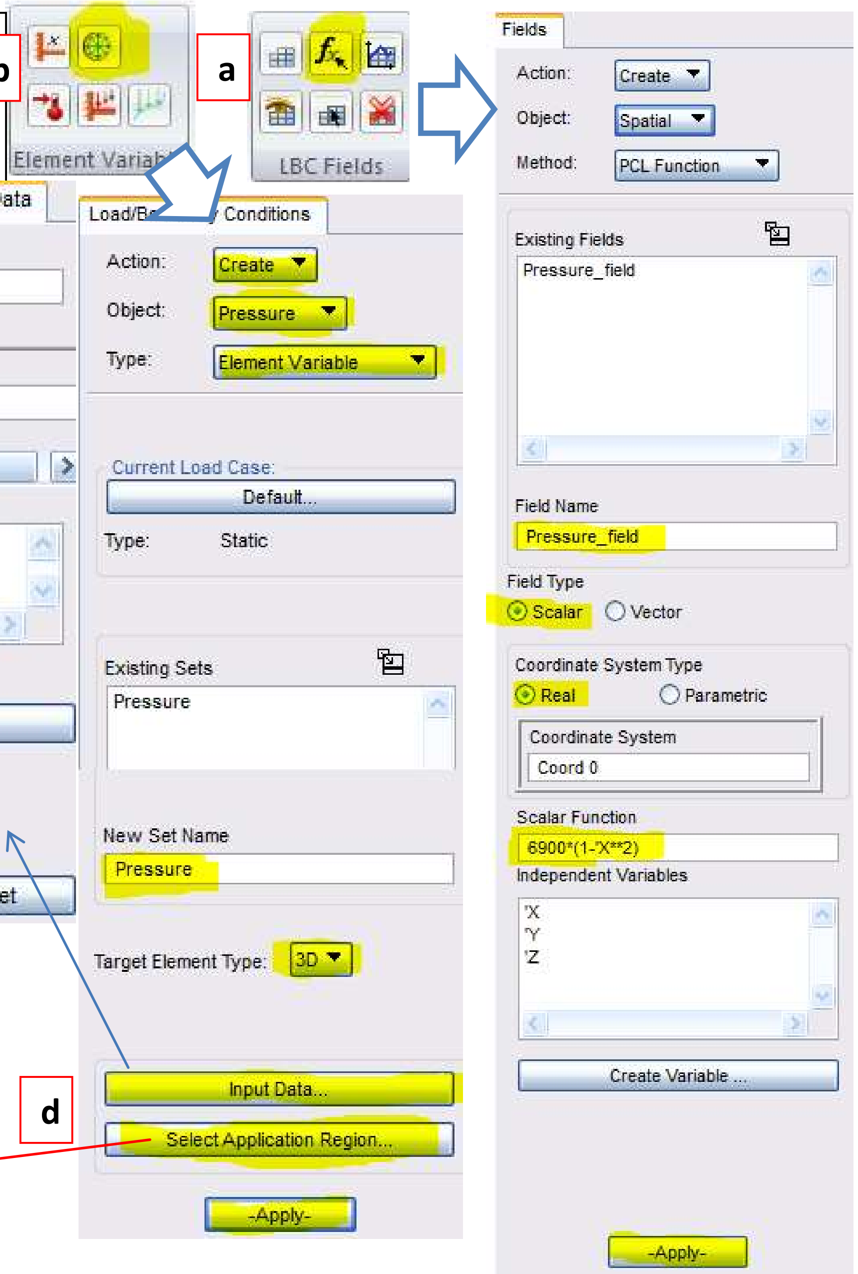
In this exercise you will create a loading condition and a constraint set for the clevis model. The base of the lug will be clamped. The hole will be under quadratically varying pressure P = 6900 *(1-x^2).

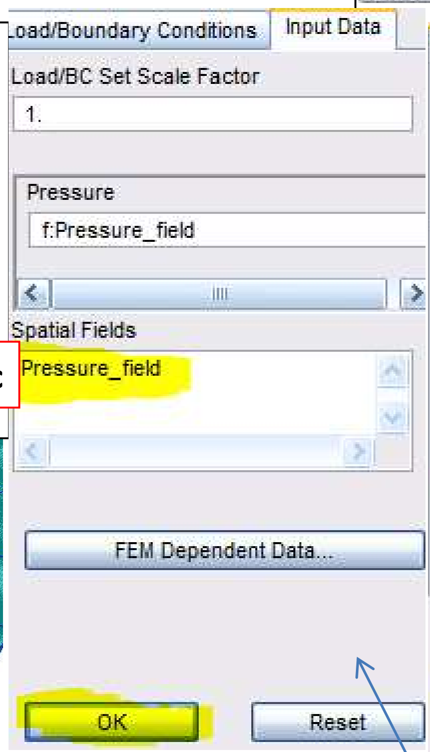
1. Creating a pressure case (ATTENTION: 'X, below, is **NOT** TYPED, it is chosen from the list of Independent Variables!!):
   a. *Load/BCs // **LBC FIELDS->Create spatial function** insert **Pressure_field** as name check **Scalar** and **Real** and insert 6900*(1-'X**2) as a Scalar Function* and hit *–Apply-*

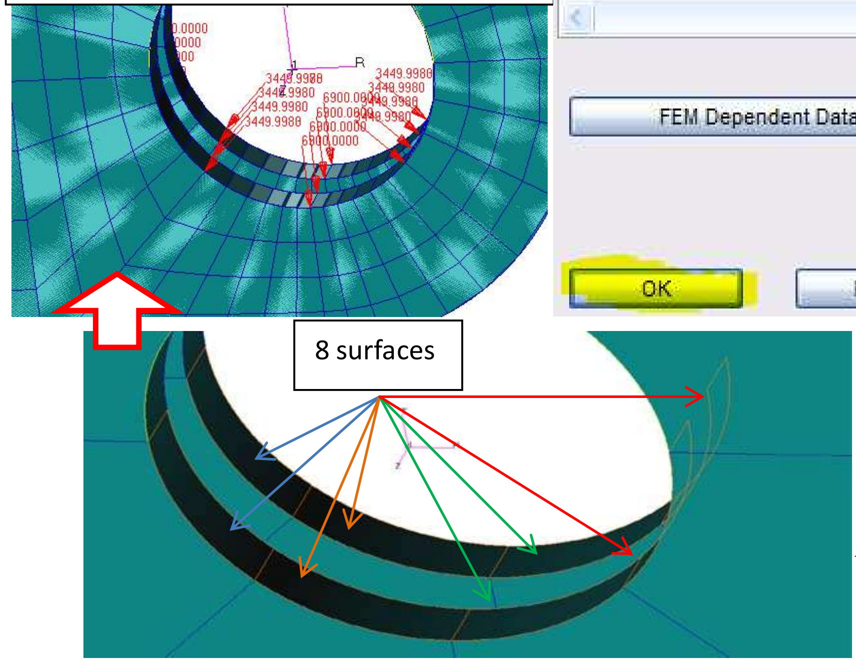
b. *Element Variable/* ***Pressure,*** Enter Pressure as a name, change target to **3D** and click *Input Data*.

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-*

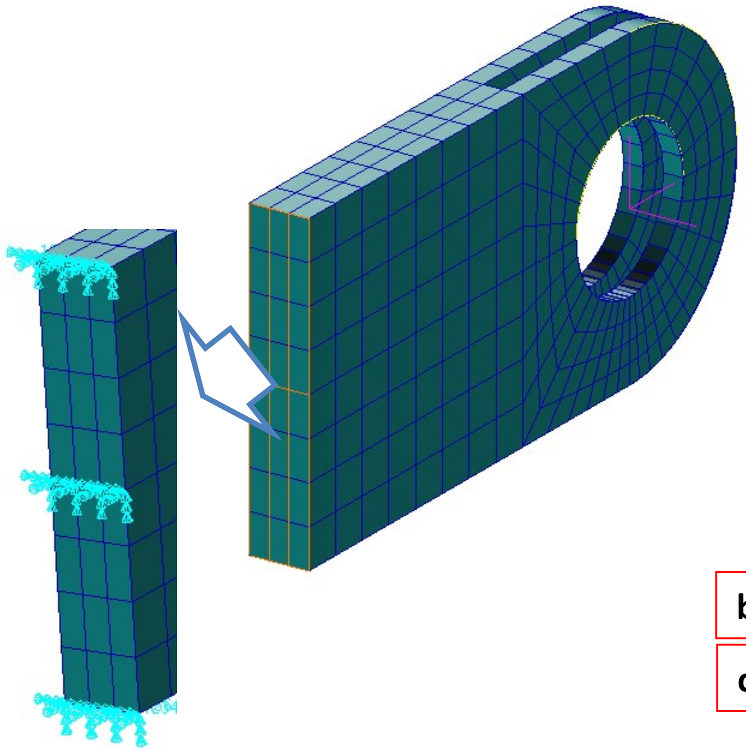


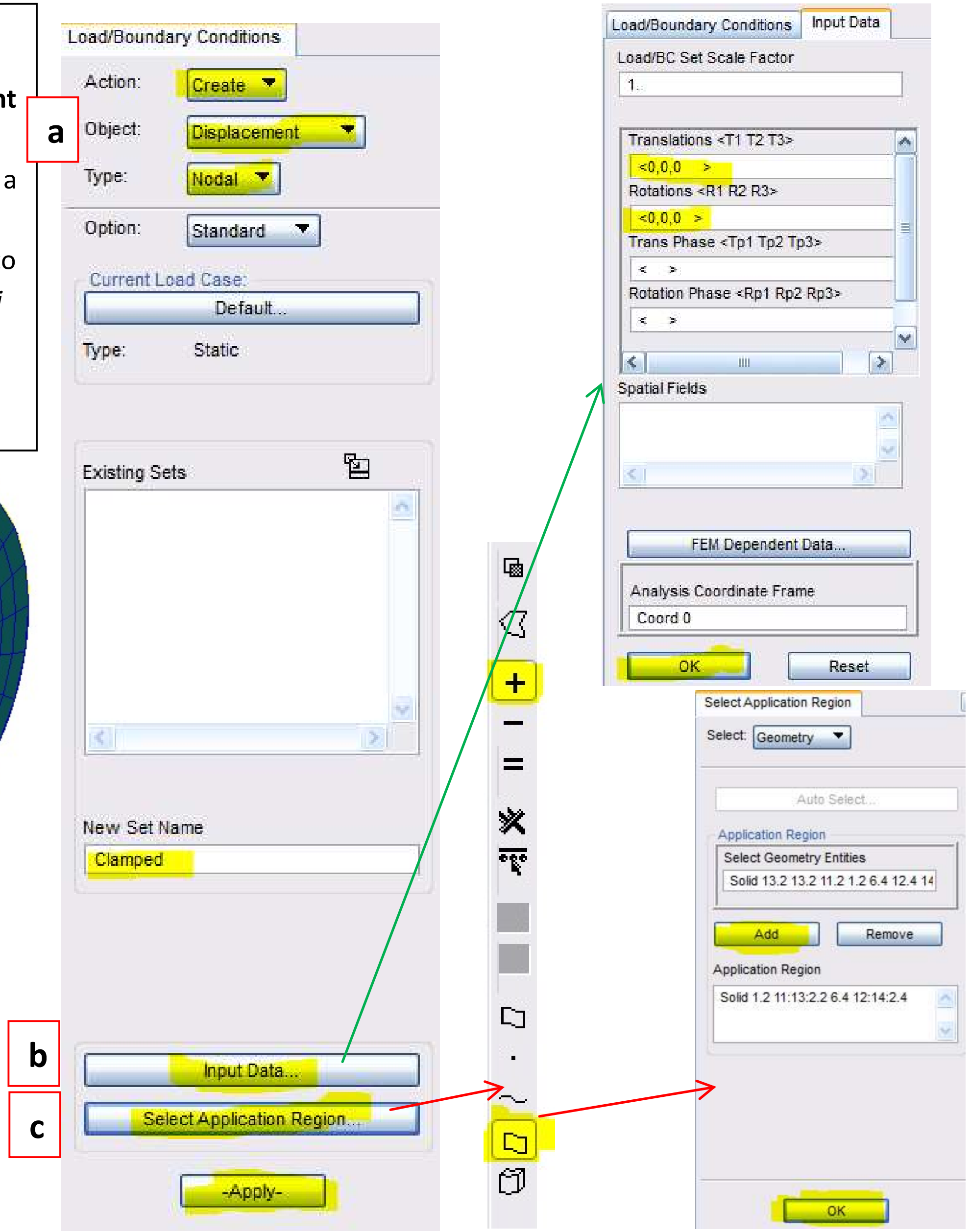




2. Create a Nodal Displacement boundary condition:

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



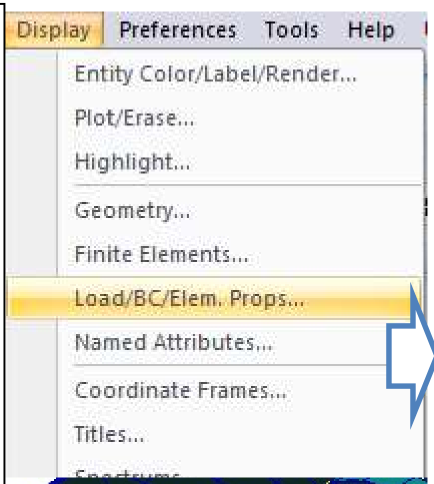


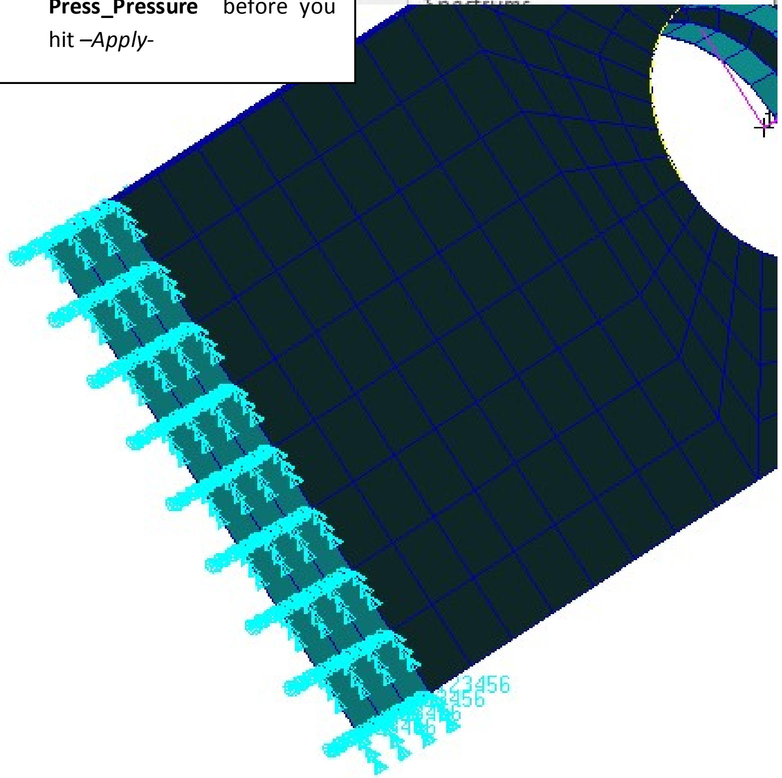
<u>Display both the displacement and force on the finite element model</u>

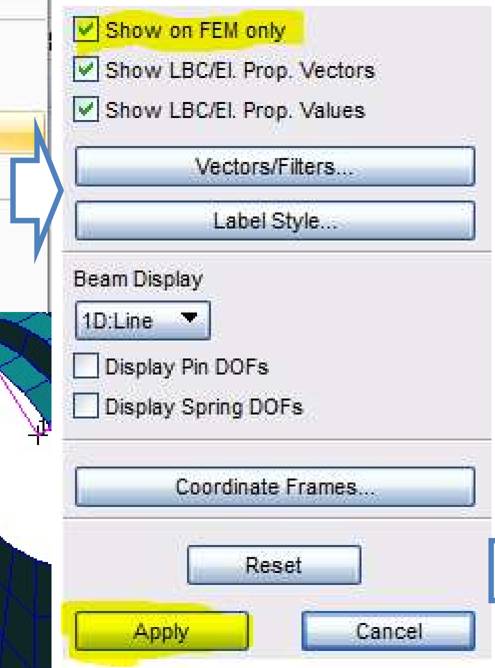
a) *Display/Load/BC/Elem. Props...* and check **Show on FEM only** and hit *–Apply-*

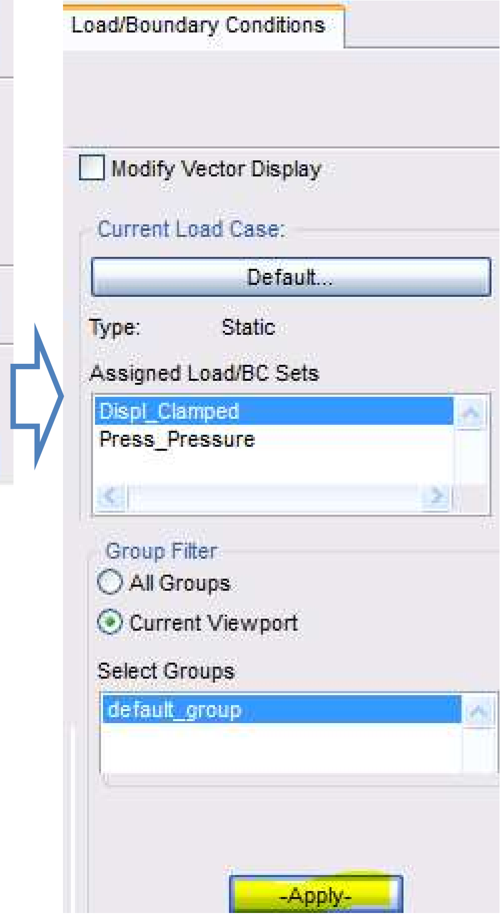
b) After that, in *Loads/BCs tab* choose **Plot markers** as a *Action*

c) Highlight **Displ_Clamped** an *–Appl* . If you want to show on the model also a pressure, highlight al **Press_Pressure** before you hit *–Apply-*





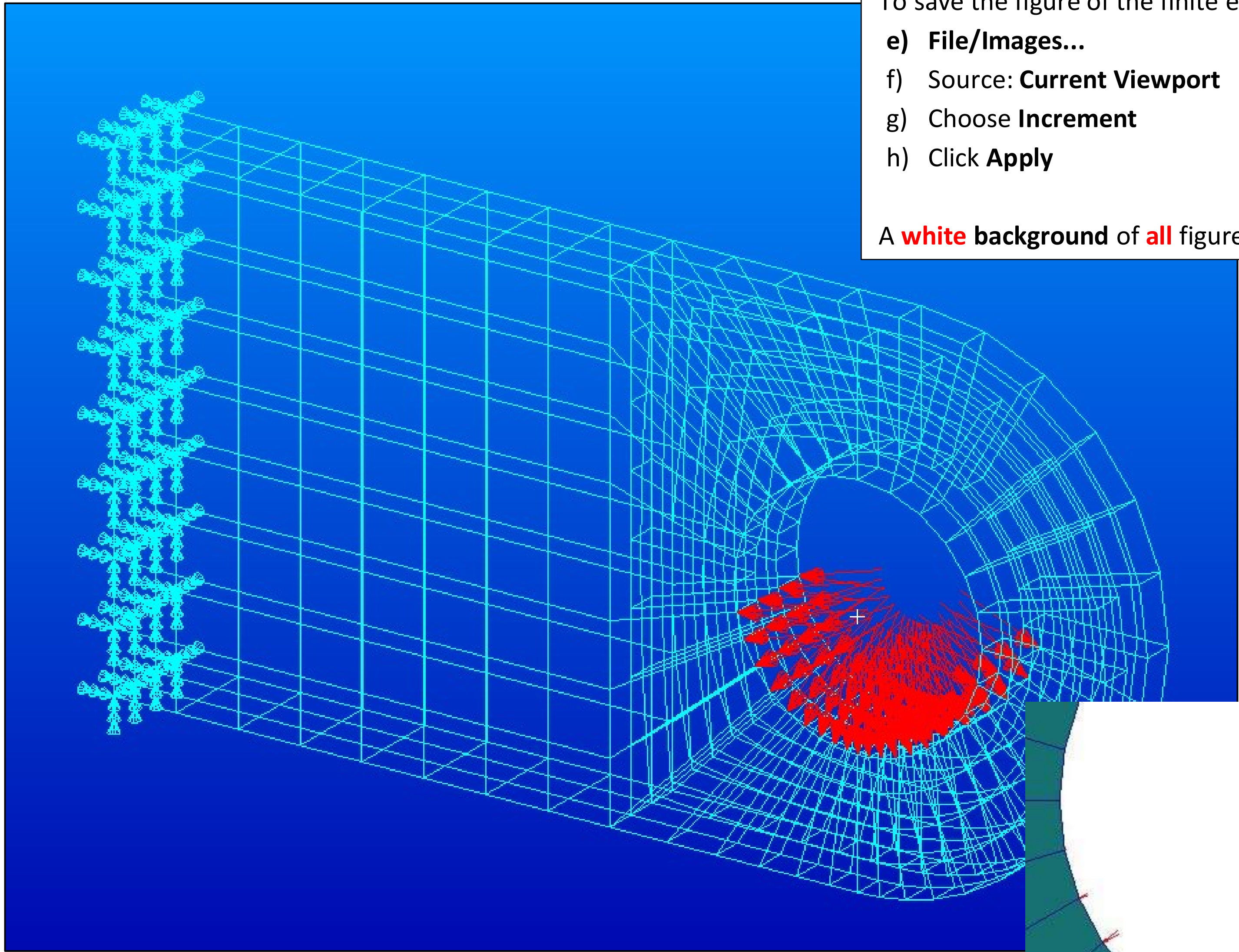


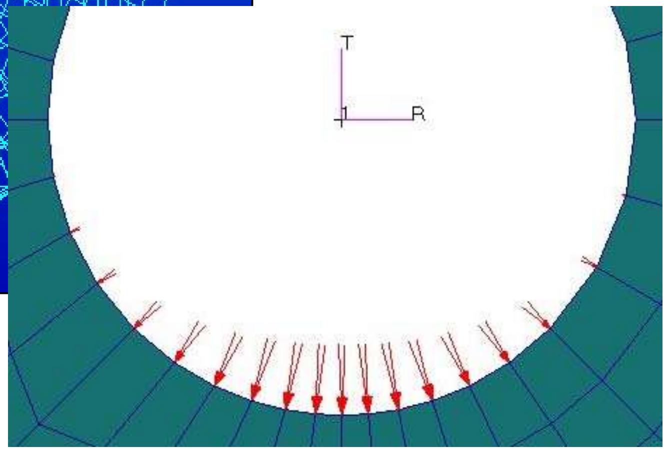


To save the figure of the finite element model:

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

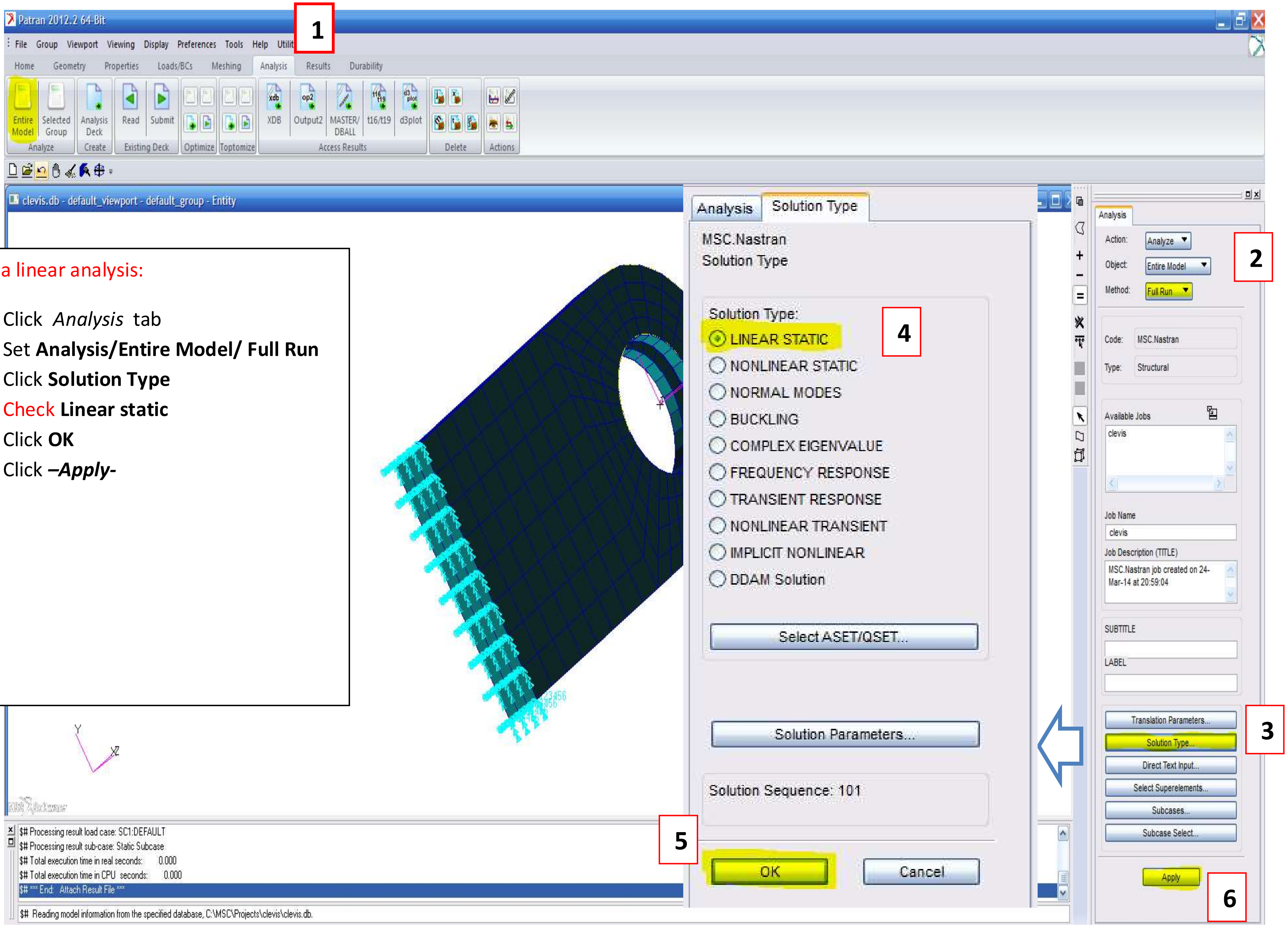
A **white background** of **all** figures is **obligatory**.

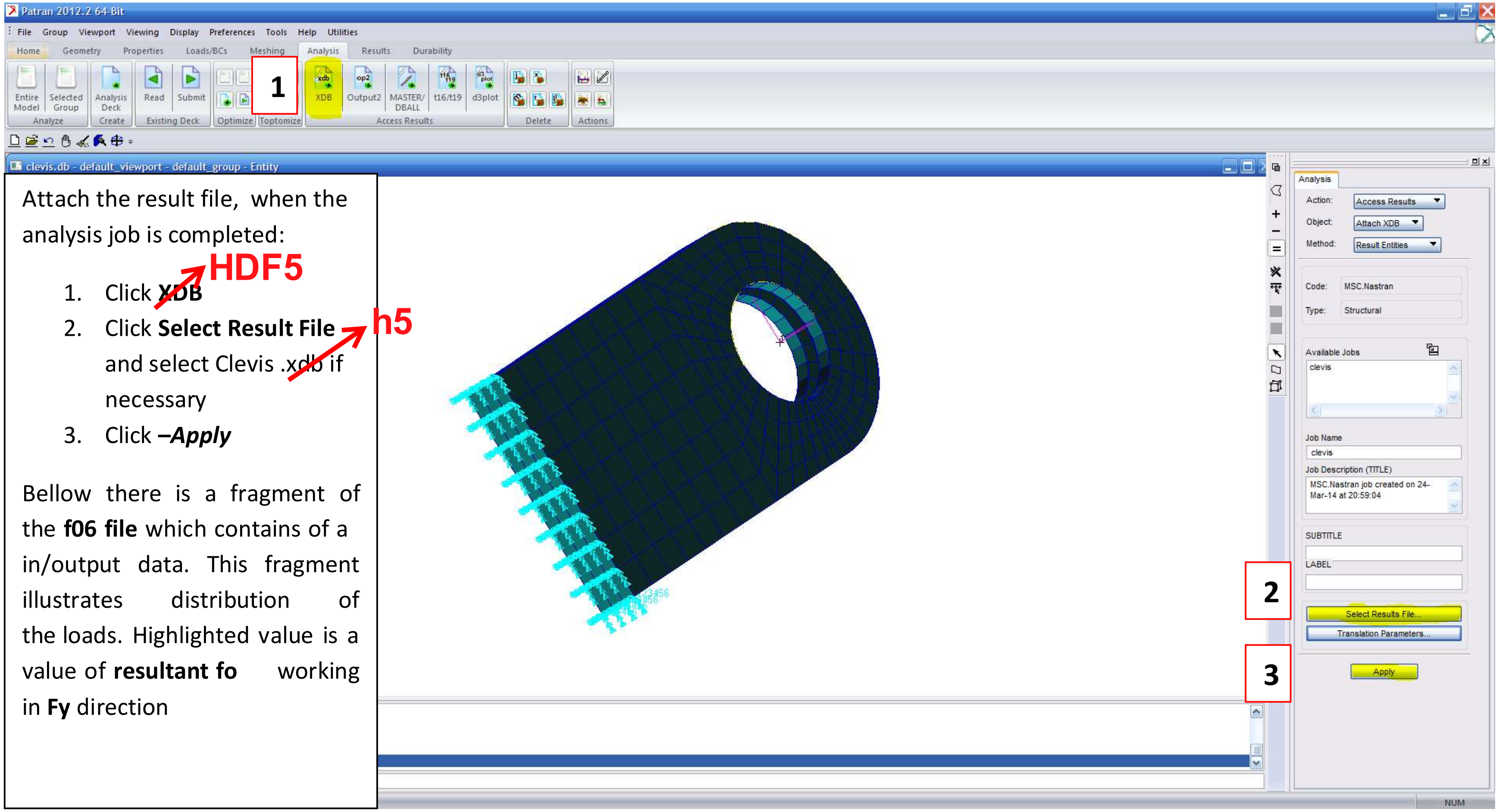




Run a linear analysis:

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-***





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

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

Bellow there is a fragment of the **f06 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

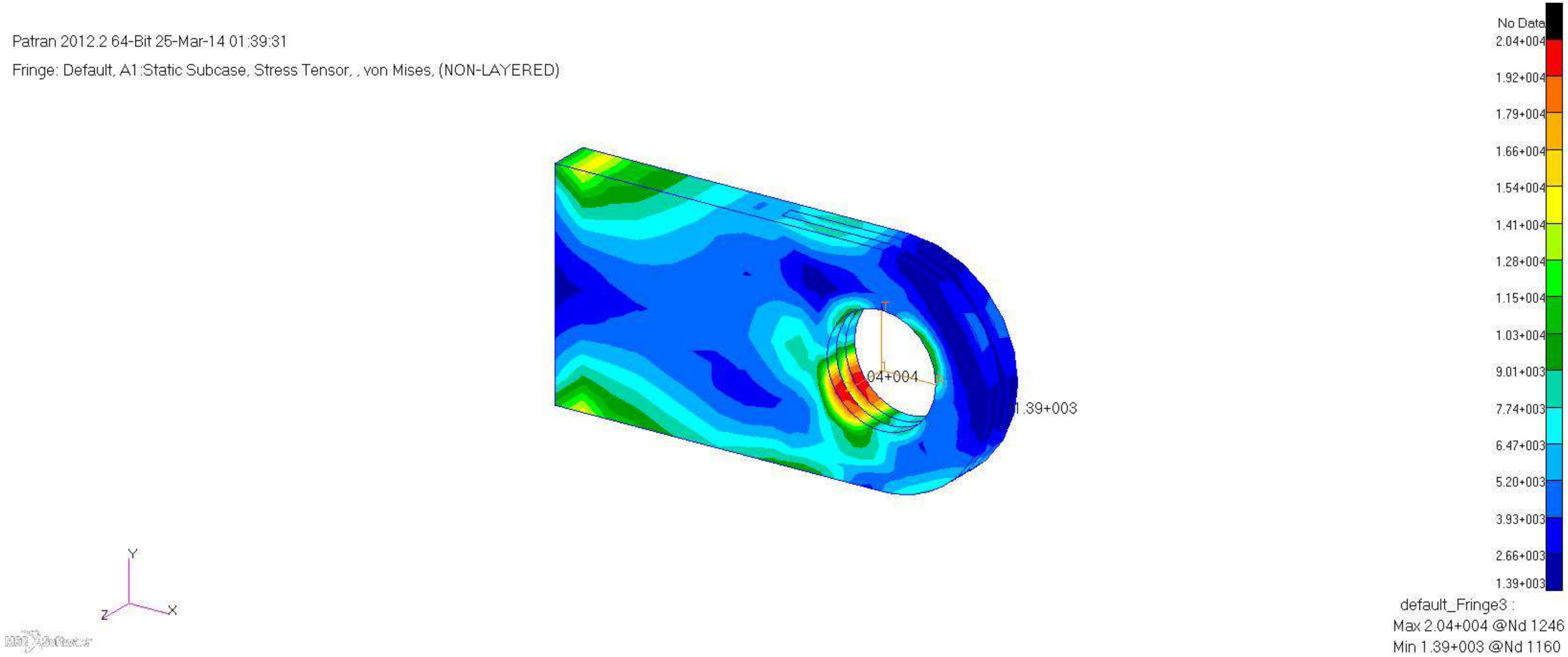
SUBCASE/ DAREA ID	LOAD TYPE	T1	T2	T3	R1	R2	R3
1	FX	-1.011008E+01	----	----	----	3.790687E+00	2.749616E+04
	FY	----	4.583406E+03	----	1.718777E+03	----	-2.750044E+04
	FZ	----	----	8.715680E-05	5.994861E-05	5.229408E-04	----
	MX	----	----	----	0.000000E+00	----	----
	MY	----	----	----	----	0.000000E+00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

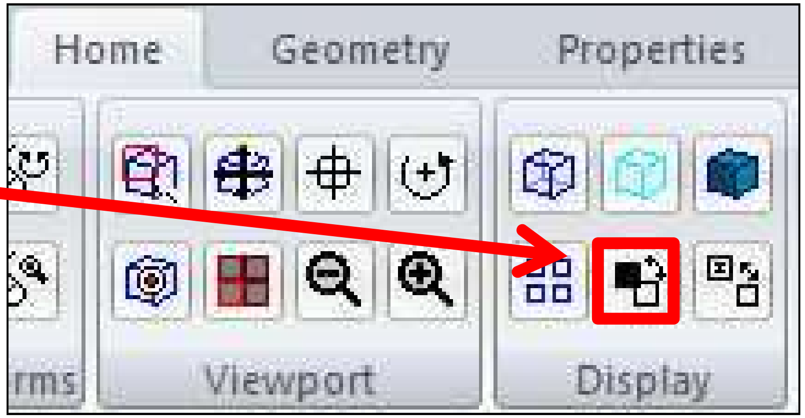


CLEVIS supplement

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

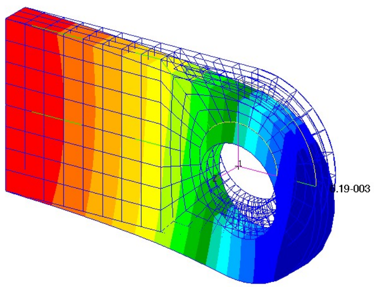


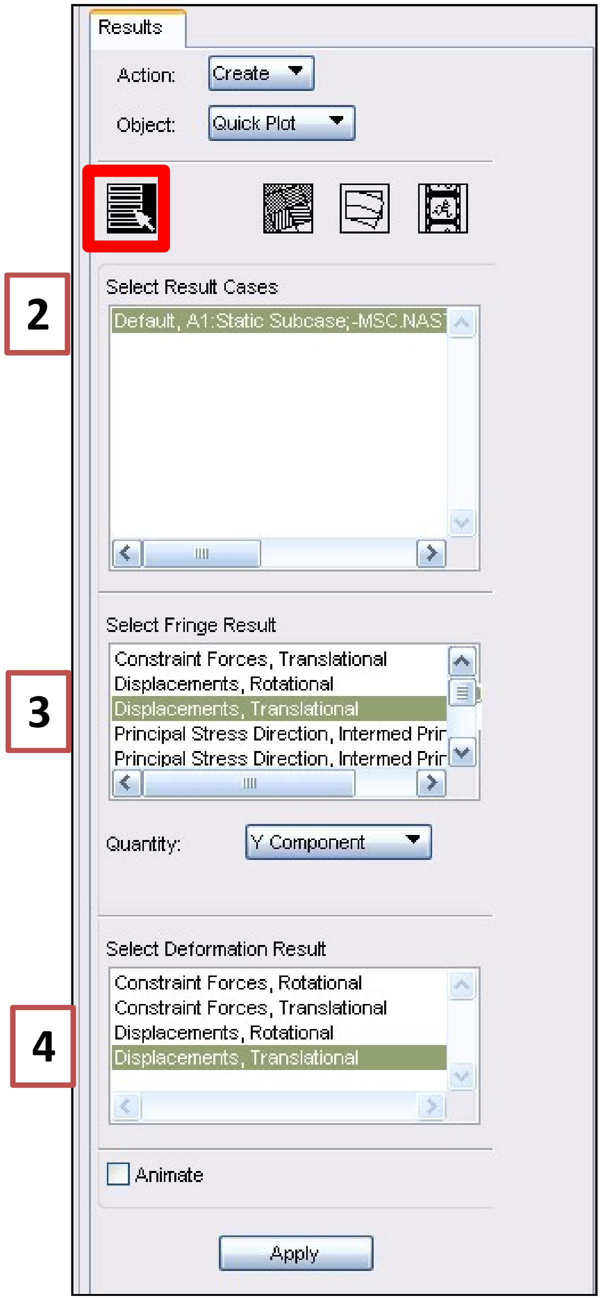
Follow the steps (**numbers** in **red frames**). → 1

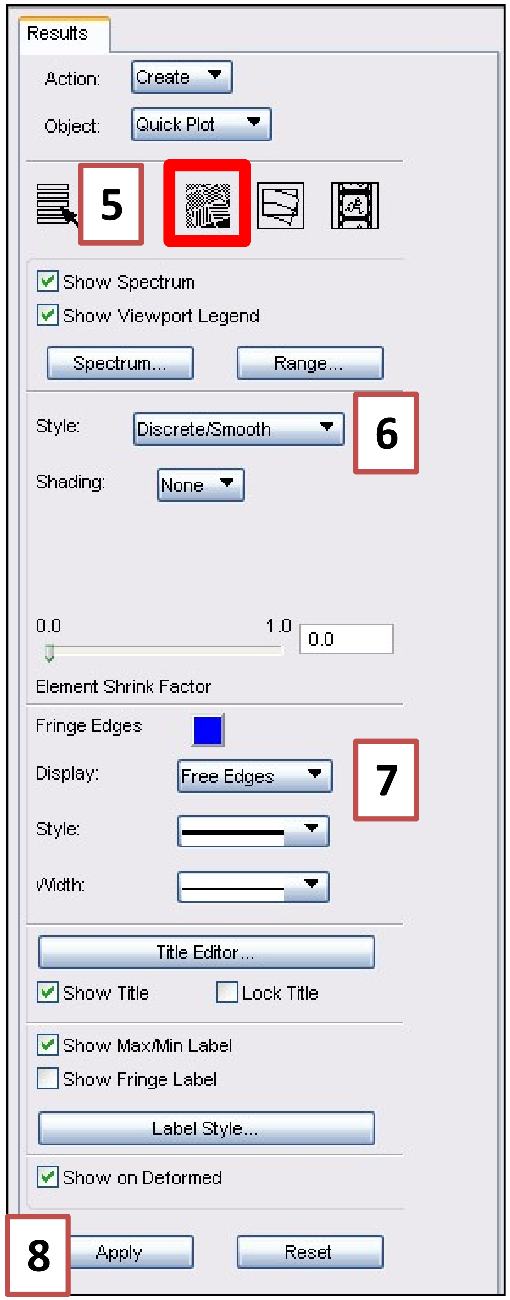
PLOT no. 1
Vertical translational displacements in Y direction

1 **Results -> Create -> Quick Plot**

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





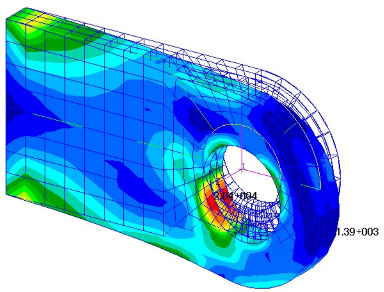


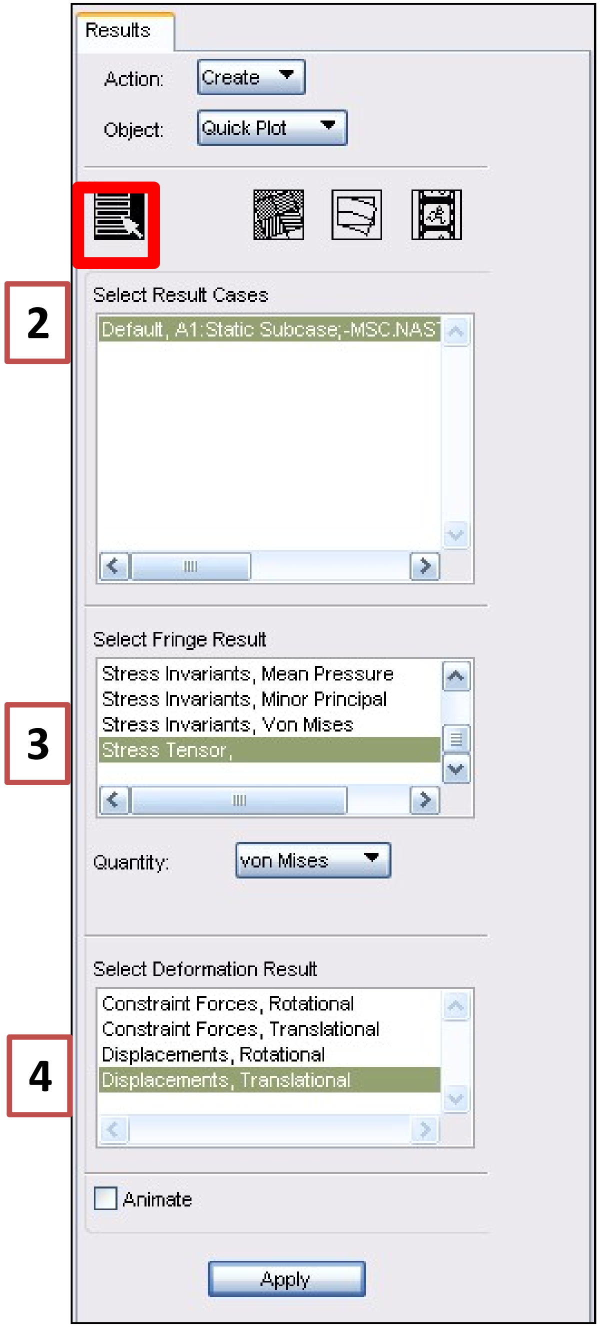
PLOT no. 2

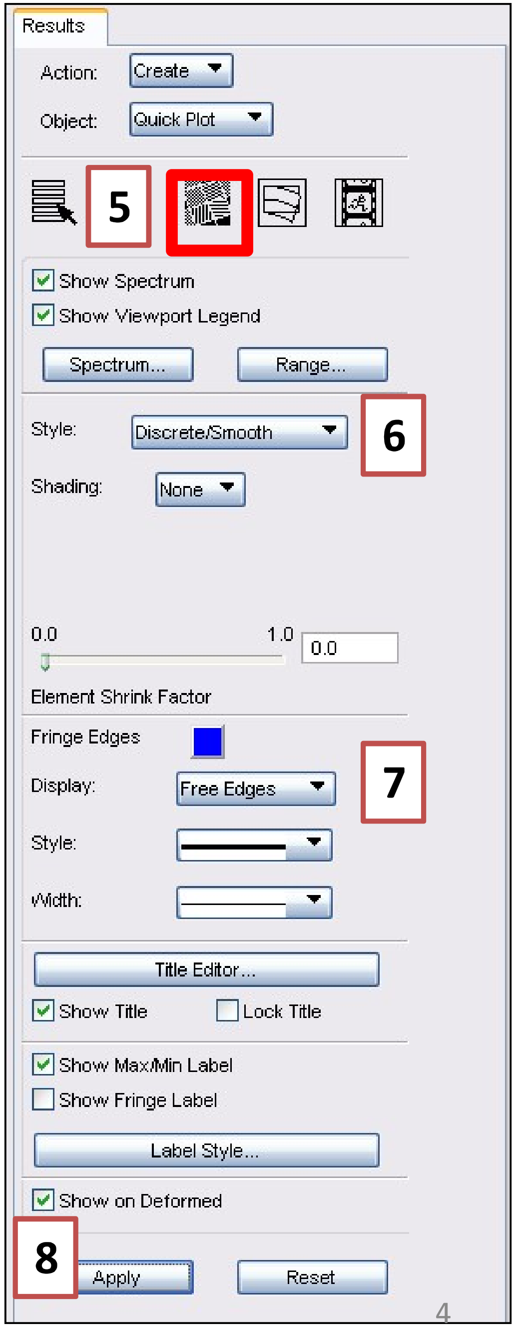
Von Mises stress σ_{equiv}

1 Results -> Create -> Quick Plot

9 File -> Images... -> Apply







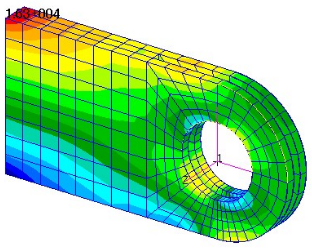
PLOT no. 3

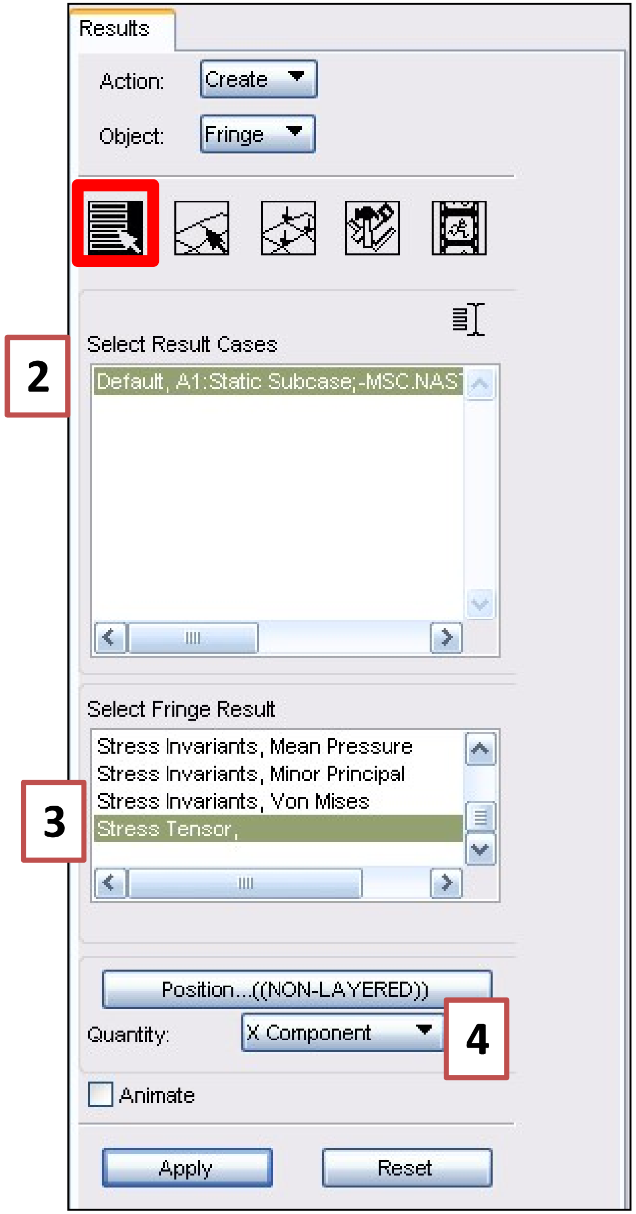
Stress in X direction σ_x

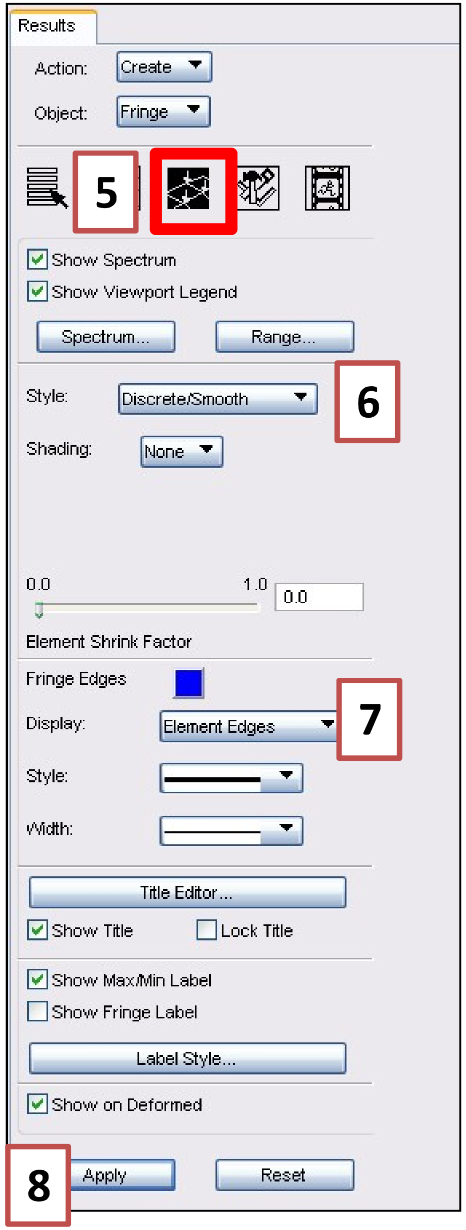
with averaging, continuous σ_x

1 Results -> Create -> Fringe

9 File -> Images... -> Apply







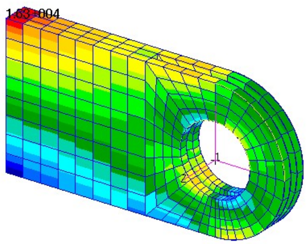
PLOT no. 4

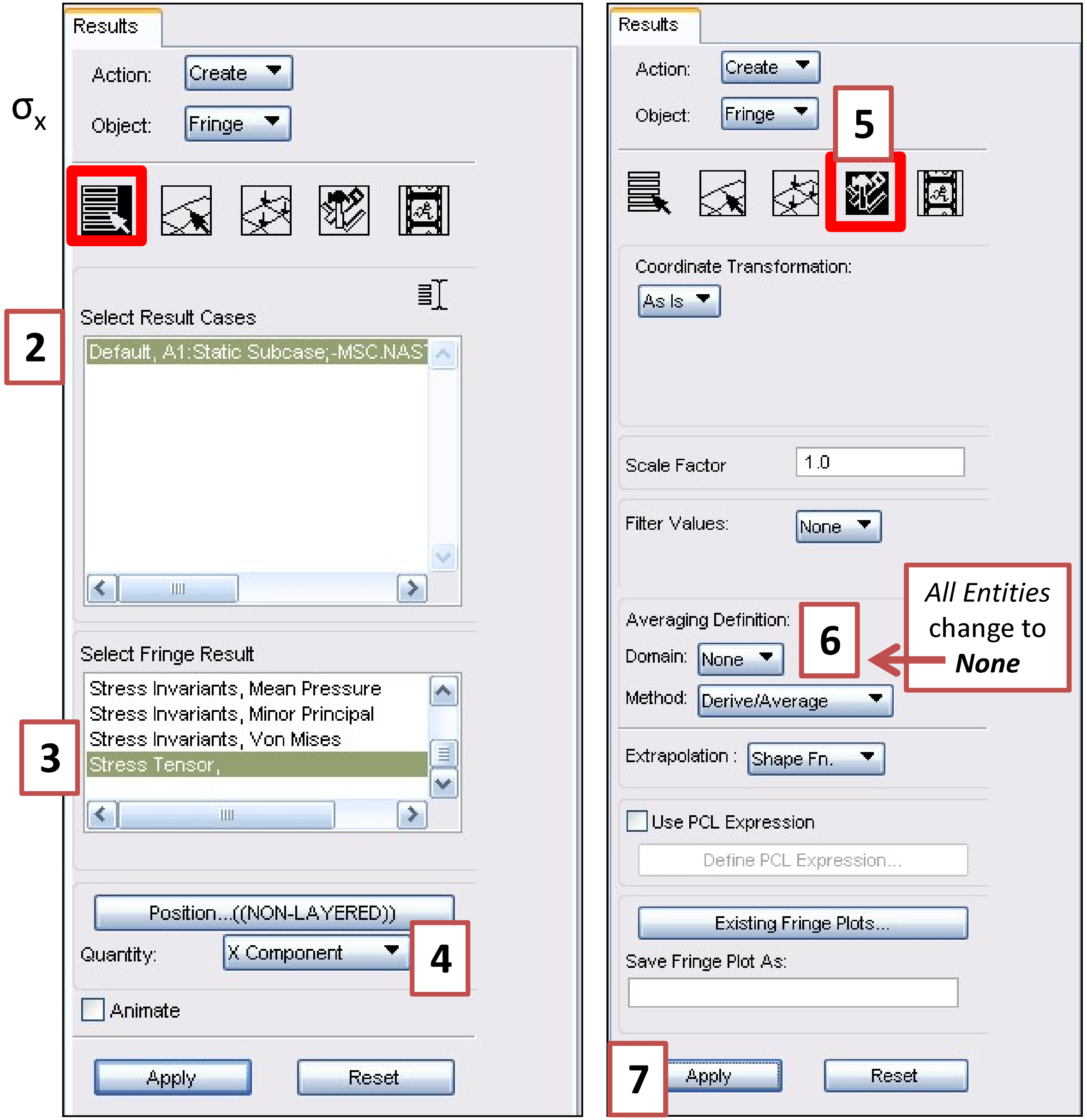
Stress in X direction σ_x

without averaging, discontinuous σ_x

1 Results -> Create -> Fringe

8 File -> Images... -> Apply



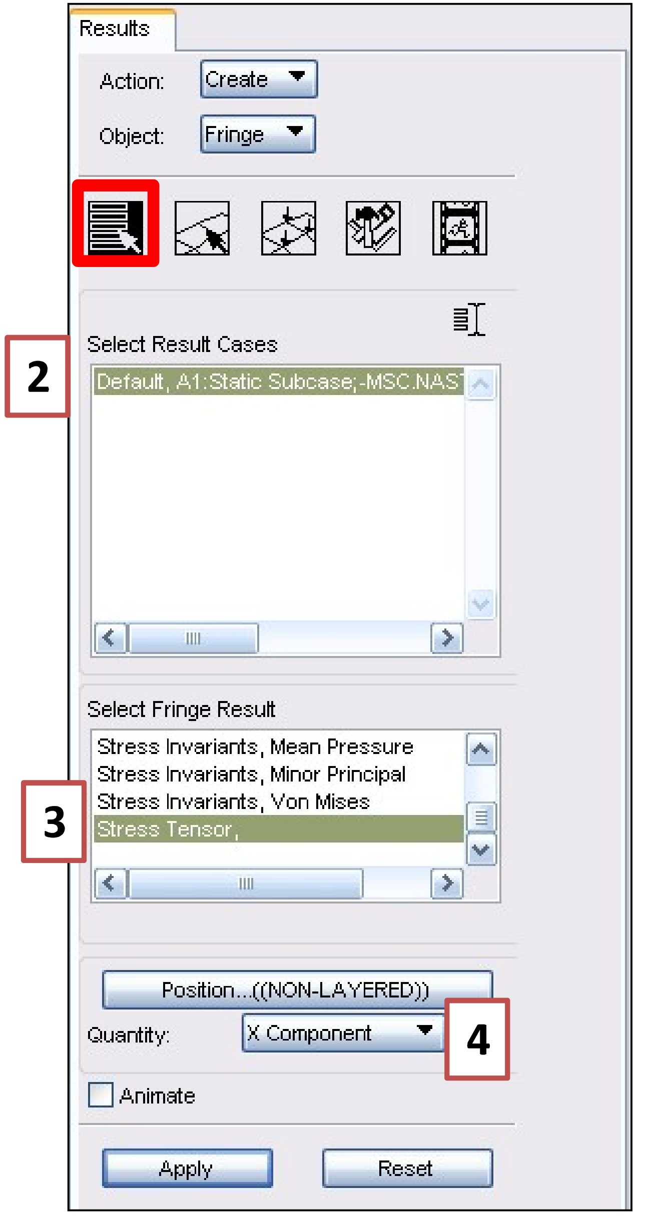


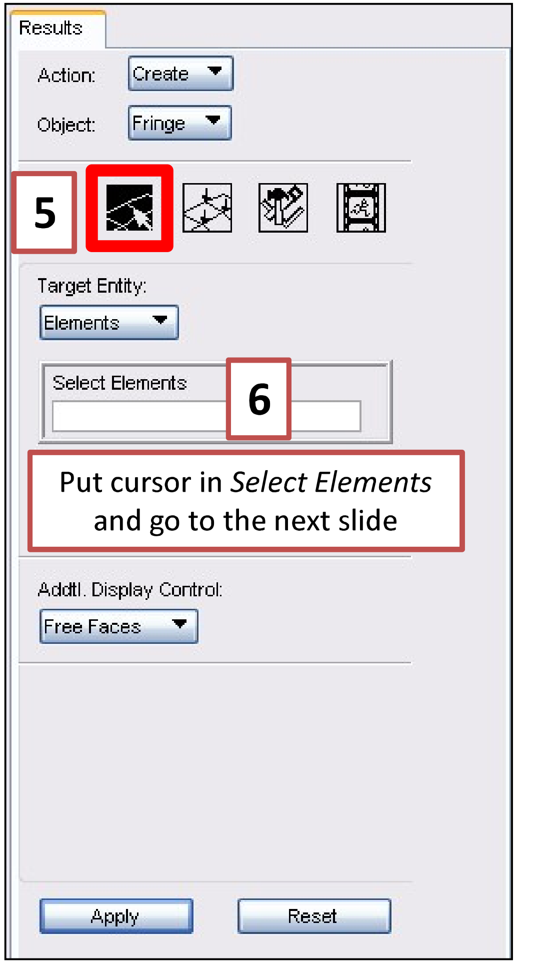
PLOTS: no. 5 and no. 6

Stress in X direction σ_x

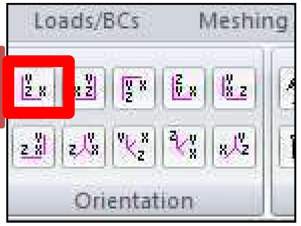
with averaging, continuous σ_x for the base of the clevis (**2** different views)

1 Results -> Create -> Fringe

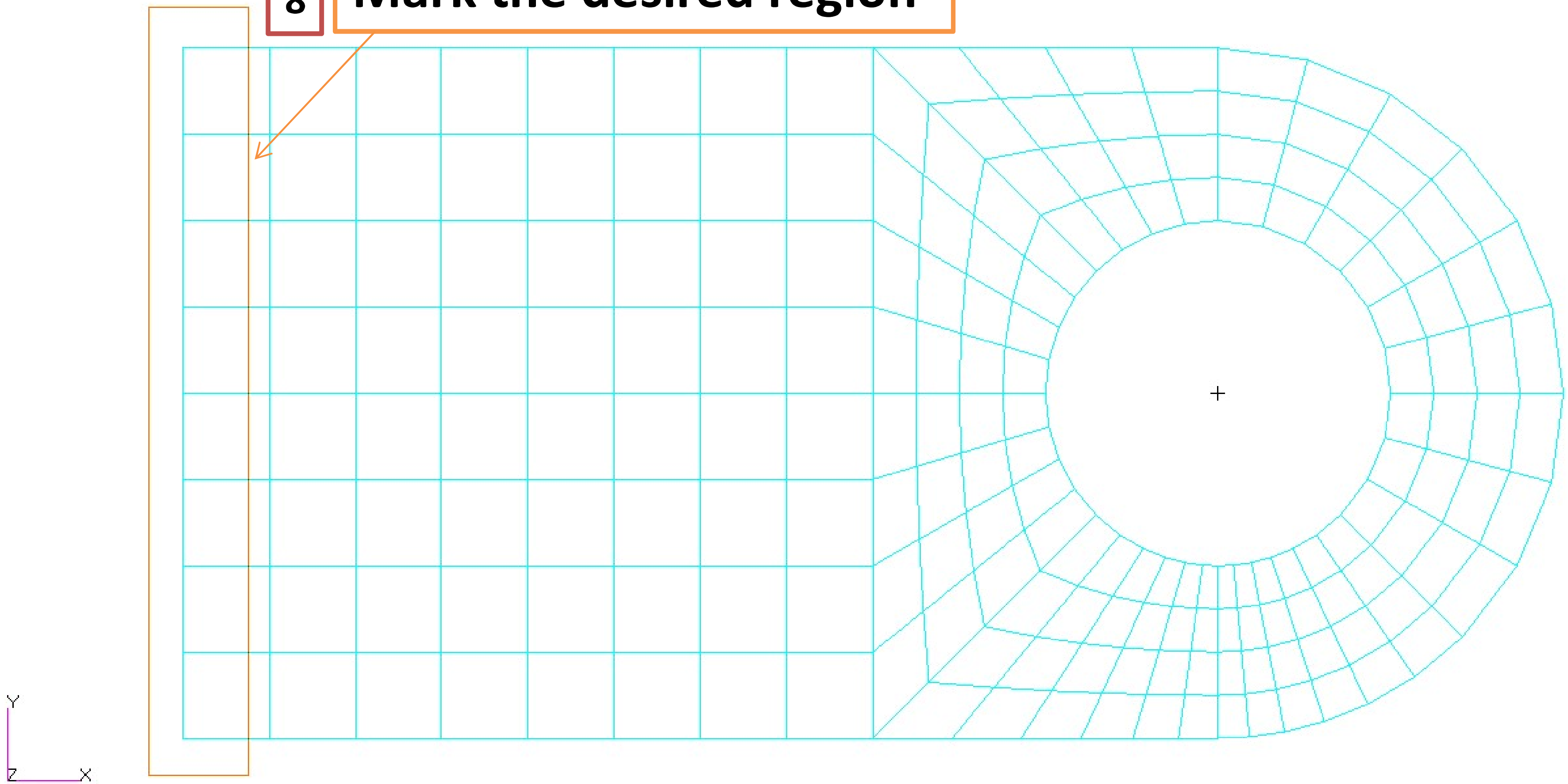




7

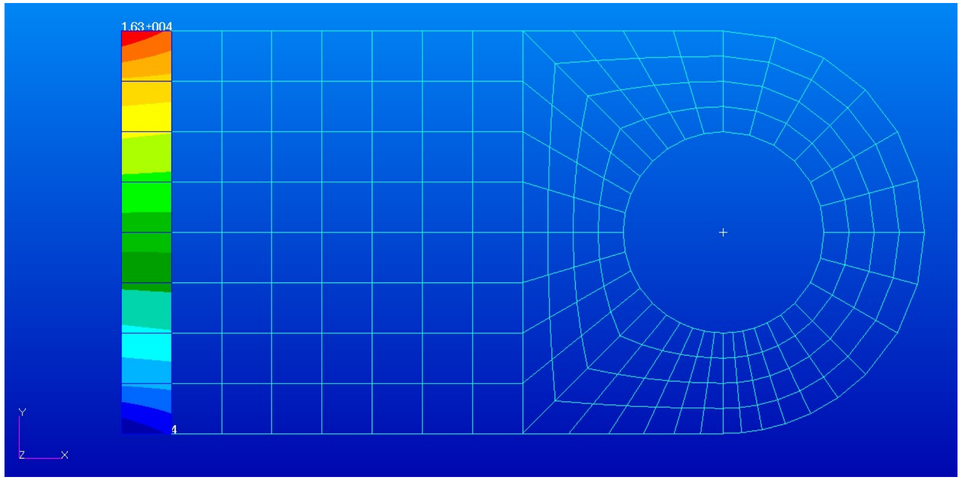


8 **Mark the desired region**

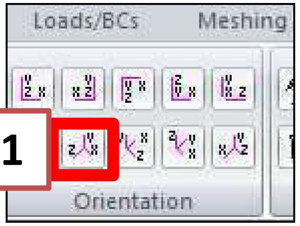


9 Click **Apply**

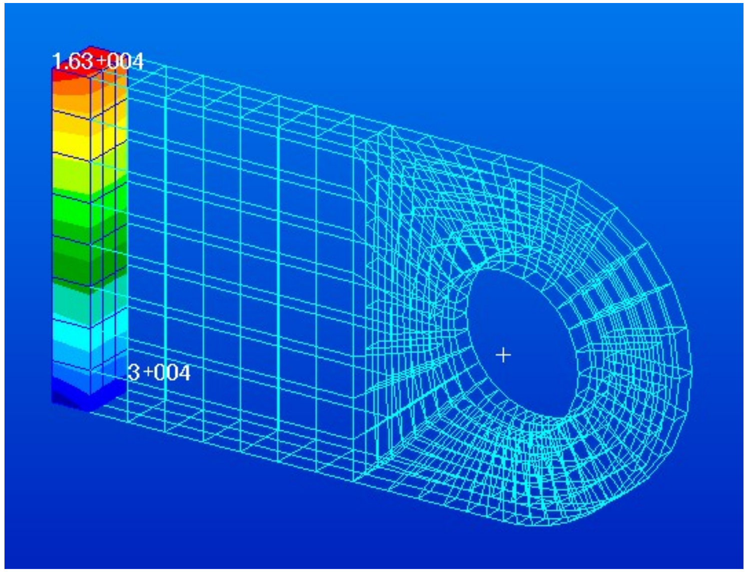
10 File -> Images... -> Apply



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



11



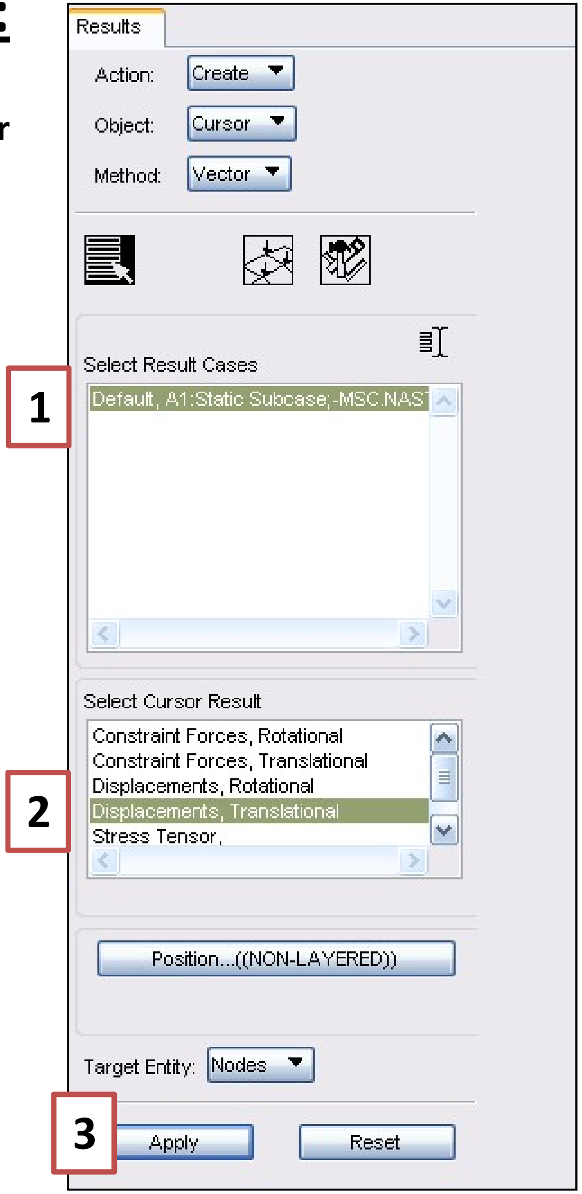
12 File -> Images... -> Apply

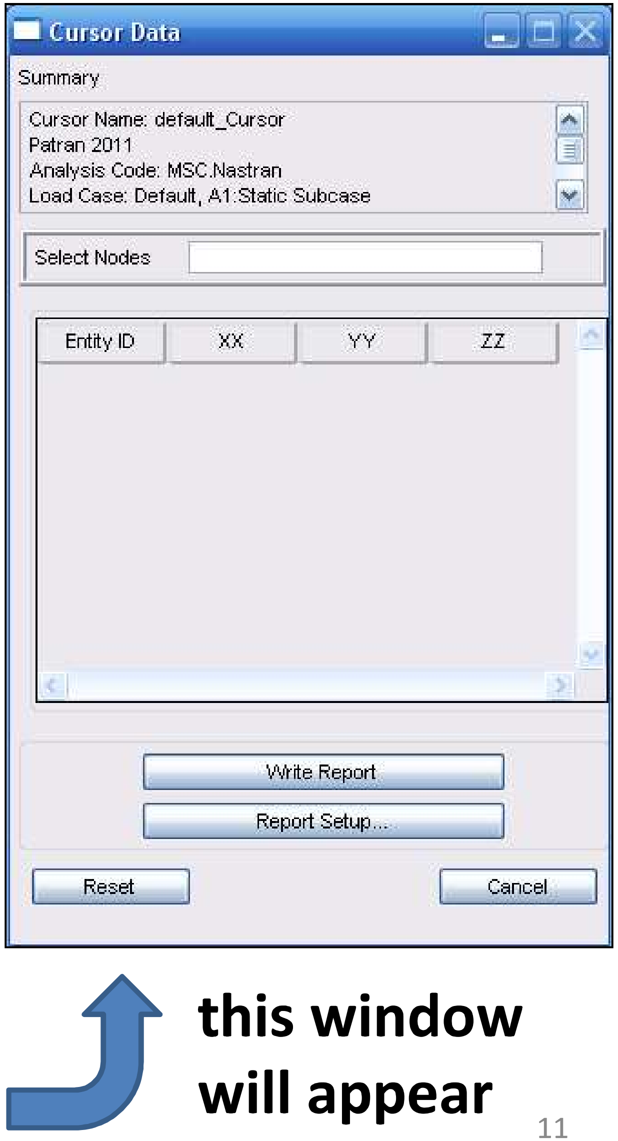
Check the value of the displacement in the direction Y of the node located on the lower surface of the hole at the distance 6 [in]:

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]:

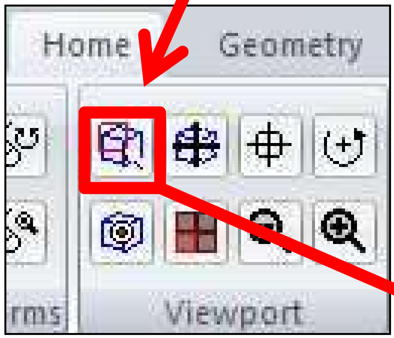
Results -> Create -> Cursor -> Vector

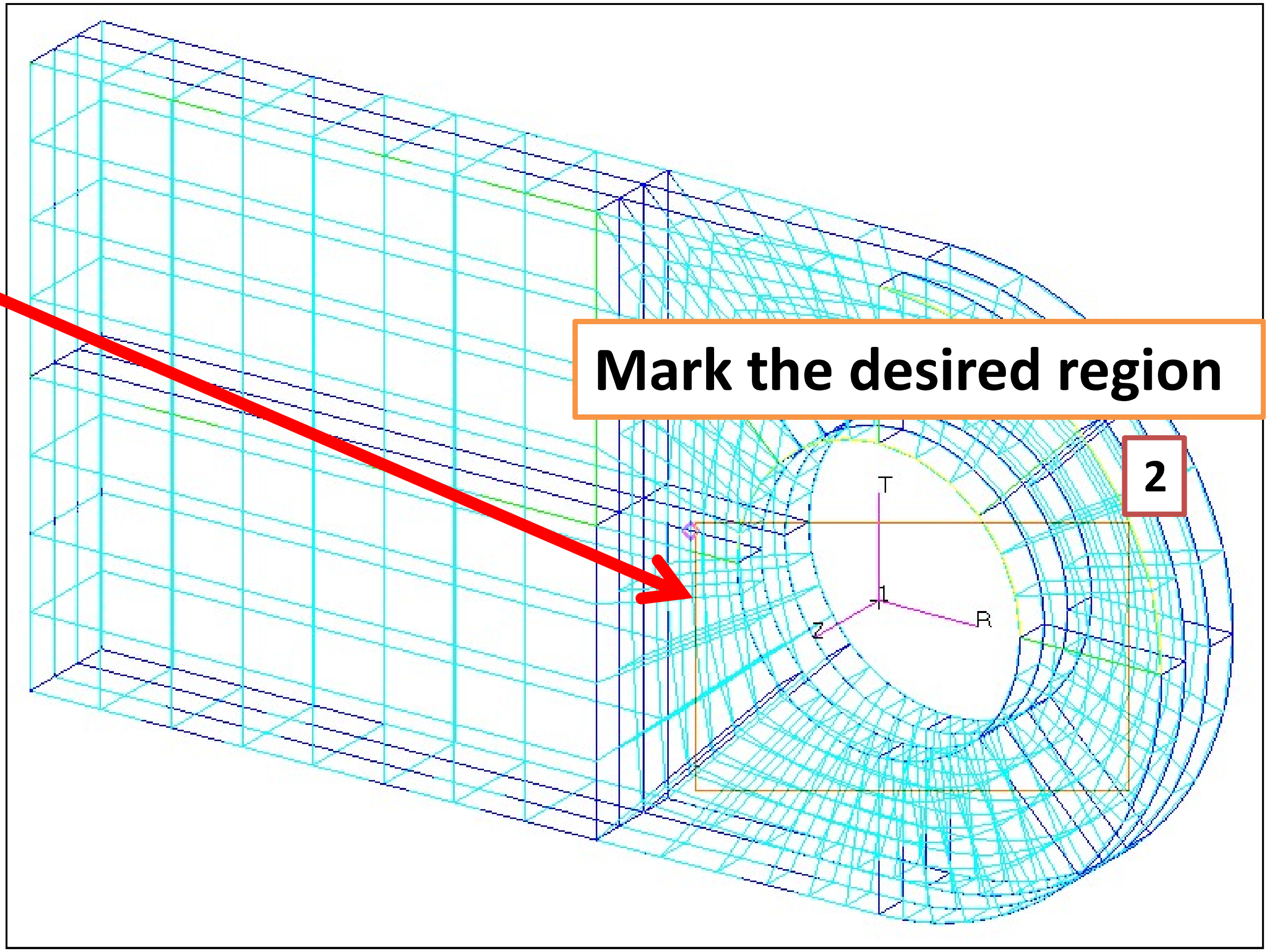




this window will appear

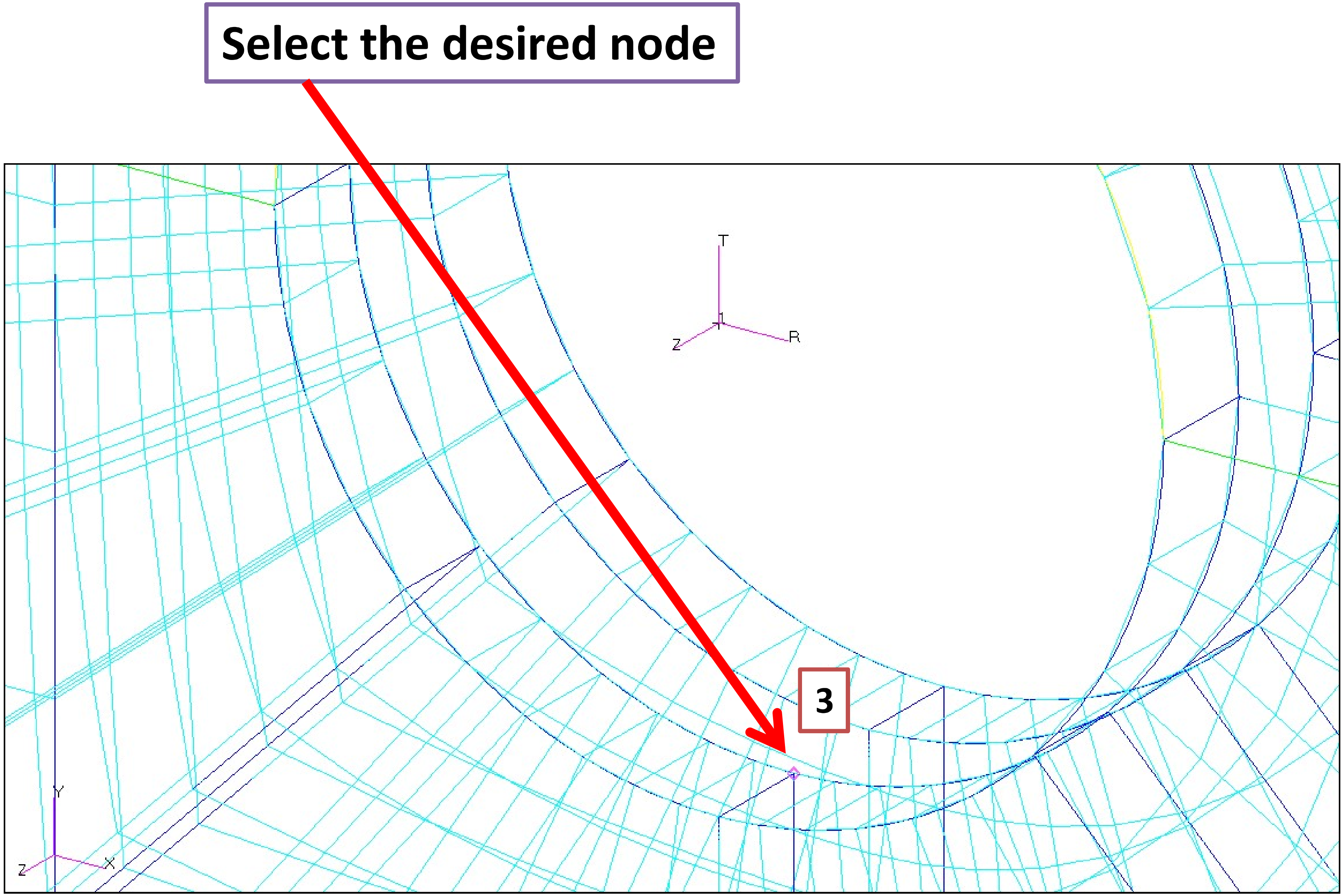
1 **View Corners**





Mark the desired region

2

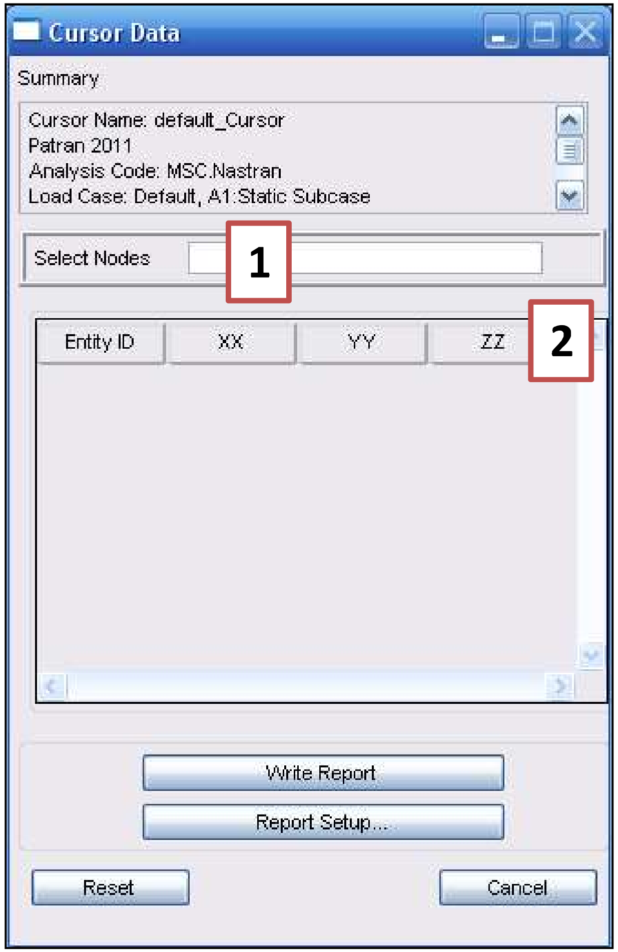

Select the desired node
3
T
Z
R
Y
Z
X

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

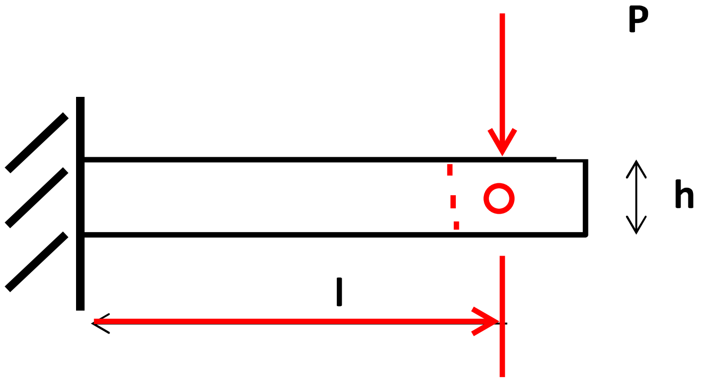


BEAM

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 ($f_{beam} = \cdots$).

Data:

$l = \cdots \ [in]$	length
$b = \cdots \ [in]$	width
$h = \cdots \ [in]$	height
$E = \cdots \ [psi]$	

$I_y = \cdots \ [m^4]$	
$P = \cdots \ [\text{lbf}]$	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 ($f_{beam} = \cdots$)
- data and calculations with proper units
- relative error calculations

c) **Conclusions**