MEE 631 CAD of Mechanical Systems

Tutorial: Performing Convergence study on Stress singularities

Instructor: Iman Salehinia, Ph.D.

TA: Mujahid Mohammed

Department of Mechanical Engineering

NIU, Spring-2017

We have encountered stress singularities while solving the bracket that we modeled for Lab Assignment-3 Tutorial-2.

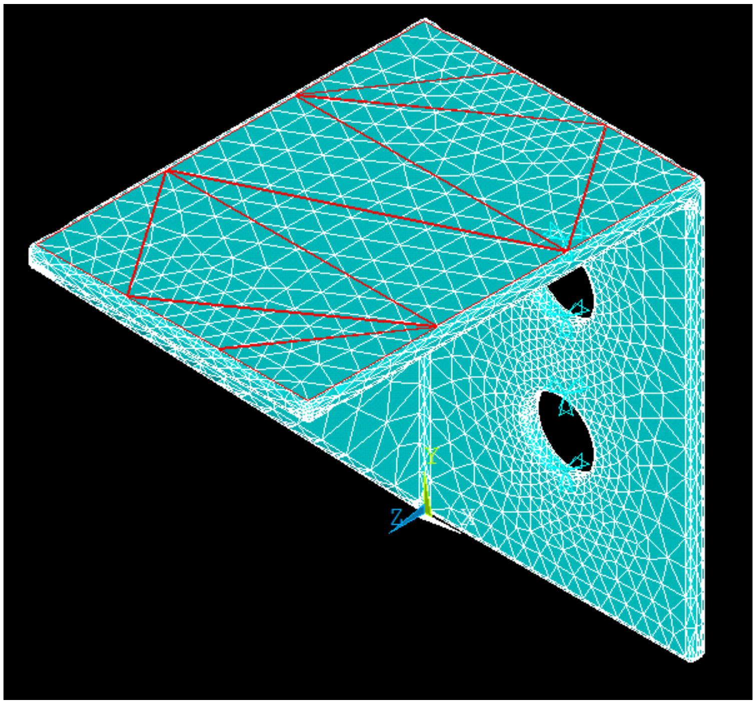


Fig-1 Image showing boundary and loading conditions on the bracket

The holes on the bracket are restricted in all directions and a pressure of 50 Psi is applied on the top surface of the bracket.

Stress singularities are the points in the finite element model where the results for the stress don't converge on mesh refinement. Stress increases at these points, as the mesh size is refined in each trial. Stress singularities occur at critical locations in the model the geometry has sharp re-entrant corners, corner of different bodies in contact, sharp edges, cracks, point restraints and point loads.

Consider solving a flat plate applied with uniform tensile stress on the ends and having a square section cut at its center. This problem could be solved considering symmetry of the geometry and loading.

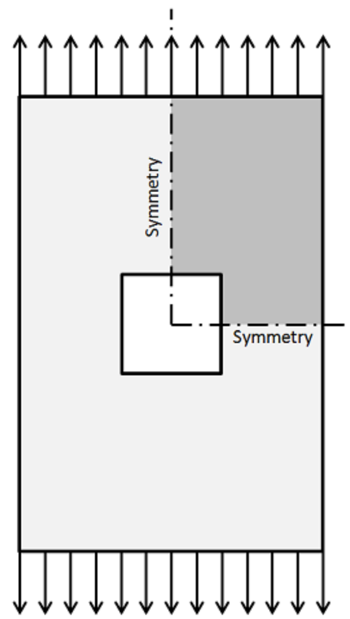


Fig. 2- Flat plate with a square cut under uniform tensile load (Frei, 2013)

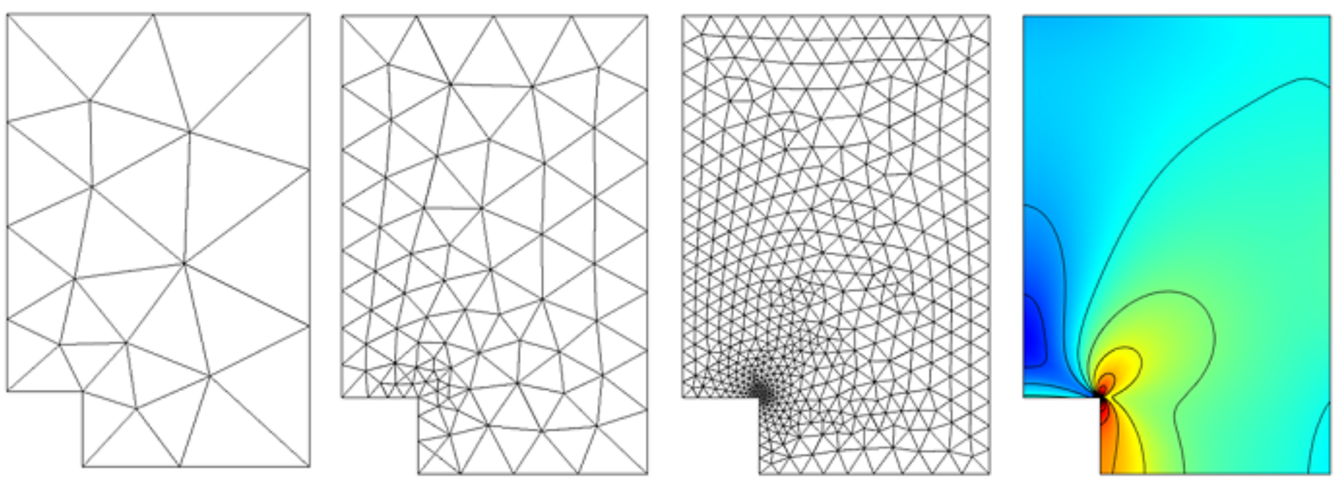


Fig.3- Adaptive mesh refinement around the corner of the square cut section (Frei, 2013)

While solving it was observed that the stress at the corner in the square cut section does not converge and increases with an increase in refinement. This situation is a stress singularity.

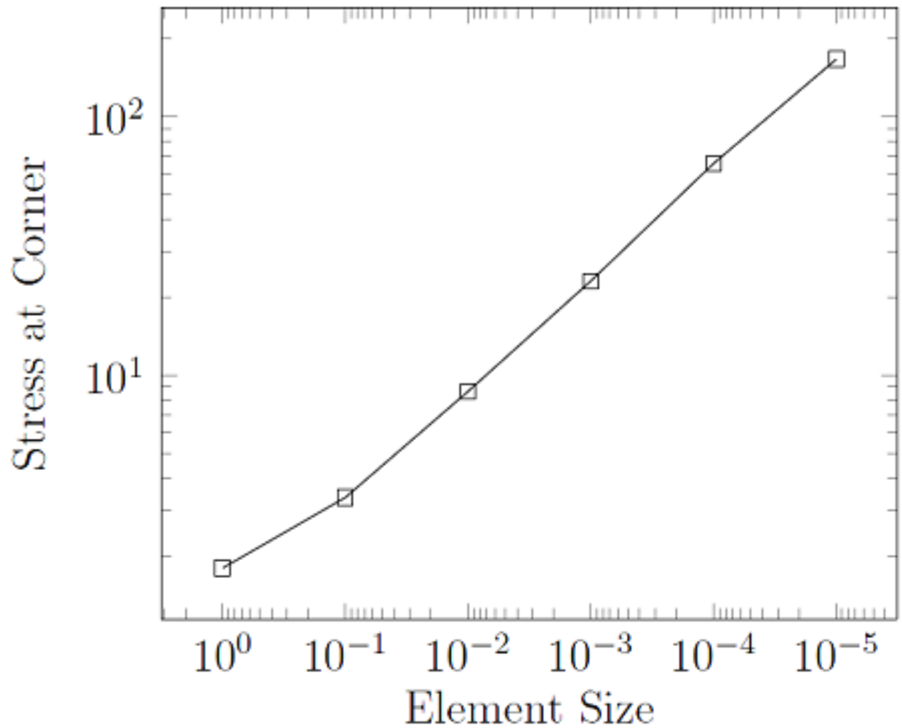


Fig. 4- Stress at corner Vs Element Size plot. The stress doesn't converge due to the presence of singularity (Frei, 2013)

Though the stress doesn't converge in a model with singularities, the displacement or directional deformation get converged due to the fact that stress is a derivative of displacement. It can be clearly observed from the contour plots that with each refinement the maximum value of stress increases which corresponds to the singularity point.

St. Venant's Principle [2] states that the effect of local disturbances to a uniform stress field remains local. This means that the increasing maximum value of stress on a model due to stress singularity is only concentrated in its vicinity and has negligible effect as we go away from the point of singularity.

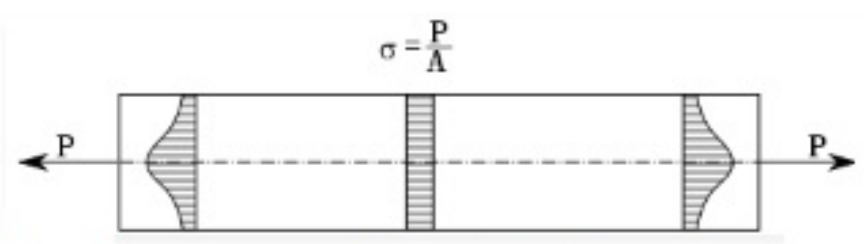


Fig. 5- Uniform tension load applied to specimen (Acin, 2015).

The stress distribution at the vicinity of the applied load is not uniform. However, the stress becomes uniform at location away from the applied load. This concept is used to study stress convergence in a model with singularities. Adding fillets at sharp reentrant will also help in getting the stress converged.

Stress convergence at a location away from the singularity could be studied in two ways by either considering variation of maximum stress which corresponds to a single node on a particular line or study the variation of stress along a particular path for different mesh refinements. It is important that the location is far enough from the stress singularity point so that the stress singularity does not exist and also close enough so that converged values are reliable for critical points. FE analyzer must try various points to make sure about the validity and reliability of the results. This is a skill that gets better via practice.

<u>**Study of convergence for a model with singularity over a node**</u>

The geometry created as per the assignment file had sharp corner at the interface of surfaces which acted as the sharp re-entrant creating one a singularity point while the other being at the center holes where the boundary conditions has been applied. In order to reduce the effect of these sharp edges a model was created in Solidworks with fillets at all the edges. The fillet at the holes provided lines (entities) where the convergence study was carried out for the stress.

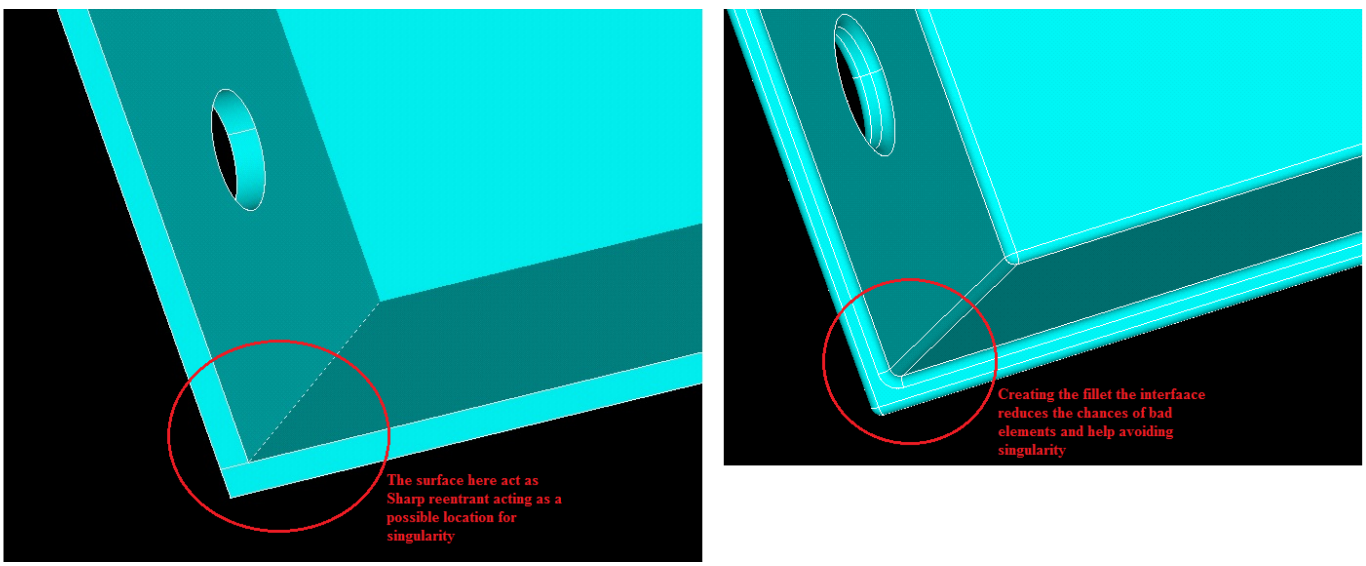


Fig. 5- Study of stress convergence considering the variation of maximum stress on nodes of a line at a distance away from the singularity.

Let's find out how to perform the convergence study.

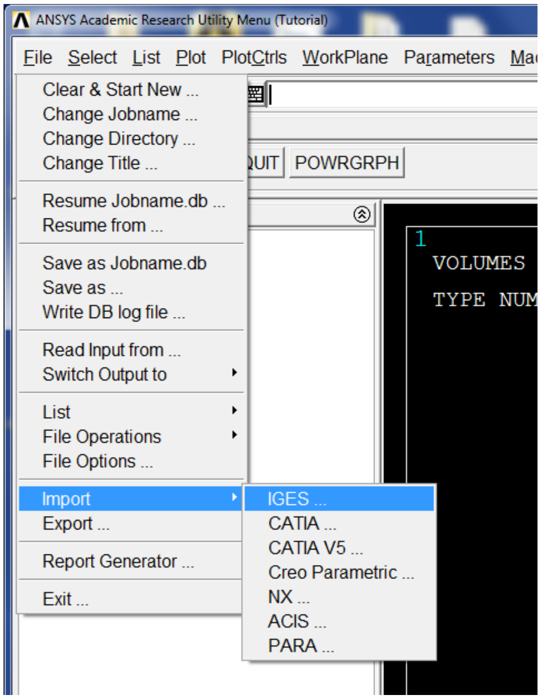
The IGES and Parasolid file for importing the geometry could be accessed through NIU Blackboard

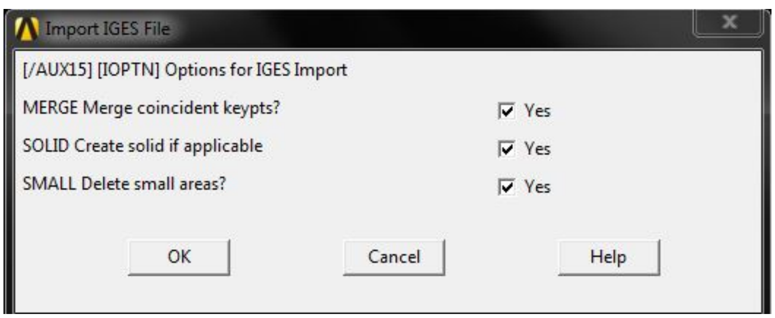
MEE631 > Content > Lecture Notes > Session for ANSYS tips and tricks.

PREPROCESSOR:

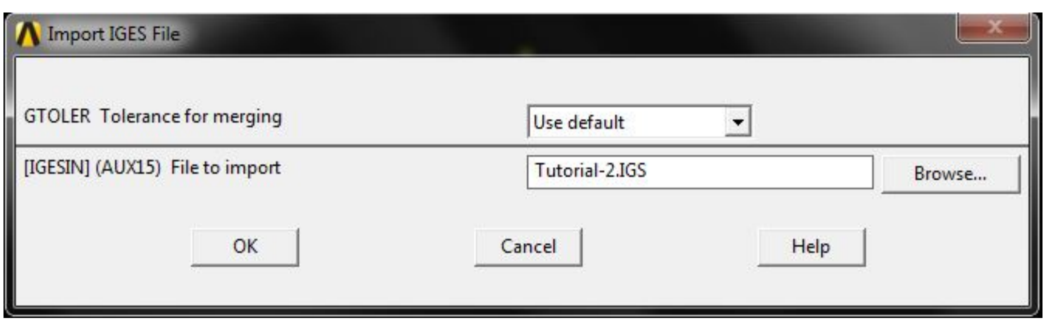
1) Import the geometry:

Utility Menu > File > Import > IGES



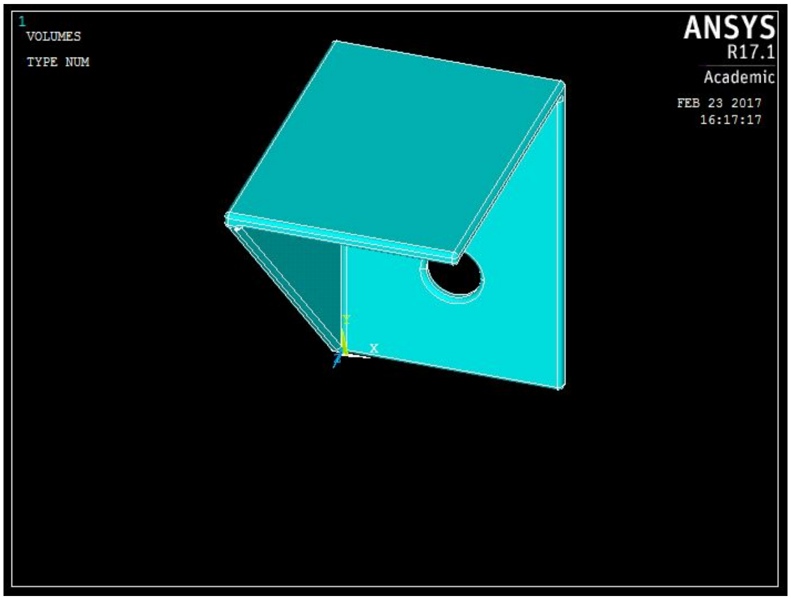


Select all options Import IGES File dialog box and click OK.



Browse to appropriate file location and select the IGES file saved on your computer, click OK.

The geometry of the modal is loaded in the workspace.

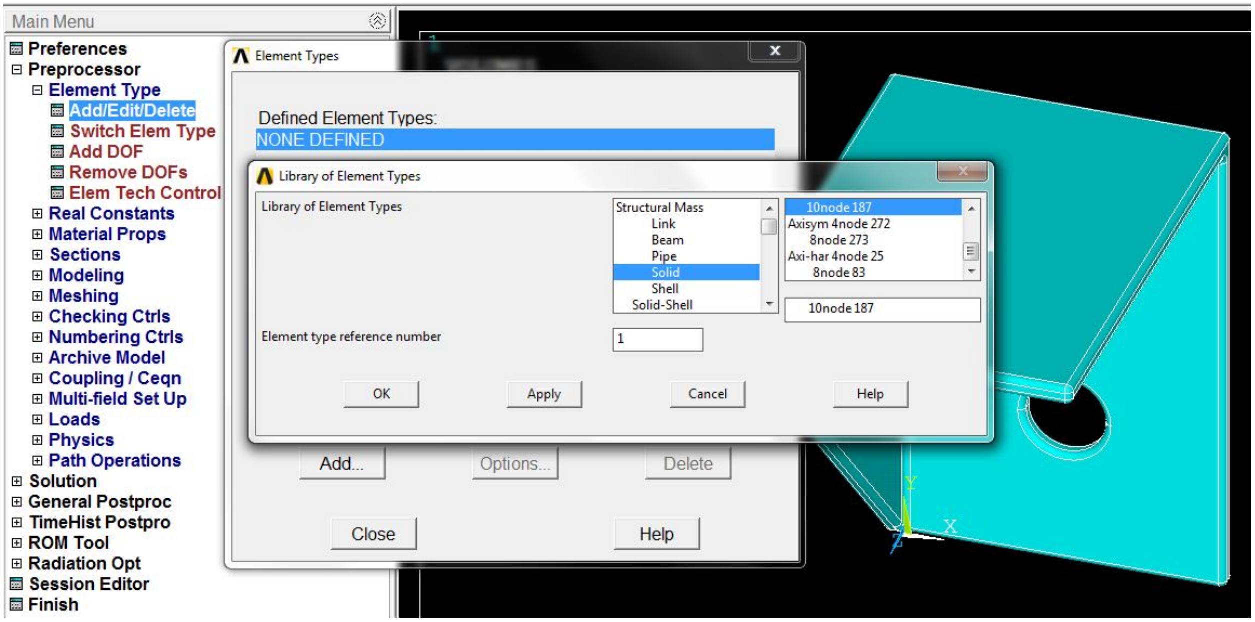


Since this geometry was modelled as a single part file there is no need to perform volume glue operation after loading the file.

2) Define Element type:

Main Menu > Preprocessor > Element Type > Add/Edit/Delete

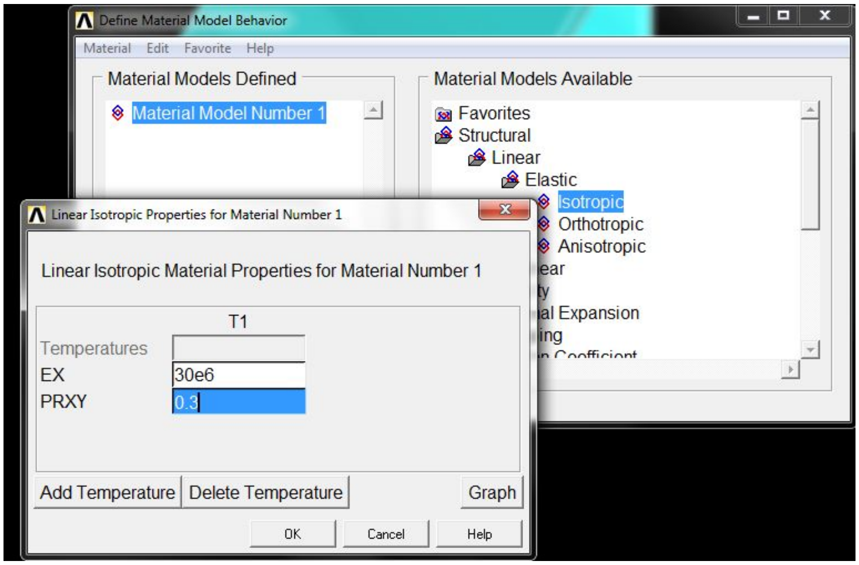
Select Solid 10 node 187 as the element type from the dialog box.



3) Define Material Properties:

Main Menu >Preprocessor > Material Properties > Structural > Linear > Elastic > Isotropic

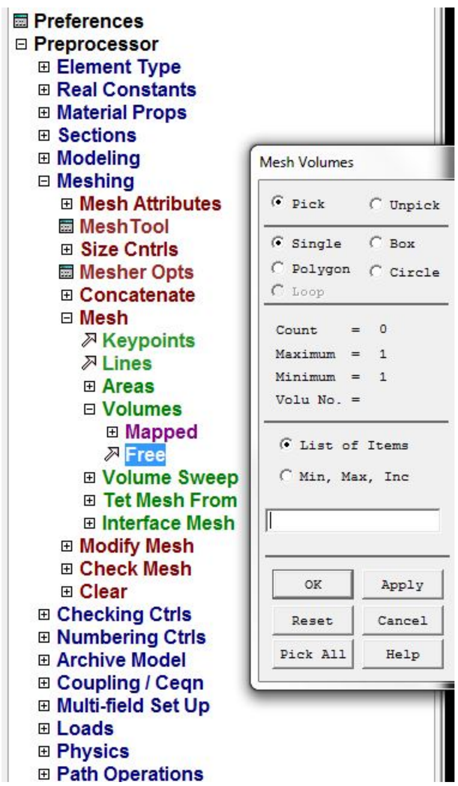
Enter 30e6 for modulus of elasticity and 0.3 for poisons ratio



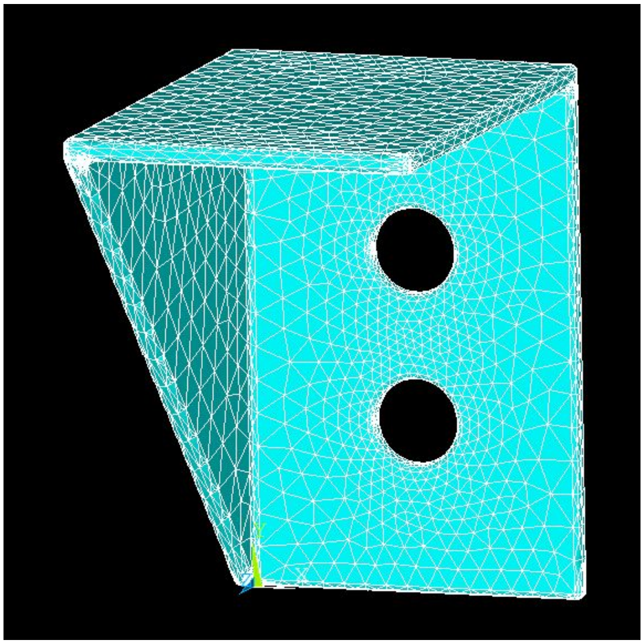
4) Meshing:

Main Menu > Preprocessor > Meshing > Mesh > Volumes > Free

Select the volume to mesh it with 10 node tetrahedral elements.



Meshed Model:



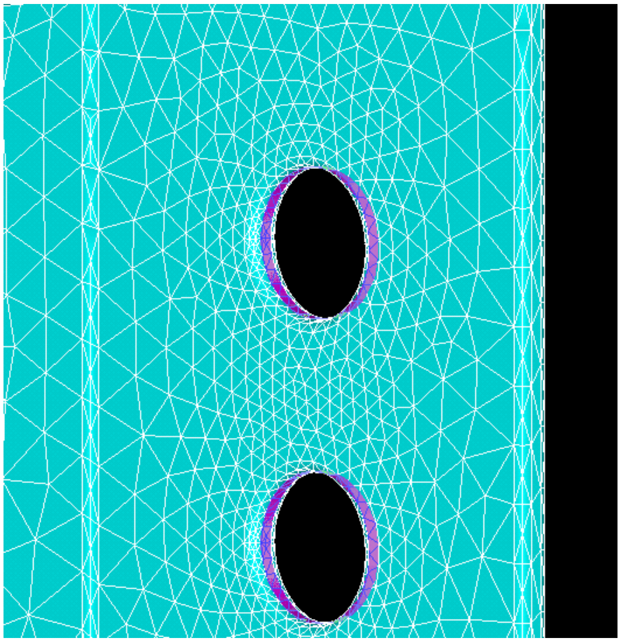
SOLUTION:

1) Define Loads

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Areas

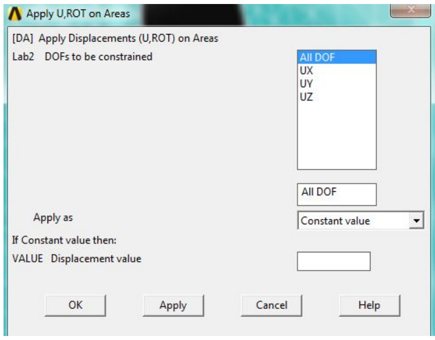
Select area on the inner surface of the holes by using the manual pick option or enter the area numbers as 65,66,73,74.

Click OK.



Care should be taken when area is being picked by picking tool that only the inner surface is selected and the fillet region is not selected.

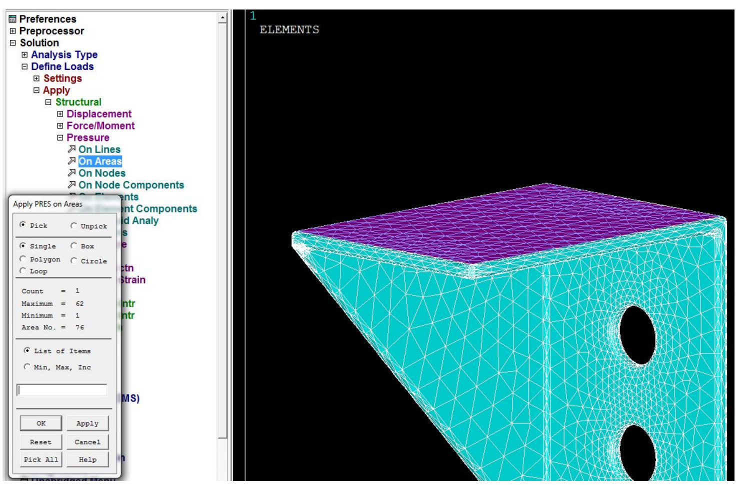
Select All DOF, Click OK



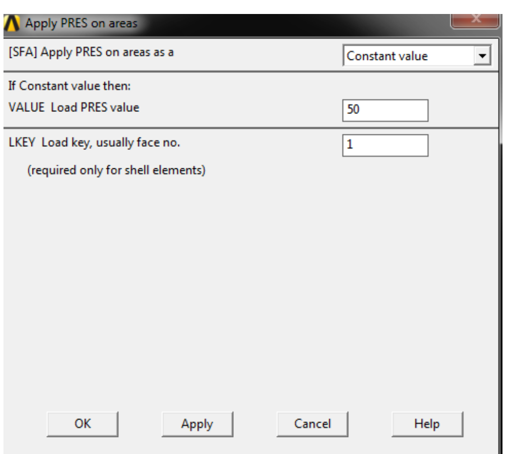
2) Apply Pressure

Main Menu > Solution > Define Loads > Apply > Structural > Pressure > On Areas

Select are on the top surface or enter area number 76 and click OK.



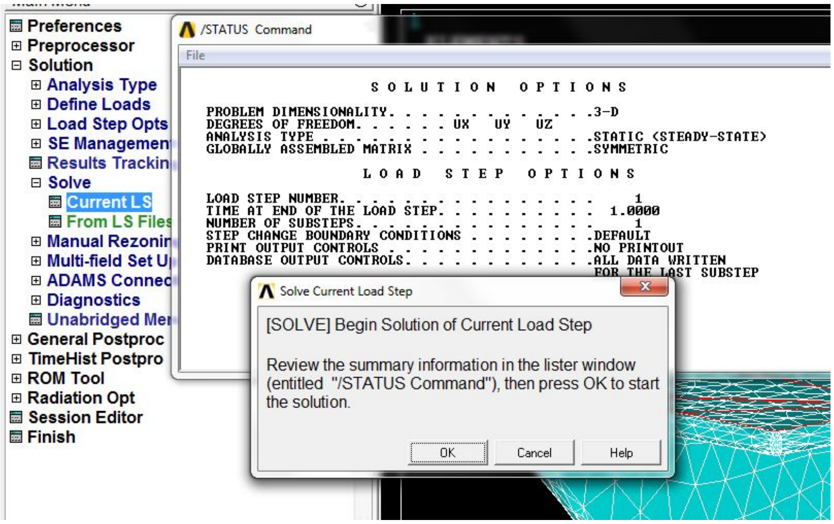
Enter 50 as the value of Pressure. Click OK



3) Solve

Main Menu > Solution > Solve > Current LS

Click Ok on the Solve Current Load Step dialog box

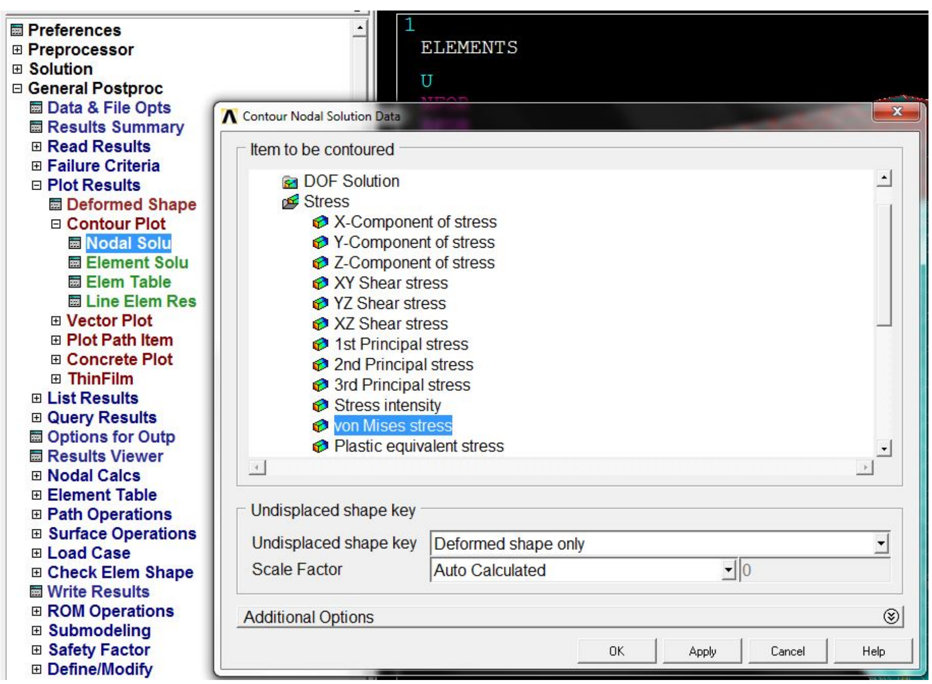


Solution is done!

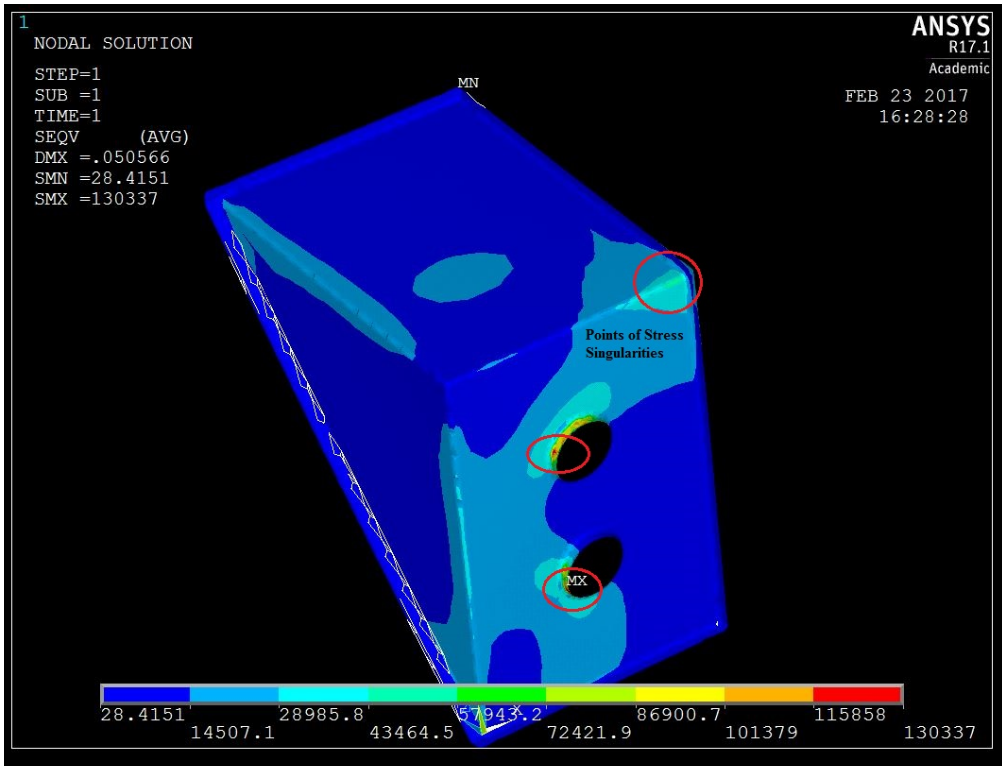
POSTPROCESSOR:

1) Von Mises Stress

Main menu > General Postprocessor > Plot Results > Contour Plot > Nodal Solution > Stress > Von Mises Stress. Click OK.



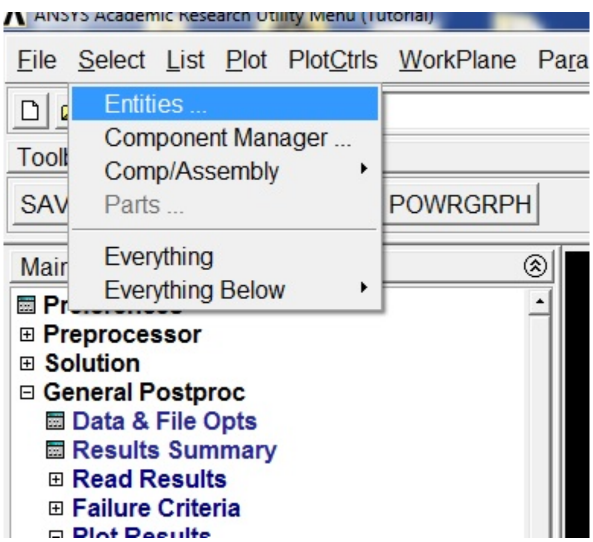
Contour Plot of Von-Mises Stress:



The singularity points are highlighted in the above image. Adding a fillet on the interface of back and top blocks helps in reducing the singularity effect which can be removed by refining the mesh appropriately.

2) Study of Von Mises Stress at a location away from the singularity.

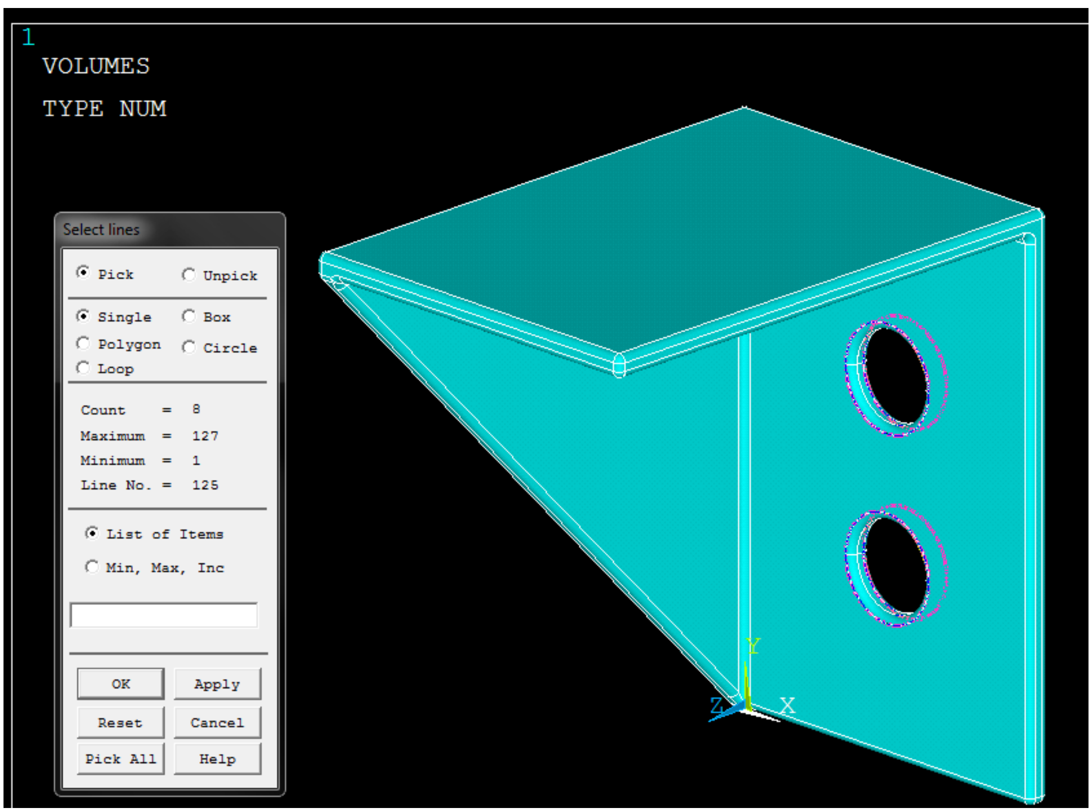
Utility Menu > Select > Entities



In the Select Entities dialog box choose “**Line**” in the first drop down menu and **“By Num/Pick”** in the second drop down menu as shown in the image below and click OK.

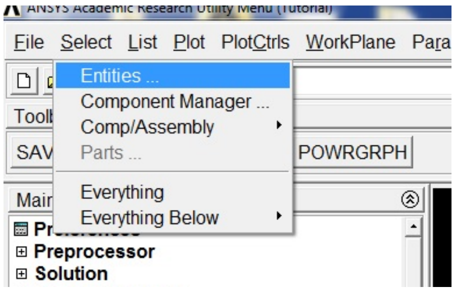
Select lines on the outer surface of the fillet either by using the picking tool or entering the values as 6, 27, 35, 55, 90, 91, 125 and 127.

Click OK.

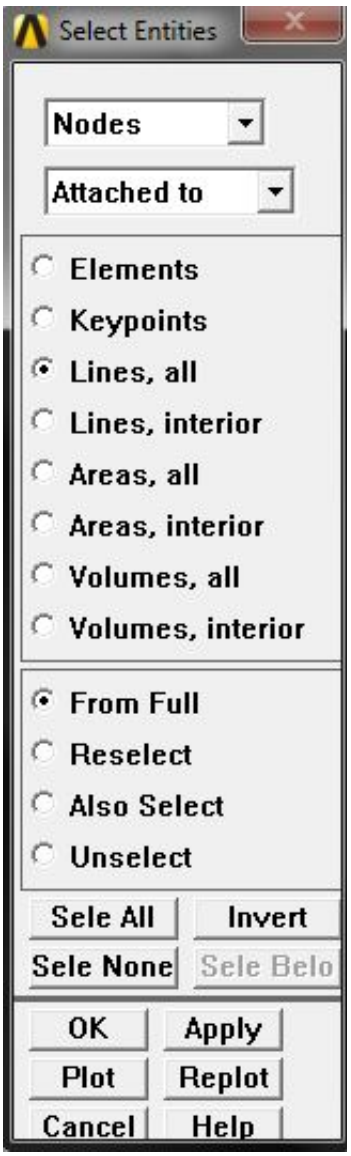


Now select the nodes attached to these lines.

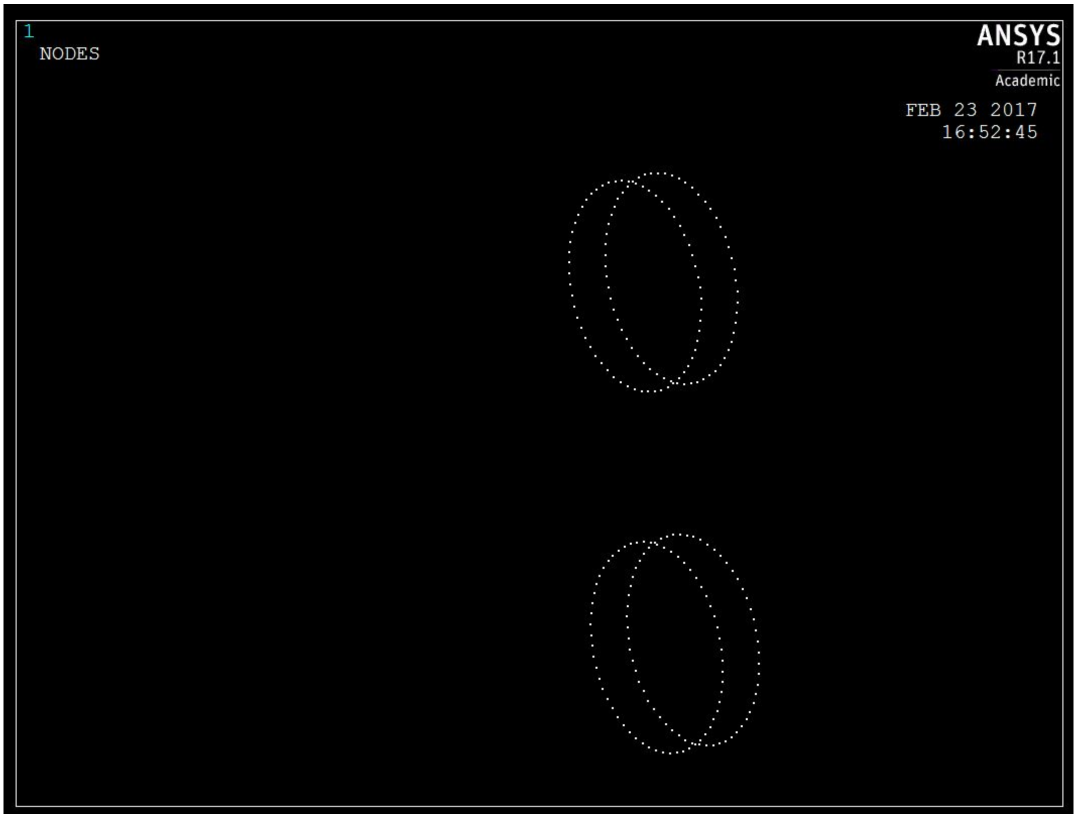
Utility Menu > Select > Entities



Choose **"Nodes"** in the first drop down menu and **"Attached to"** in the second drop down menu and select **"Line, all"** as the option Click OK.



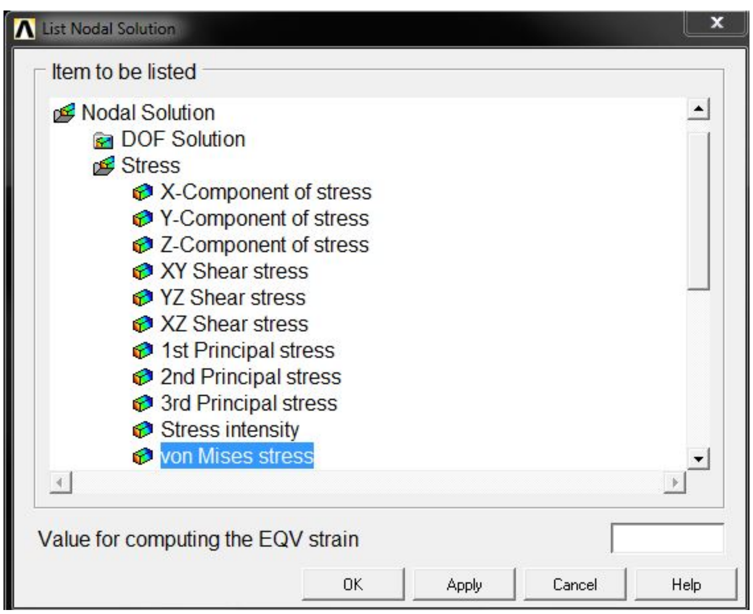
Nodes on these lines are selected.



3) Reading von Mises stress on these nodes.

Utility Menu > List > Results > Nodal Solution

Select Stress in the dialog box then Von Mises stress and click OK.



A text file will open containing the stress on the nodes that are currently selected.

At the bottom of the file Maximum and Minimum stress are listed with the node numbers

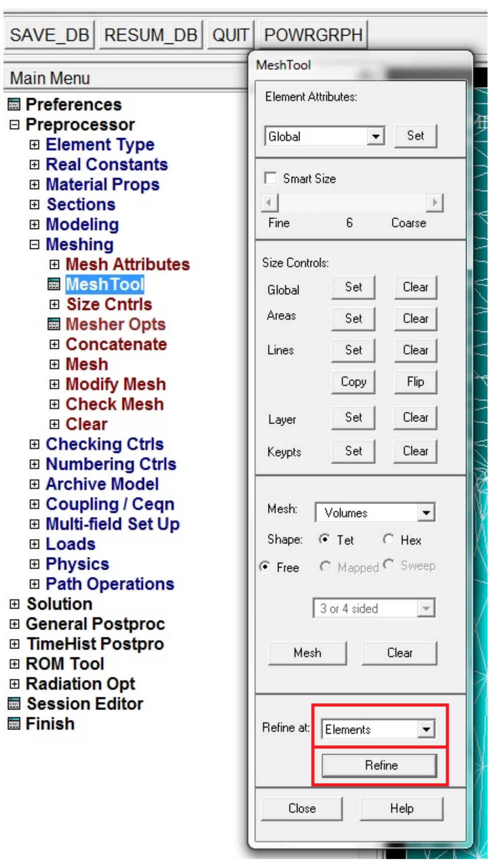
	S1	S2	S3	SINT	SEQU
MINIMUM VALUES					
NODE	1592	1592	482	1629	1629
VALUE	-1391.4	-20795.	-37902.	1094.3	984.69
MAXIMUM VALUES					
NODE	18	24	20	482	482
VALUE	37057.	25514.	2533.3	41792.	$\boxed{39416.}$ Maximum Von Mises Stress

Refining the mesh:

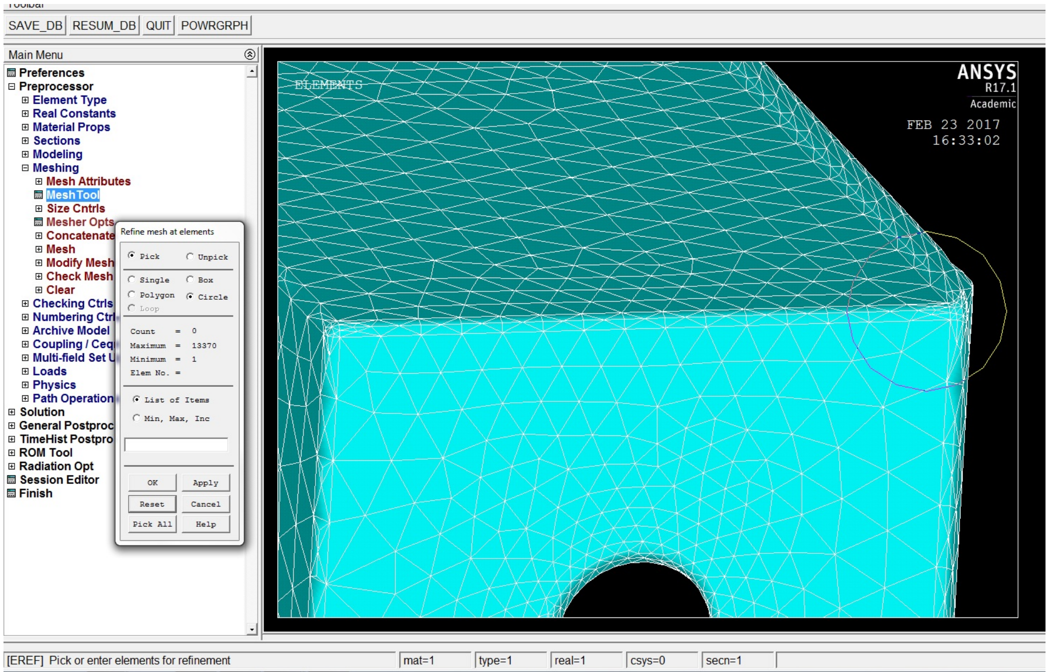
Since the stress singularity occurs at a particular region on the model it is not wise to re-mesh the whole geometry with finer elements. Local refinement of the mesh would help in obtaining fine elements near the singularity point with considerable increase in the number of elements.

Main Menu > Pre Processor > Meshing > Mesh Tool

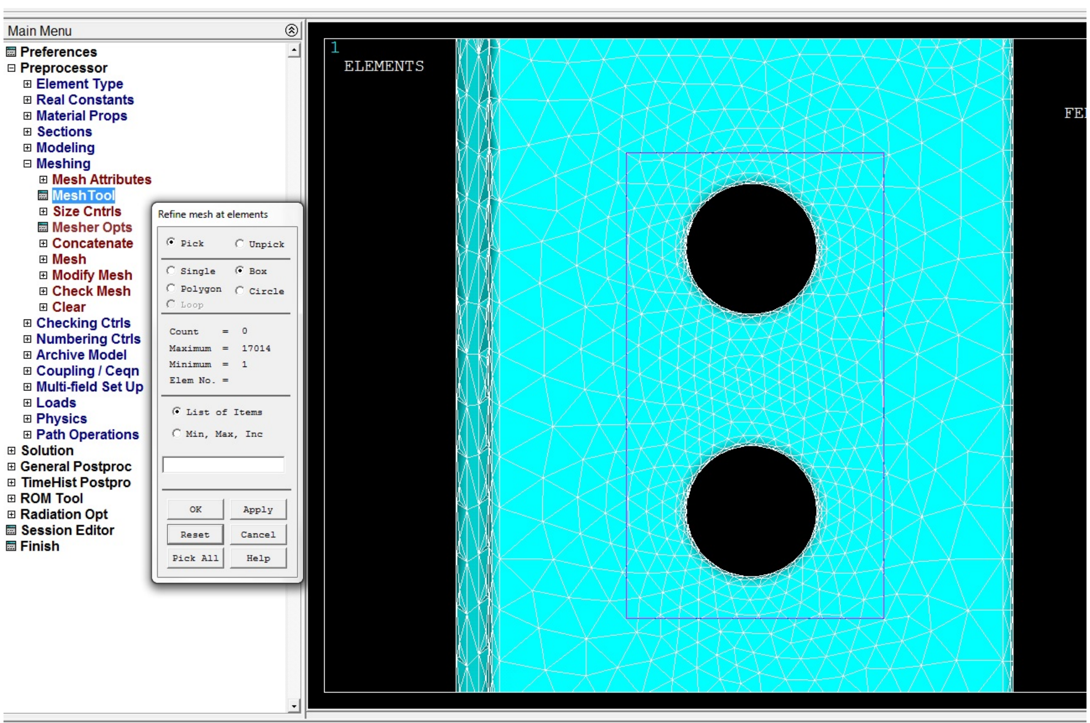
Make sure the drop down (Refine at) at the bottom reflects elements and click Refine.



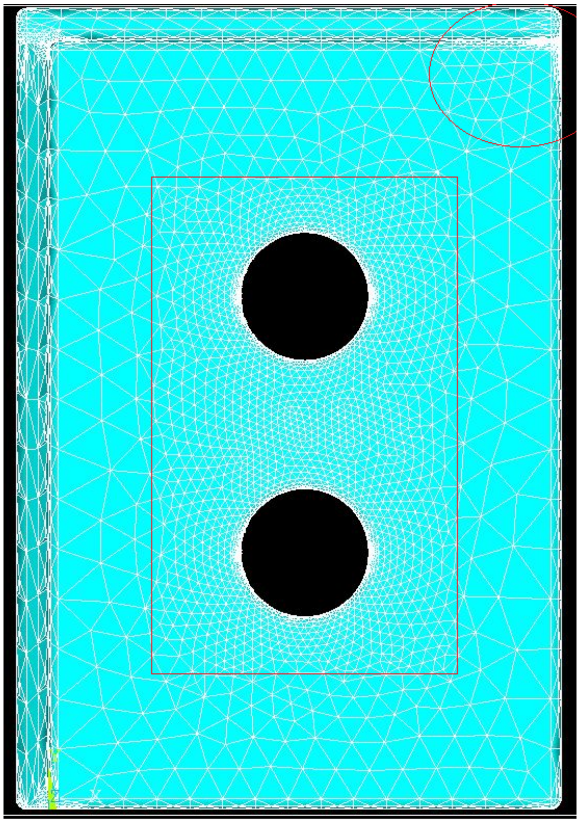
In pick option select the picking option as circle and select the elements at the top right corner as shown in the image. Click Ok



Again select the refine option from the mesh tools dialog box and select the elements around the circle either by a choosing a box option or by circle option. Click OK



Refined model is shown below:



Solve the model again

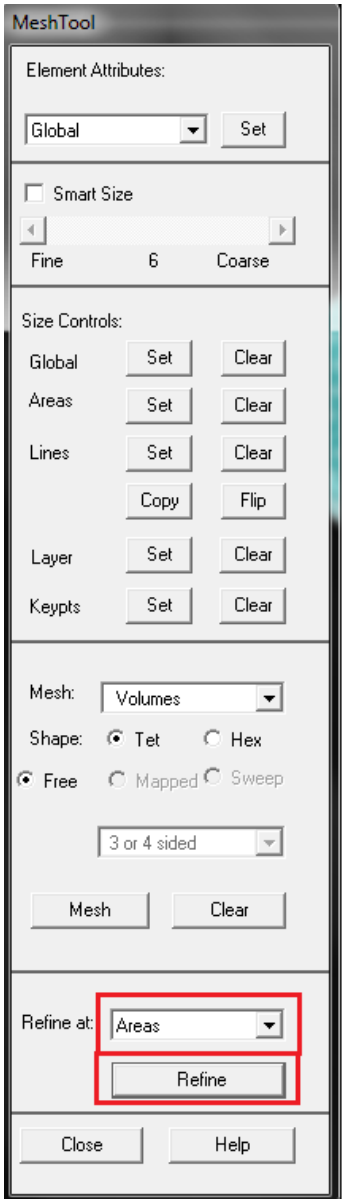
Main Menu > Solution > Solve > Current LS

Click OK.

Read the results by following the steps from the Post Processing section for different mesh in order to perform the convergence study.

Note: Since the elements are selected arbitrarily around the area of interest it is not possible to replicate the same exact results for a particular trial with same set of loading conditions and refinement condition.

Sometimes mesh refinement by selecting elements might give an error and no refinement is allowed. Under such circumstances mesh refinement could be carried out by selecting the refine at **“Area”** in Mesh tool drop down menu at the bottom of mesh tool and clicking **“Refine”** to select the Area where the refinement is required. You may also click on the “set” bottom on the “size control” in “mesh tool” to enter the maximum element size on an area.



Studying Convergence over a path

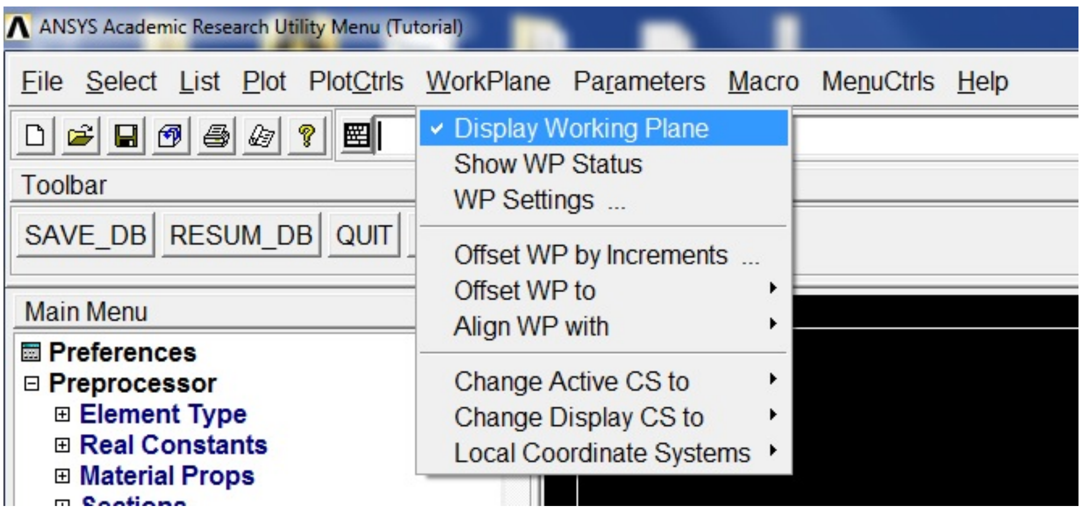
Convergence of stress could be studied over a path. Path operations option in the general postprocessor helps in plotting the variation of stress along the path and this variation when compared with different mesh sizes would help in studying the convergence.

Path is defined by a set of nodes in ANSYS. When the desired path of study is along an edge of the geometry there won't be any issue to define a path or study convergence as for each mesh and its refinement there would be nodes available along the edge to define the path and study convergence. When it comes to study variation of stress along a path which is not on an edge the position of nodes varies with each remeshing and could result in plotting the variation of stress along a different location with each remeshing. In order to overcome this, the volume of the model should be divided along the path of interest and glued back in order to have nodes along that particular path for each refinement.

Here we will learn how to create a path on the model and proceed towards convergence study. The desired path is from the left end of the fillet on lower hole to the side-wall of the model. The volume is divided into two pieces and glued back in order to create a line along which the path will be determined. There are various ways for dividing a volume into smaller volumes, however we only cover one of those ways in this tutorial, i.e. dividing a volume by cutting it using workplane.

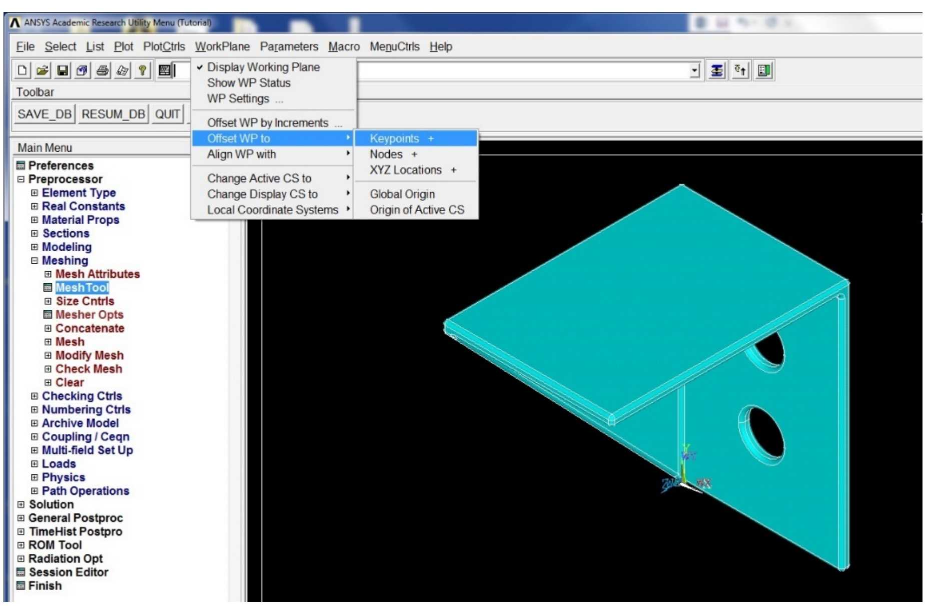
- Display workplane coordinate system.

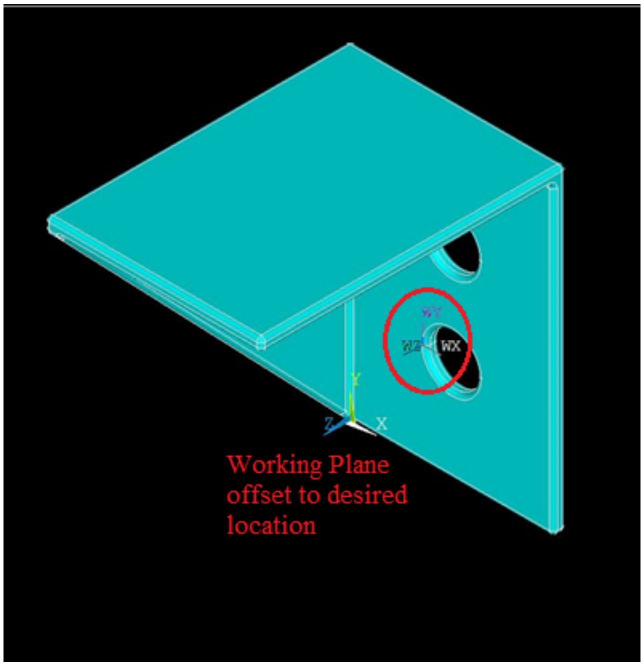
Utility Menu > Workplane > Display Working Plane



Move Workplane at the left end of the lower hole.

Utility Menu > WorkPlane > Offset WP to > Keypoints +

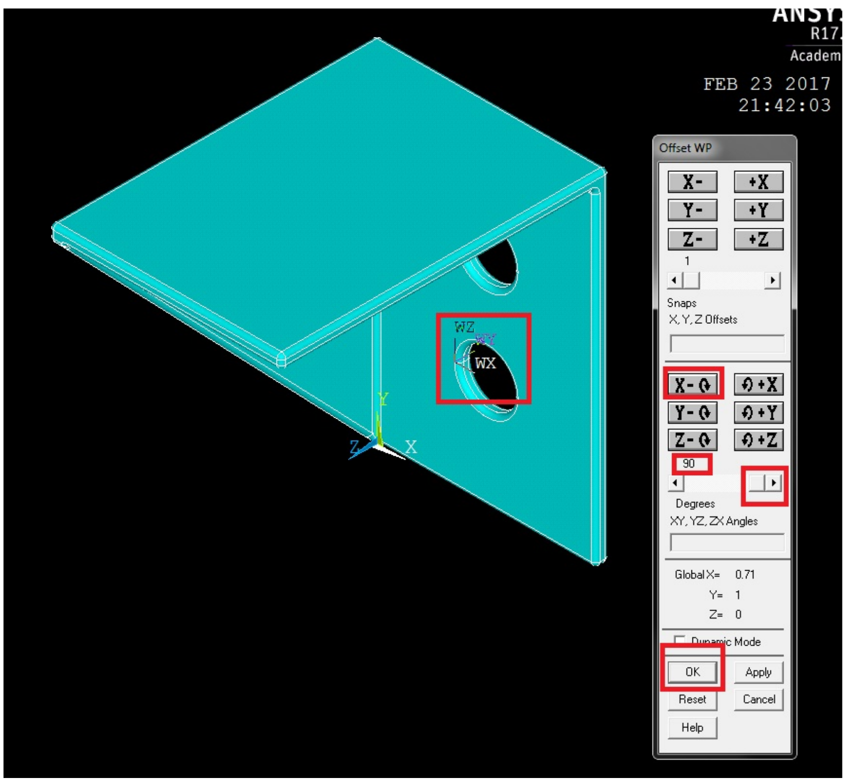




Now the working plane is at the desired location but it is parallel to the desired path line thus needs to be rotated in order to divide the volume by the working plane. Note that the default workplane is x-y.

Rotate the Working plane by 90 degrees about X axis.

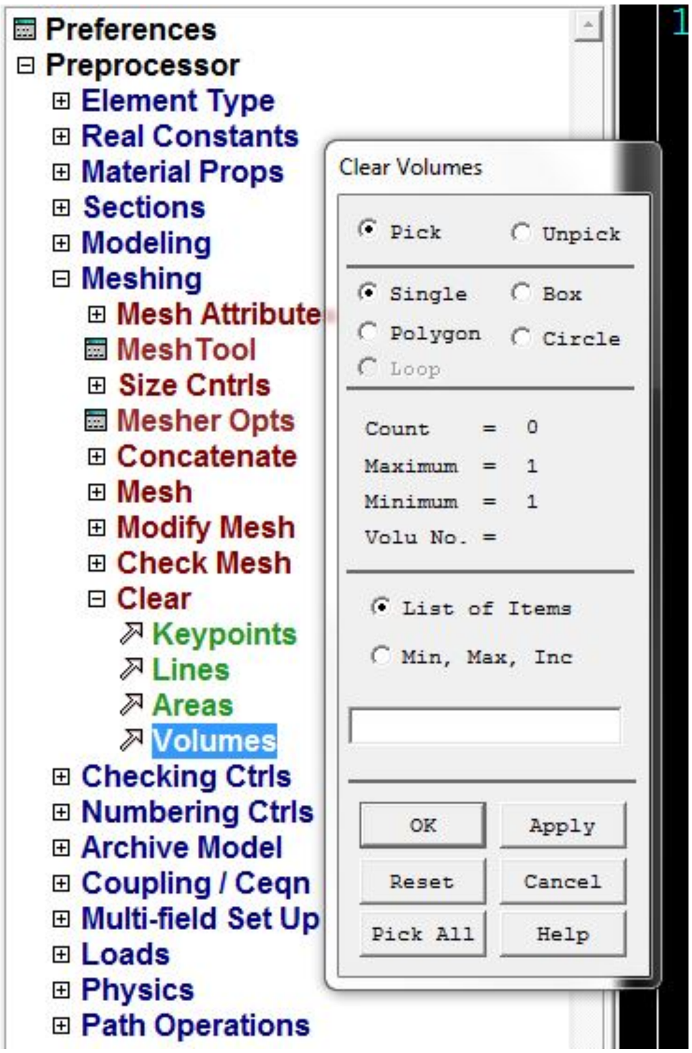
Move the slider to right till it says 90 then click -X or +X, Click OK.



Before dividing the volume by the working plane it is necessary to clear the mesh as a meshed volume cannot be divided. It makes sense, right?!

Main Menu > Preprocessor > Meshing > Clear > Volumes

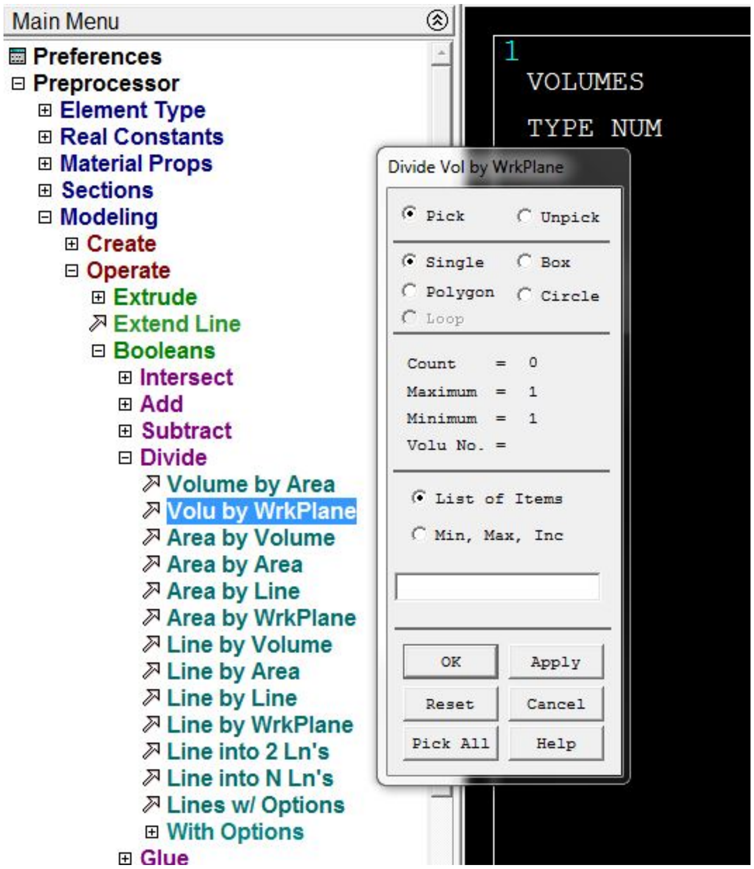
In the pick menu select Pick All. Click OK.



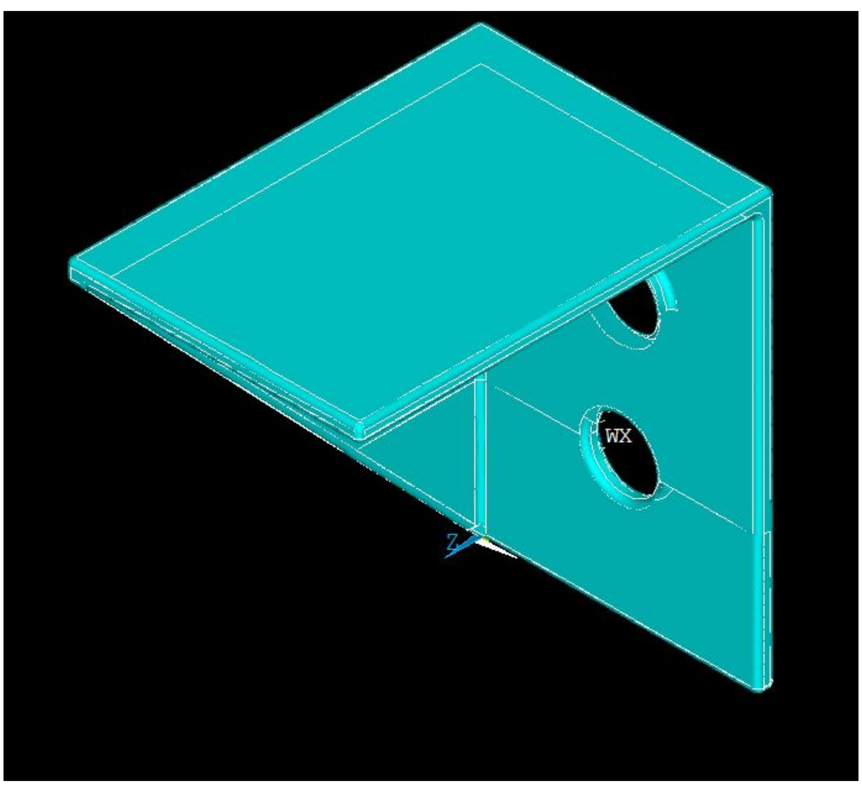
Dividing the volume by the working plane.

Main Menu > Preprocessor > Modelling > Operate > Booleans > Divide > Volu by WrkPlane

Select Pick All, click OK.



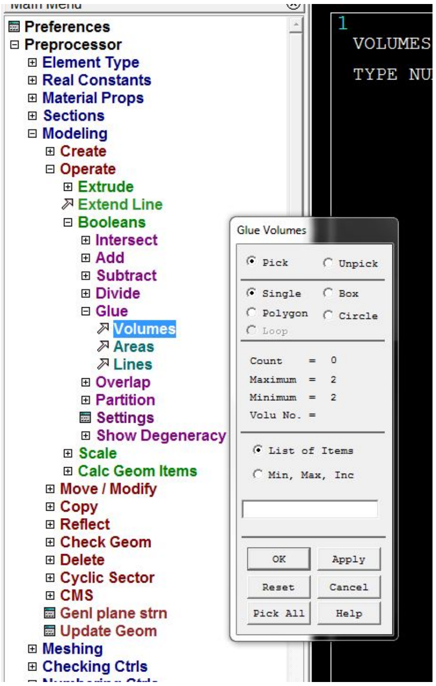
Divided Volumes.



Glue all volumes

Main Menu > Preprocessor > Modelling > Operate > Booleans > Glue > Volumes

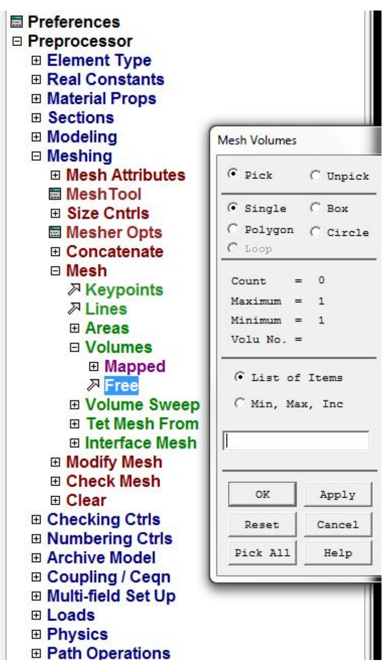
Select Pick All, Click OK.



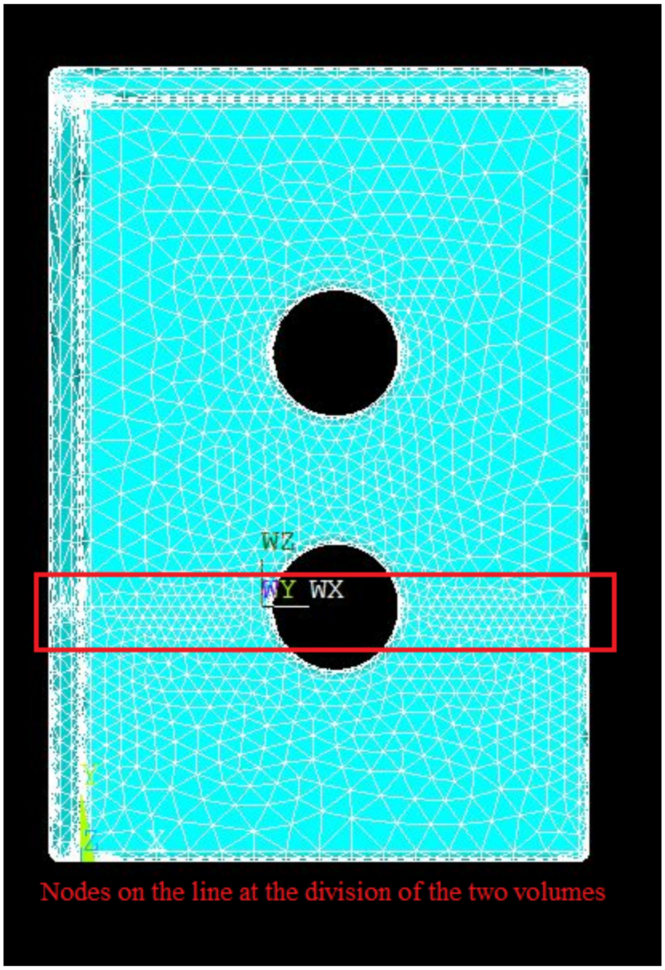
Meshing the volumes

Main Menu > Preprocessor > Meshing > Volumes > Free

Select Pick All, Click OK.

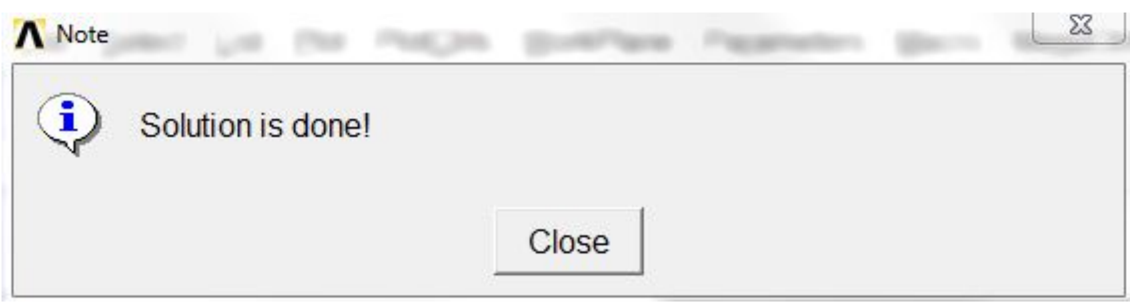


Meshed Volume with nodes along the desired path



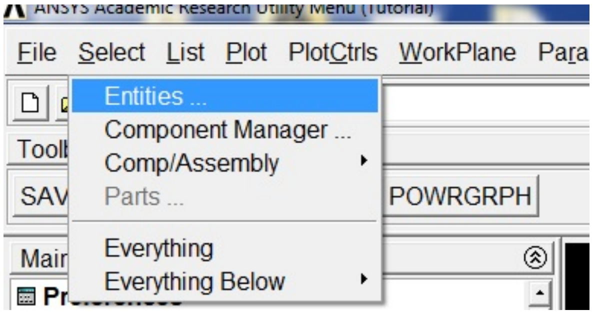
Since loads are already defined on the areas directly solving would generate the results. If the loads are not determined please check the Solution section above.

Main Menu > Solution > Solve > Current LS

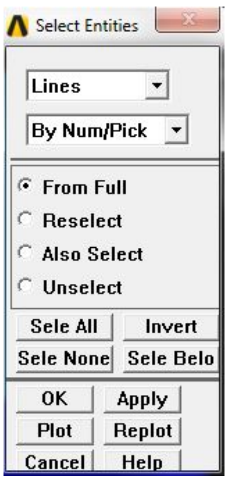


Select the line along the which the path is to be defined.

Utility Menu > Select > Entities

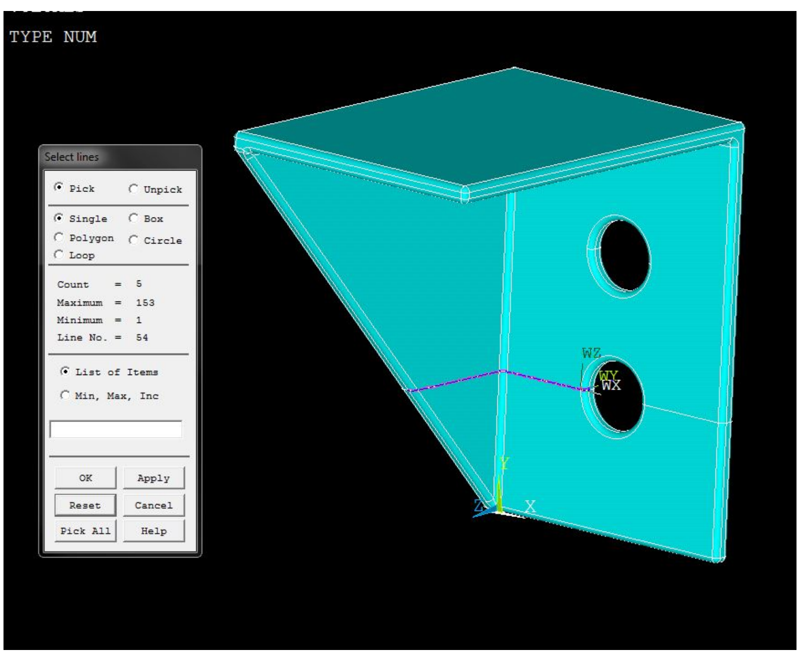


In the Select Entities dialog box choose "**Line**" in the first drop down menu and **"By Num/Pick"** in the second drop down menu as shown in the image below and click OK.



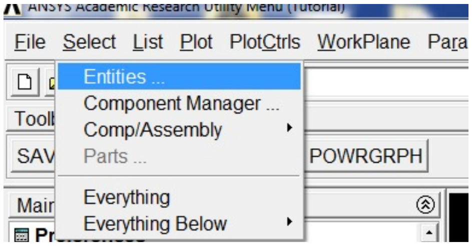
Select lines horizontal to the base starting at the outer surface of the fillet and moving towards left till the side wall by using the picking tool.

Click OK.

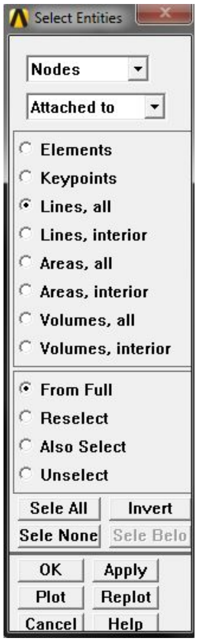


Now select the nodes attached to this lines.

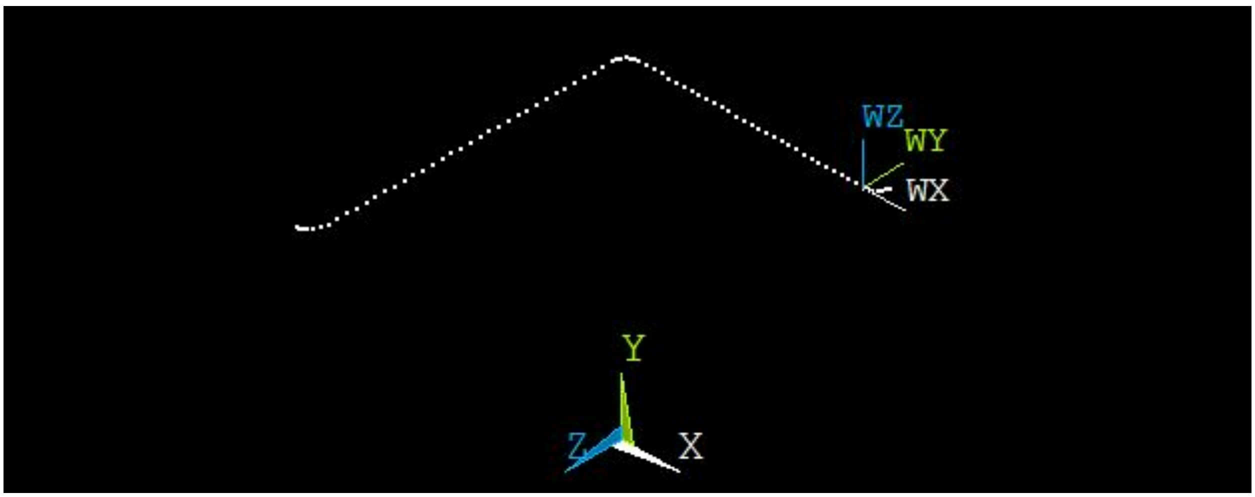
Utility Menu > Select > Entities



Choose **"Nodes"** in the first drop down menu and **"Attached to"** in the second drop down menu and select **"Line, all"** as the option Click OK.



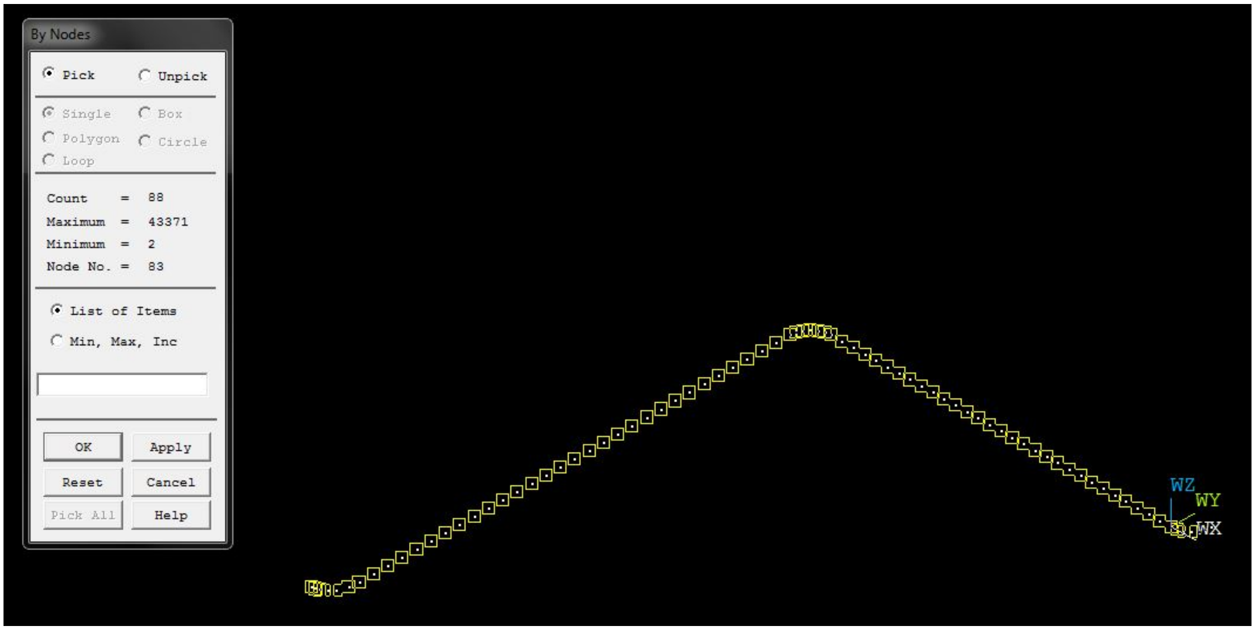
Nodes on the lines selected.



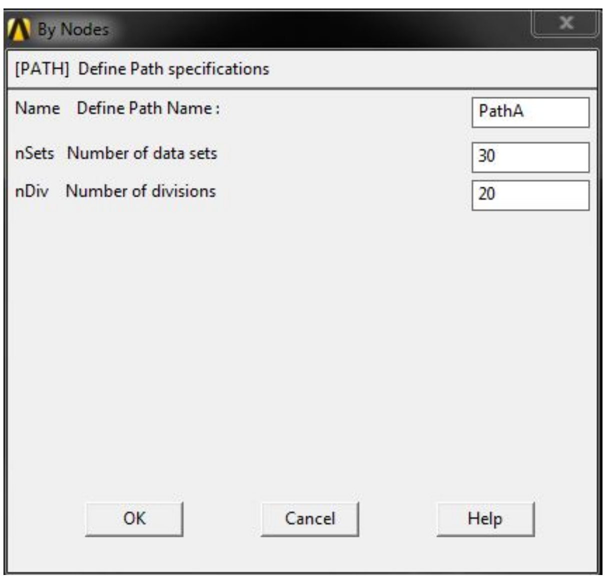
Define Path

Main Menu > General Postprocessor > Path Operations > Define Path > By Nodes

Pick all the nodes one by one in order to determine the sequence of nodes along the path.



Write “PathA” in the Name Define path name, click OK.

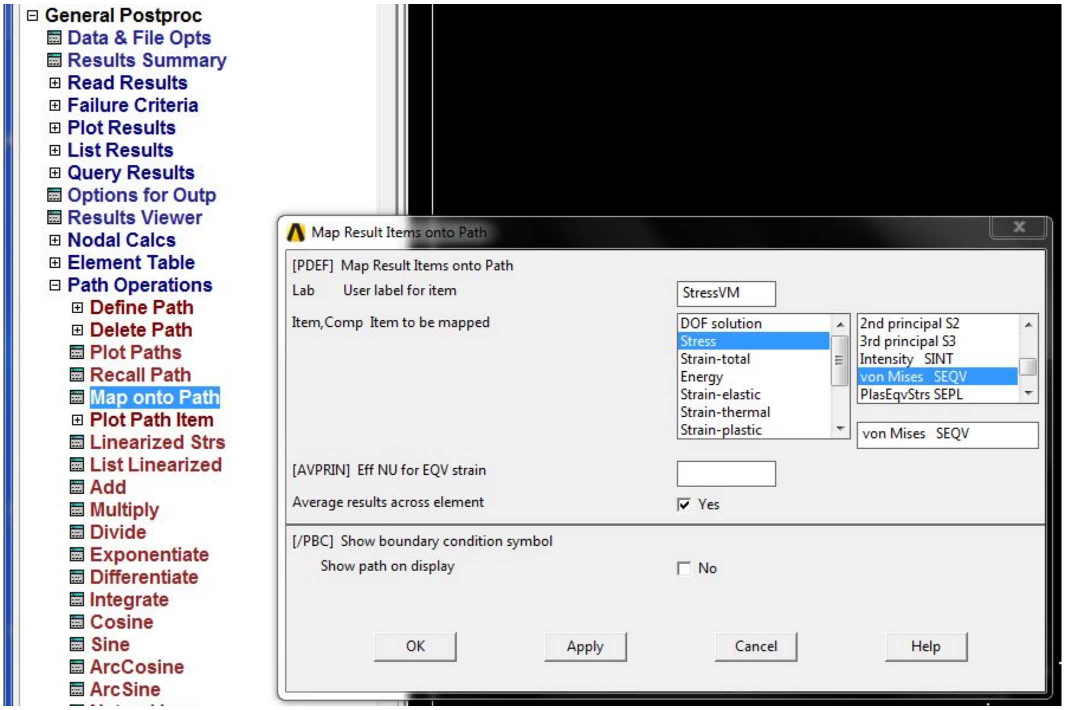


Selecting the results to map onto plot.

Main Menu > General Postprocessor > Path Operations > Map onto Path

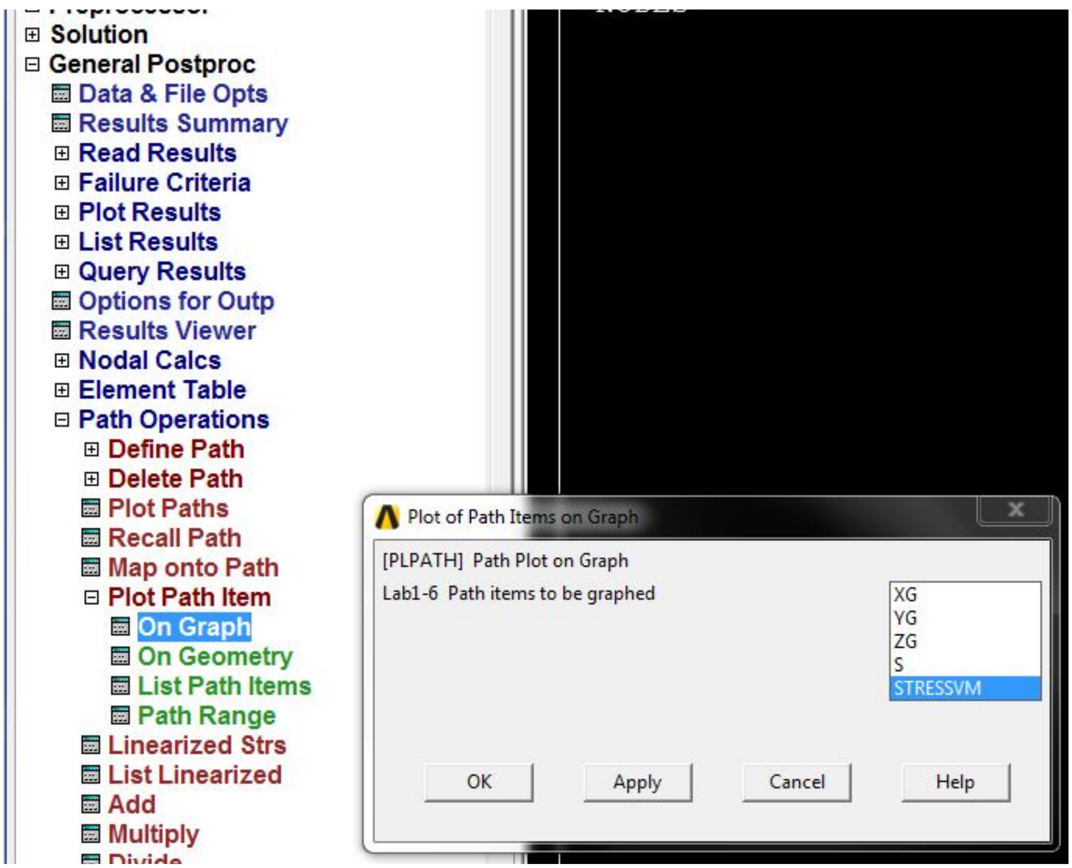
Write “StressVM” as User Label

Select “Stress” in the left menu and “von Mises” in the right menu. Click OK.

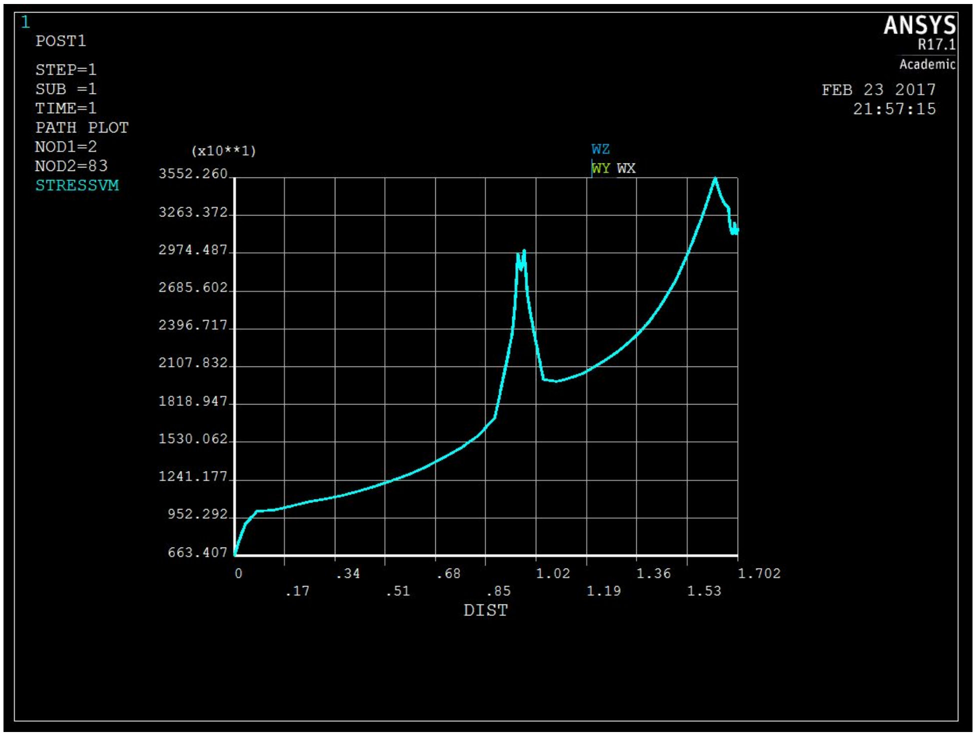


Main Menu > General Postprocessor > Path operations > Plot Path Item > On Graph

Select “STRESSVM”, click OK.

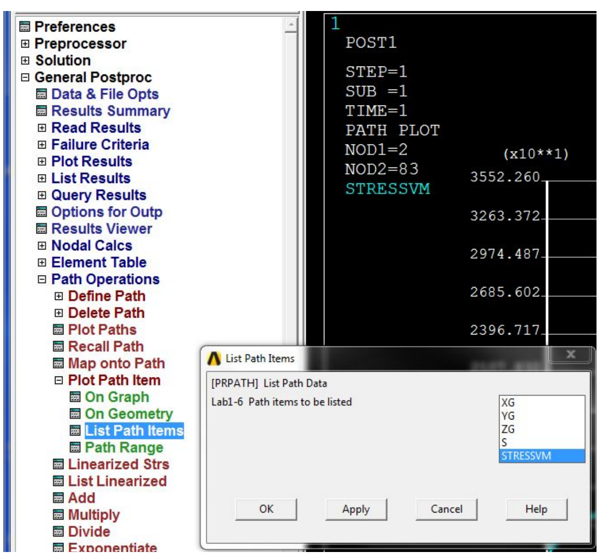


Variation of Von Mises Stress along the path.



To obtain the list file in order to redraw the plot in excel select List Path Items.

Main Menu > General Postprocessor > Path Operations > Plot Path Items > List Path Items



Click file and save the file to import the data in Microsoft Excel to plot the variation of stress along the path.

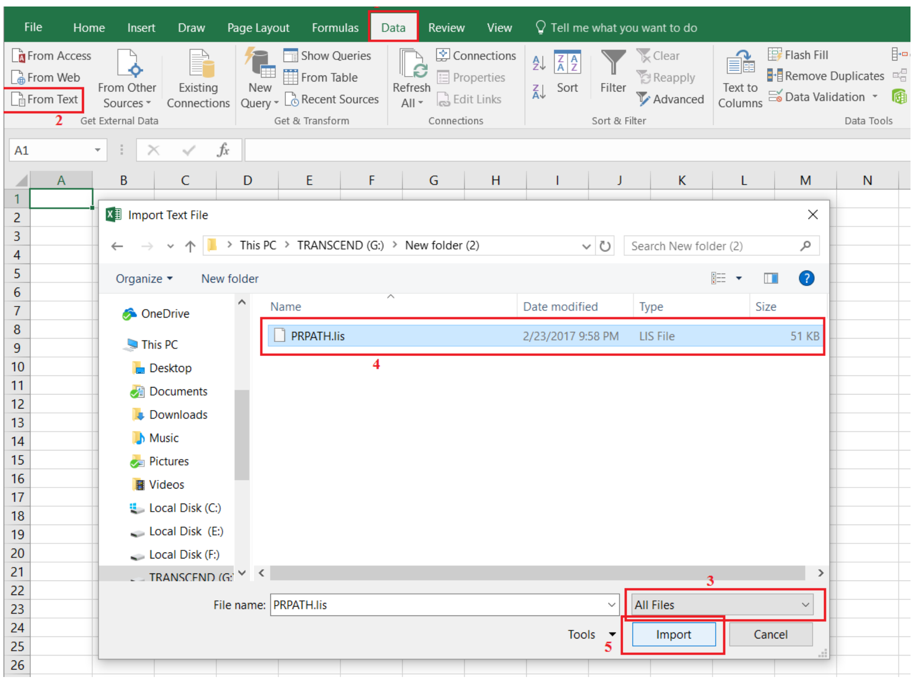
This procedure of reading results should be repeated for different meshes and plot each trial result as a series on a graph using Microsoft excel in order to study convergence there is no need to divide the volume for each mesh refinement.

Importing text data into Microsoft Excel.

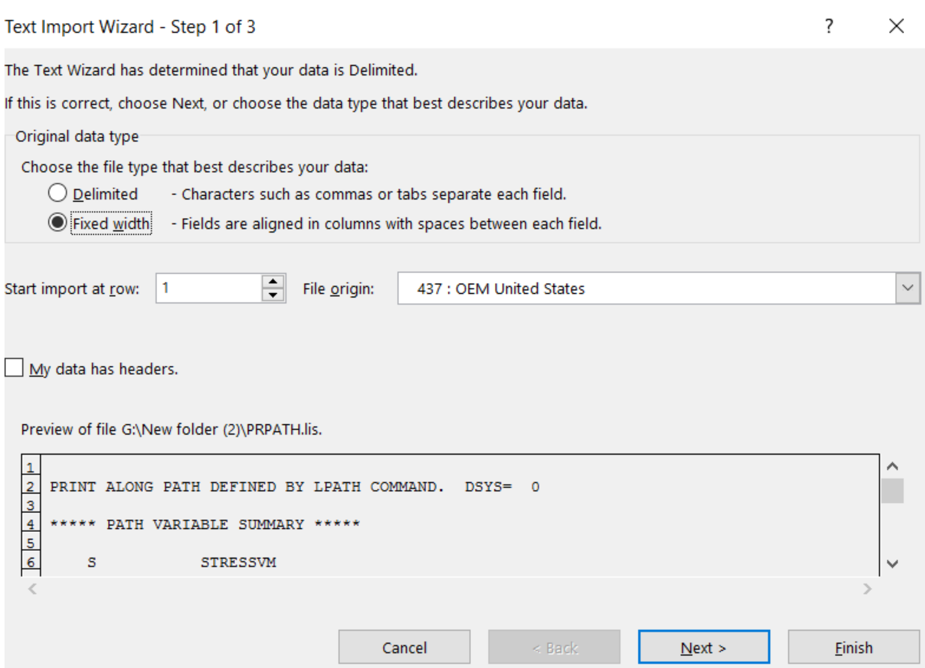
Open an Excel worksheet

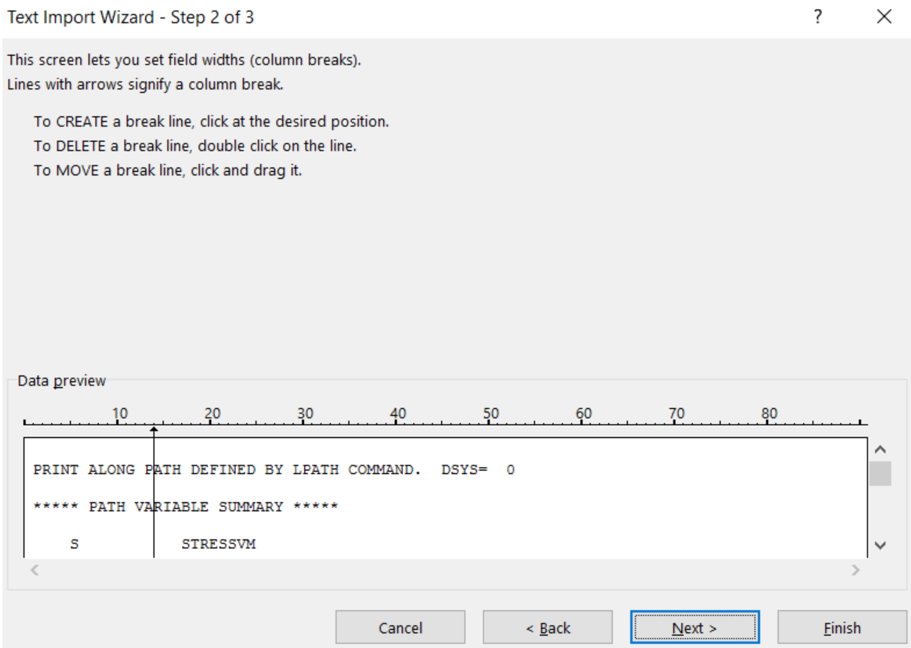
Utility Menu > Data > From Text

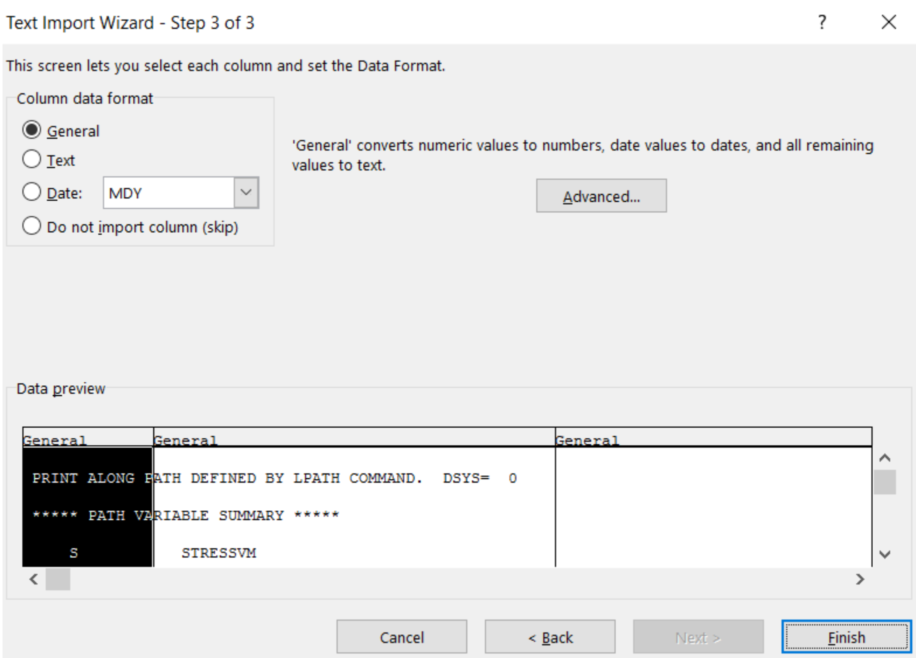
Browse to the file location, if file not visible change the display “txt files” to “All files”, select the required file and click Import

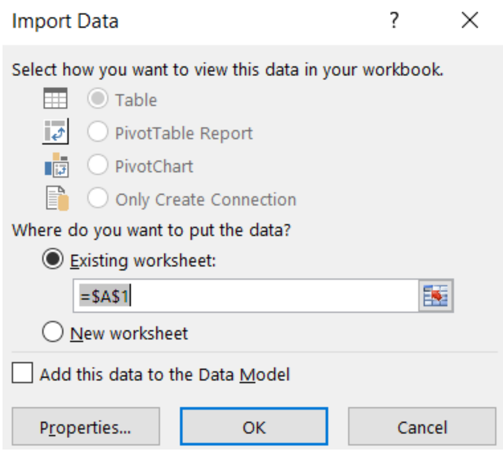


Follow the instructions as per the images shown below in order to import data in proper columns



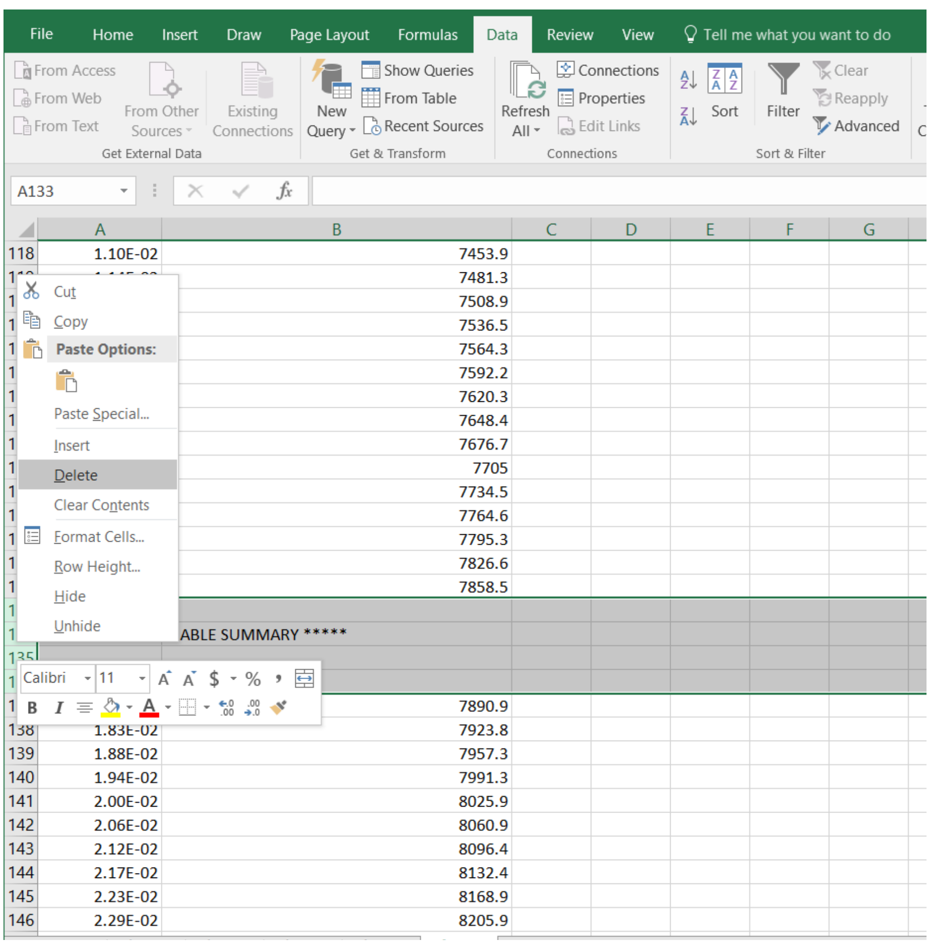

Text Import Wizard - Step 2 of 3
This screen lets you set field widths (column breaks).
Lines with arrows signify a column break.
To CREATE a break line, click at the desired position.
To DELETE a break line, double click on the line.
To MOVE a break line, click and drag it.
Data preview
10 20 30 40 50 60 70 80
PRINT ALONG PATH DEFINED BY LPATH COMMAND. DSYS= 0
***** PATH VARIABLE SUMMARY *****
S STRESSVM
Cancel
< Back
Next >
Finish


Text Import Wizard - Step 3 of 3
This screen lets you select each column and set the Data Format.
Column data format
General
Text
Date: MDY
Do not import column (skip)
'General' converts numeric values to numbers, date values to dates, and all remaining values to text.
Advanced...
Data preview
General General General
PRINT ALONG PATH DEFINED BY LPATH COMMAND. DSYS= 0
***** PATH VARIABLE SUMMARY *****
S STRESSVM
Cancel
< Back
Next >
Finish



After successfully completing above procedure the data gets imported in respective columns

But there would be unwanted additional rows in between which should be deleted in order to get the plot of the variation of stress along the path.



Some good reading material on stress singularities can be accessed through following references:

1) Stress singularities, stress concentrations and mesh convergence: http://www.acin.net/2015/06/02/stress-singularities-stress-concentrations-and-mesh-convergence/

2) How to Identify and Resolve Singularities in the Model when Meshing: COMSOL Blog https://www.comsol.com/blogs/how-identify-resolve-singularities-model-meshing/

3) Singularities in Finite Element Models: Dealing with Red Spots: COMSOL Blog https://www.comsol.com/blogs/singularities-in-finite-element-models-dealing-with-red-spots/

References

Acin, M. (2015, June 2). *Stress singularities, stress concentrations and mesh convergence*. Retrieved from http://www.acin.net/2015/06/02/stress-singularities-stress-concentrations-and-mesh-convergence/

Frei, W. (2013, October 29). *COMSOL Blog*. Retrieved from https://www.comsol.com/blogs/how-identify-resolve-singularities-model-meshing/