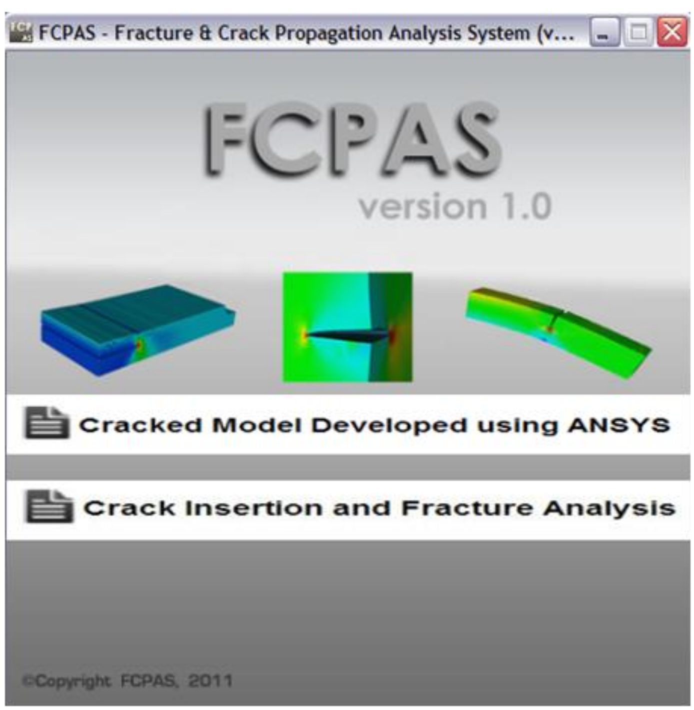
F C P A S

Fracture and Crack Propagation Analysis System

Version 1.0

Software & Tutorial Document

March 2011



Tutorial Document for Fracture Analysis with FCPAS

Introduction

In this tutorial, some basic linear elastic fracture mechanics examples are included that demonstrate usage of FCPAS (**F**racture and **C**rack **P**ropagation **A**nalysis **S**ystem) to solve three-dimensional fracture problems. The generation of models, meshing and application of boundary conditions and loads are done using the commercially available finite element software, ANSYS™. Then, a converter program is used to convert the finite element model information, boundary conditions and loads data from ANSYS™ [1] format into FCPAS format. Finally, FCPAS Solver is run to solve the crack problem and determine the stress intensity factors for the problem of interest. The following chart shows the general algorithm and file structure of process (Figure T.1).

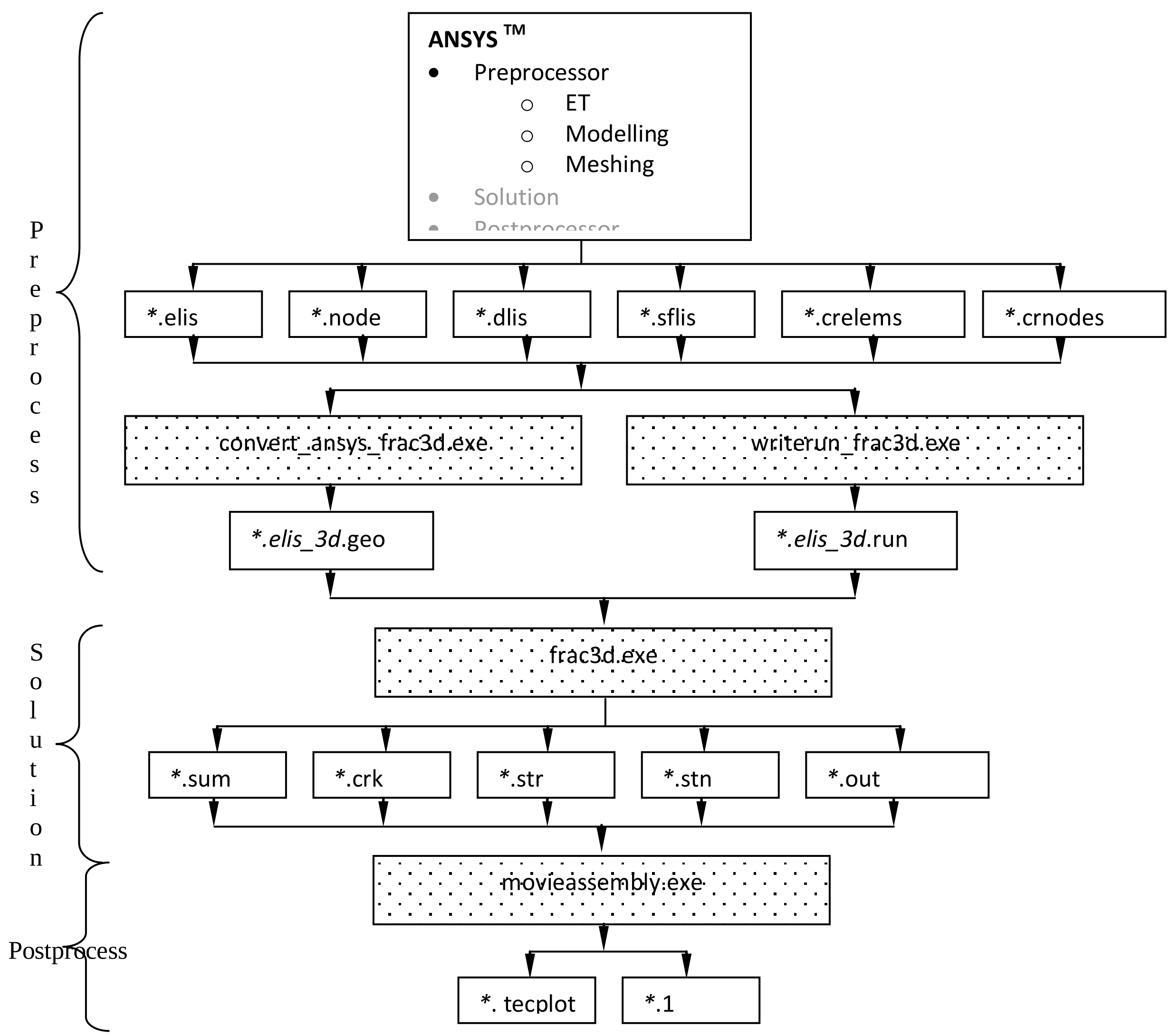


Figure T.1 General flow of the analysis process.

The steps from ANSYS™ model generation to solving the crack problem in FRAC3D are explained in detail. The problems included in this tutorial are:

1. Two-dimensional mode-I central crack in a large isotropic medium,
2. Mode-I crack in a Compact Tension, C(T) test specimen,
3. Mode-I central crack in a finite-thickness plate.

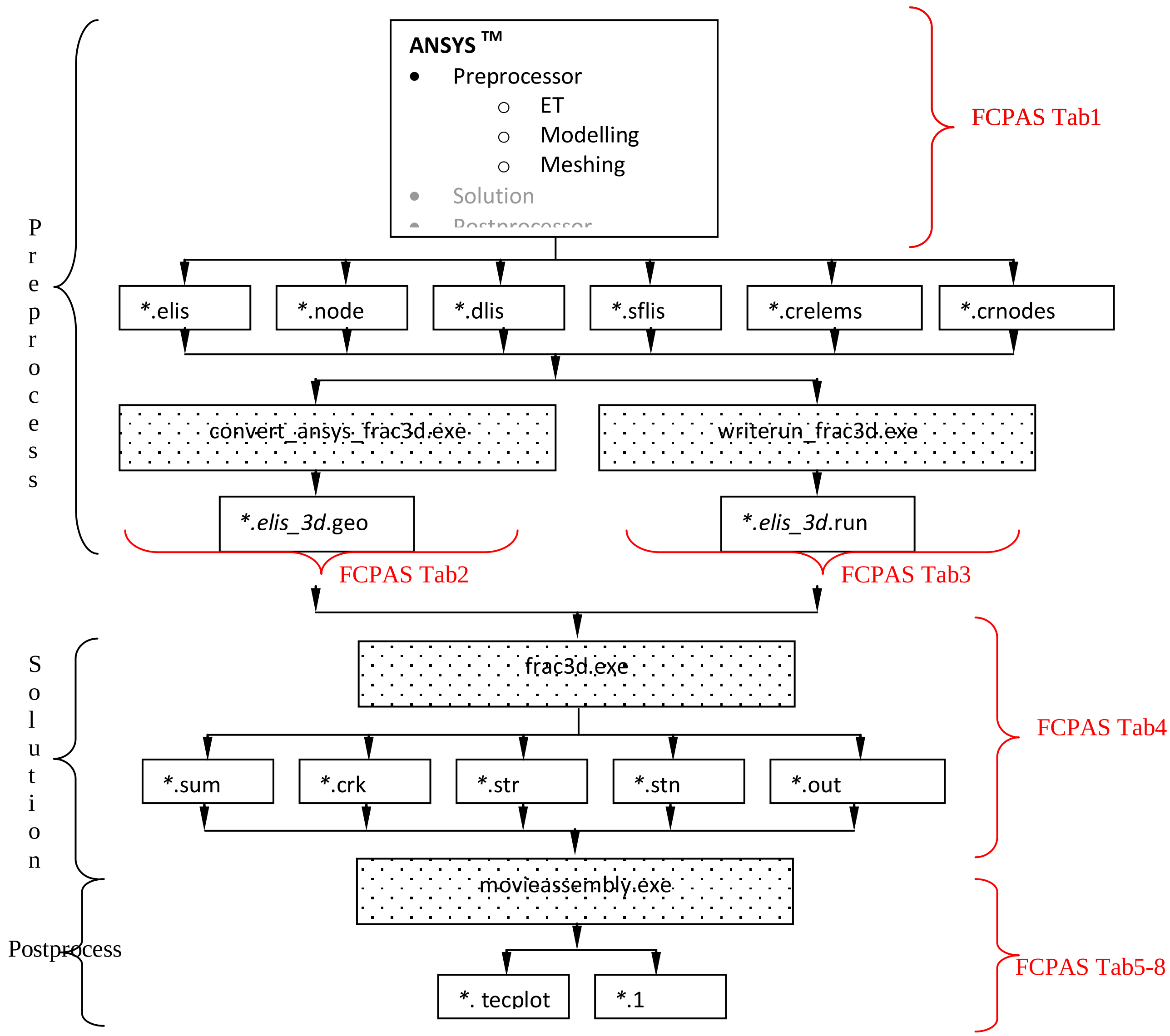
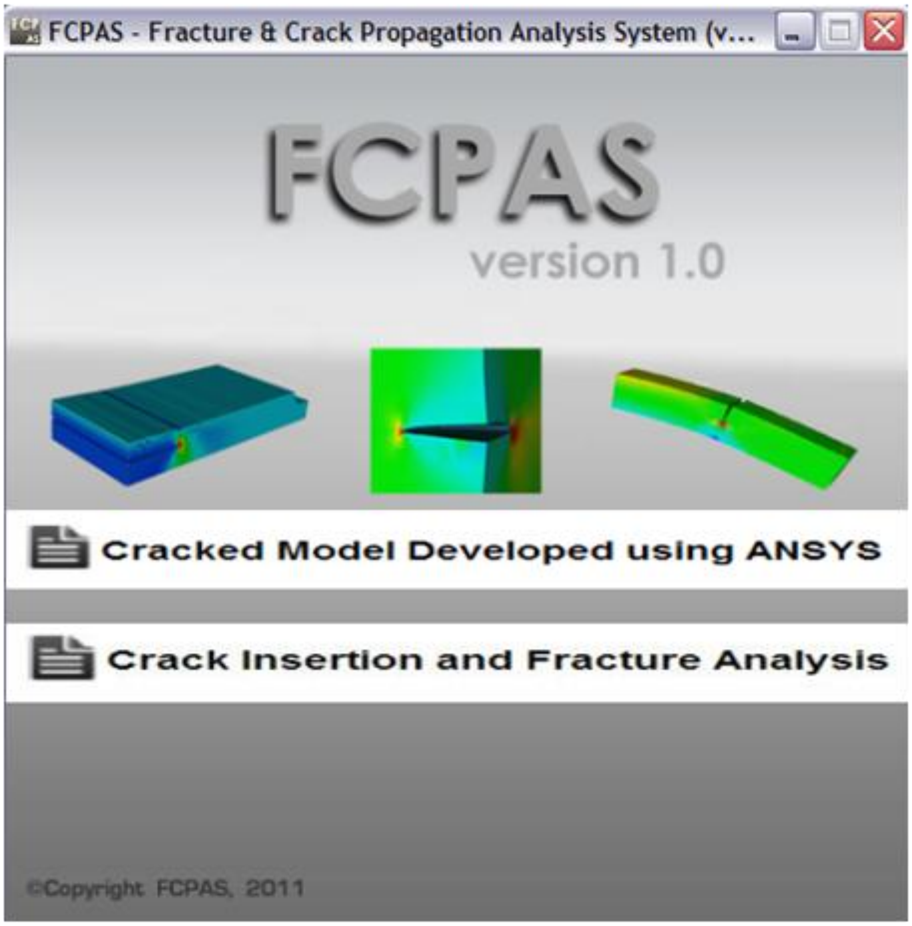


Figure T.2 General algorithm of FCPAS

FCPAS's GUI (Graphical User Interface) allows the user to follows the process in Figure T.3 in an orderly and user-friendly manner. This is the first version of the software and currently linear fracture analysis is available.



Cracked Model Developed using ANSYS : If you click this button, you can work with **Cracked Model Developed with ANSYS**.

Crack Insertion and Fracture Analysis : If you click this button, you can work with **Crack Insertion and Fracture Analysis.**

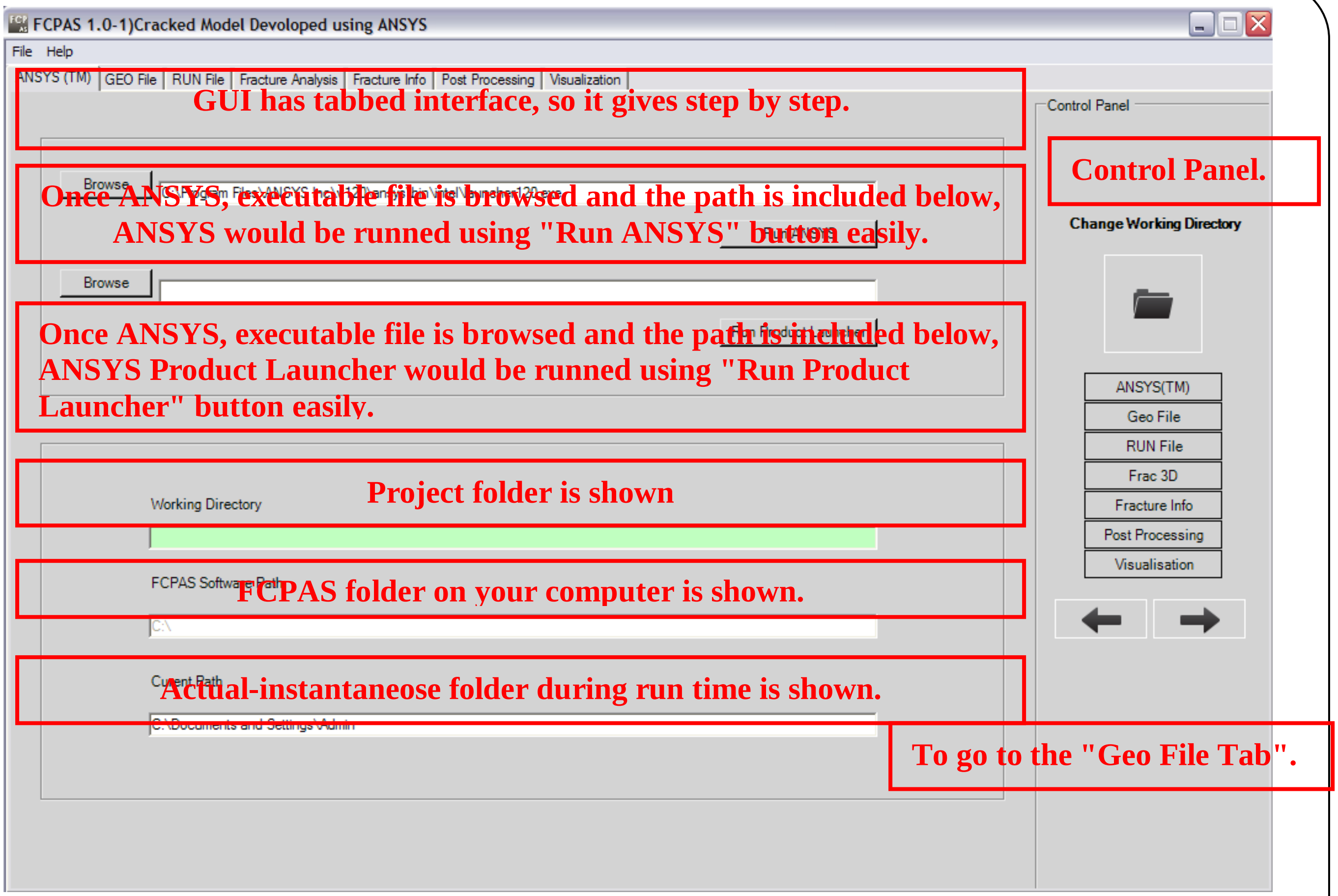


Figure T.3. FCPAS graphical user interface

When you click first button, the above form comes up.

EXAMPLE.1. Two-Dimensional Mode-I Central Crack in a Large Isotropic Plate

T.1.1. Problem Description

Consider the infinite domain in Figure T.4a containing a central crack and subjected to uniform tensile pressure loading perpendicular to the crack plane. We can model this problem as a plate in tension with a central crack as shown in Figure T.4b. Due to symmetry in the problem; only a quarter model is analyzed as shown in Figure T.4c. The plate is made of steel with Young's modulus E = 200 GPa and Poisson's ratio ν = 0.33. Let width to be $2w$ = 20 m, height is $2h$=20 m, a = 1 m and σ_0= 1 Pa. The objective is to compute the mode-I stress intensity factor (SIF).

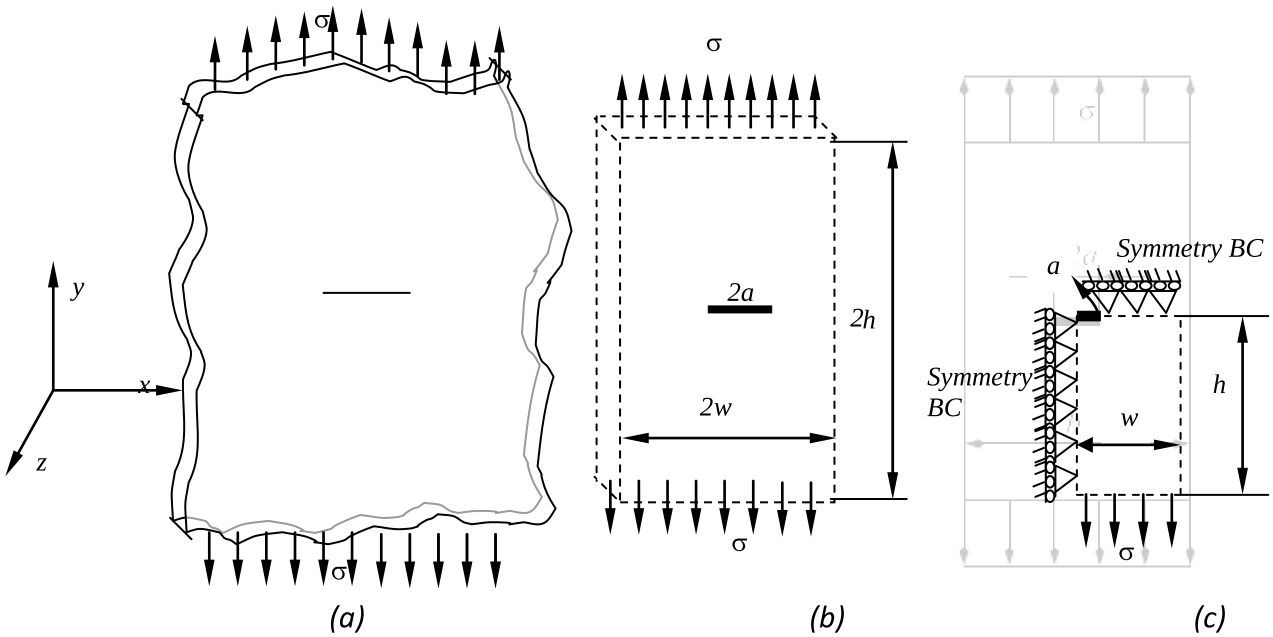


Figure T.4 Through-thickness crack in a large plate.

Note that for this problem, analytical solution is given by;

$K_1 = \sigma_0 \sqrt{\pi a}$, where σ_0 = Stress (1 Pa), *a*: Half of crack length (1 meter). Use of this solution yields K_I = 1.77 *Pa* $\sqrt{m}$.

T.1.2 Assumptions

- Linear elastic fracture mechanics (LEFM).
- Plane strain problem.

$$\varepsilon_{zz} = 0 = \frac{\partial w}{\partial z} = 0$$

$$\varepsilon_{zz}(x, y) = 0$$

In the ANSYS™ tab of the FCPAS, we browse “"C:\Program Files\ANSYS Inc\v120\ansys\bin\intel\launcher120.exe”.

The above directory location may change depending on the version of Ansys being used.

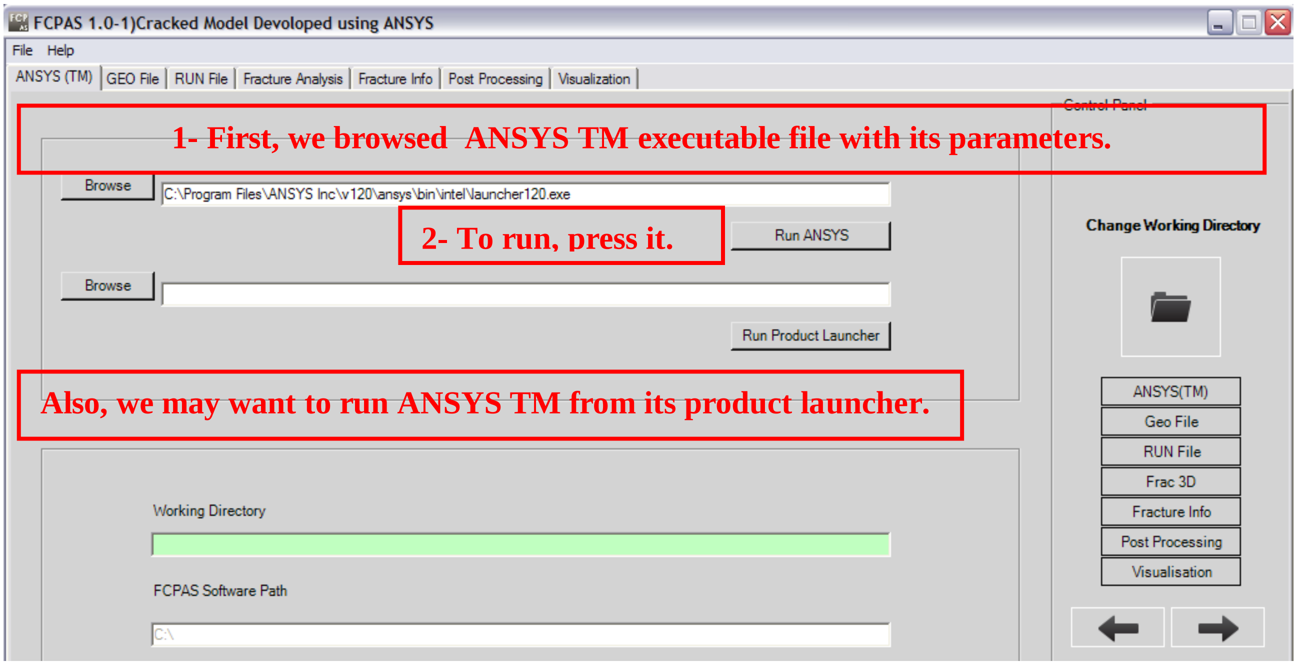


Figure T.5 ANSYS^{TM} tab of FCPAS.

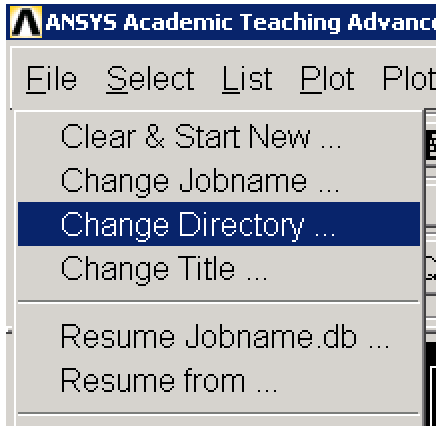
T.1.3 Generation of the Finite Element Model within ANSYS^{TM}

We will model this problem as a two-dimensional model plane strain model by taking into account the symmetries in horizontal and vertical directions. Also, in the out-of-plane direction, we will use one layer three-dimensional elements. To do this, we will first mesh the back face of the domain with area (2D) elements and extrude the mesh into the third direction. To do this, we will use PLANE82 and SOLID95 elements from the ANSYS^{TM} element library. Note that ANSYS^{TM} Help is very useful tool to identify and select the suitable elements for the problem of interest.

Preprocessing

Change Directory

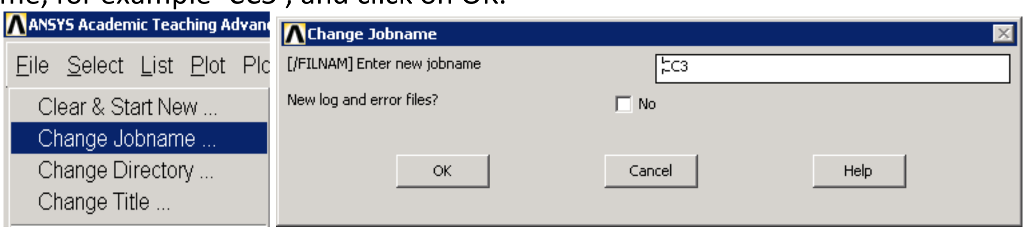
Before starting modeling, create a folder in which you would like to perform analyses & change directory to this folder.



Give the Job a Name

Utility Menu>File>Change Jobname ...

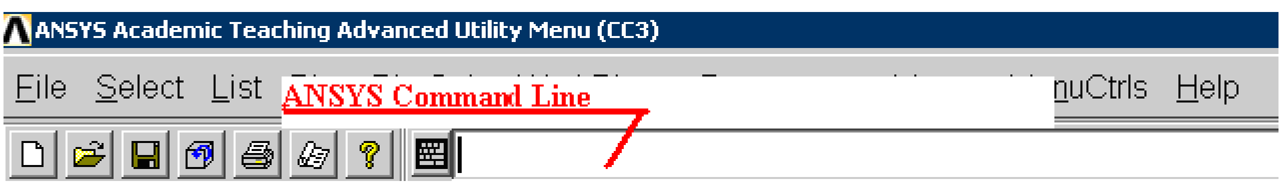
Enter a name, for example `*CC3*', and click on OK.



Define Element Type

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

This brings up the 'Element Types' window. Click on the Add... button. The 'Library of Element Types' window appears. Highlight Plane 82 `Plane-8node 82' and solid95 `Solid-20node 95'. Click on OK or in command line, use **(ET,1,82)**[1], **(ET,2,95).**

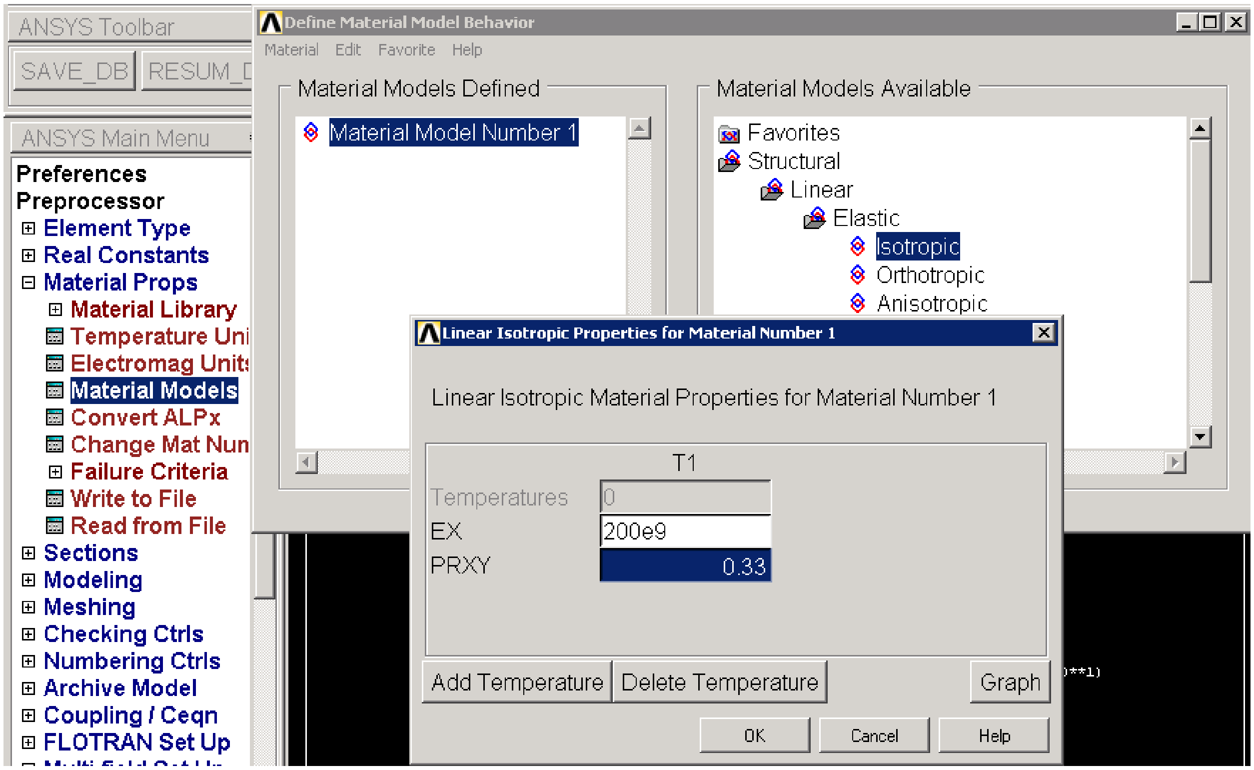


Define Material Properties

Main Menu>Preprocessor>Material Props>Material Models

On the right side of the `Define Material Model Behavior' window that opens, double click on `Structural', then `Linear', then `Elastic', finally `Isotropic'. Enter in values for the Young's modulus (EX = 200E9) and Poisson's ratio (PRXY = 0.33) of the plate material.

[1] PLANE82 element provides us both "plane strain or stress" options.

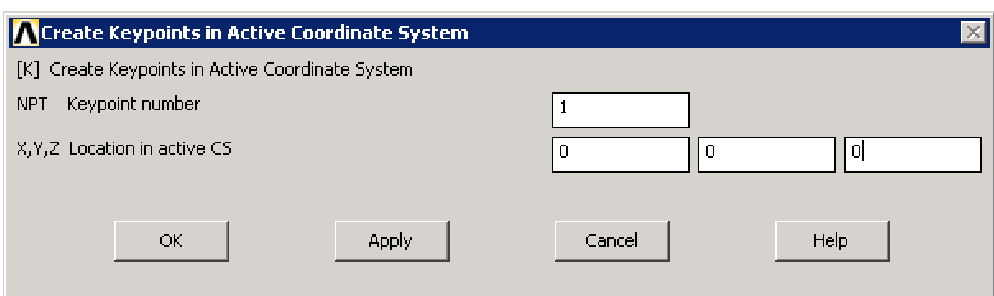


Define Keypoints

Main Menu>Preprocessor>Modeling>Create>Keypoints>In Active CS

We are going to create 5 keypoints given in the following table:

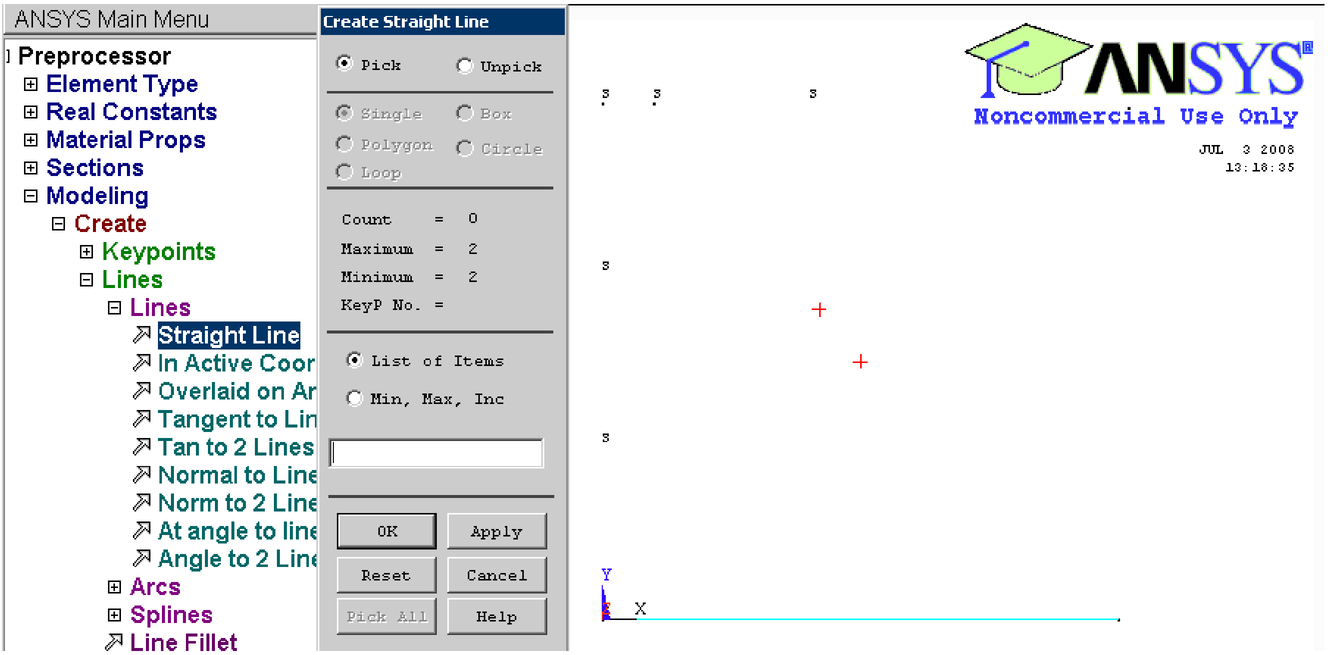
Keypoints	X [m]	Y [m]	Z [m]
1	0	0	0
2	10	0	0
3	10	10	0
4	1	10	0
5	0	10	0



Define Line Segments

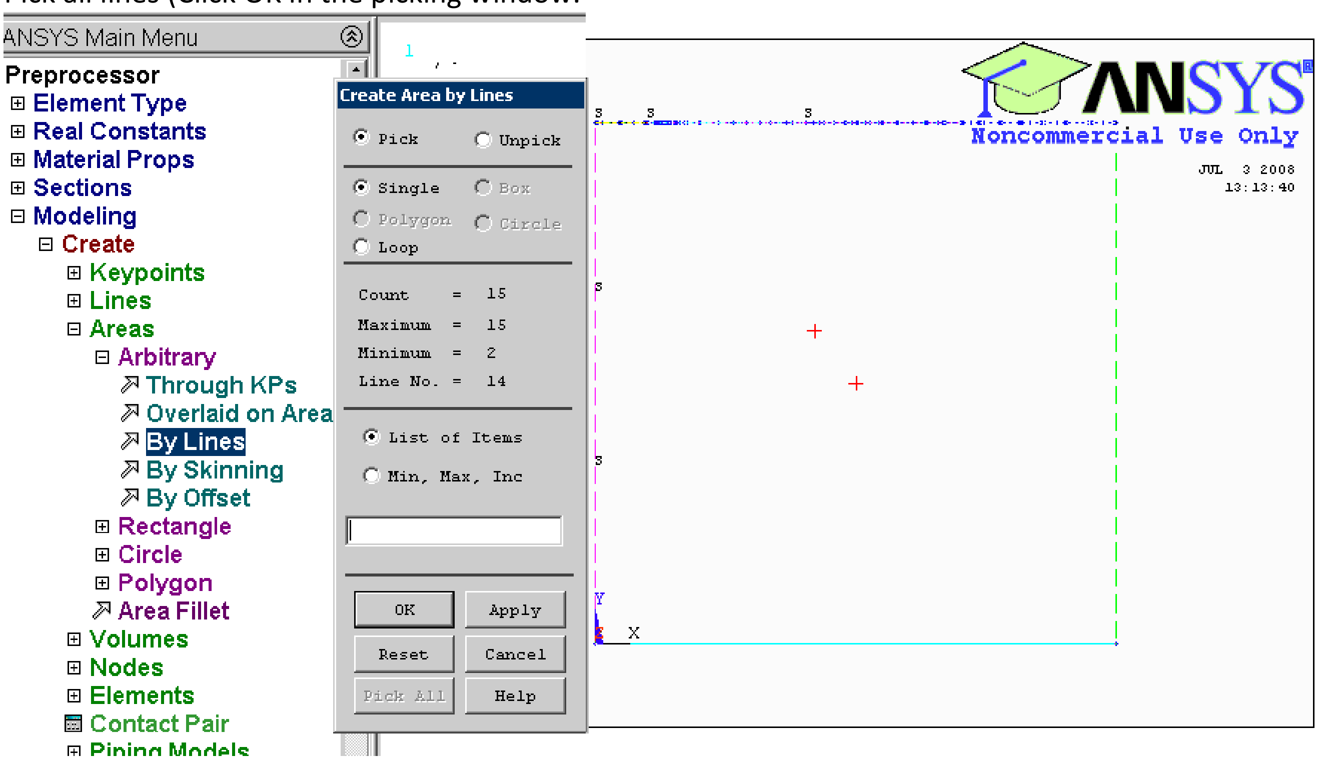
Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Straight Line

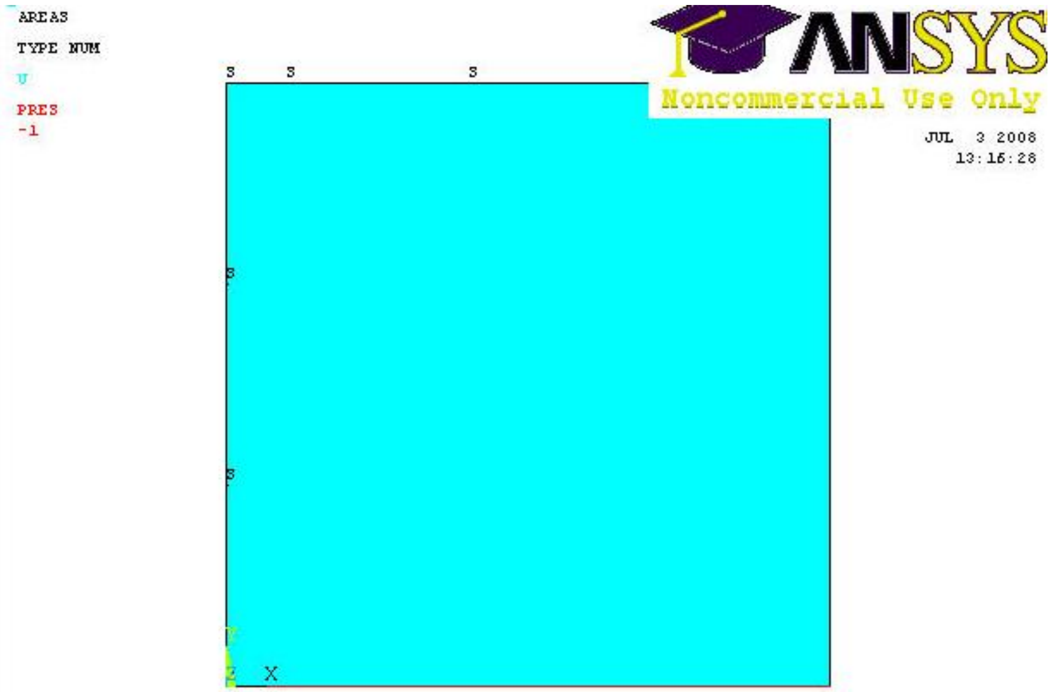
This is required to create the models boundary lines, successively like first 1 to 2, 2-3 and finally 5to 1.



Create the Area

Main Menu>Preprocessor>Modeling>Create>Areas>Arbitrary>By Lines
Pick all lines (Click OK in the picking window.



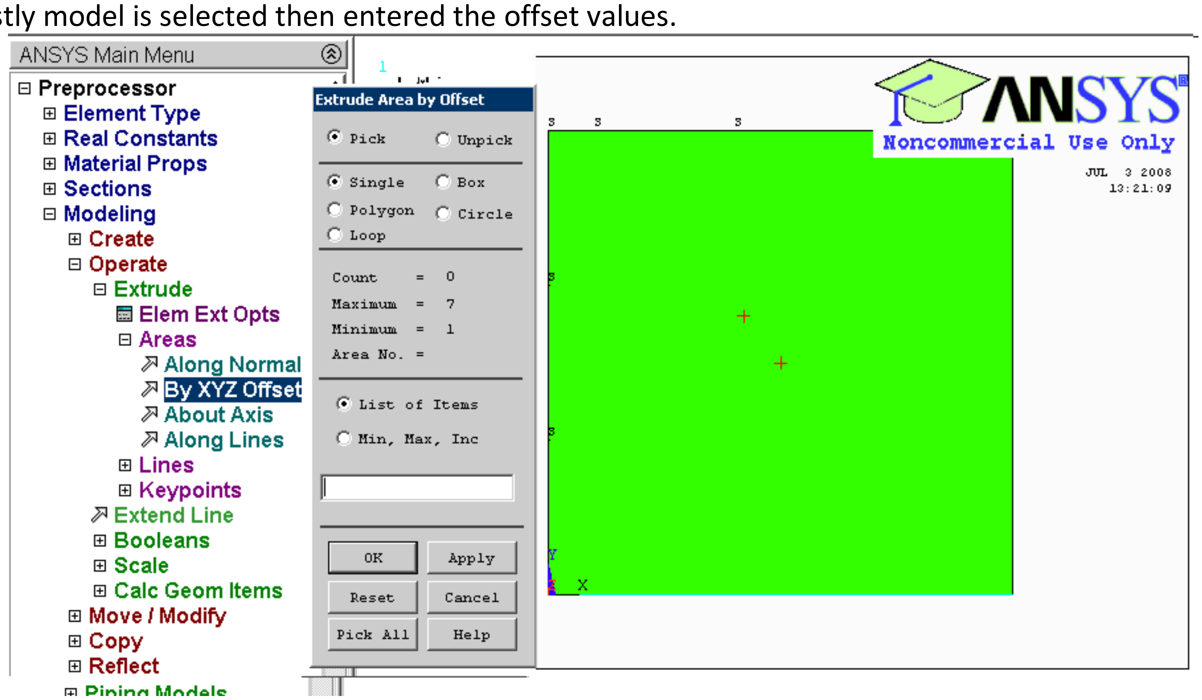


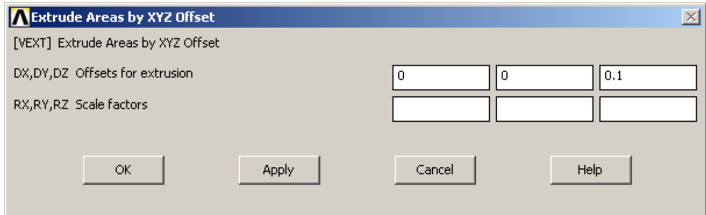
EXTRUSION

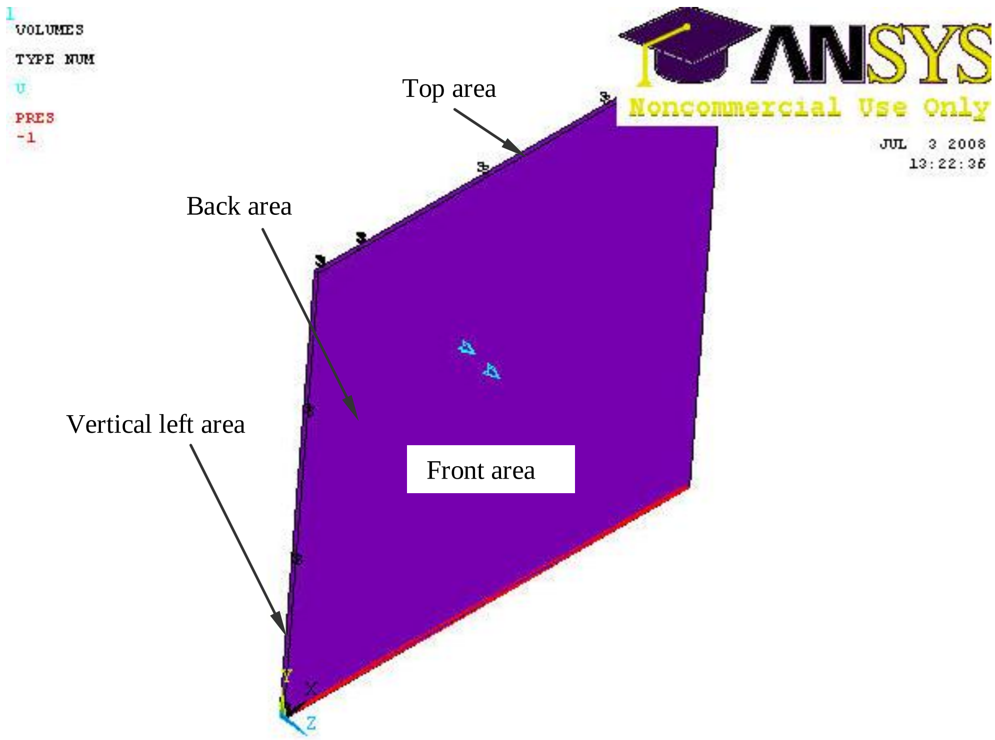
To create volume, we can use easily extrusion property by 0.1 unit through normal direction (*z* axes)

Main Menu>Preprocessor>Modeling>Operate>Extrude>Areas>By XYZ Offset

Firstly model is selected then entered the offset values.





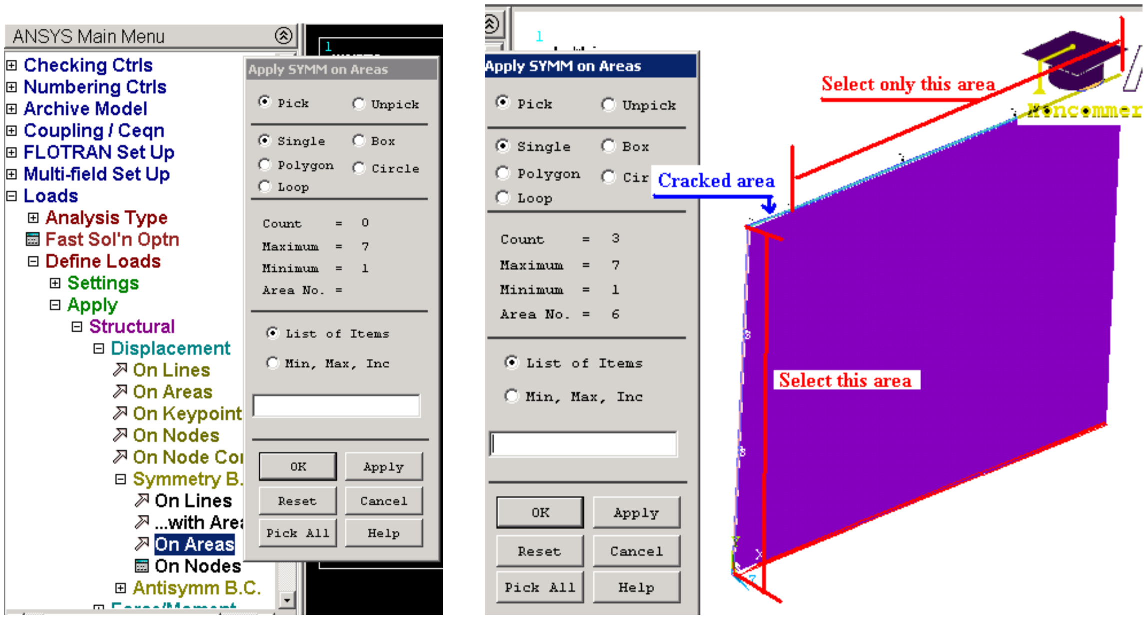


Apply Boundary Conditions

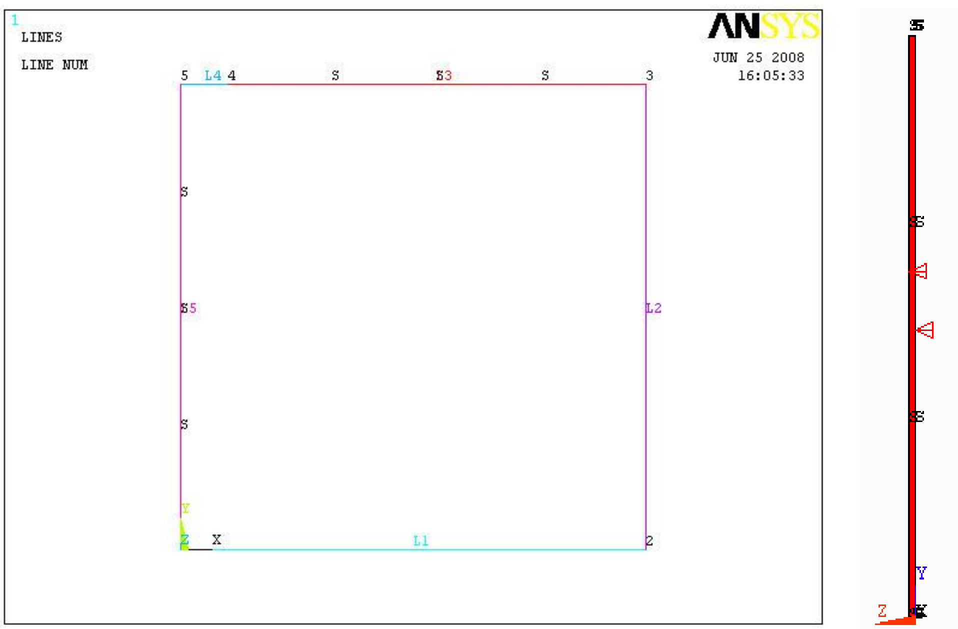
Because of the symmetry, our system has the following BC's:

- On symmetry areas
  - Left Area [from (0,0) to (0,10)] U_x =0
  - Top Area [from (10,10) to (1,10)] U_y =0
- On back area, constrain U_z (U_z=0)
- On front area, constrain U_z (U_z=0)

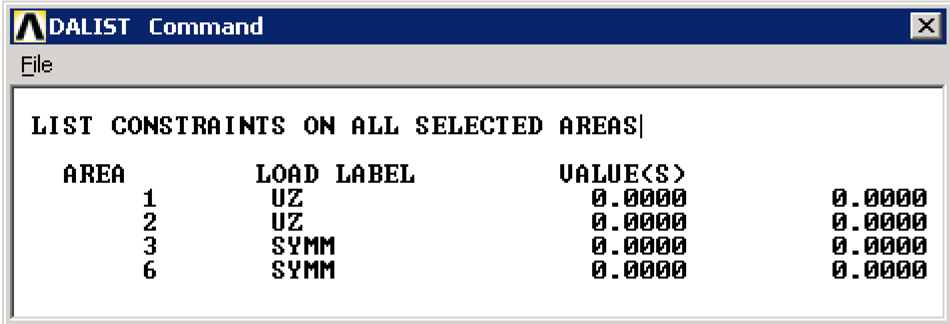
Apply the displacement constrains using;
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>on Areas> on back area and front area or **DA, p** for area BC.



Be careful in selecting. To get accurate selection, you can use perspective views using **ctrl+right buttons**.



To check the applied boundary conditions on areas, **DALIST** is used in the command line

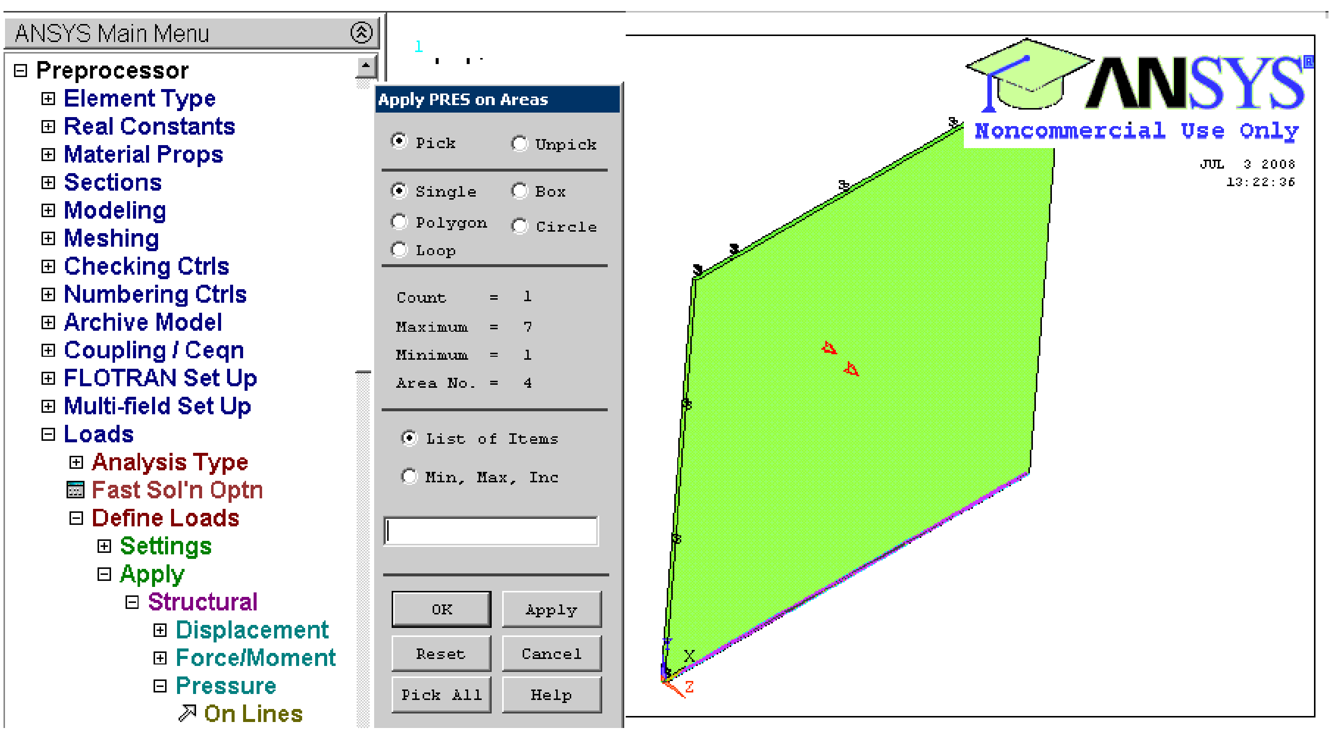


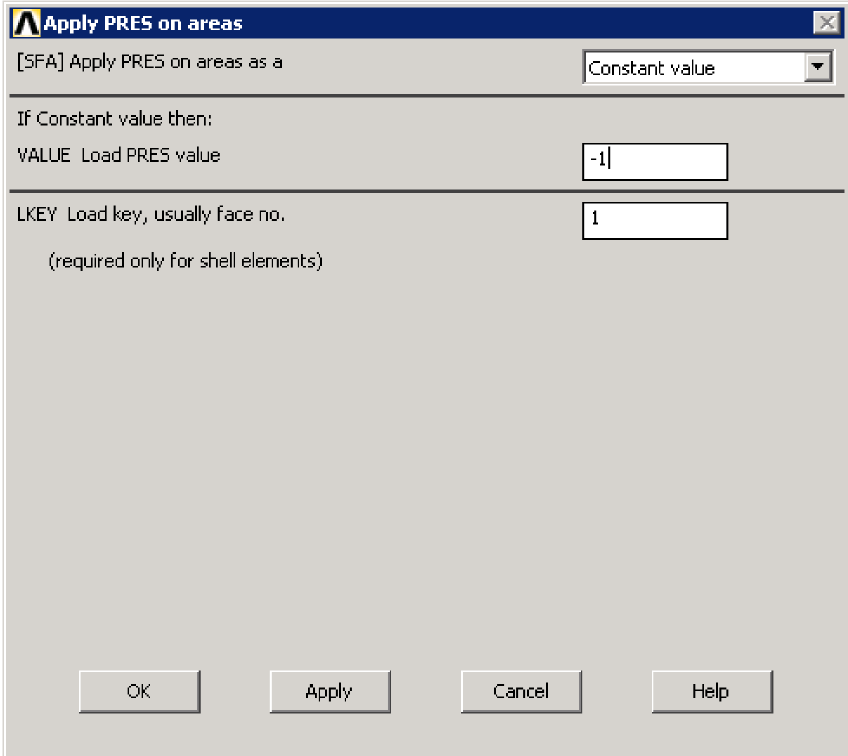
AREA	LOAD LABEL	VALUE(S)	
1	UZ	0.0000	0.0000
2	UZ	0.0000	0.0000
3	SYMM	0.0000	0.0000
6	SYMM	0.0000	0.0000

Apply Loads

Now we will apply the distributed surface forces (pressure).
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Areas
Carefully pick the bottom area (at *z*=0 and *y*=0) and then click OK in the picking window or **sfa, p** to apply area pressure.

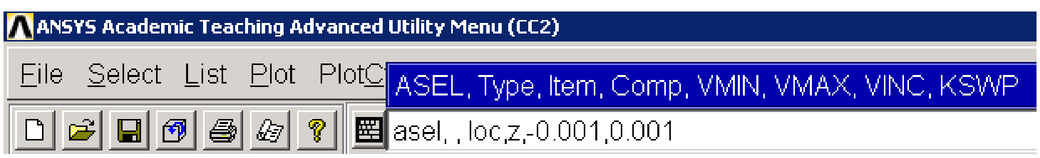
Select `Constant value' enter `-1' for `Load PRES value', then click OK (Negative sign is for tensile loading).





Meshing the Model

Back area has to be meshed first. Before meshing, we can select the back area (template area) with **ASEL** command using the coordinate (location) option.



In the above command, *Asel* is used to select a subset of areas that are located in the coordinate range specified. To be able to select the back area only, very small distance is given between the minimum and maximum coordinate in *z* direction only.

asel, , loc,z,-0.001,0.001

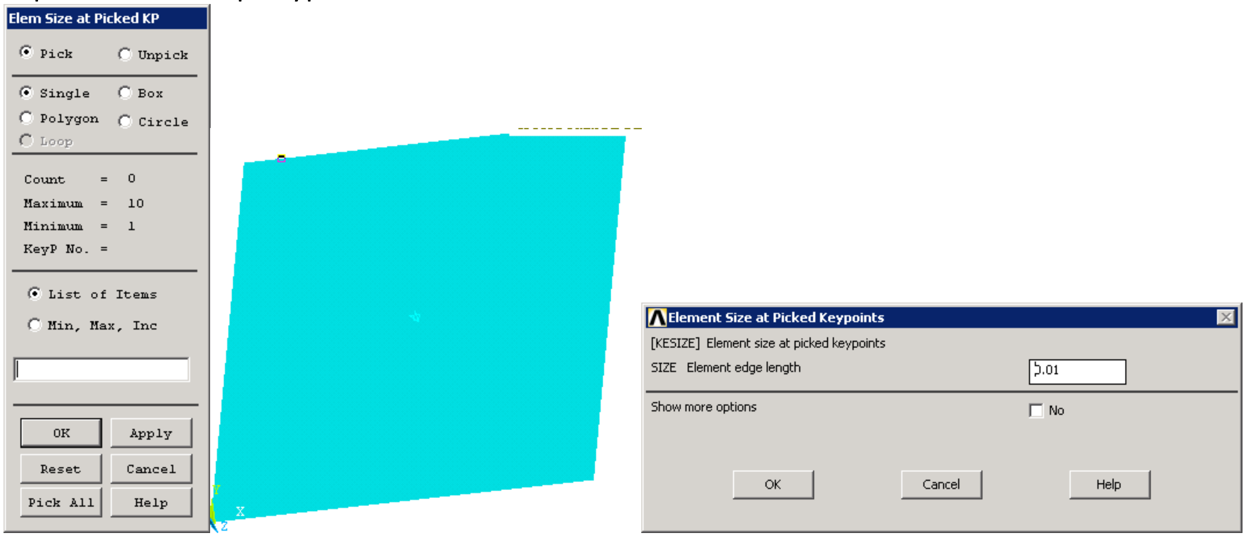
aplot

Meshing

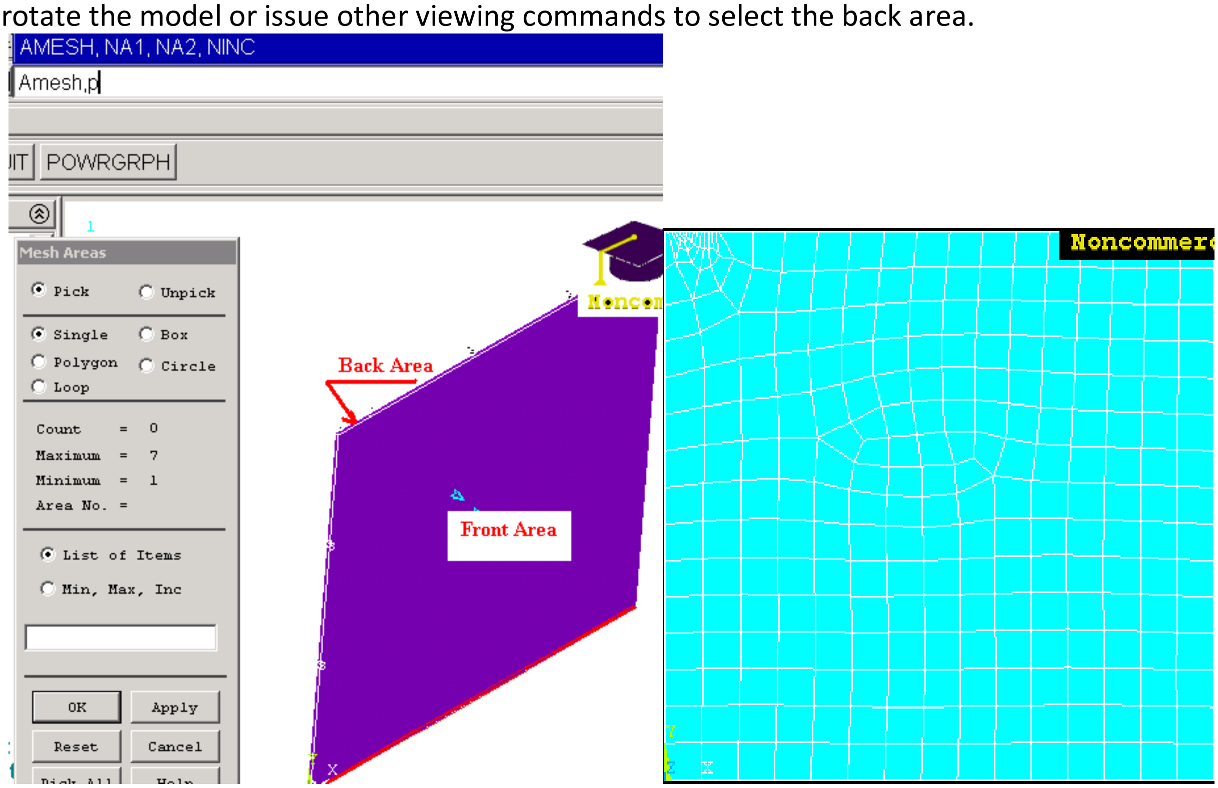
To obtain accurate fracture solution, we need to generate fine mesh near the crack tip. For this, we can use the KESIZE command to specify element size at the crack tip keypoint. First zoom into the crack tip region. Then issue the command

kesize, p

and pick the crack tip keypoint and write 0.01 as the element size value SIZE in the window.

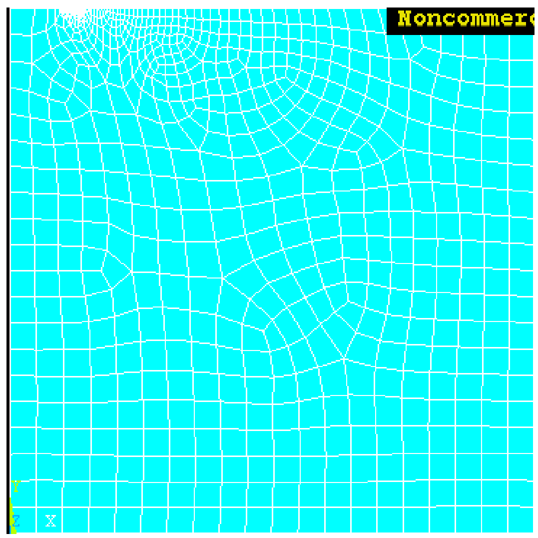


Amesh,p ' Select the back area exactly. Be careful in selecting. Use, zoom in, perspective view or rotate the model or issue other viewing commands to select the back area.

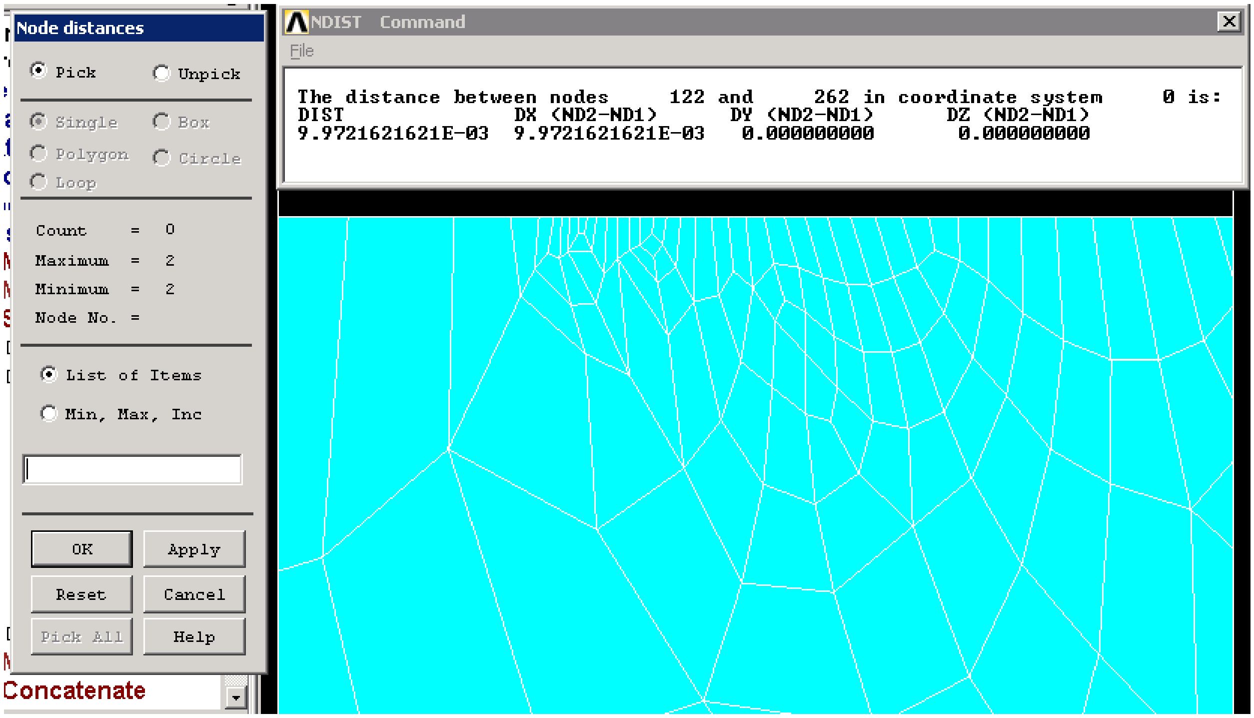


First, zoom into the crack region and use **Ksel** to locate the exact location of the crack tip, since we will measure the element edge sizes ahead of and behind the crack tip. To do this, we use the *ndist* command (**ndist,p**) and measure the crack tip edge size. This gives us 6.67e-1, which is coarser than the required size. Therefore, we require finer mesh. Now, we try a specified global element size. **Esize,0.5** global size value is entered as 0.5.

Mesh again the back area; **Amesh,p**



We may check the distance again using the **NDIST** command. This gives us 9.972e-3, which is fine enough (i.e., 1/100th of the crack length).

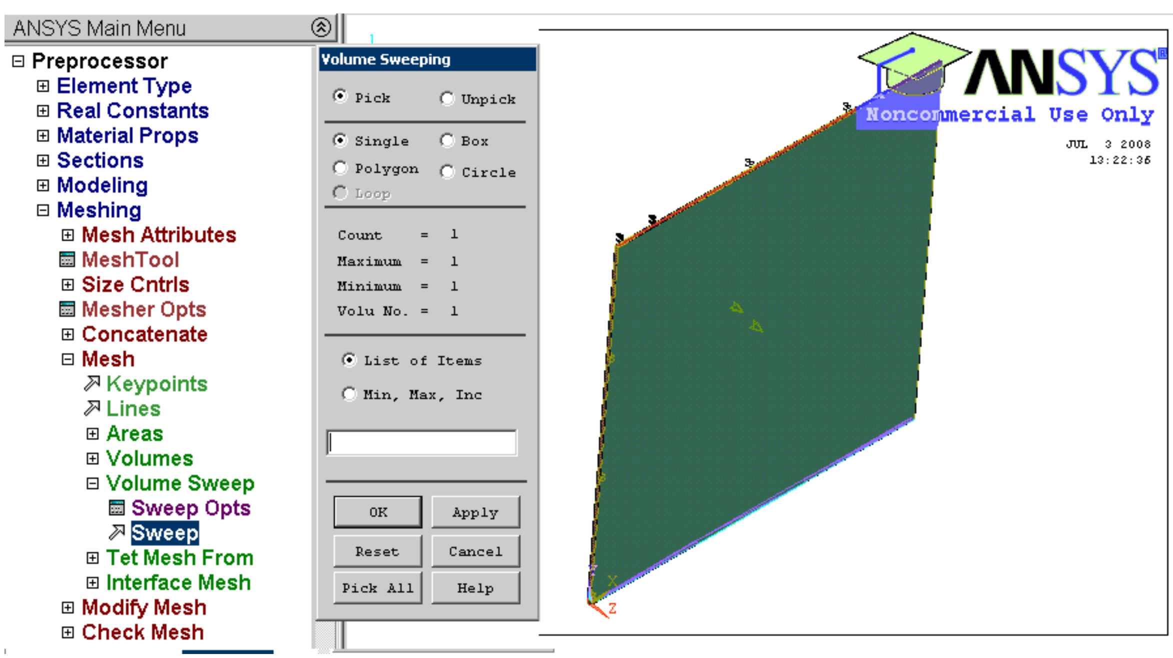


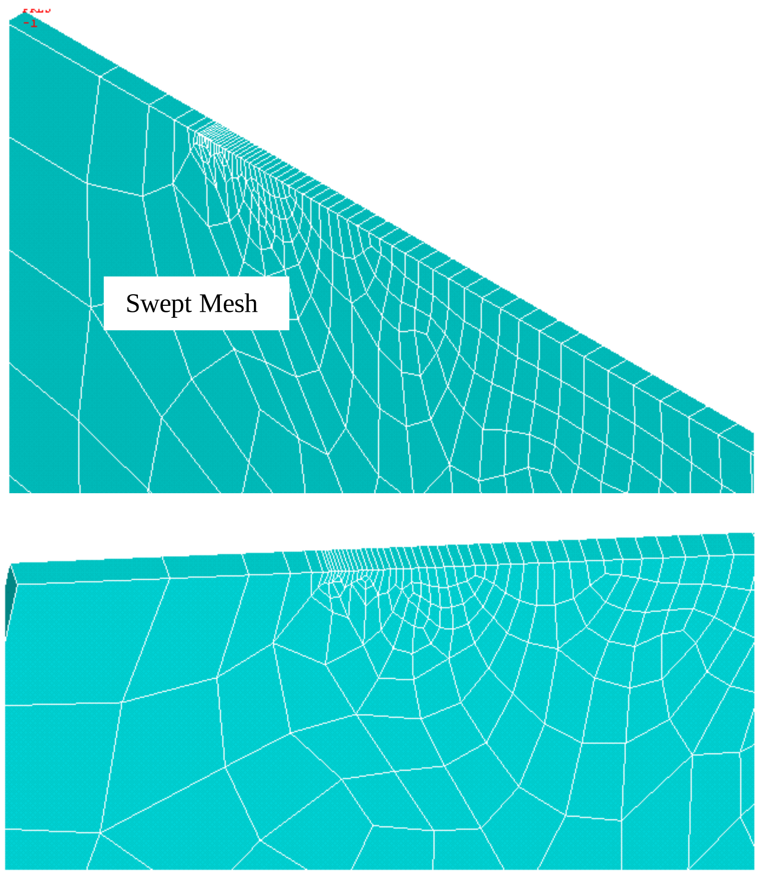
As can be seen above, the specified element size is achieved.

Volume Sweeping

Now we will extrude the 2D mesh in to the third dimension by one element in this direction.

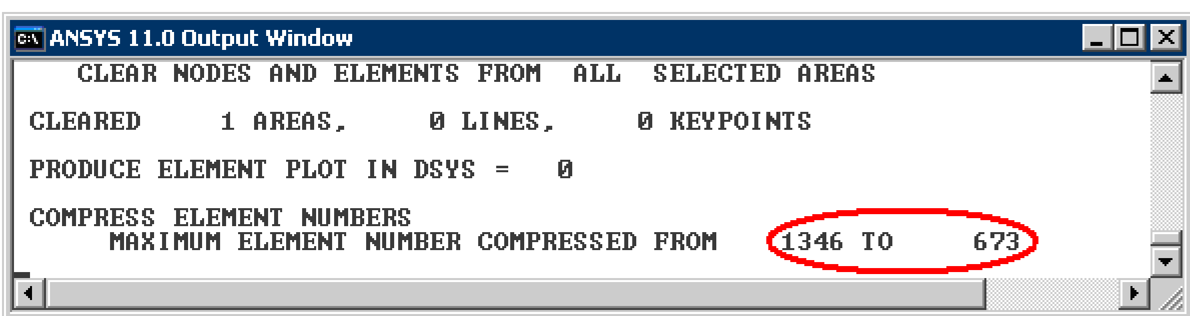
Main Menu>Preprocessor>Meshing>Mesh>Volume>Volume sweep>Sweep



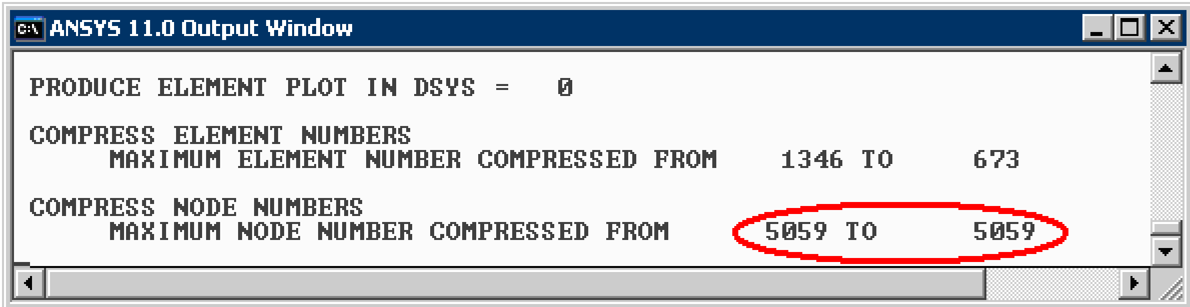


Volume scraping gives us prismatic mesh in the bounded volume, with respect to area mesh (template mesh). After sweeping the 2D mesh to generate the 3D mesh, we need to delete 2D elements, since FRAC3D requires 3D elements only.

To do this, we use **Aclear, all** ' Area mesh is deleted. Because of this, a gap occurs in the sequence of element numbers. To remove the gap we use, **Numcmp, elem**

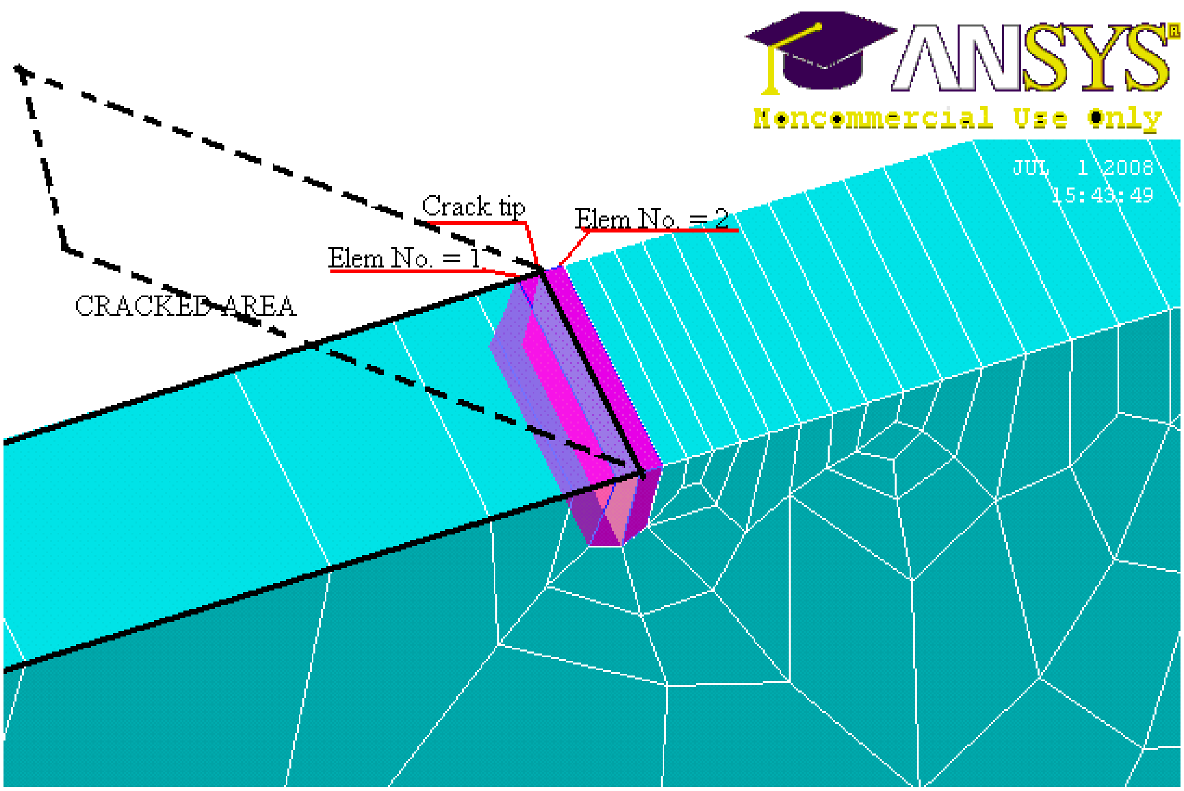


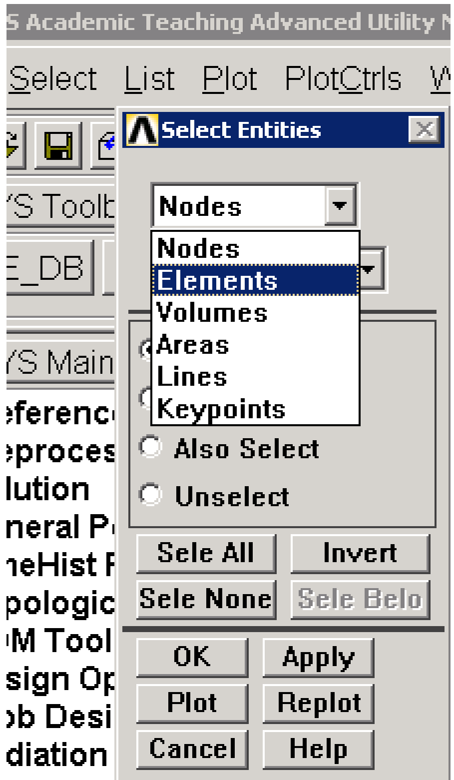
To make sure no gap exists in the node numbers as well, we use **Numcmp, node**



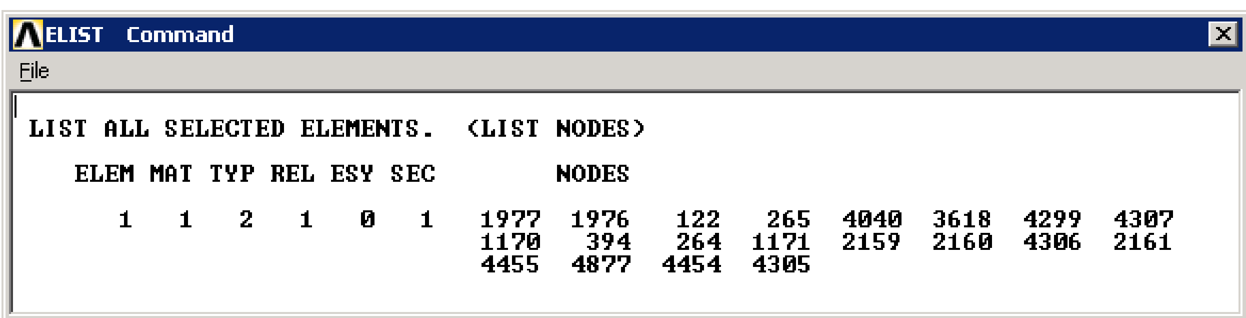
Definition of Crack for FRAC3D

Now, we need to provide crack tip element and node number information for FRAC3D analysis. Zooming into the crack region, we can find the elements and nodes located at the crack tip. (See detailed crack tip definition requirements in this tutorial for which elements and nodes to be selected). We need to identify the crack tip element on the bottom crack surface (with respect to the chosen local coordinate system) immediately behind the crack tip. Using, **Select-Entities-Elements,** form the main menu (or Esel, p command) try to select the elements at the crack tip. For this, move your mouse pointer near to crack tip region. We have to select the element which is both at crack tip and on the bottom crack surface (as defined by the local coordinate system at the crack tip). You can see that the crack tip element number is **1**.



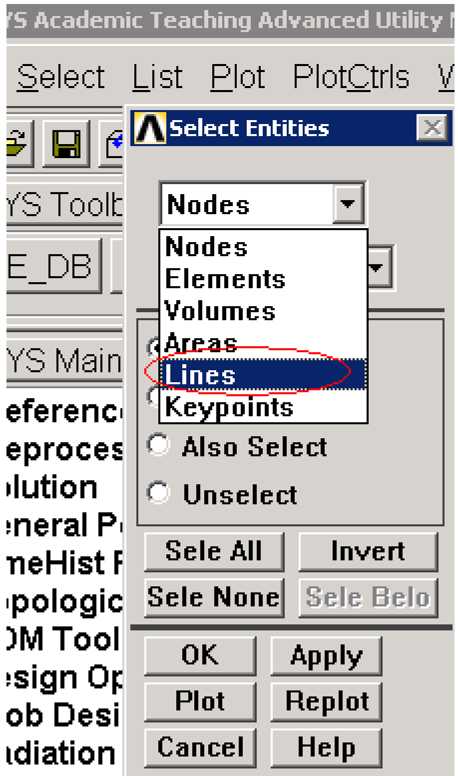


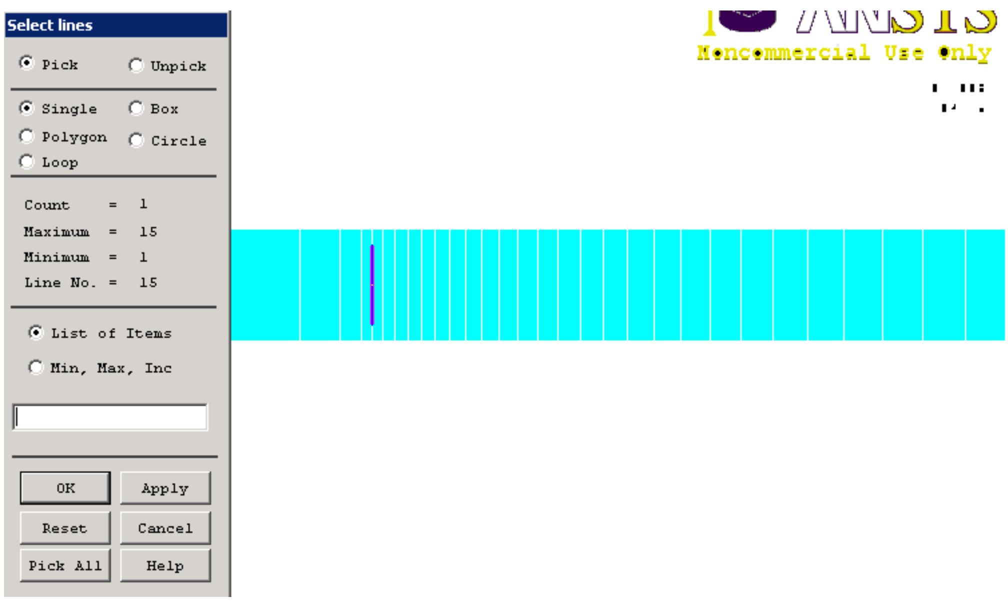
After element selection, we list the selected element and save the file FRAC3D analysis. Use **Elist** to see and save the list as a file.

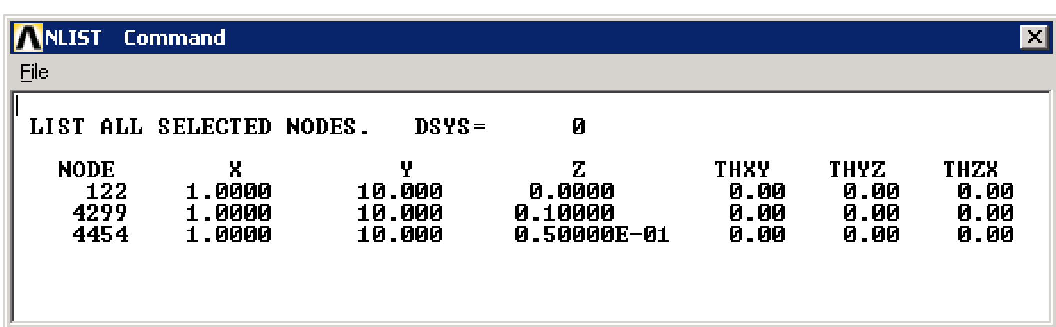


The element information (for the crack tip elements) is saved from the Elist window as ***cc3.crelems***

In addition to crack tip elements, we also need to select nodes on the crack front. To be able to select the nodes of the crack front, it is required to select the crack tip line and then select the nodes associated with this line. Using, **Select-Entities-Lines**, crack tip line is selected. Then, **NSLL, S, 1** is used to select all the nodes along the selected crack front line. Using **Nlist**, we can see that the crack front nodes numbers are: **122, 4299, 4454.**

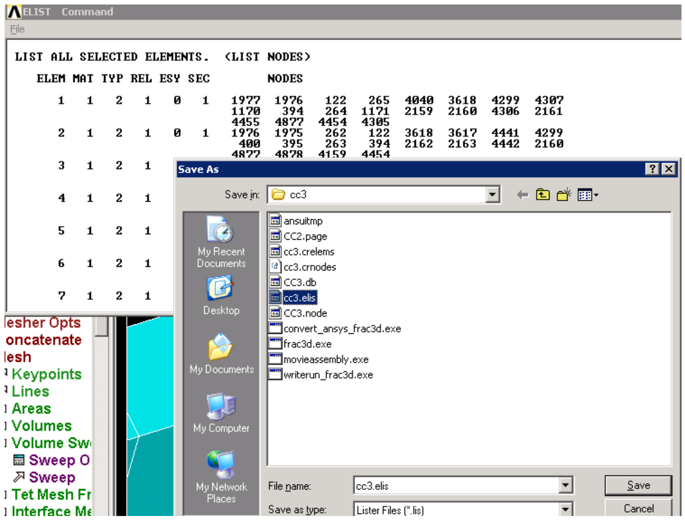

Select Entities
Nodes
Nodes
Elements
Volumes
Areas
Lines
Keypoints
Also Select
Unselect
Sele All
Invert
Sele None
Sele Belo
OK
Apply
Plot
Replot
Cancel
Help


Select lines
Pick
Unpick
Single
Box
Polygon
Circle
Loop
Count = 1
Maximum = 15
Minimum = 1
Line No. = 15
List of Items
Min, Max, Inc
OK
Apply
Reset
Cancel
Pick All
Help
Noncommercial Use Only


NLIST Command
File
LIST ALL SELECTED NODES. DSYS= 0
NODE X Y Z THXY THYZ THZX
122 1.0000 10.000 0.0000 0.00 0.00 0.00
4299 1.0000 10.000 0.10000 0.00 0.00 0.00
4454 1.0000 10.000 0.50000E-01 0.00 0.00 0.00

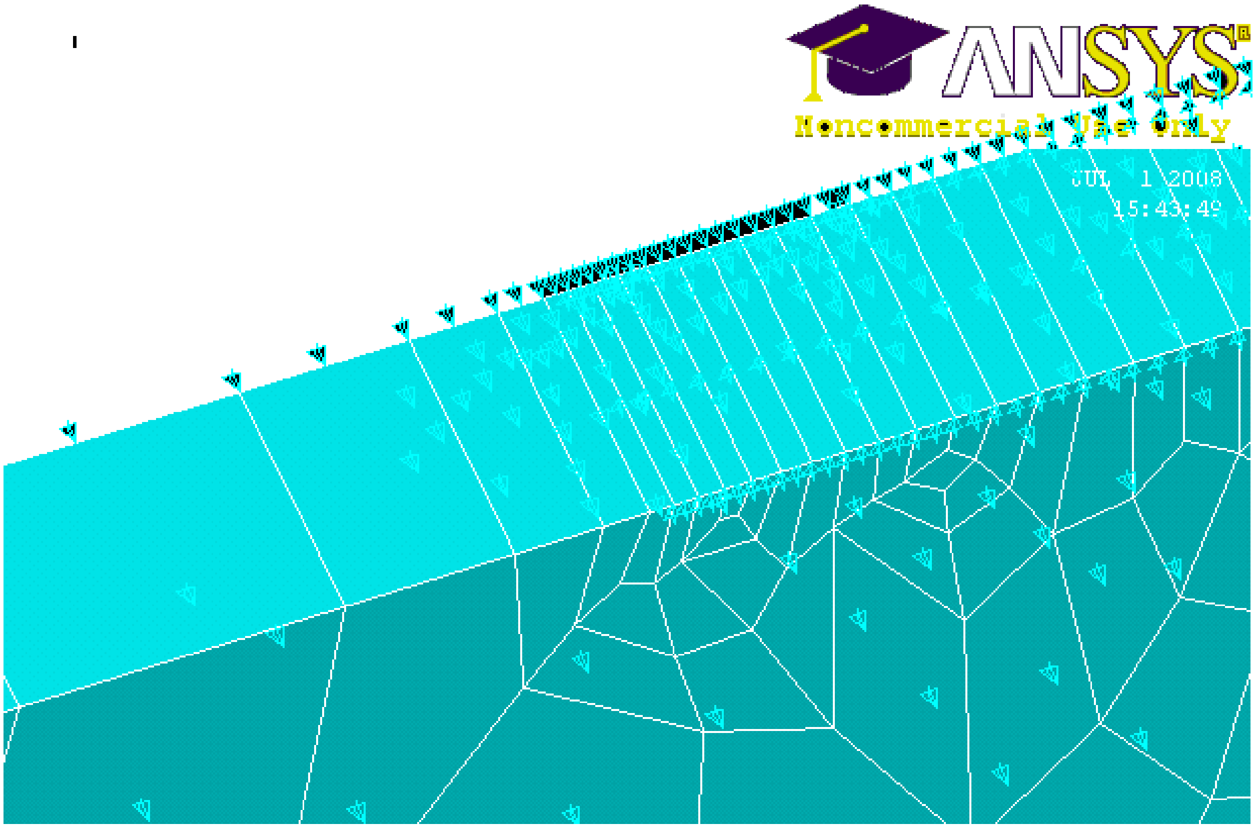
The coordinates of the selected nodes (the crack tip nodes) are saved from the **Nlist** window as ***cc3.crnodes***

Also using **Select-Everything,** all elements in model are saved in file, ***cc3.elis.***

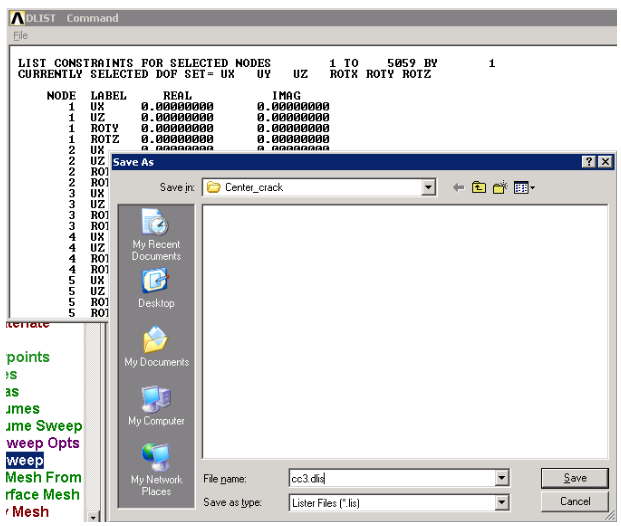


Also using **nwrite** all nodes are saved as ***cc3.node*** automatically in current working directory.

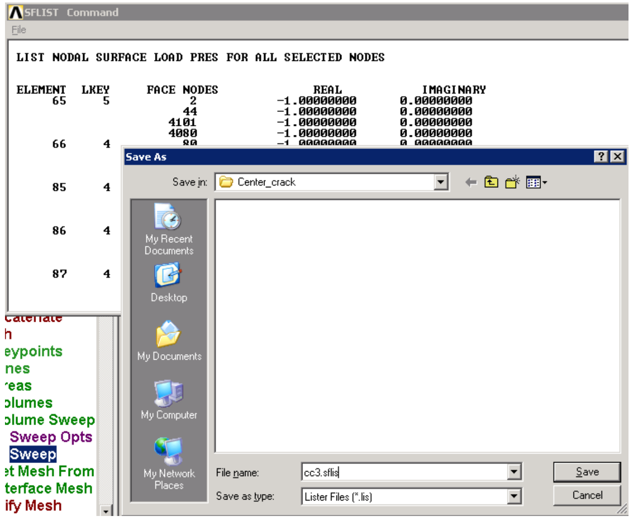
SBCTRAN is used to transfer solid model loads and boundary conditions to the FE model. Loads and boundary conditions on unselected keypoints, lines, areas, and volumes are not transferred.
sbct



Using **dlist** displacement BC's are saved as ***cc3.dlis.***



Using **sflist** pressure loads on elements are saved as ***cc3.sflis.***



Now, we completed all modeling steps in ANSYS^{TM}. We are ready to convert all model information into FRAC3D format using the converter program.

T.1.4 Converting ANSYS Model into FCPAS Format (Generation of *cc3.geo* File)

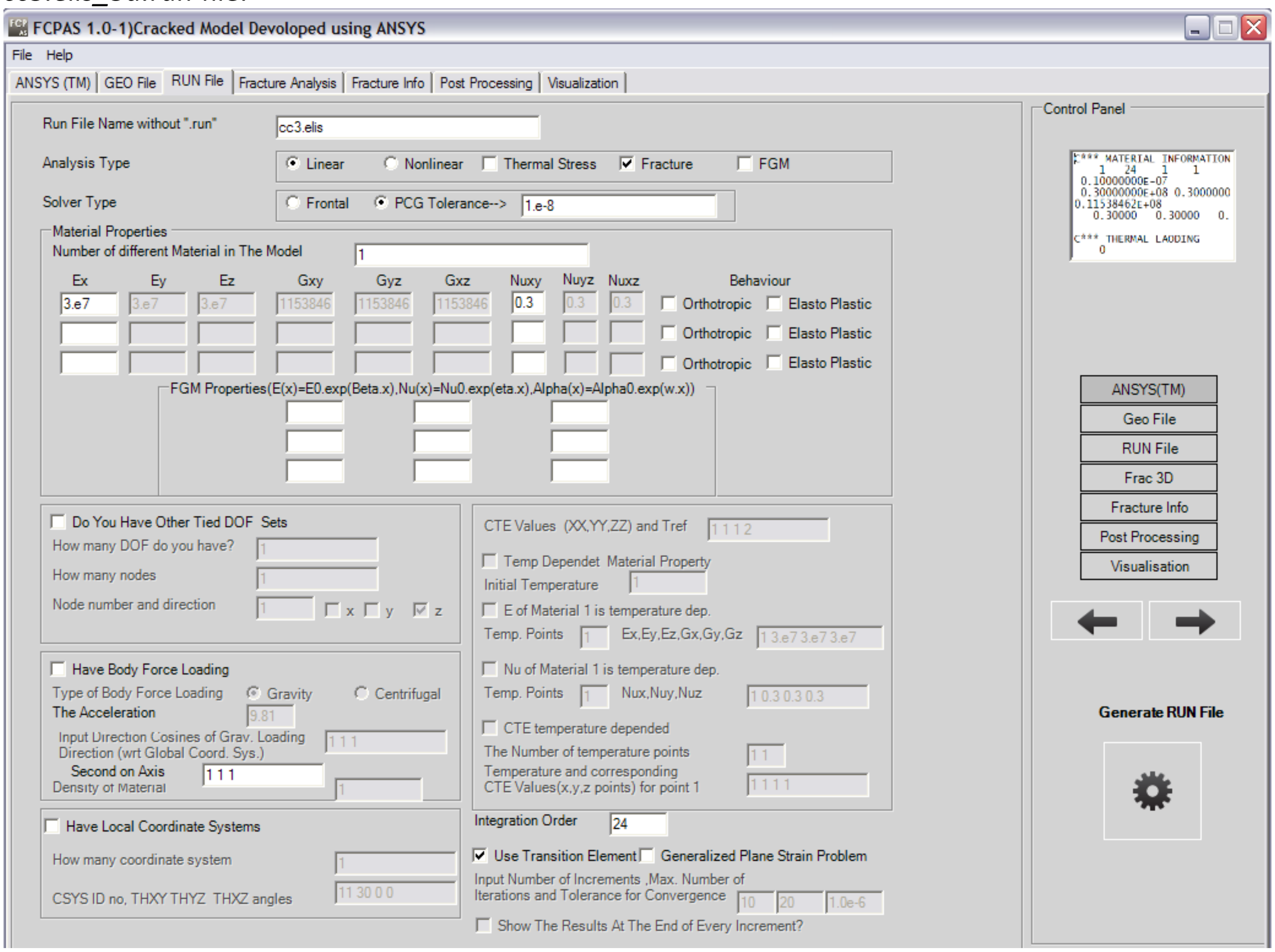
FCPAS requires its model information in a specific format. To convert ANSYS™ model files into FCPAS format, we can use the *convert_ansys_fcpas.exe* program. The converter program can be run by typing, its path in MSDOS prompt or from the "Geo File" tab from FCPAS GUI.

Input file names can be selected by "Browse" buttons. "Generate Geo file" creates *cc3.elis_3d.geo* file. To go to Run File preparation, press "Next Step".

T.1.5 Generation of *.run File

Now, we need to generate a run file which is also required for fracture analysis. We use *writerun_fcpas.exe* or FCPAS GUI to generate *.run file (*cc3.elis_3d.run* file). The **.run* file contains analysis type, material properties, solver type tolerances, body forces and local coordinate systems data.

Parameters can be selected by clicking the objects in the tab. "Generate Run file" creates *cc3.elis_3d.run* file.



To pass into "Frac3D" tab, press "Next Step".

T.1.6 Running FRAC3D (FCPAS Solver)

To run the FRAC3D, three kinds of input files are required;

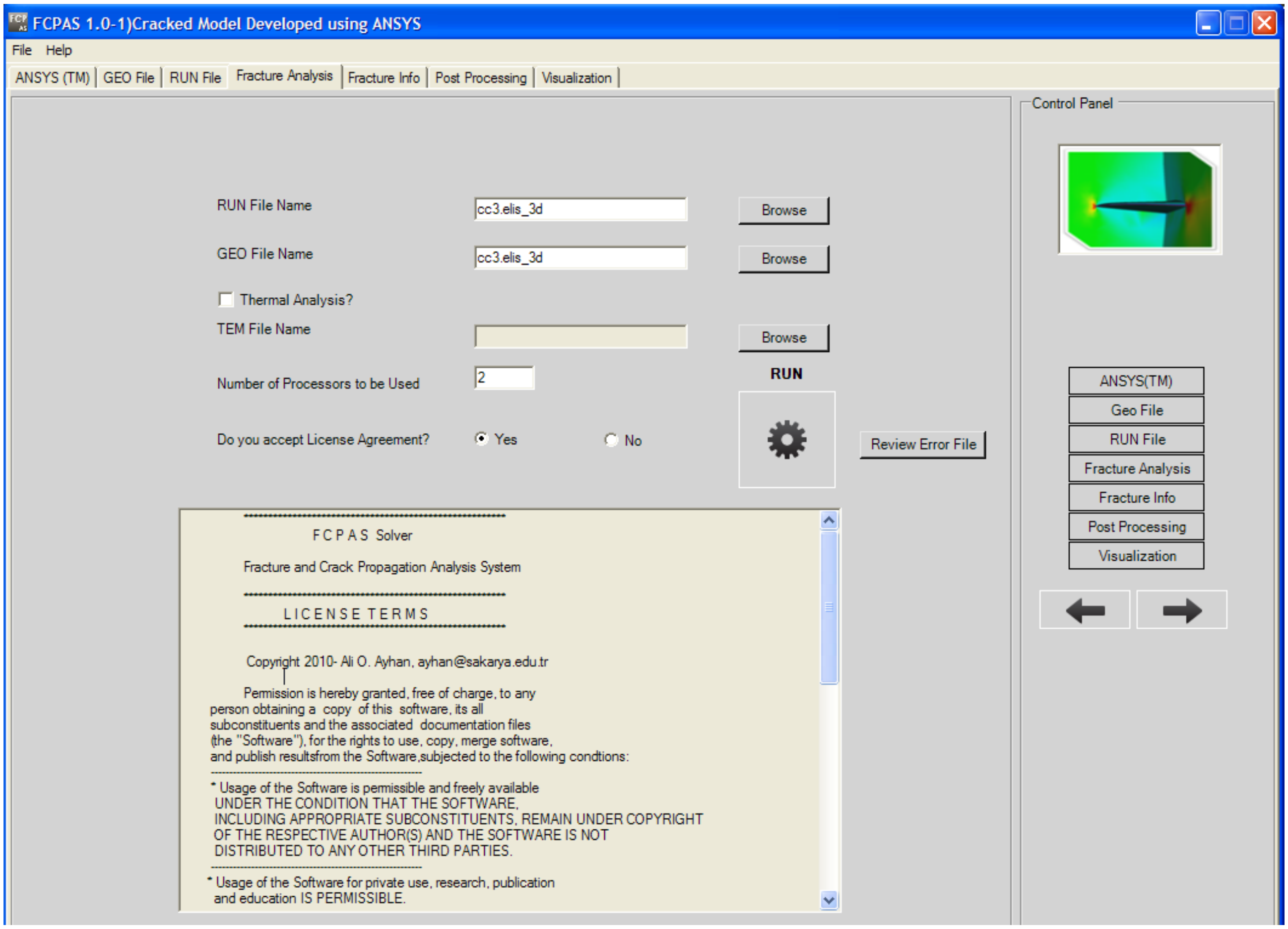
*.run (compulsory)
*.geo (compulsory)
*.tem (optional)

FRAC3D gives the results in the following output files;

*.out
*.str
*.stn
*.crk

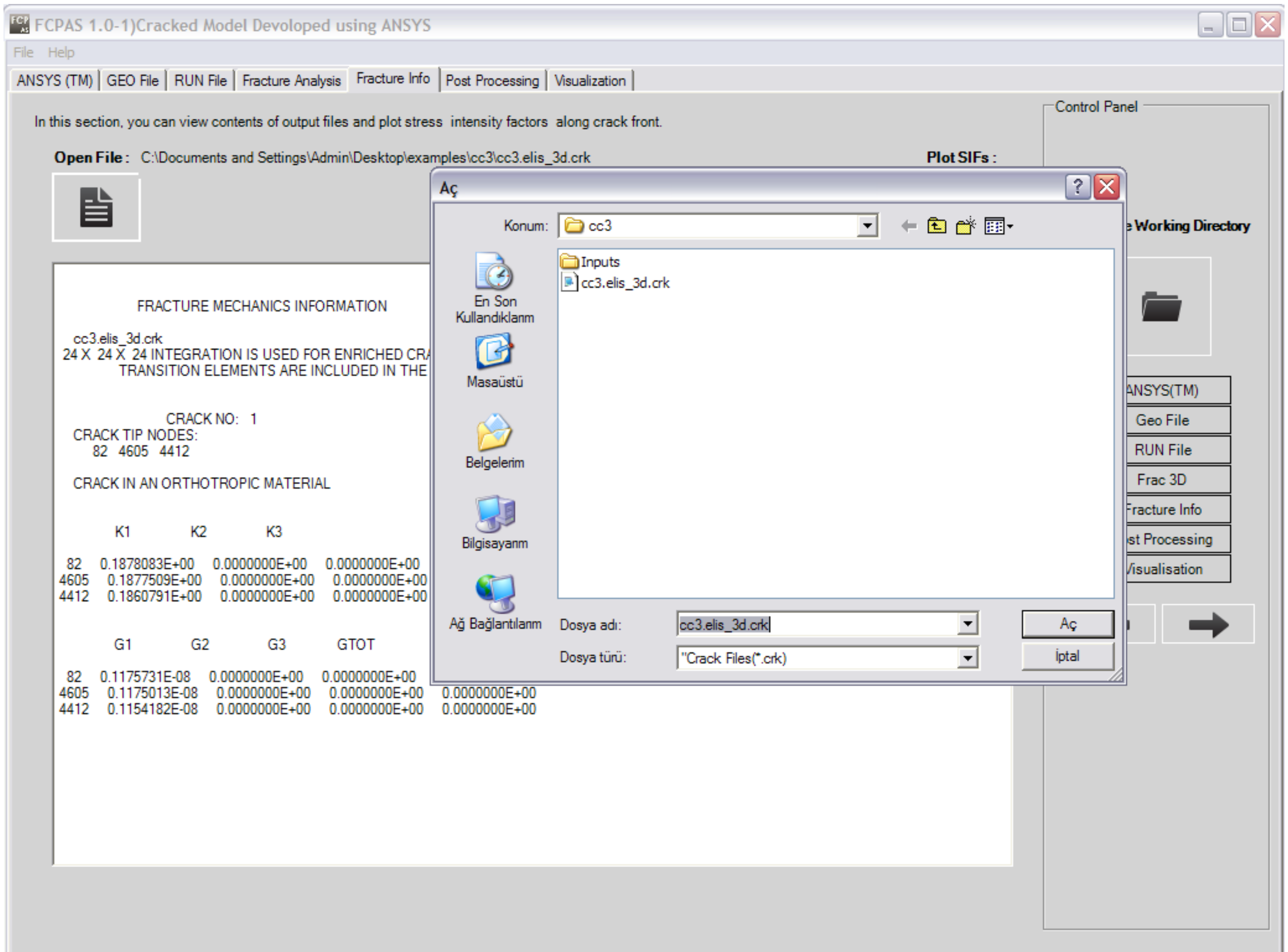
Now, we are ready to run FRAC3D. To do this we can use *frac3d.exe*. When running FRAC3D, geo and run files names have to be entered. The following table shows the steps and input for this specific problem.

Select the *cc3.elis_3d.geo, cc3.elis_3d.run*, and *cc3.elis_3d.tem* (if required) files by browsing and press run button to run the *Frac3D.exe* in the background.

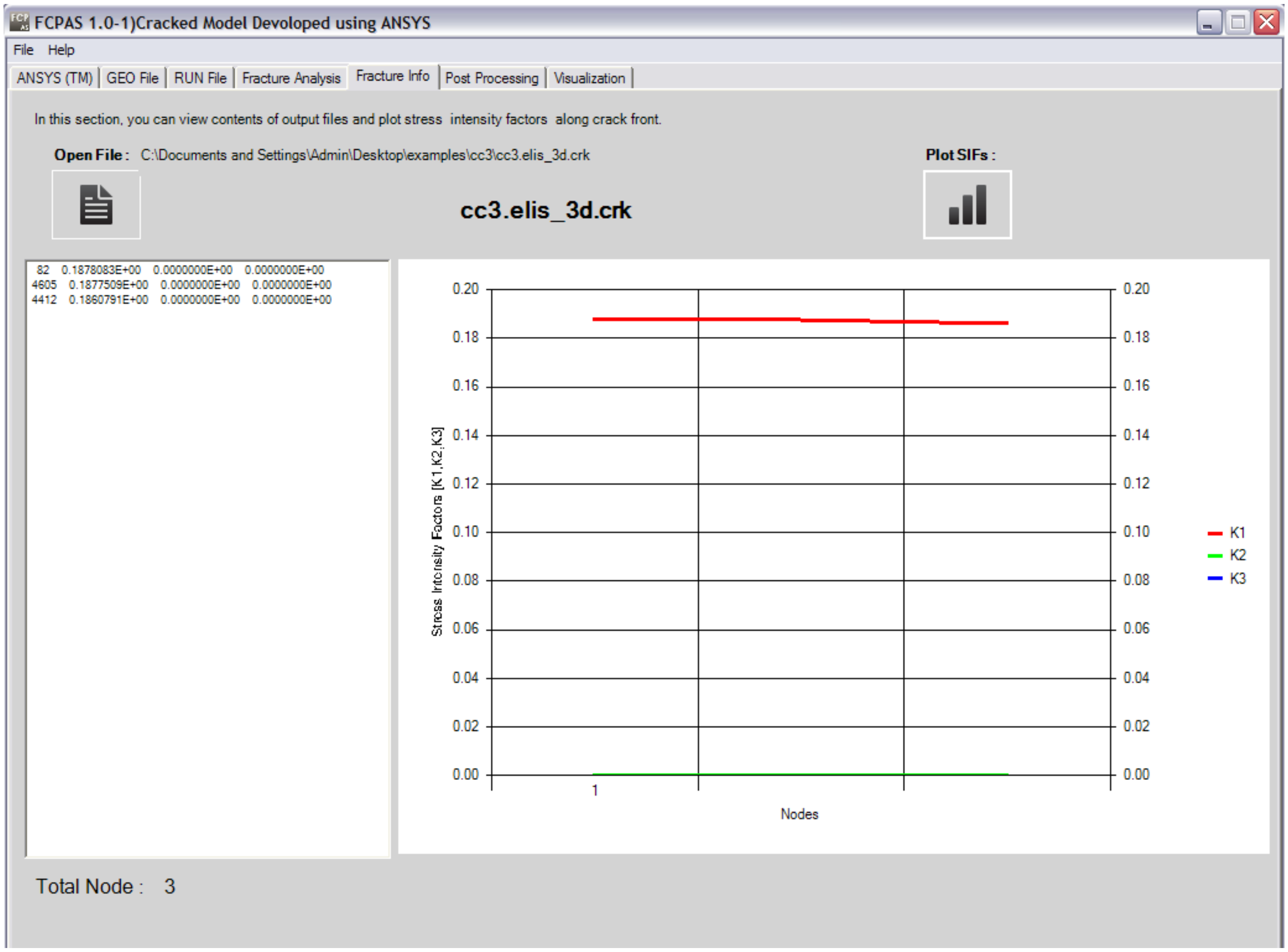


REVİEW EROR FİLE toolu olan yeni görüntüyü al

After finishing the process, output files can be seen in the “Fracture Info” tab. In the “Fracture Info” tab you can browse anyfile to see its content and plot the K_1, K_2 and K_3 data in an x-y graph.

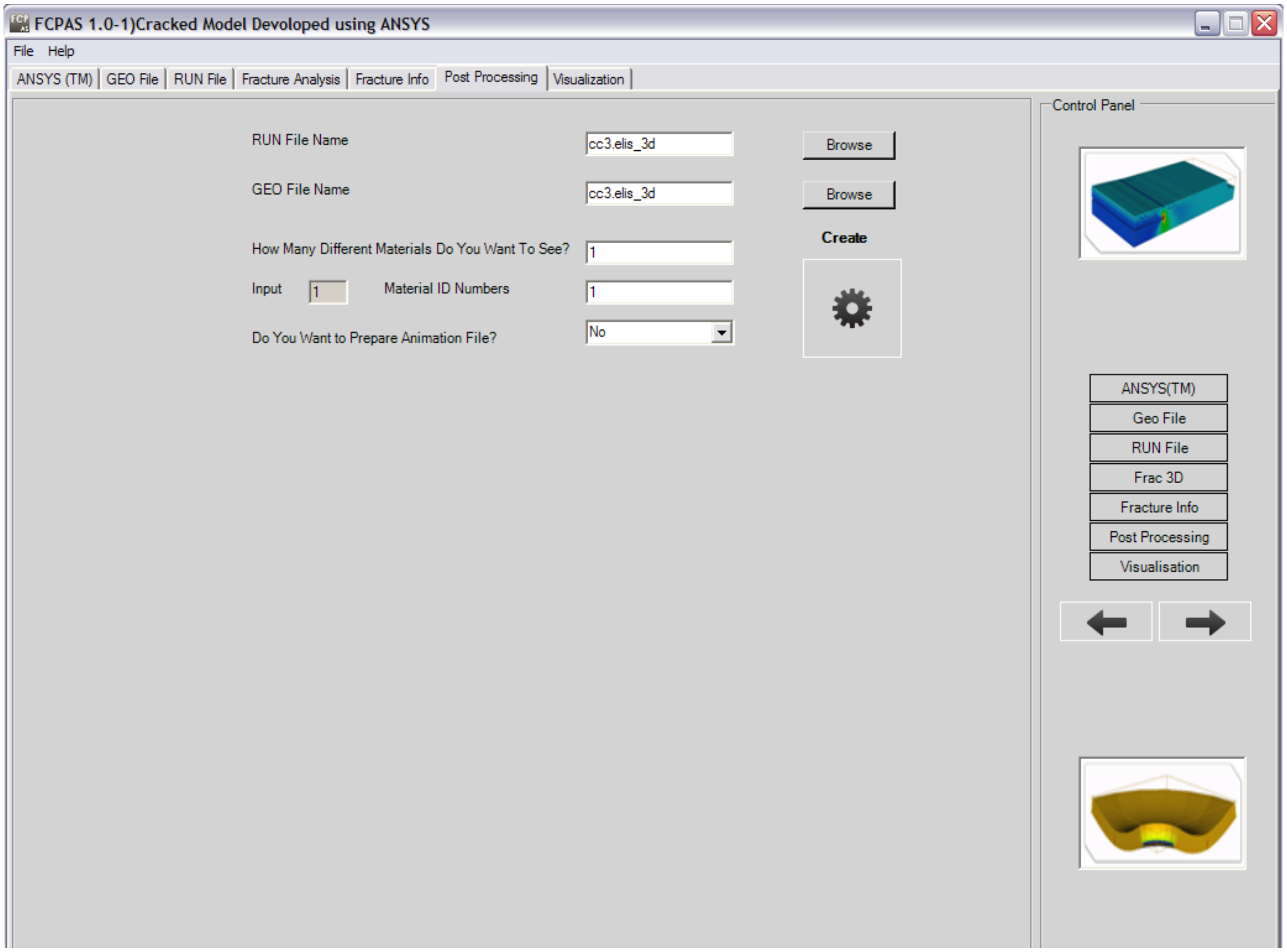


To plot the K_1, K_2 and K_3 data in a graph, just press "Plot SIF's" button.

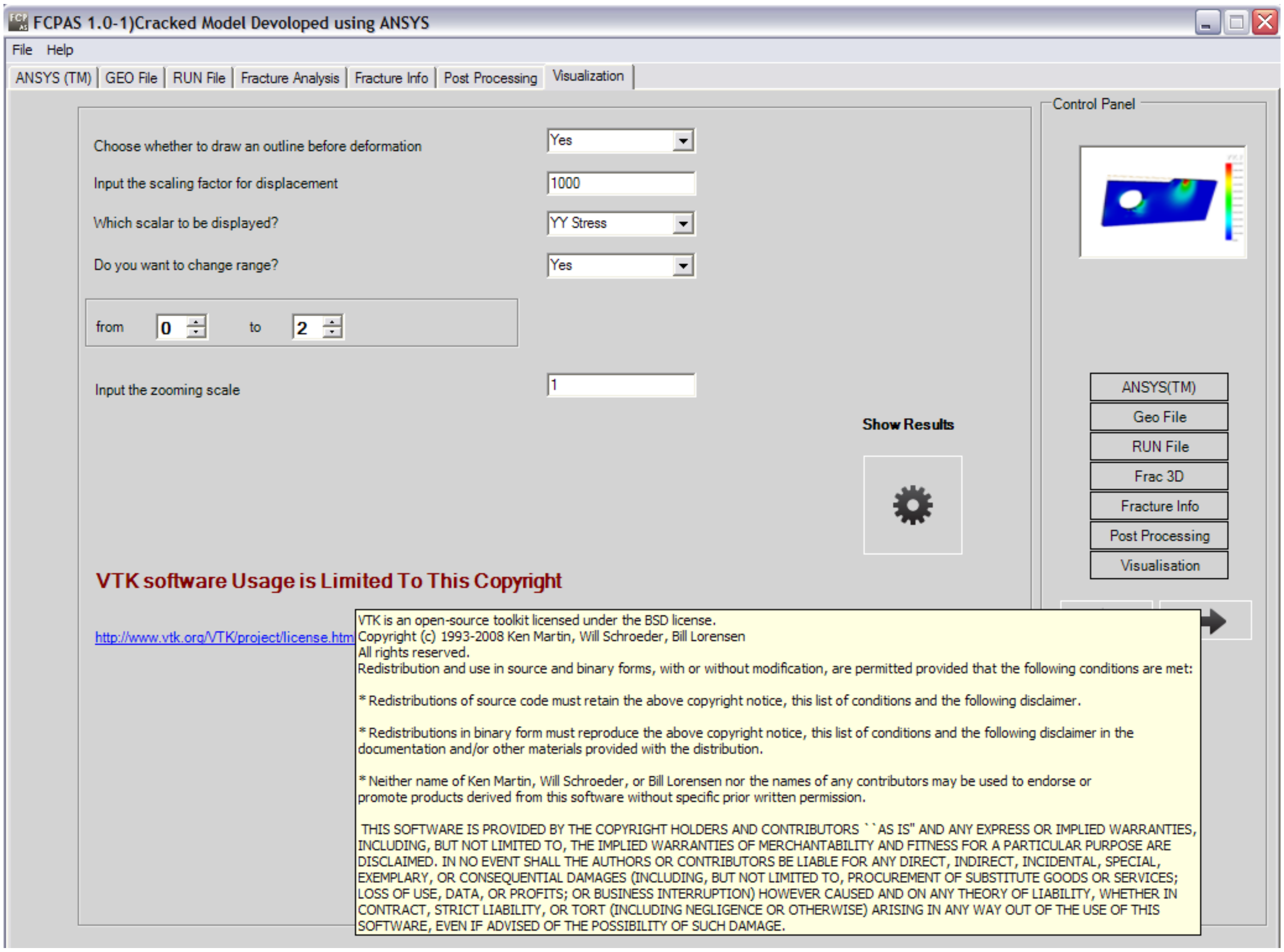


We can see that the computed K_1 (1.814) is ~%2 higher than the analytical solution.

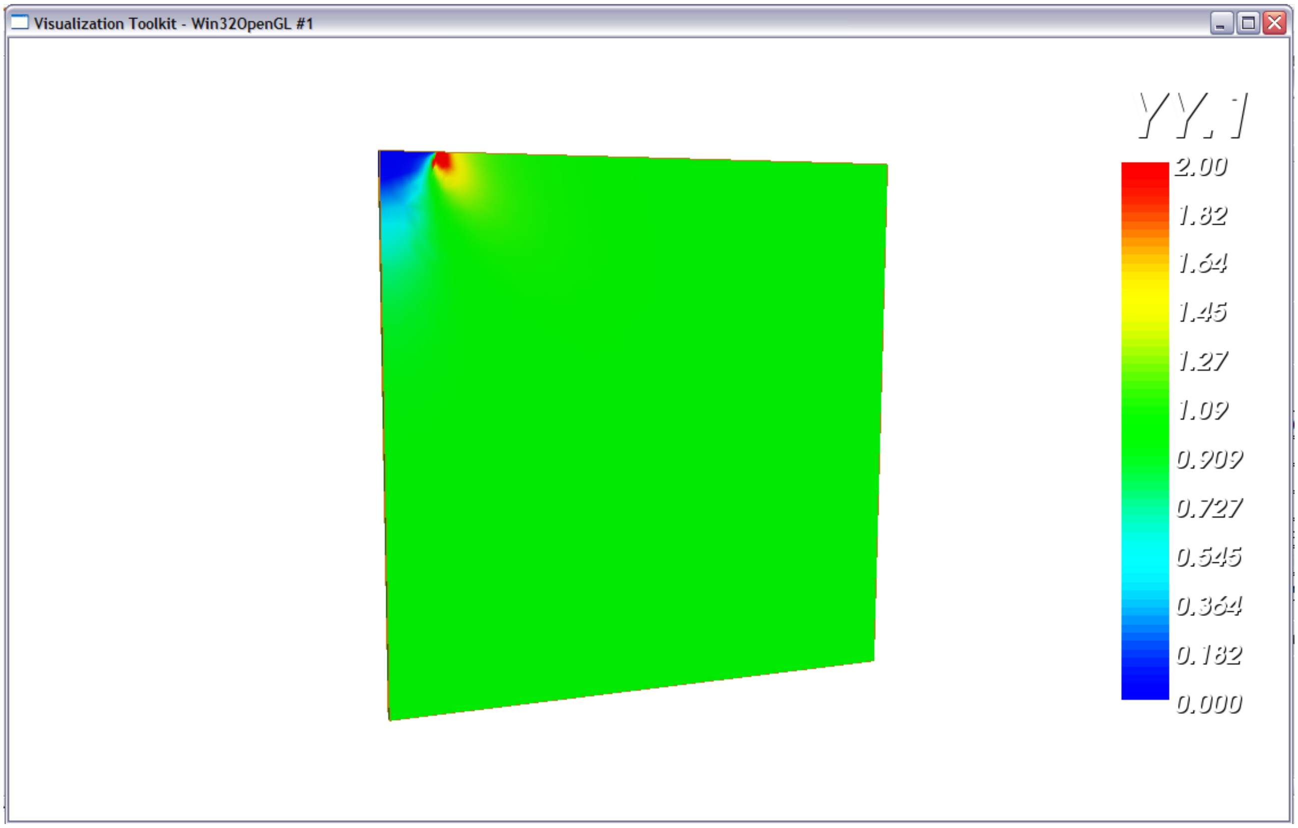
T.1.7 Post-processing of FRAC3D Results



T.1.8 Visualization of FRAC3D Results



To see the Cracked Model results, choose the parameter you would like to contour plot and press "Show Results" button.



EXAMPLE.2. Three-Dimensional Mode-I C(T) Specimen

T.2.1. Problem Description

Toughness is the ability of a material to resist fracture. The general factors, affecting the toughness of a material are temperature, strain rate, relationship between the strength and ductility of the material and presence of stress concentration (notch) on the specimen surface. Compact Tensile (CT) specimen is one specimen type to measure fracture toughness of a material. In this example, we will model a crack of length a=27.5mm in a CT specimen and compute the mode I stress intensity factor (SIF) along the crack front. The material is Al-7075 with Elasticity Modulus = 70 GPa, σ_Y= 95 MPa, P= 1 N, ν= 0,33, ρ= 2,81 g/cm^3. The dimensions are given in Figure D.38. These dimensions are in mm.

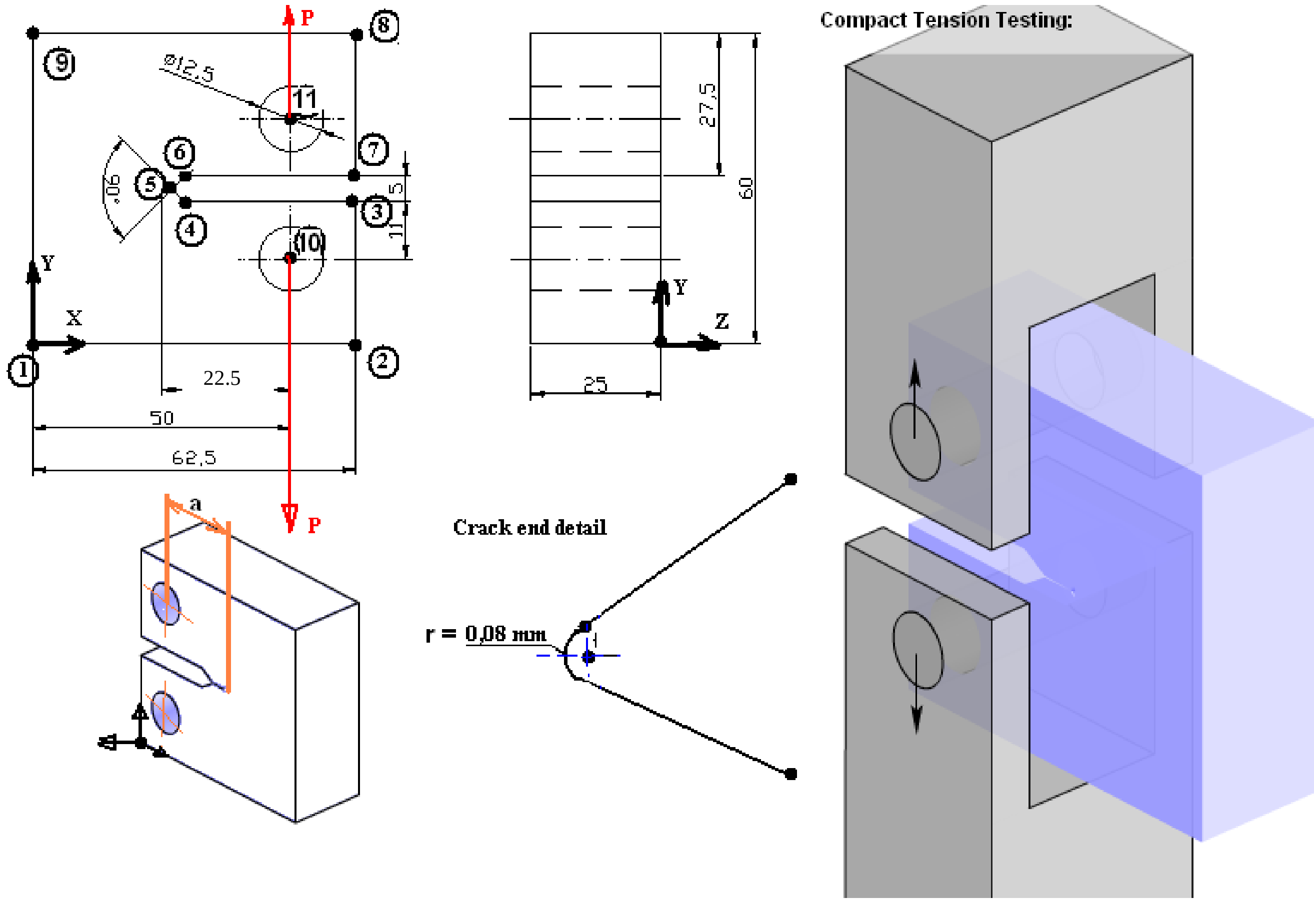


Figure D. 2 CT specimen and its dimensions

In FCPAS ANSYS™ tab, we run ANSYS™ program.

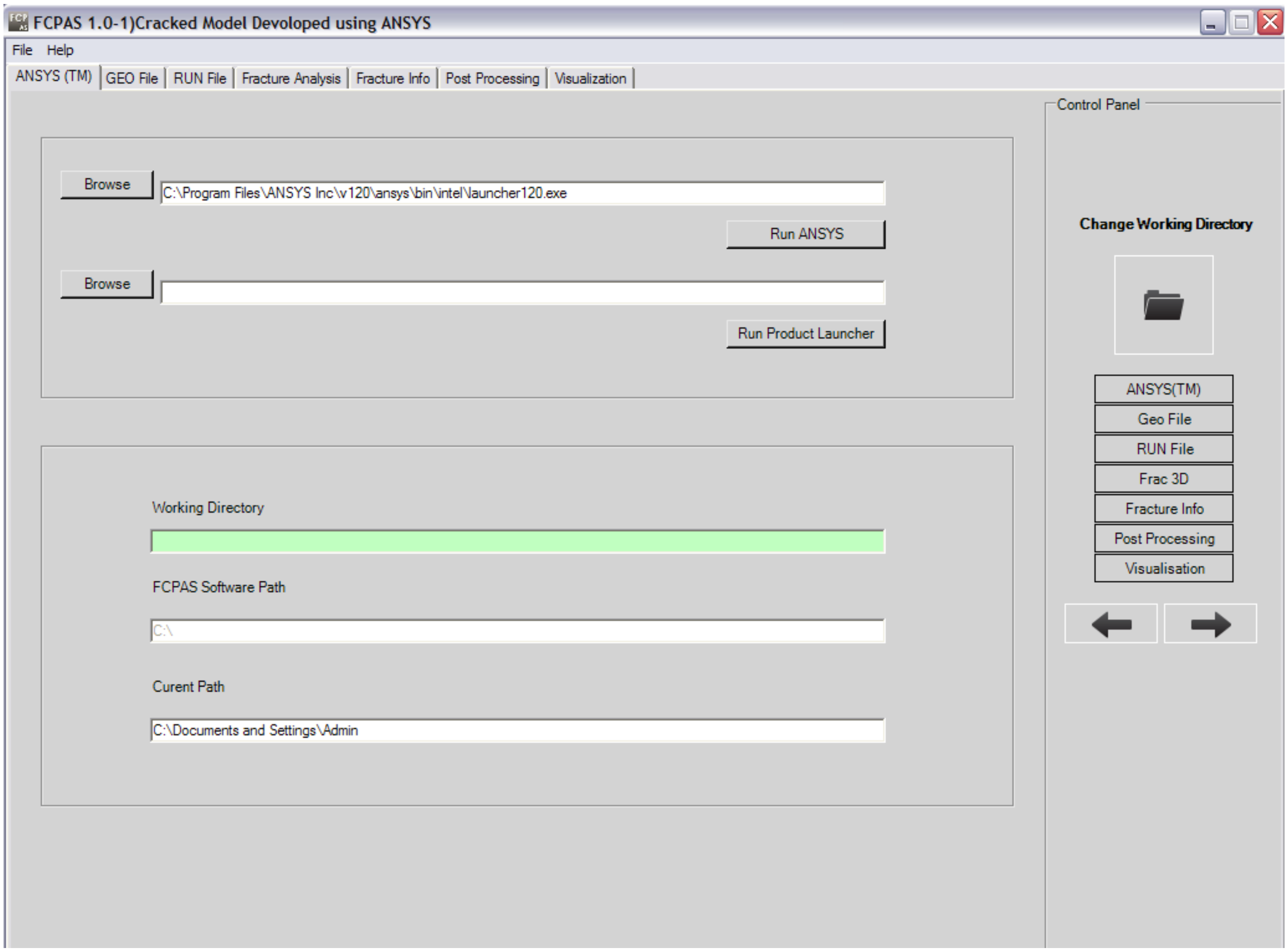


Figure T.7. FCPAS graphical user interface

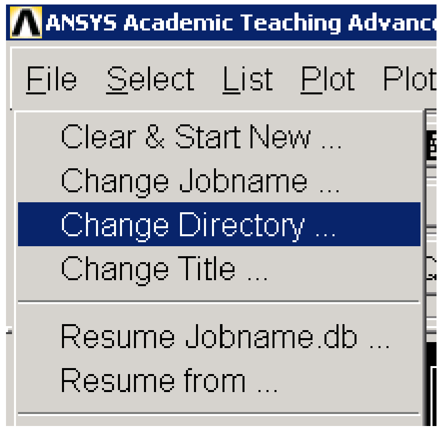
T.2.2. Generation of the Finite Element Model within the ANSYS^{TM} Preprocessor

First of all, we must take into account the problem type, we will model this as a 3D problem. Also, due to the symmetry of the problem, only analysis of a half model is needed. We will model this three-dimensional problem using multi-layers of 3D elements in the out of plane direction. To do this, we will first mesh the back face of the domain with area (2D) elements and extrude the mesh into the third direction. We will use PLANE82 and SOLID95 elements from the ANSYS^{TM} element library [3]. Note that ANSYS^{TM} Help is very useful tool to identify and select the suitable elements for the problem of interest.

Preprocessing

Change Directory

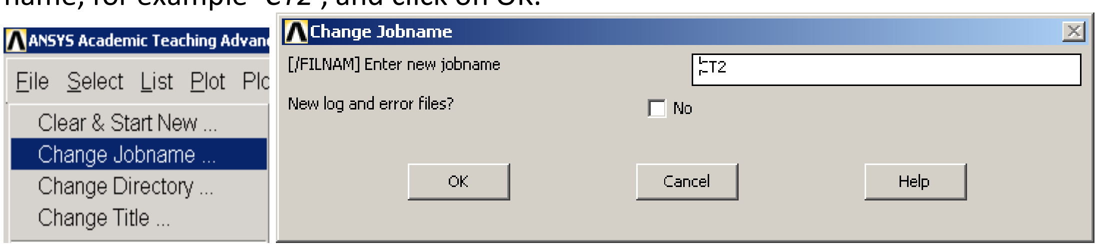
Before starting the model, create a folder in which you would like to work&change directory to this folder.



Give the Job a Name

Utility Menu>File>Change Jobname...

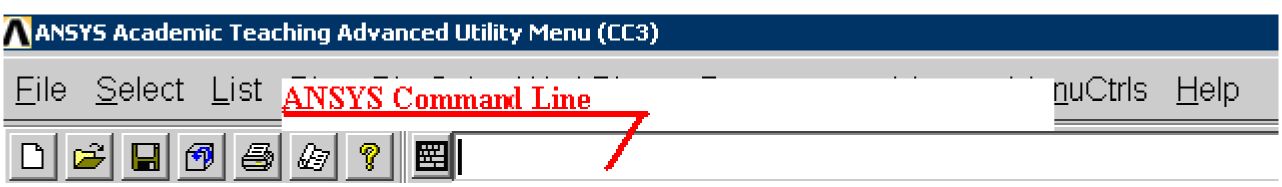
Enter a name, for example `*CT2*', and click on OK.



Define Element Type

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

This brings up the 'Element Types' window. Click on the Add... button. The 'Library of Element Types' window appears. Highlight "PLANE82-8 node 82"and "SOLID95-20 node 95". Click on OK or in command line, use **(ET,1,82)**[2], **(ET,2,95).**



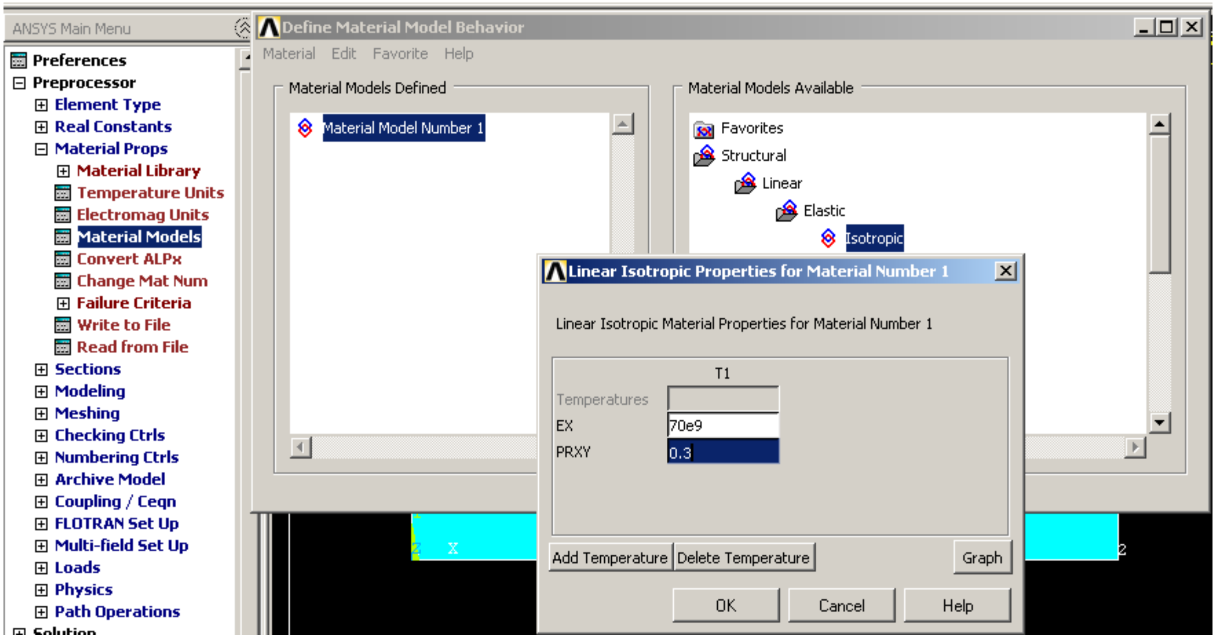
Define Material Properties

Main Menu>Preprocessor>Material Props>Material Models

[2] PLANE82 element provides us both "plane strain or stress" options.

On the right side of the `Define Material Model Behavior' window that opens, double click on `Structural', then `Linear', then `Elastic', finally `Isotropic'. Enter in values for the Young's Modulus (EX = 70E9) and Poisson's ratio (PRXY = 0.33) of the plate material. Or in command line, use

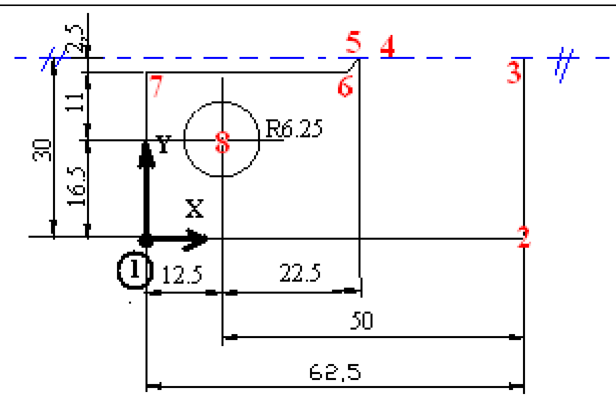
MP,EX,1,70e9
MP,PRXY,1,0.33



Define Keypoints

Main Menu>Preprocessor>Modeling>Create>Keypoints>In Active CS

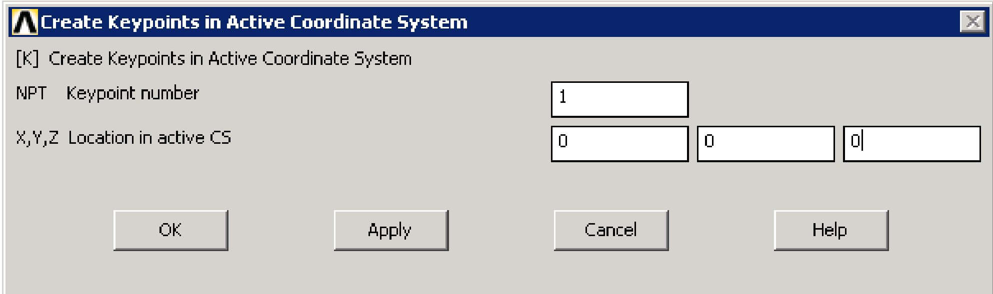
We are going to create 5 keypoints given in the following table:

KEYPOINTS	LOCATIONS			
	X [m]	Y [m]	Z [m]	
1	0	0	0	
2	0.0625	0	0	
3	0.0625	0.03	0	
4*	0.04	0.03	0	
5	0.035	0.03	0	
6	0.0325	0.0275	0	
7	0	0.0275	0	
8**	0.0125	0.0165	0	

*: If sharp crack length is 2 mm, then coordinates must be (0.037, 0.03, 0). In this case it is 5 mm.
**: This keypoint is the center of the hole and going to be used in Hole (circle) placing section.

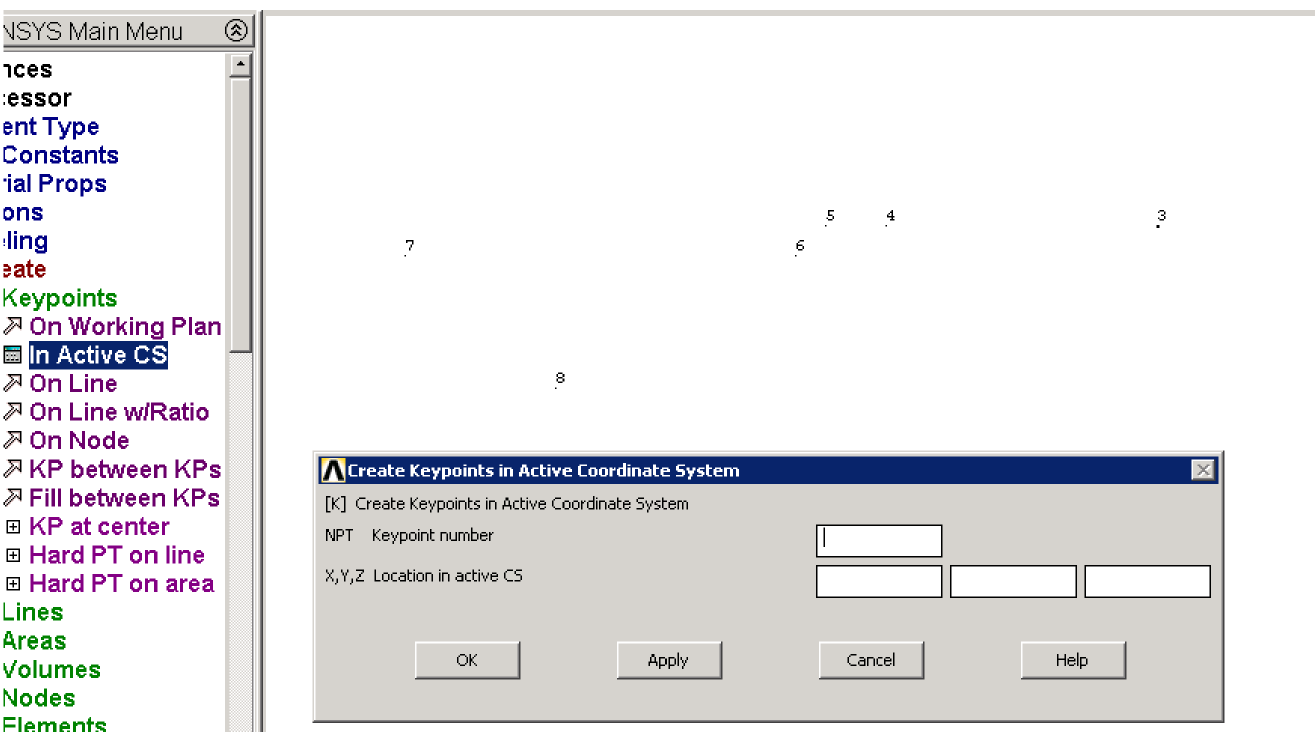
K,1,0,0,0,
K,2,0.0625,0,0,
K,3,0.0625,0.030,0,
K,4,0.04,0.03,0,

K,5,0.035,0.03,0,
K,6,0.0325,0.0275,0,
K,7,0,0.0275,0,
K,8,0.0125,0.0165,0,



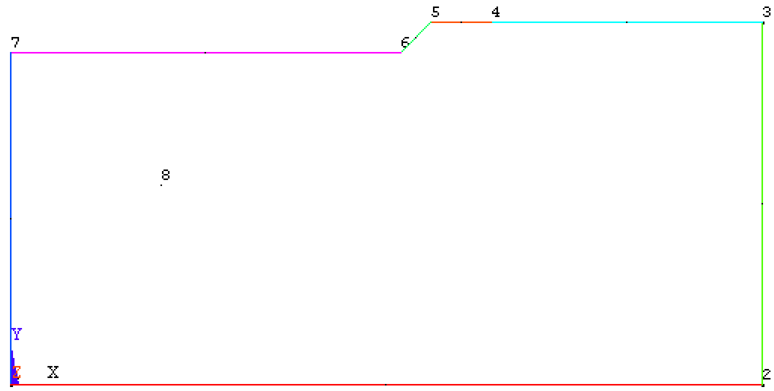
Define Line Segments
Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Straight Line

Clicking keypoints from 1 to 2, 2 to 3…, lines can be drawn. Last line must be KP7 to KP1. Keypoint 8 is for only creating the hole. For now keypoint 8 will not be used.



Or,

LSTR, 1, 2
LSTR, 2, 3
LSTR, 3, 4
LSTR, 4, 5
LSTR, 5, 6
LSTR, 6, 7
LSTR, 7, 1

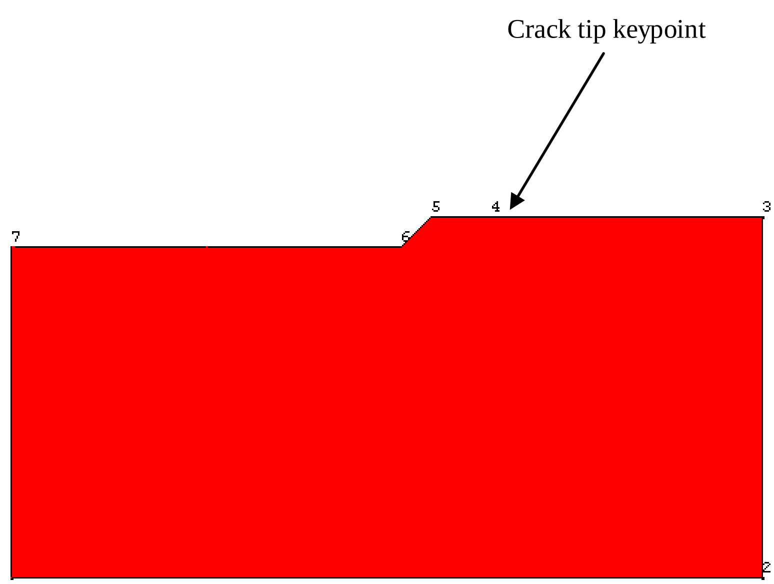


Create the Area

Main Menu>Preprocessor>Modeling>Create>Areas>Arbitrary>By Lines

Pick all lines (Click OK in the picking window). Or;

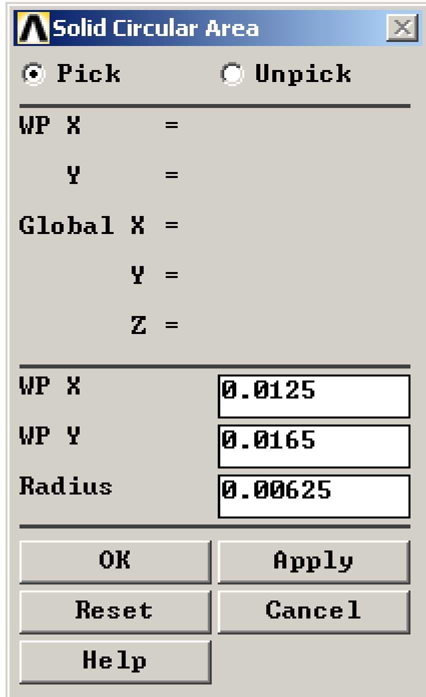
```
FLST,2,7,4
FITEM,2,1
FITEM,2,2
FITEM,2,3
FITEM,2,4
FITEM,2,5
FITEM,2,6
FITEM,2,7
AL,P51X
```

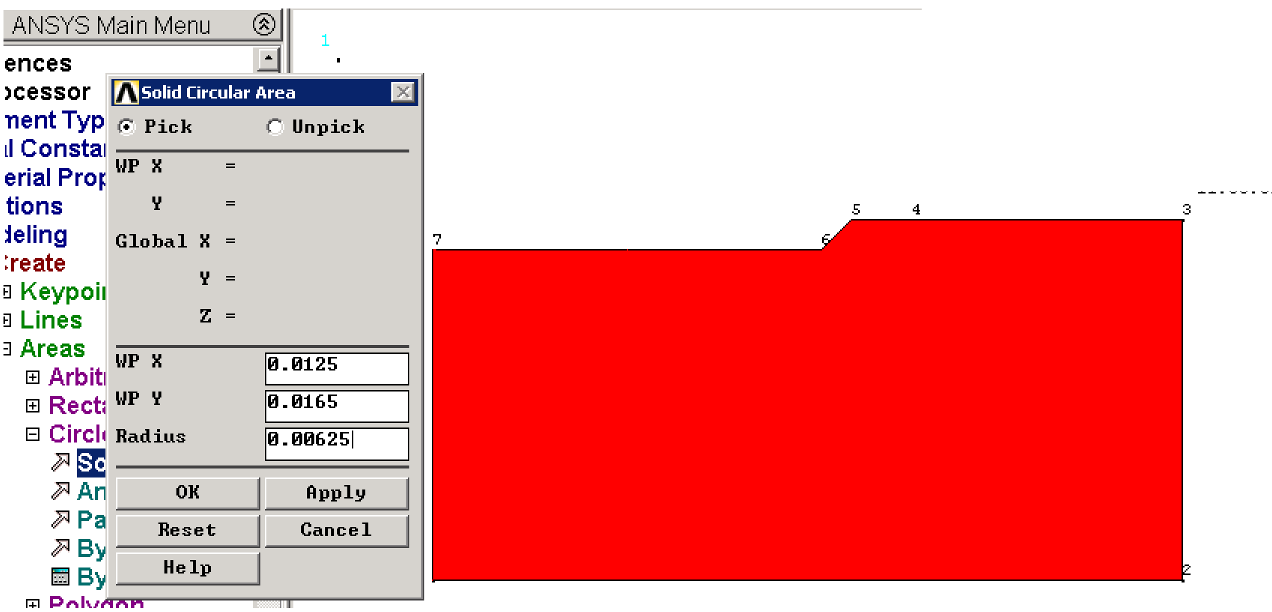


Hole (circle) placing

Main Menu>Preprocessor>Modeling>Create>Areas>Circle>Solid Circle

Click keypoint 8 and enter radius value as 0.00625m. Or; CYL4,0.0125,0.0165,0.00625

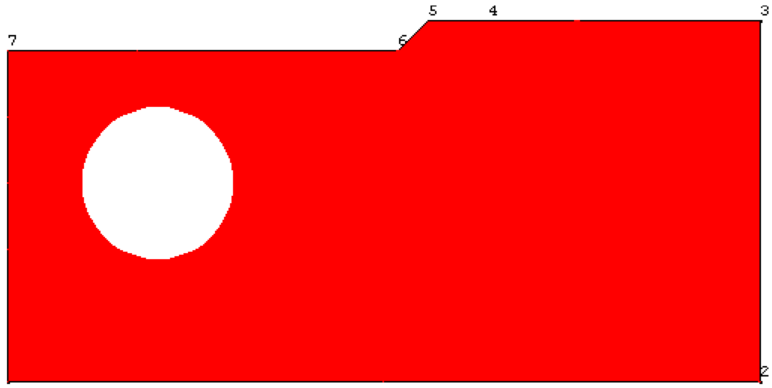




Subtracting hole area

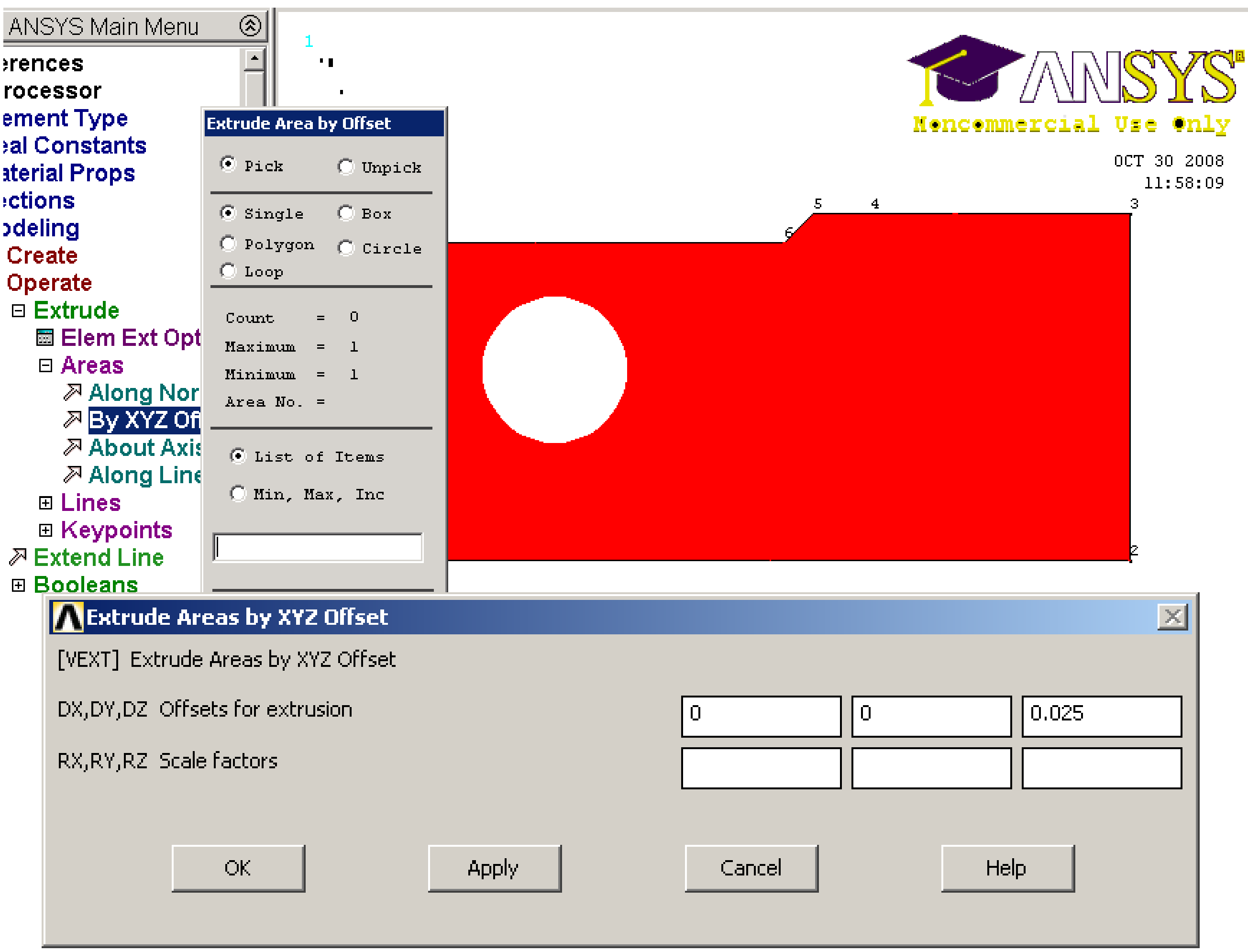
Main Menu>Preprocessor>Modeling>Operate>Booleans>Subtract>Areas

Firstly select the whole area and hit Ok then select the bigger area and click apply and then hole area that its center point is on keypoint8 is selected and click OK. Or use ASBA, 1, 2 and hit OK.



EXTRUSION

To create volume, we can easily extrude the area by 0.025m in the normal direction (*z* axes)
Main Menu>Preprocessor>Modeling>Operate>Extrude>Areas>By XYZ Offset
First, model is selected then the extrusion distance is entered.

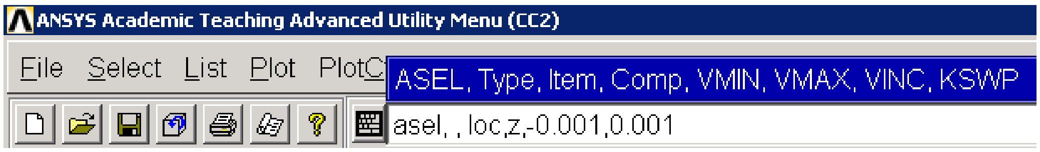


Or;

FLST,2,1,5,ORDE,1
FITEM,2,3
VEXT,P51X, , ,0,0,0.025,,,,

Meshing the Model

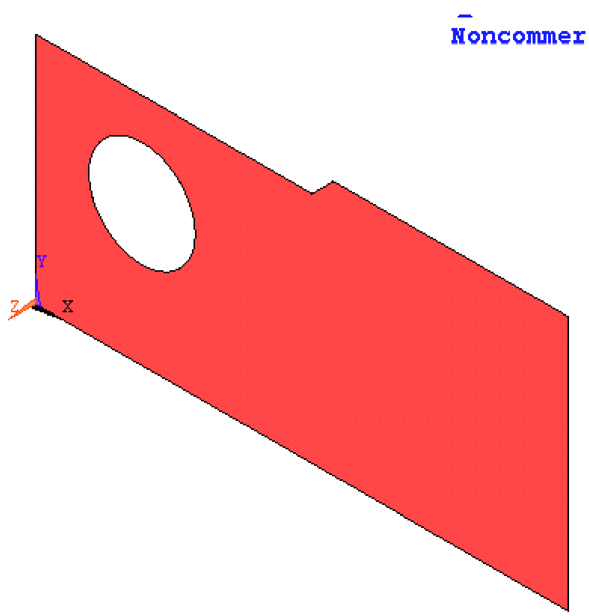
Back or front area has to be meshed first. Before meshing we can select the back area (template area) with **ASEL** command using the coordinate (location) option.



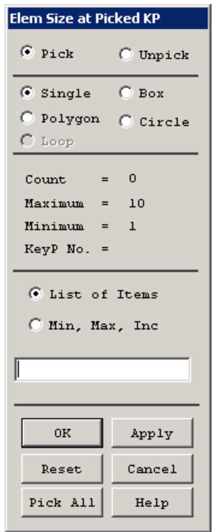
In the above command *Asel* is used to select a subset of areas that are located in the coordinate range specified. To be able to select the back area, very small distance is given between the minimum and maximum coordinate in *z* direction.

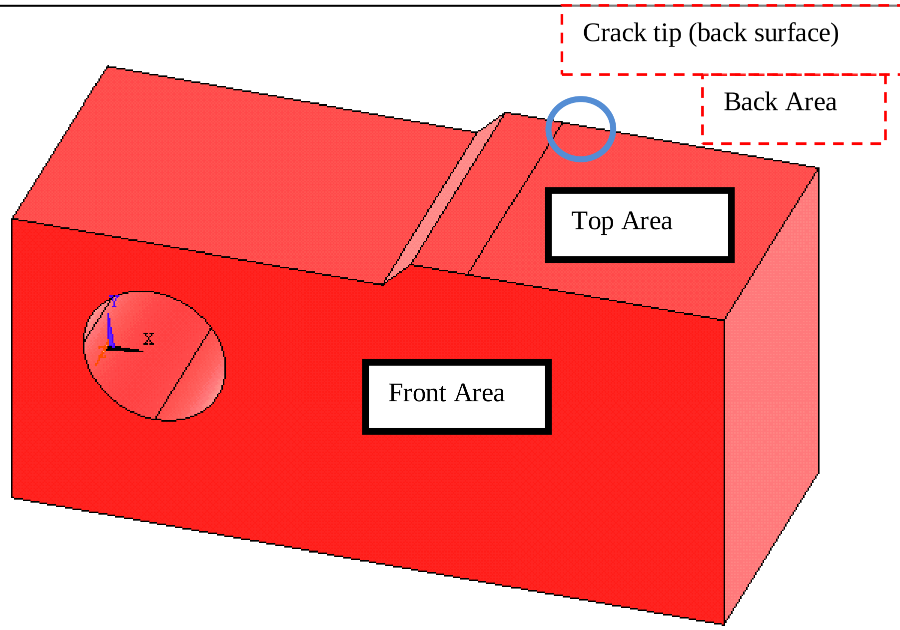
asel, , loc,z,-0.001,0.001

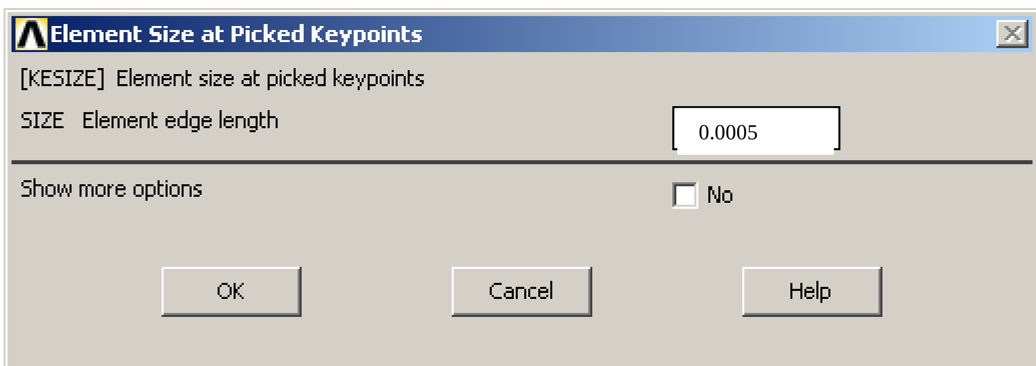
aplot



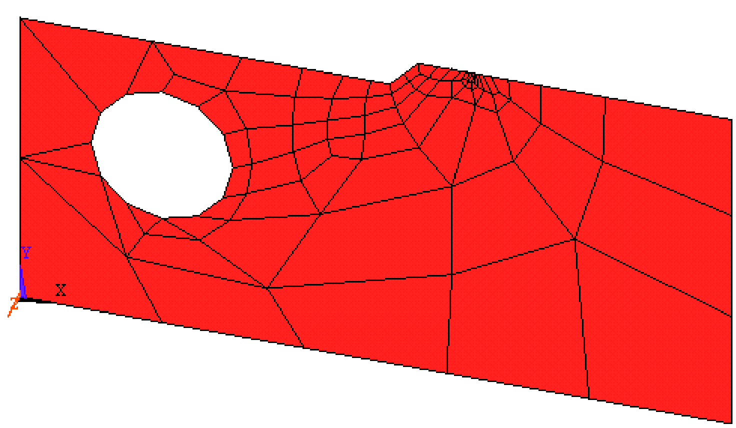
To obtain accurate fracture solution, we need to generate fine mesh near the crack tip. For this, we can use the KESIZE command to specify element size at the crack tip keypoint. First zoom into the crack tip region. Then issue the command
kesize, p
and pick **the crack tip keypoint on the back area** and write 0.0005m as the element size value SIZE in the window.







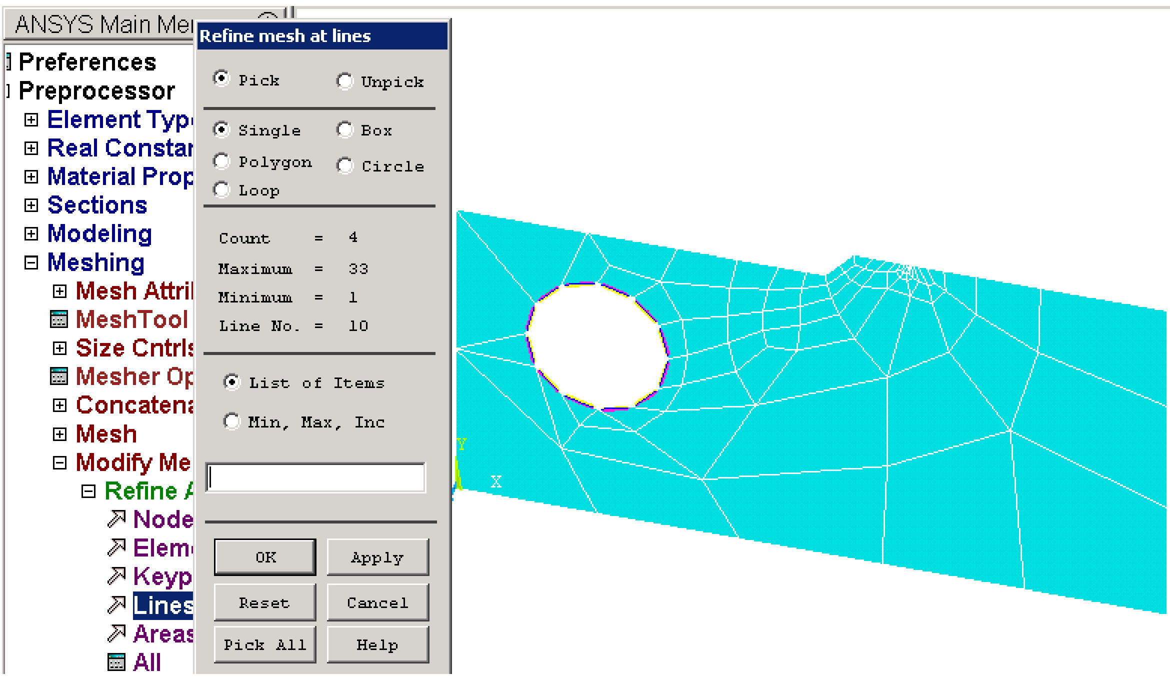
Amesh,all ' Select the back area exactly. Be careful, in selecting. Use, zoom in, perspective view or rotate the model or issue other viewing commands to select the back area.

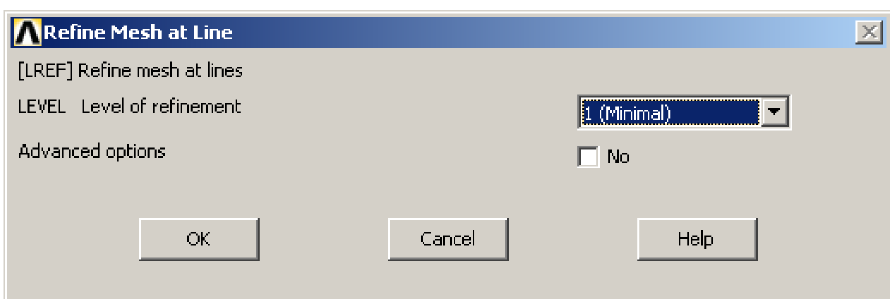


For better meshing of the hole be must refinement using
Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Lines
Use isometric view when you select hole edges. Choose 1(Minimal) value.

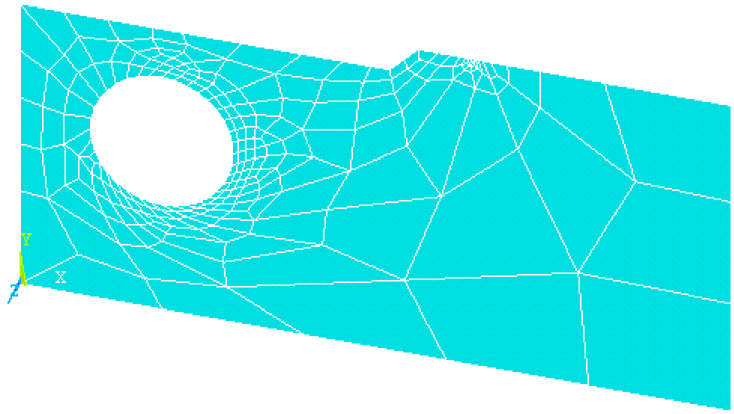
Note: All meshing and refining processes must be done in this stage. Because volume sweeping gives us hexahedral type volume mesh. Once volume sweeping is done, there is no way to return back to 2D mesh again, unless all mesh is deleted. Therefore all refinement and tuning processes on meshing stage must be finished at this point.

Now, we will refine the hole circumference. When asked by the program select all lines on the hole circumference.

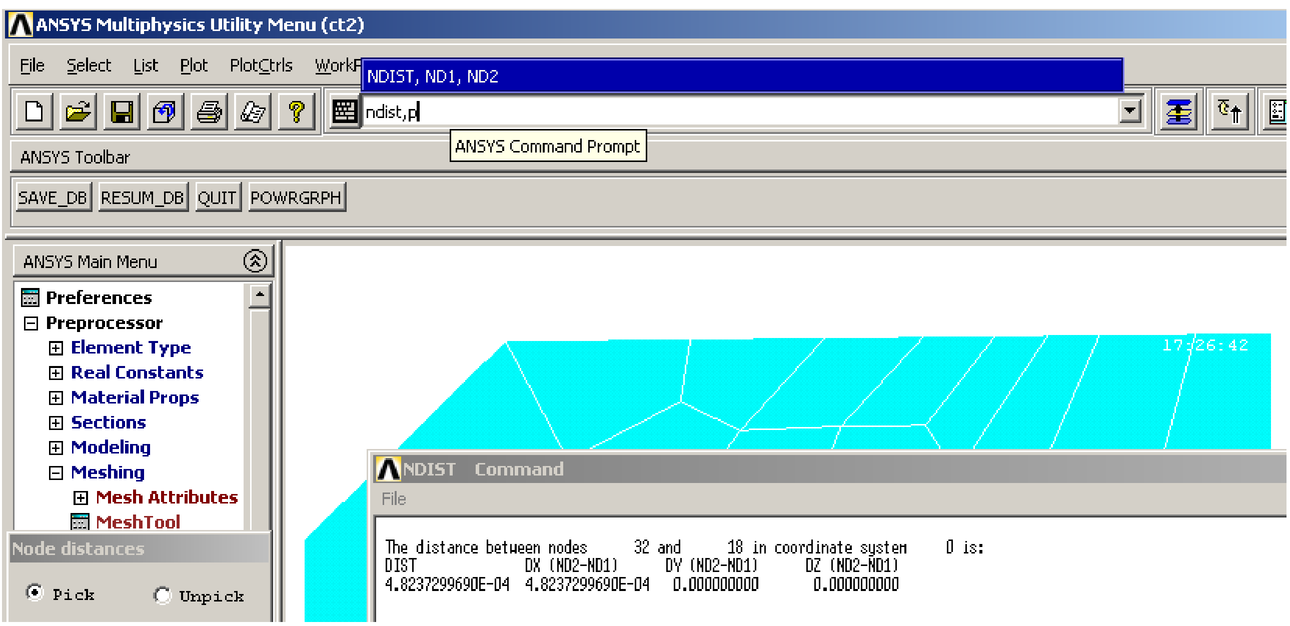




We will do a minimum level of refinement.



Next, let us check the element size near the crack tip. First, zoom into the crack region and use **Ksel** to locate the exact location of the crack tip, since we will measure the element edge sizes ahead of and behind the crack tip. To do this, we use the *ndist* command (**ndist,p**) and measure the crack tip edge size. This gives us 4.8237299690E-04, which is fine enough.



<u>**Note: If inter-node distances had not been done fine enough, we would have returned back to the refinement process again after cleaning the mesh.**</u>

As can be seen above, the specified element size is achieved.

Applying Boundary Conditions

Because of the symmetry, our system has following BC's:

1. On symmetry area (Top Area): [from (0.0225,0.03,0) to (0.0625,0.03,0)] U_y =0
2. On corner 1, constrain U_x=U_y=U_z=0
3. On corner 2, constrain U_x (to avoid the rotation about the Y axis. Note that: we could choose the corner 3 (U_z=0), instead.)

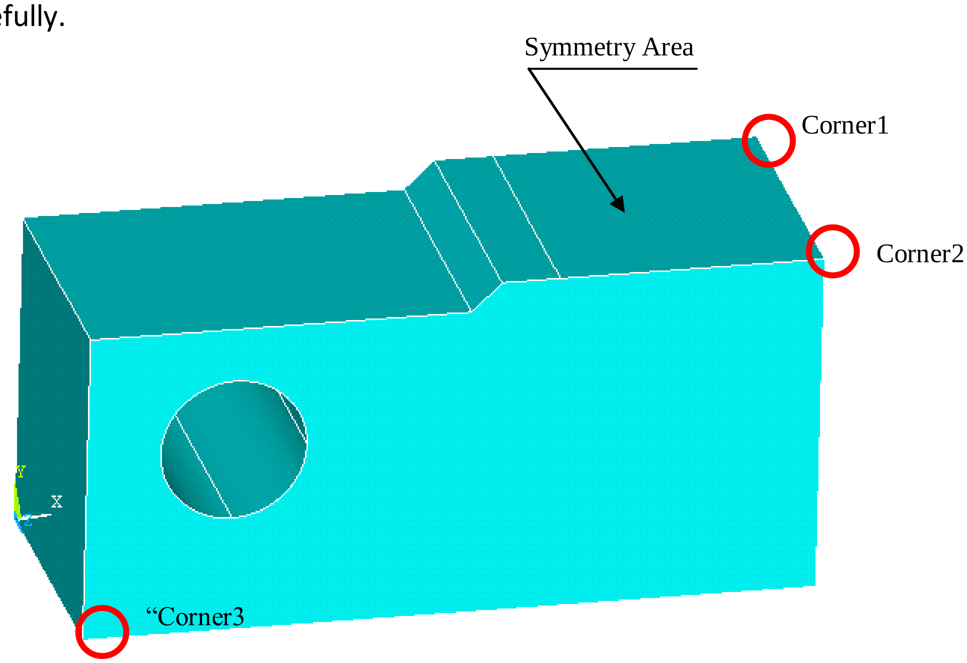
1. Apply the displacement constraint on symmetry area

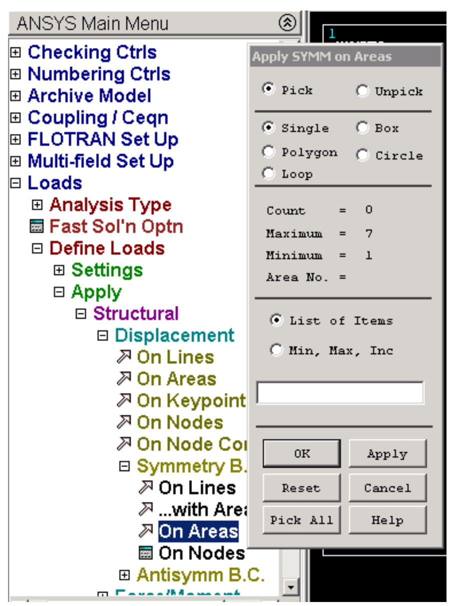
ALLSEL,ALL

aplot

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>on Area or **DA, p** for area BC.

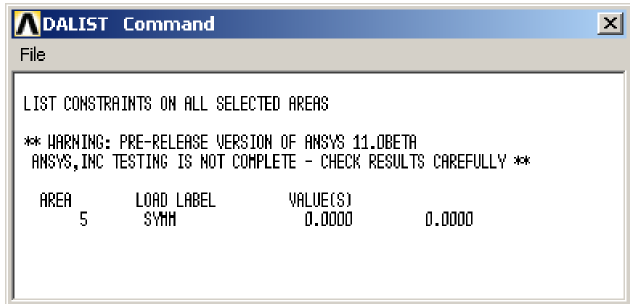
Symmetric area is top area of our model. This area must be constrained in *y* direction. Select the symmetry area carefully.





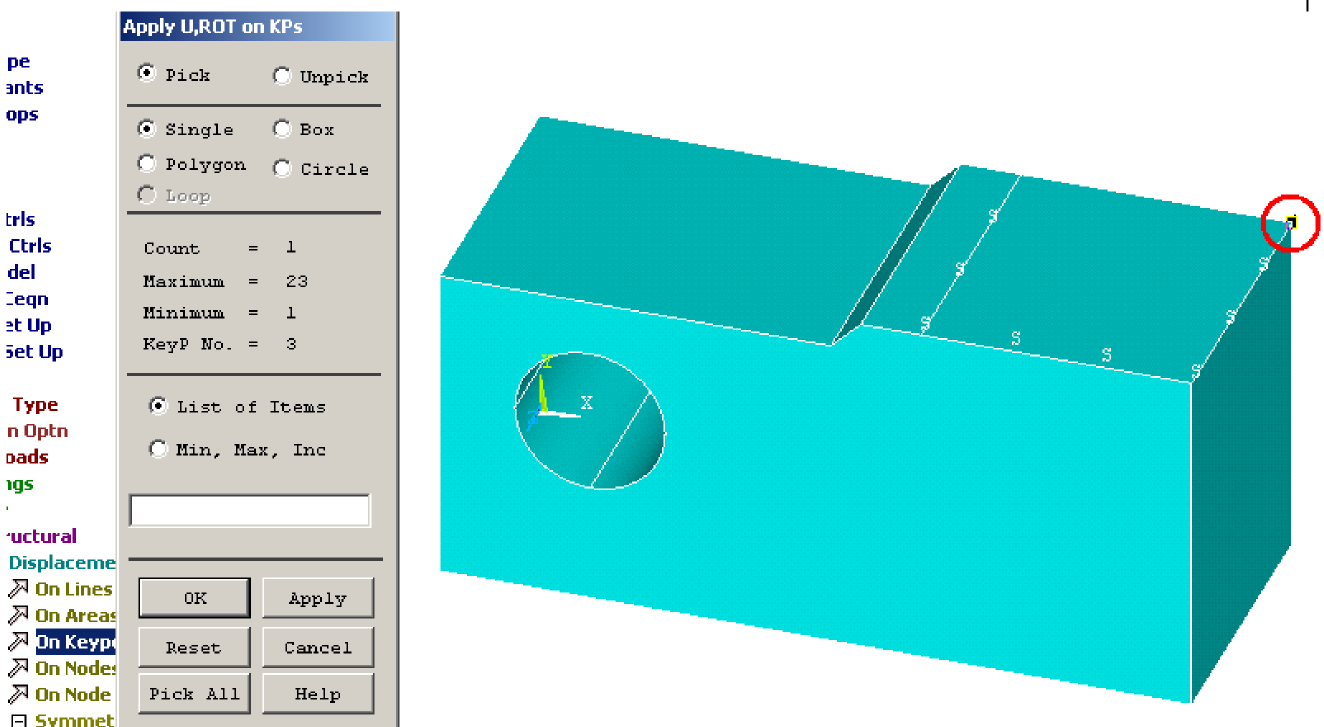
Be careful, when selecting areas. To get accurate selection you can use perspective views using **ctrl+right button**.

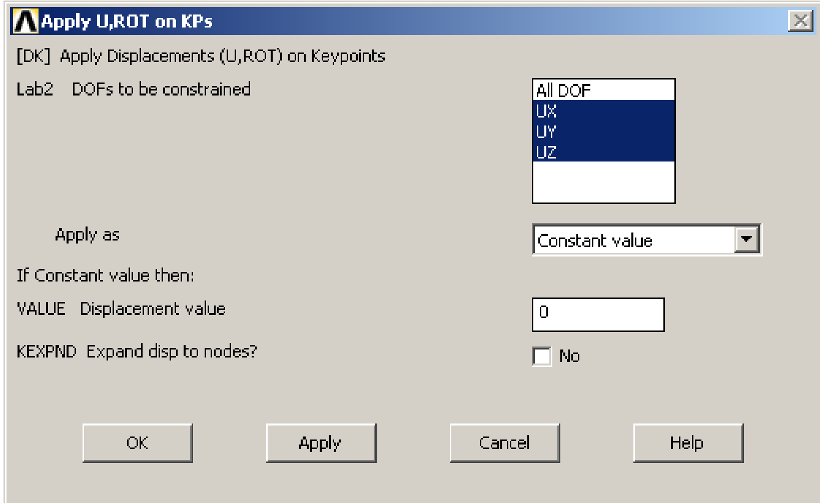
To check the applied boundary conditions on areas, **DALIST** is used in the command line



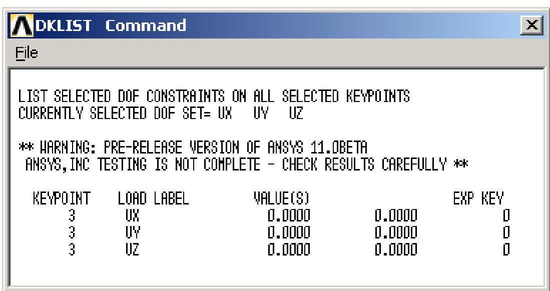
2. Apply the displacement constraint on corner 1, constraint U_x=U_y=U_z=0

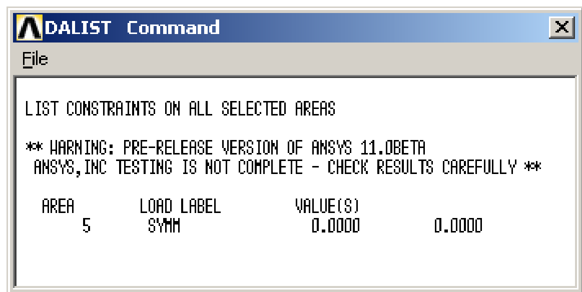
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>on Keypoints



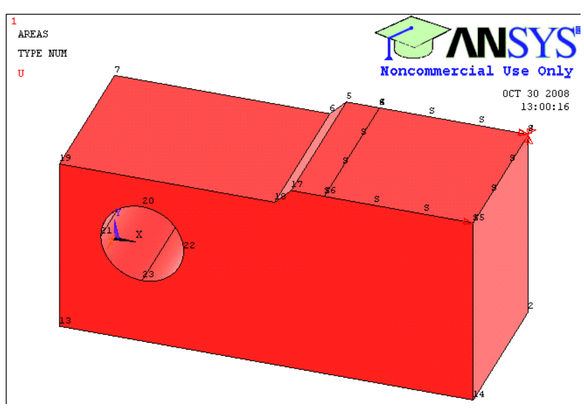


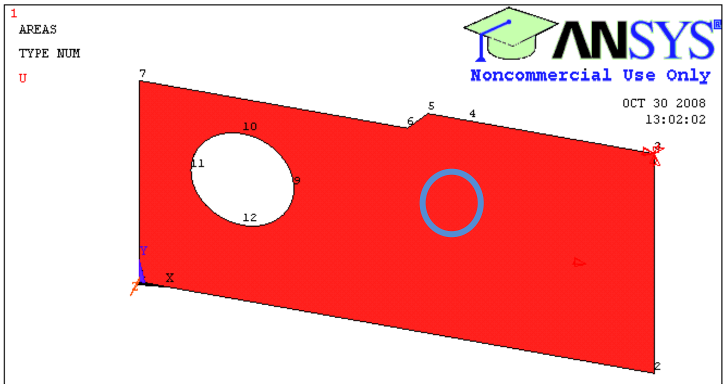
To check the applied boundary conditions on areas and keypoints, **DKLIST** and **DALIST** is used in the command line



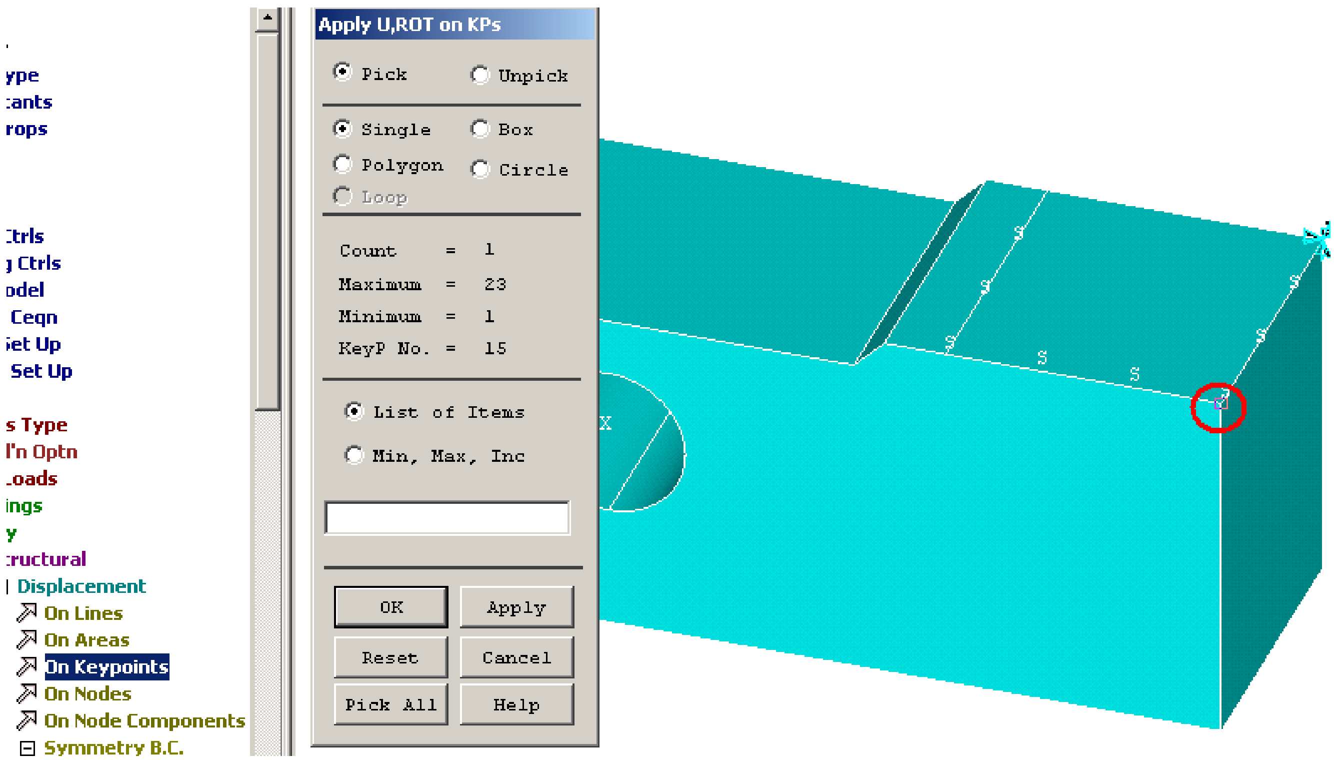


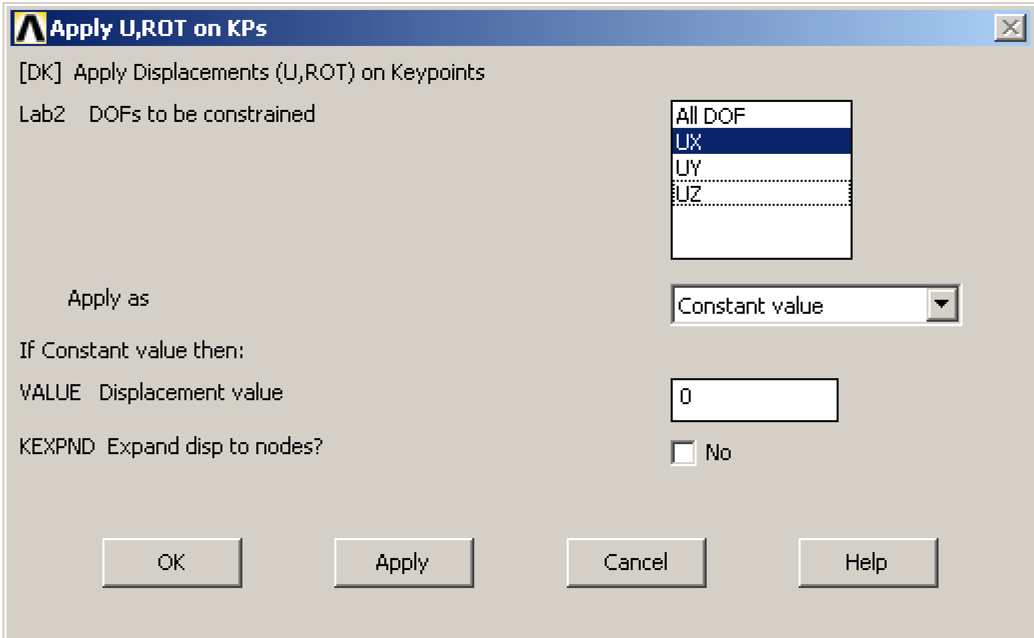
3. Apply the displacement constraint on corner 2, constraint U_x



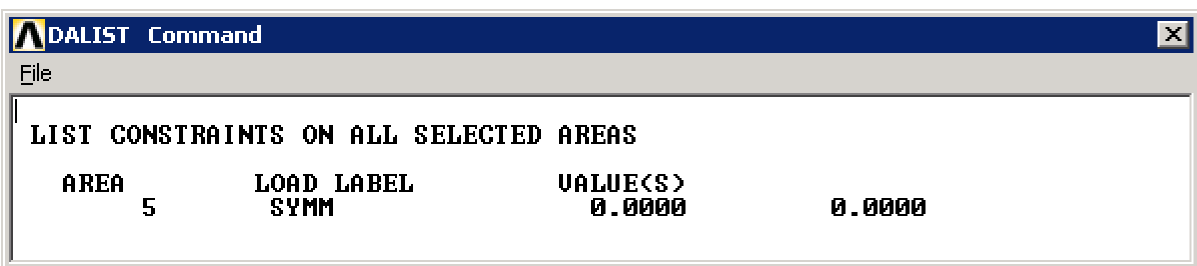


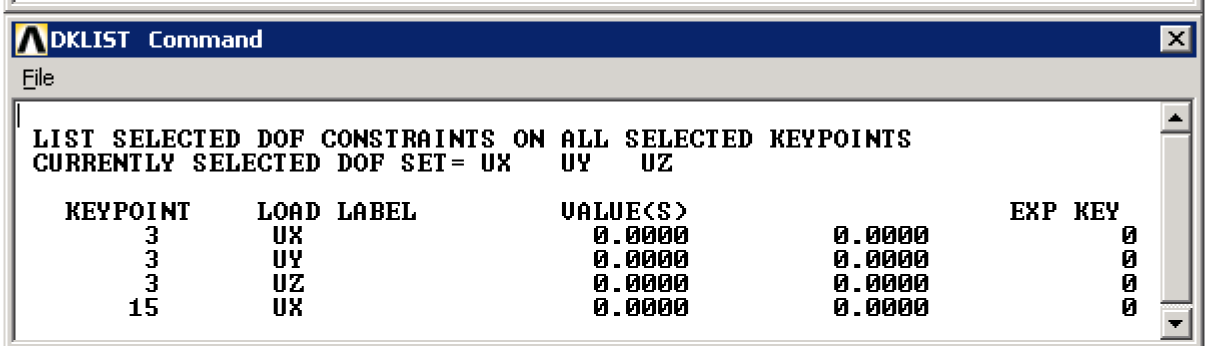
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>on Keypoints





Now let's list the last BC's.





Or;

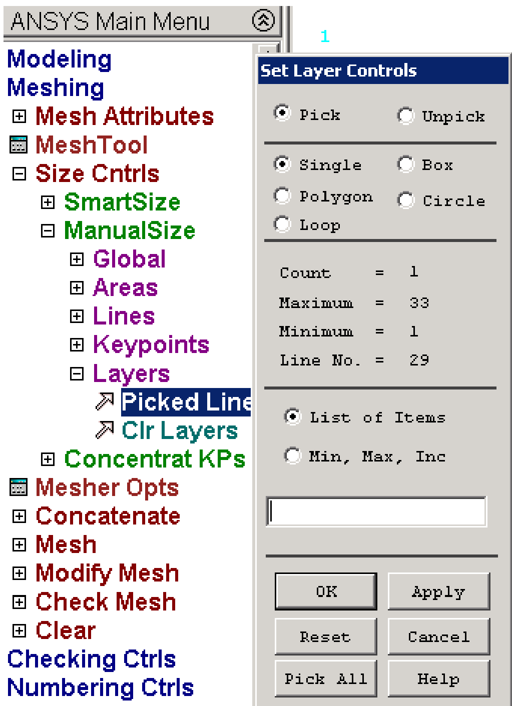
```
DA,      5,SYMM
FLST,2,1,3,ORDE,1
FITEM,2,3
FLST,2,1,3,ORDE,1
FITEM,2,3
!*
/GO
DK,P51X, ,0, ,0,UX,UY,UZ, , , ,
FLST,2,1,3,ORDE,1
FITEM,2,15
```

!*
/GO
DK,P51X, ,0, ,0,UX, , , , , ,

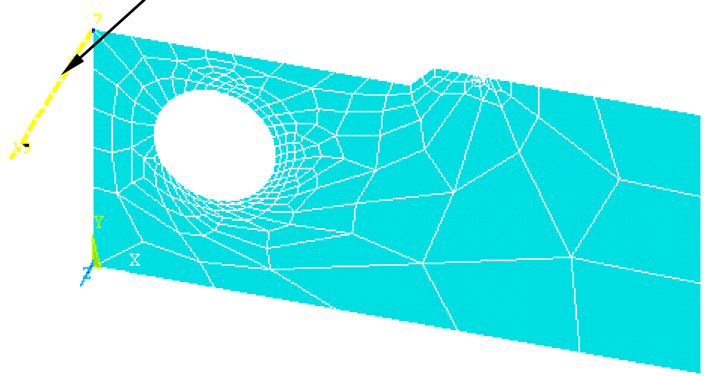
Volume Sweeping

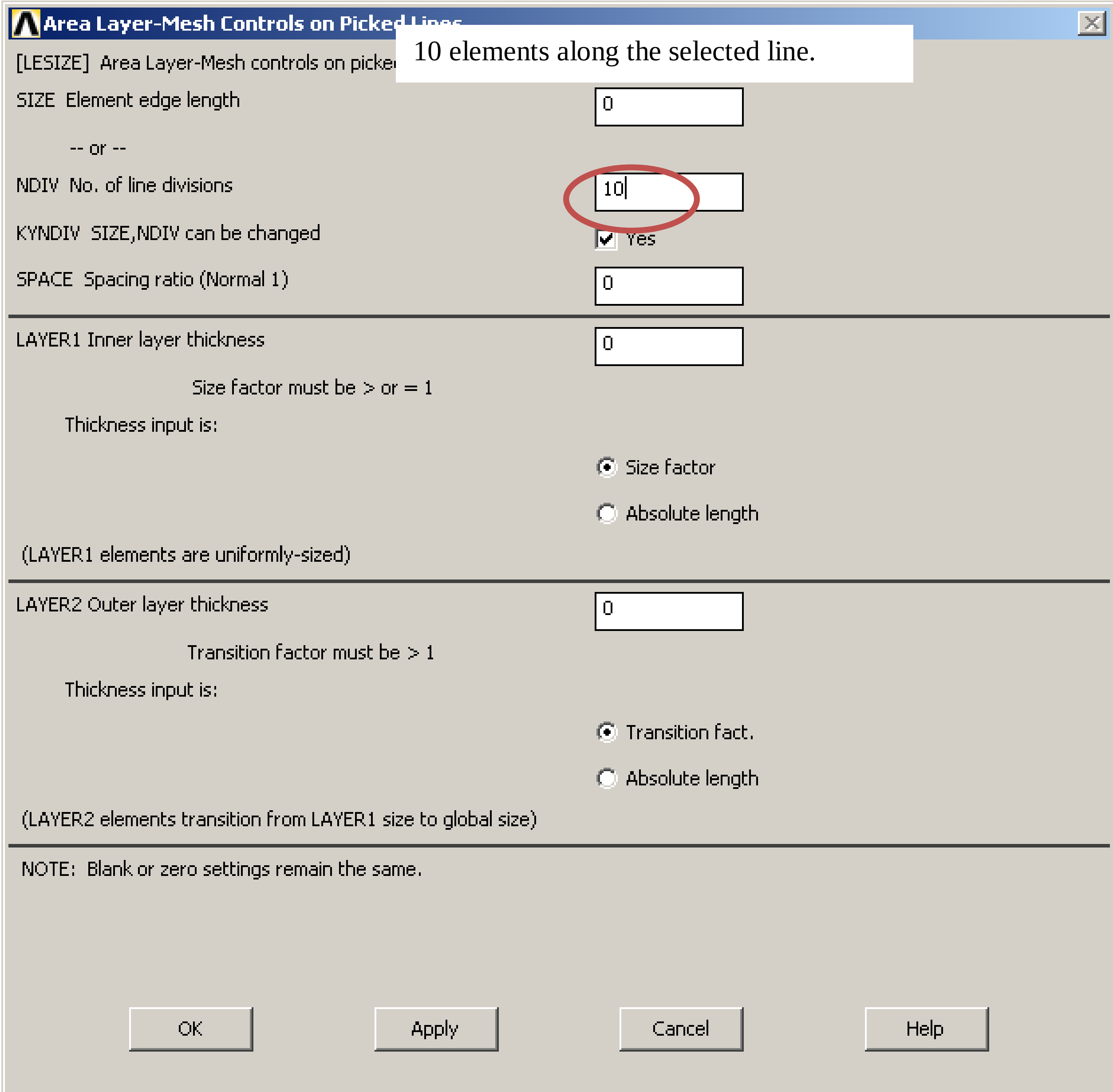
First, let's define layer size for the mesh to extruded.

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Layers>Picked Line

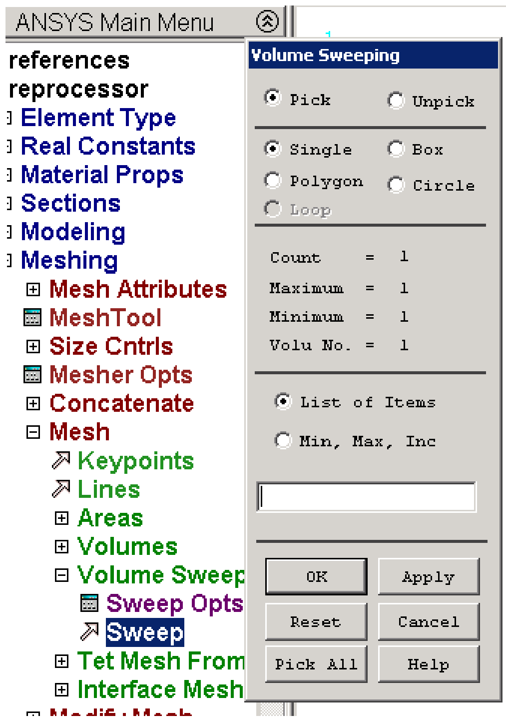


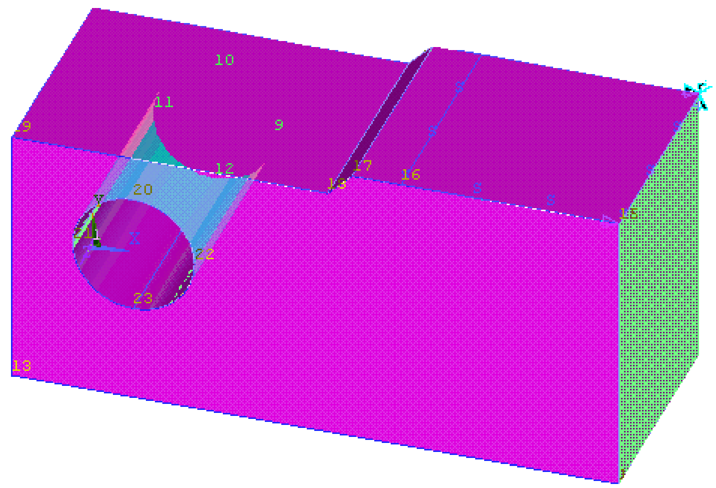
Select the on eline in *z* direction

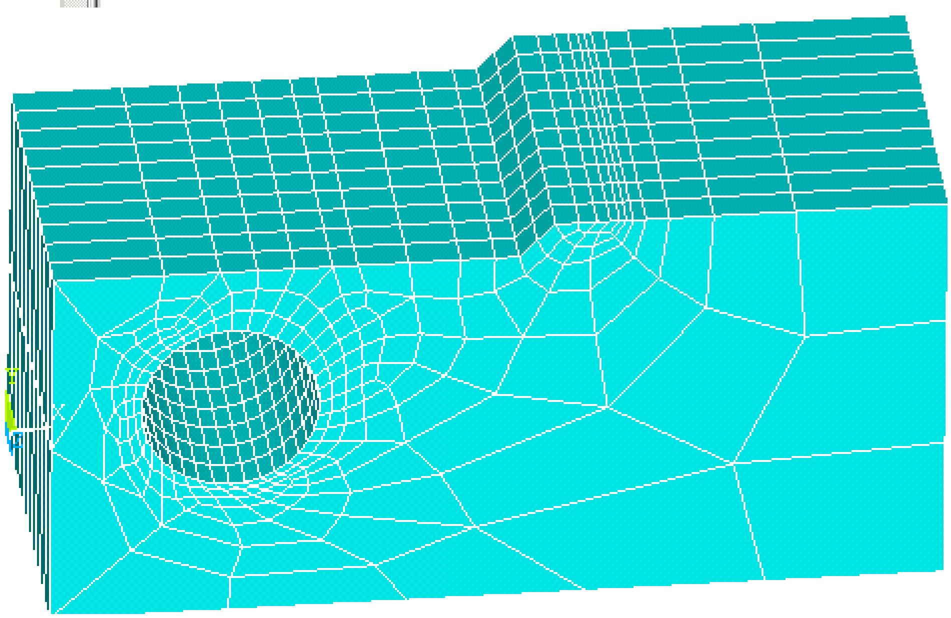




Now we will extrude the 2D mesh in *z* direction and 20 layers will be generated in this direction.
Main Menu>Preprocessor>Meshing>Mesh>Volume>Volume sweep>Sweep



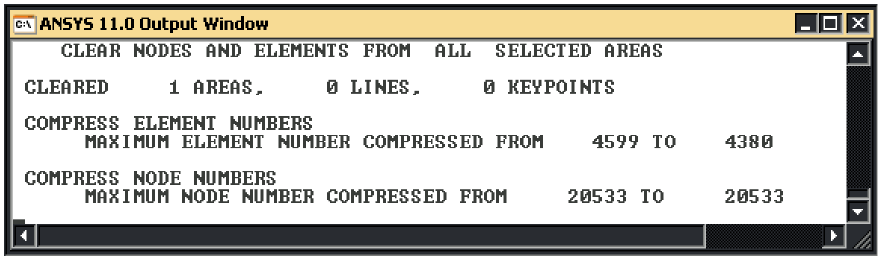




Volume sweeping gives us prismatic mesh in the bounded volume, with respect to area mesh (template mesh). After sweeping the 2D mesh to generate the 3D mesh, we need to delete 2D elements, since FRAC3D requires 3D elements only.

To do this, we use **Aclear, all** ' Area mesh is deleted. Because of this, a gap occurs in the sequence of element numbers. To remove the gap we use, **Numcmp, elem**

To make sure no gap exists in the node numbers as well, we use **Numcmp, node**



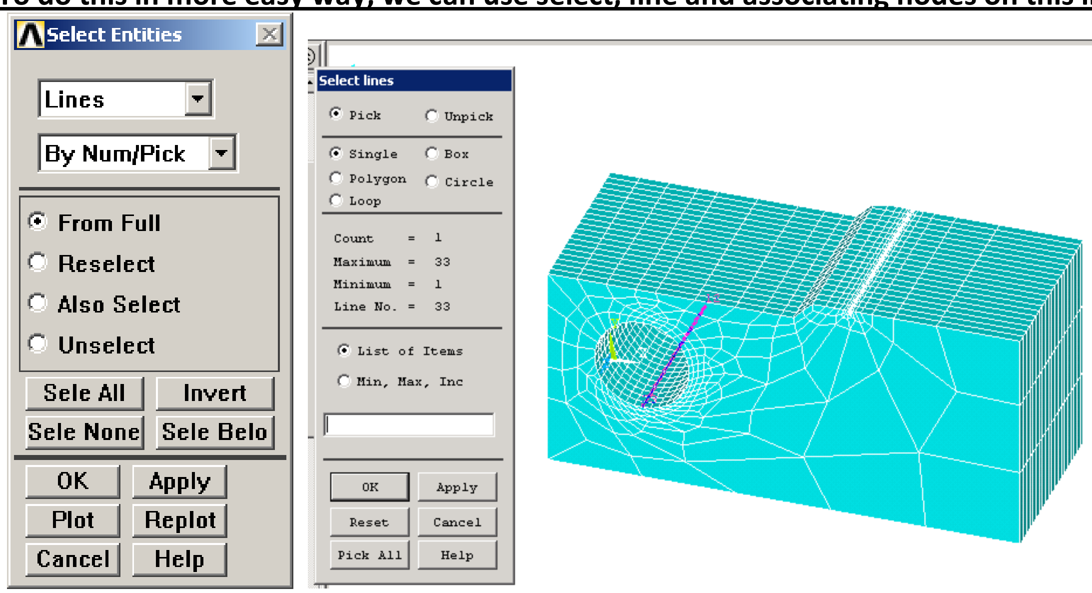
Applying Loads

Now we will apply the pin loads on a line along inner surface of the hole. To do this, we can apply concentrated forces on nodes located on these lines (We can not apply concentrated forces on line entities).

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>On nodes

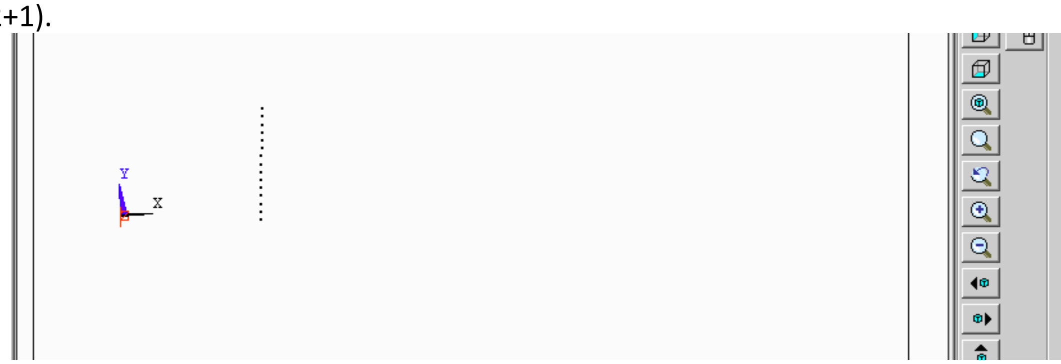
Carefully pick the nodes on the bottom line and then click OK in the picking window.

<u>NOTE: To do this in more easy way, we can use select, line and associating nodes on this line.</u>

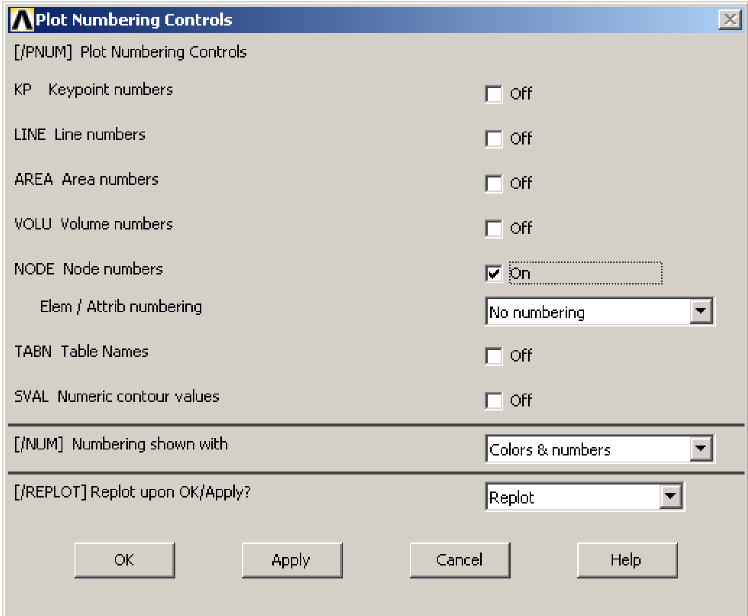


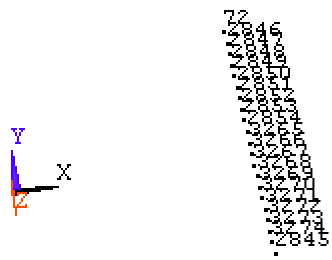
nsll,s,1
nplot

This select all nodes associated with the line selected). Note that 41 nodes are selected (20 layersx2+1).

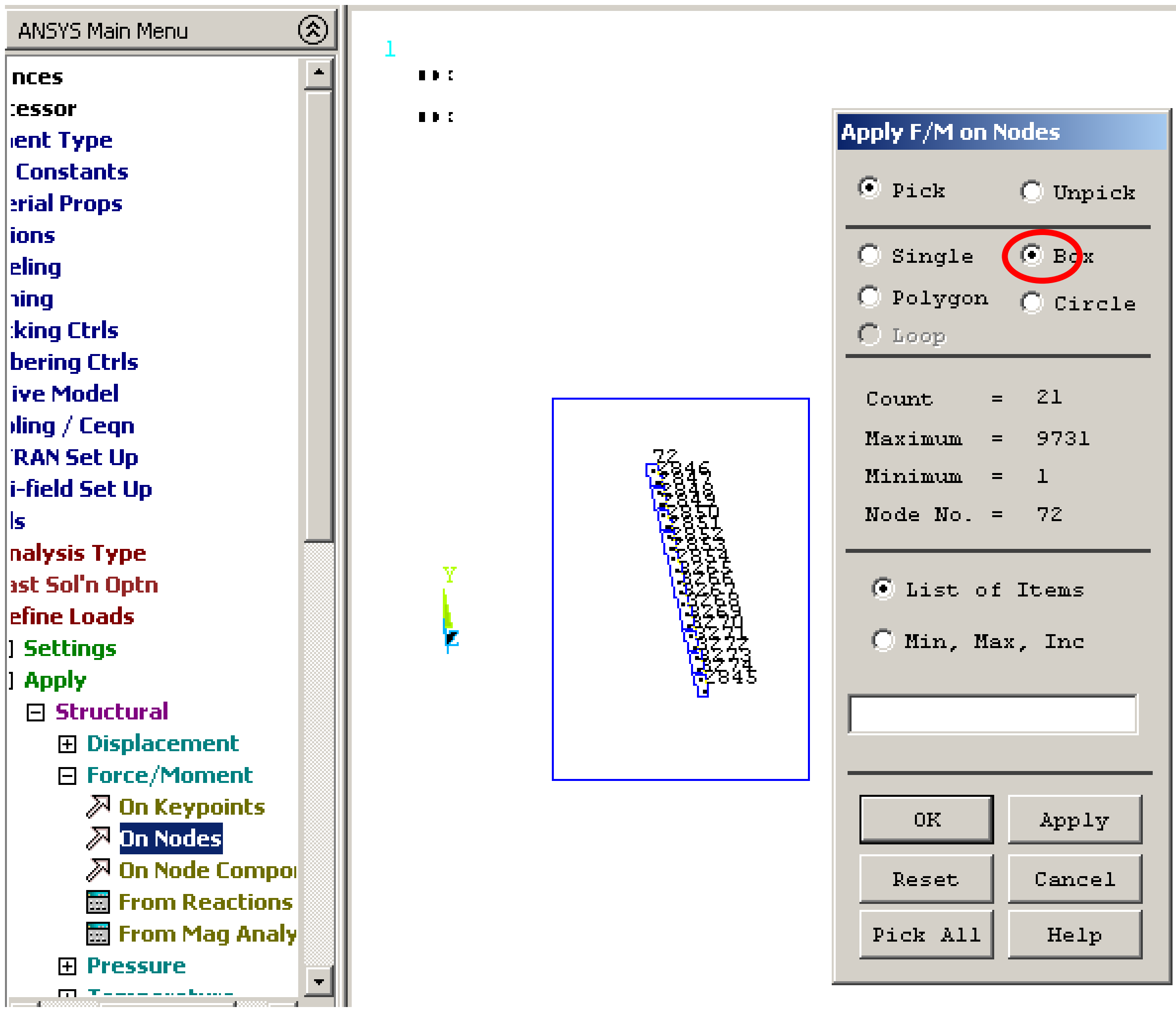


Use *menu-pltctrl-numbering* and *switch* the nodes *on* to show the numbers of the nodes.

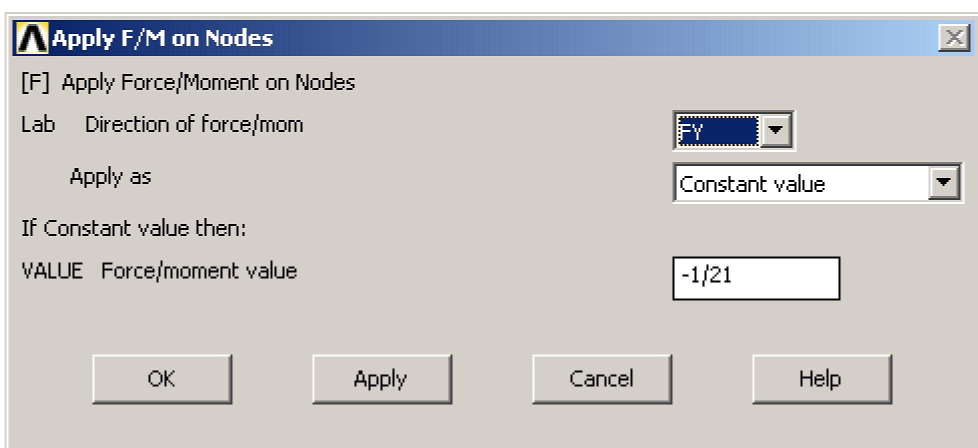




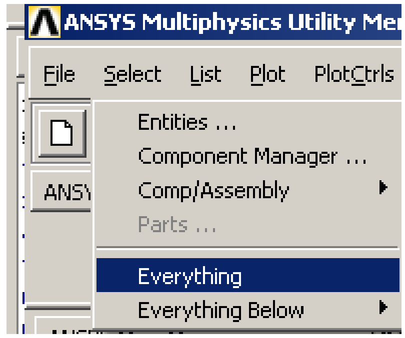
Now we will apply the force to hole’s baseline.
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>On nodes

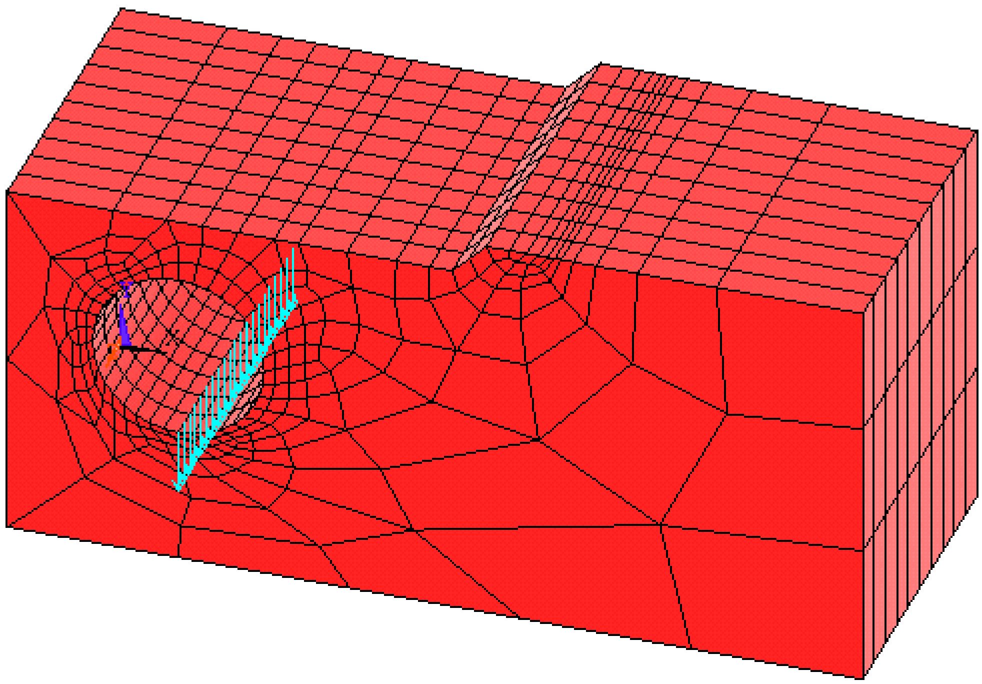


Select F_y direction, constant value and enter "-1/41" for load value, then click OK. Note that; force is applied on the 41 nodes.



Select-everything. Eplot

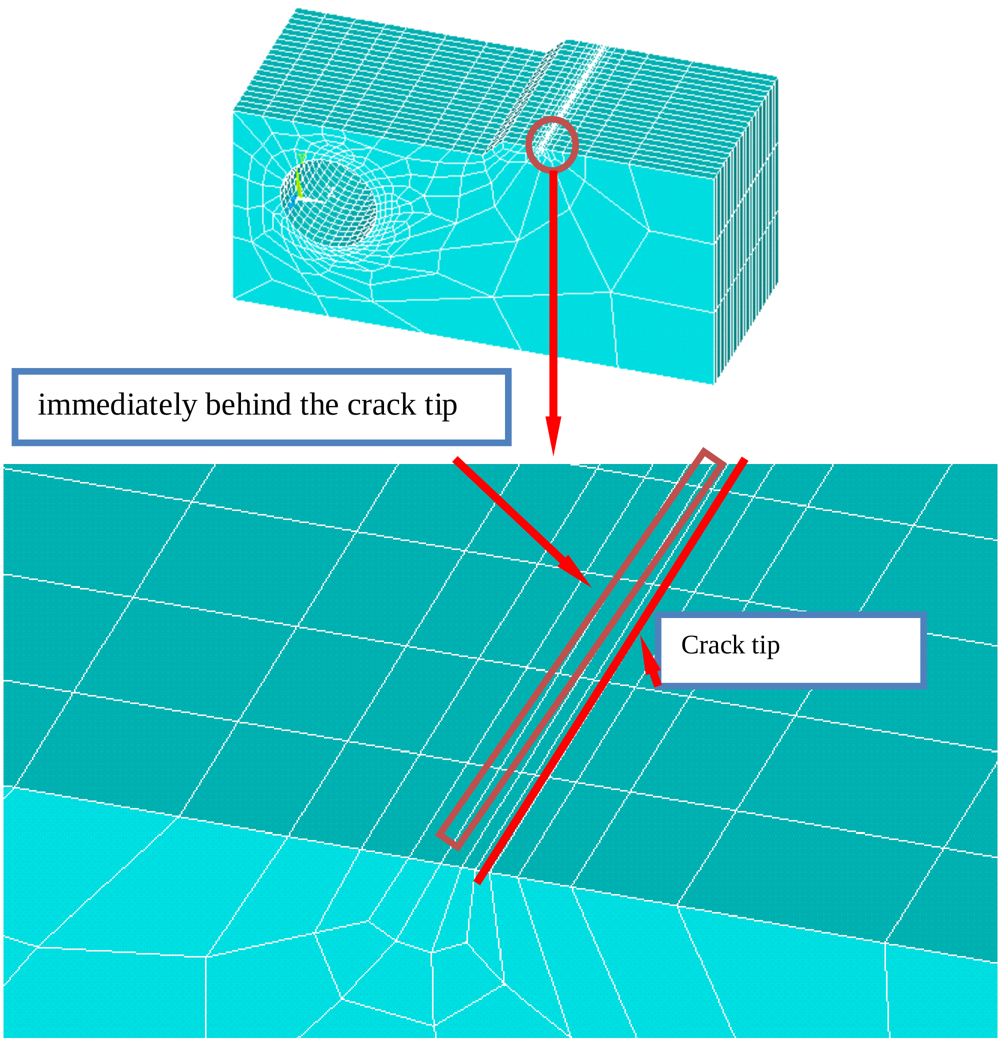

ANSYS Multiphysics Utility Me
File
Select
List
Plot
PlotCtrls
ANSY
Entities ...
Component Manager ...
Comp/Assembly
Parts ...
Everything
Everything Below



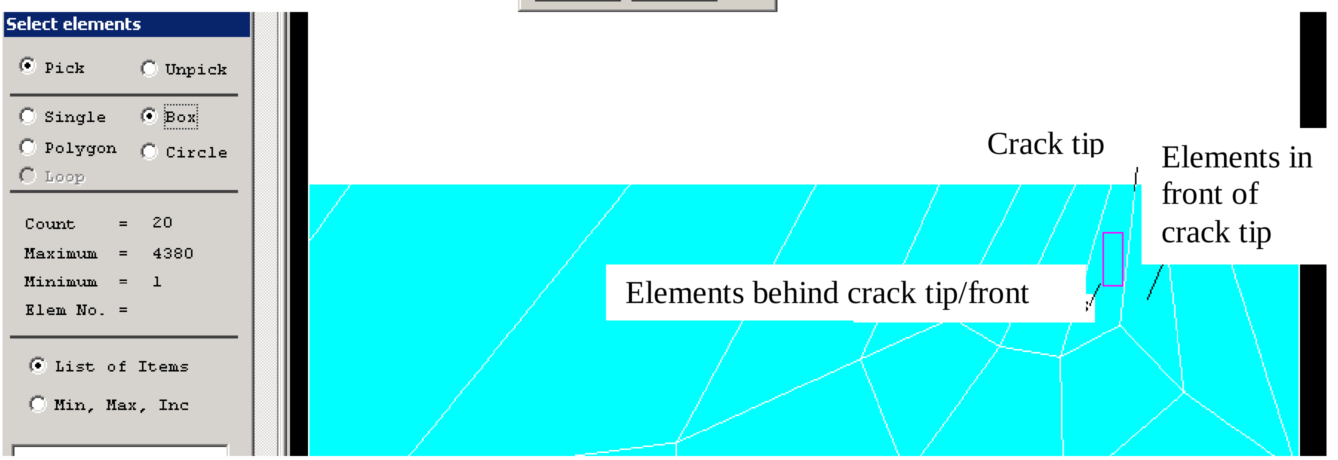
Definition of Crack for FRAC3D

Now, we need to provide crack tip element and node number information for FRAC3D analysis. Zooming into the crack region, we can find which element and which nodes are located at the crack tip. (See detailed crack tip definition requirements in this tutorial for which elements and nodes to be selected). We need to identify the crack tip element on the bottom crack surface (with respect to chosen local coordinate system) immediately behind the crack tip. Using, **Select-Entities-Elements,** form the main menu (or Esel, p command) try to select the elements at the crack tip. For this, move your mouse pointer near to crack tip region. We have to select the element which is both at crack tip and on the crack surface.

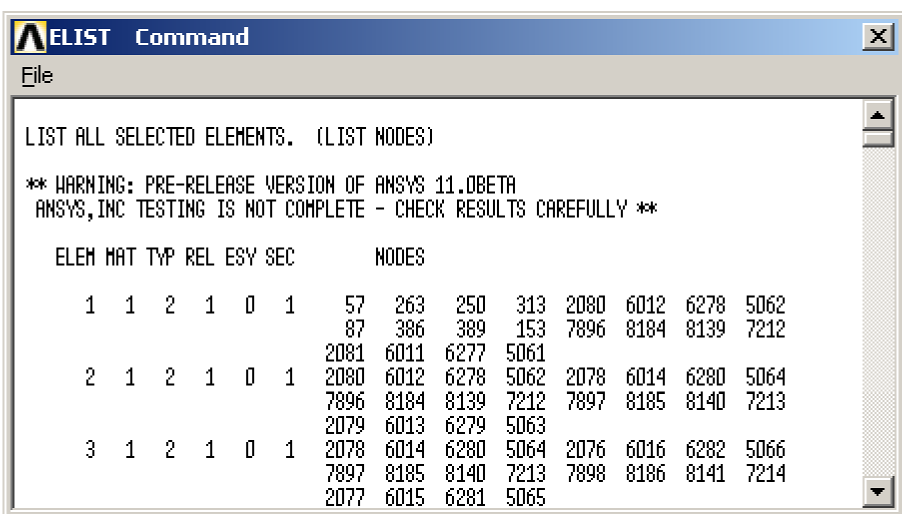
Vplot
eplot



Use *menu-pltctrl-numbering* and switch the nodes off to not show the numbers of the nodes.

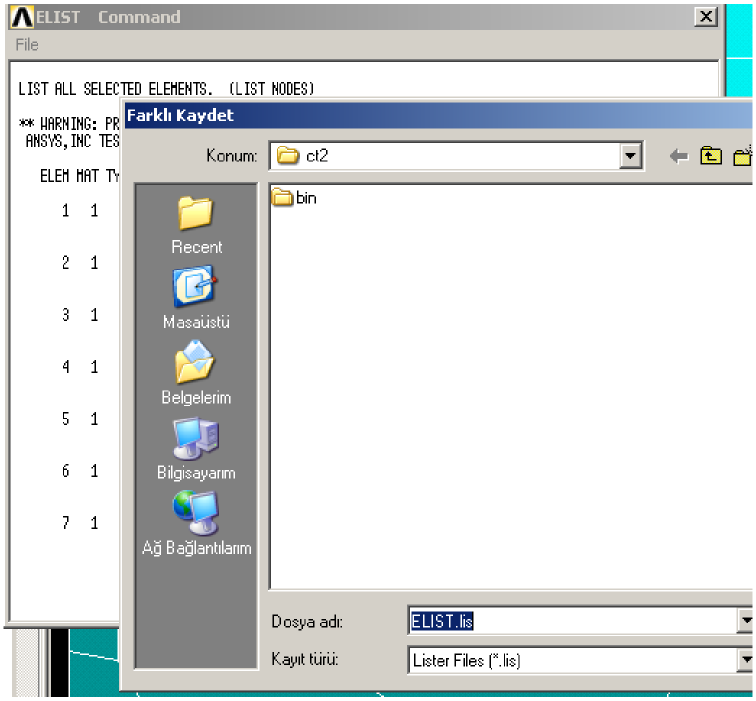


After element selection, we list the selected element and save the file for FRAC3D analysis. Use **Elist** to see and save the list as a file.

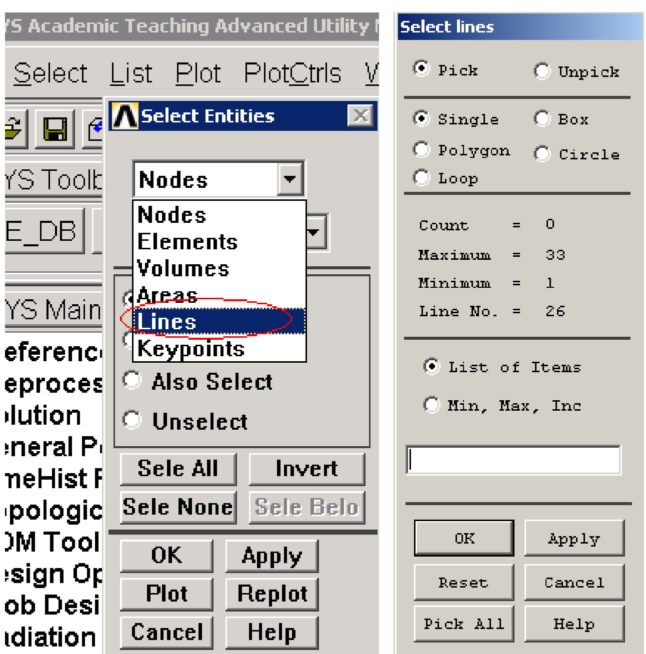


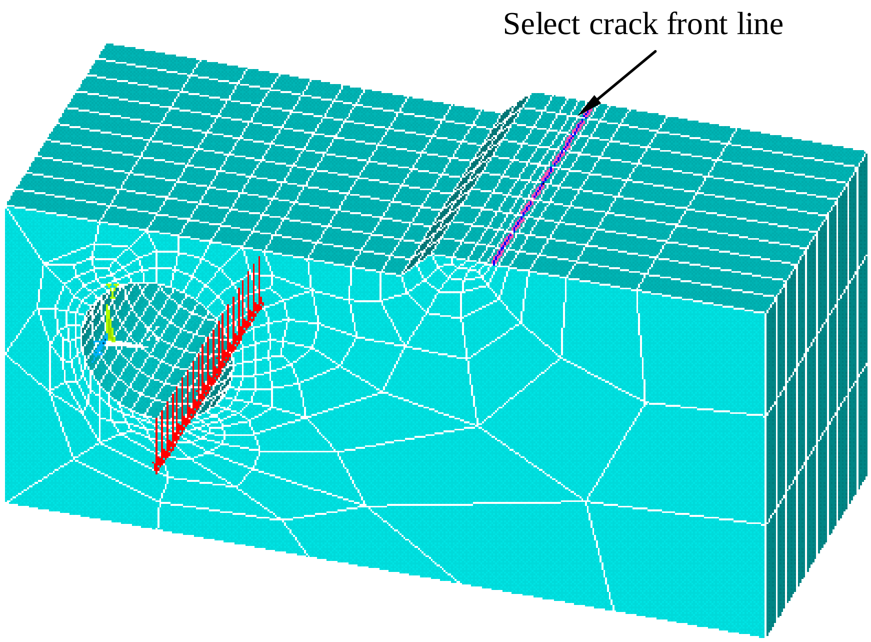

```
LIST ALL SELECTED ELEMENTS.  (LIST NODES)

** WARNING: PRE-RELEASE VERSION OF ANSYS 11.0BETA
 ANSYS,INC TESTING IS NOT COMPLETE - CHECK RESULTS CAREFULLY **

   ELEM MAT TYP REL ESY SEC        NODES

      1   1   2   1   0   1     57   263   250   313  2080  6012  6278  5062
                                87   386   389   153  7896  8184  8139  7212
                              2081  6011  6277  5061
      2   1   2   1   0   1   2080  6012  6278  5062  2078  6014  6280  5064
                              7896  8184  8139  7212  7897  8185  8140  7213
                              2079  6013  6279  5063
      3   1   2   1   0   1   2078  6014  6280  5064  2076  6016  6282  5066
                              7897  8185  8140  7213  7898  8186  8141  7214
                              2077  6015  6281  5065
```

The element information (for the crack tip elements) is saved from the Elist window as ***ct2.crelems***

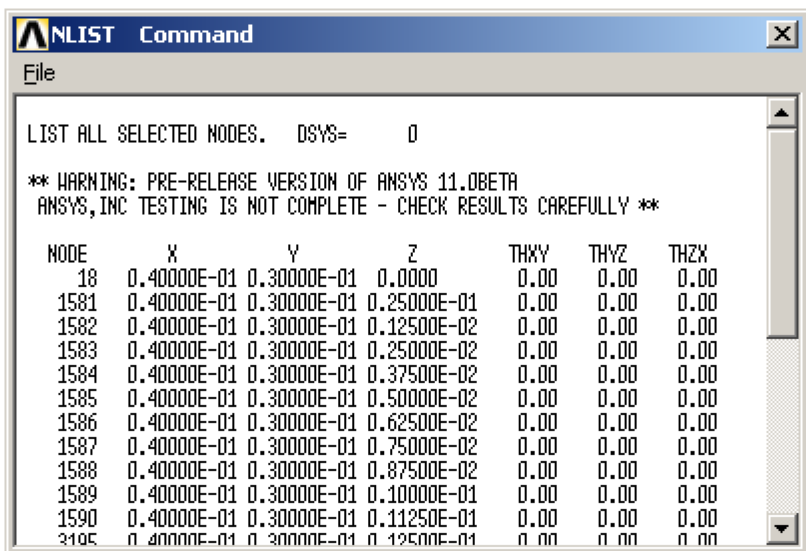


In addition to crack tip elements, we also need to select nodes on the crack front. To be able to select the nodes of the crack front, it is required to select the crack tip line and then select the nodes associated with this line. Using, **Select-Entities-Lines**, crack tip line is selected. Then, **NSLL, S, 1** is used to select all the nodes along the selected crack front line. Using **Nlist**, we can see that the crack front nodes numbers are: **18,2199,….**





NSLL, S, 1
nlist

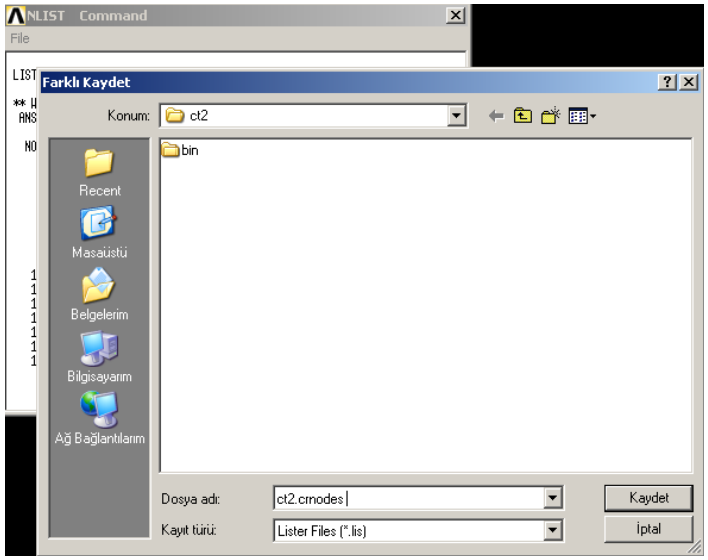


NLIST Command

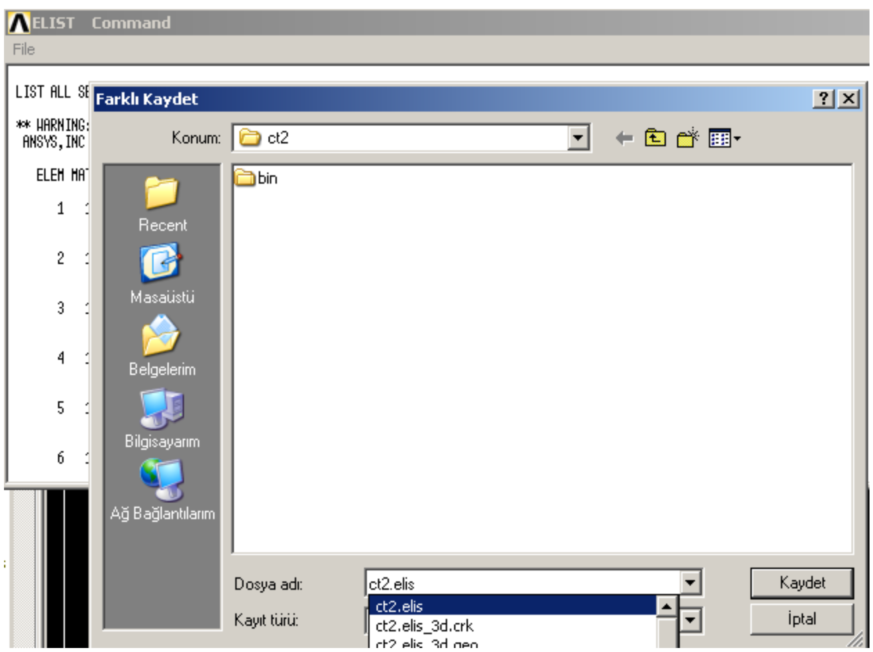
File

```
LIST ALL SELECTED NODES.   DSYS=      0

** WARNING: PRE-RELEASE VERSION OF ANSYS 11.0BETA
 ANSYS,INC TESTING IS NOT COMPLETE - CHECK RESULTS CAREFULLY **

  NODE        X             Y             Z           THXY     THYZ     THZX
    18   0.40000E-01 0.30000E-01  0.0000          0.00     0.00     0.00
  1581   0.40000E-01 0.30000E-01 0.25000E-01      0.00     0.00     0.00
  1582   0.40000E-01 0.30000E-01 0.12500E-02      0.00     0.00     0.00
  1583   0.40000E-01 0.30000E-01 0.25000E-02      0.00     0.00     0.00
  1584   0.40000E-01 0.30000E-01 0.37500E-02      0.00     0.00     0.00
  1585   0.40000E-01 0.30000E-01 0.50000E-02      0.00     0.00     0.00
  1586   0.40000E-01 0.30000E-01 0.62500E-02      0.00     0.00     0.00
  1587   0.40000E-01 0.30000E-01 0.75000E-02      0.00     0.00     0.00
  1588   0.40000E-01 0.30000E-01 0.87500E-02      0.00     0.00     0.00
  1589   0.40000E-01 0.30000E-01 0.10000E-01      0.00     0.00     0.00
  1590   0.40000E-01 0.30000E-01 0.11250E-01      0.00     0.00     0.00
```

The coordinates of the selected nodes (the crack tip nodes) are saved from the **Nlist window** as ***ct2.crnodes***



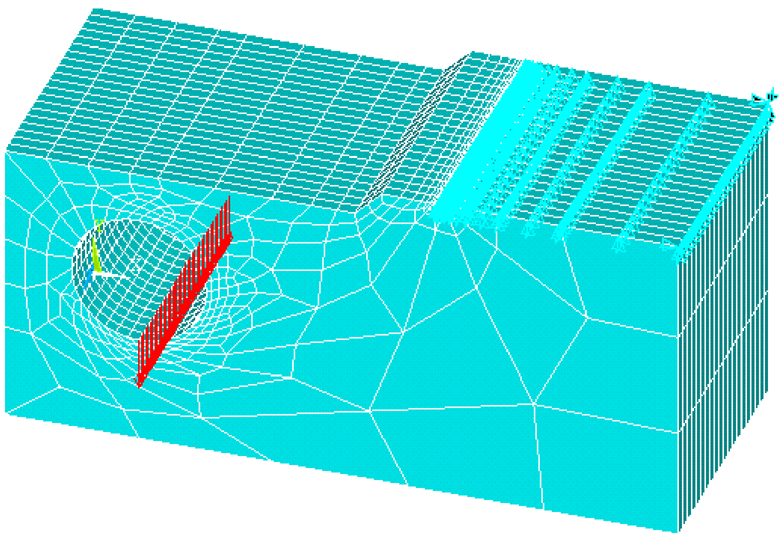
Also using **Select-Everything** the whole element list is saved ***ct2.elis***
Allsll,all
elist



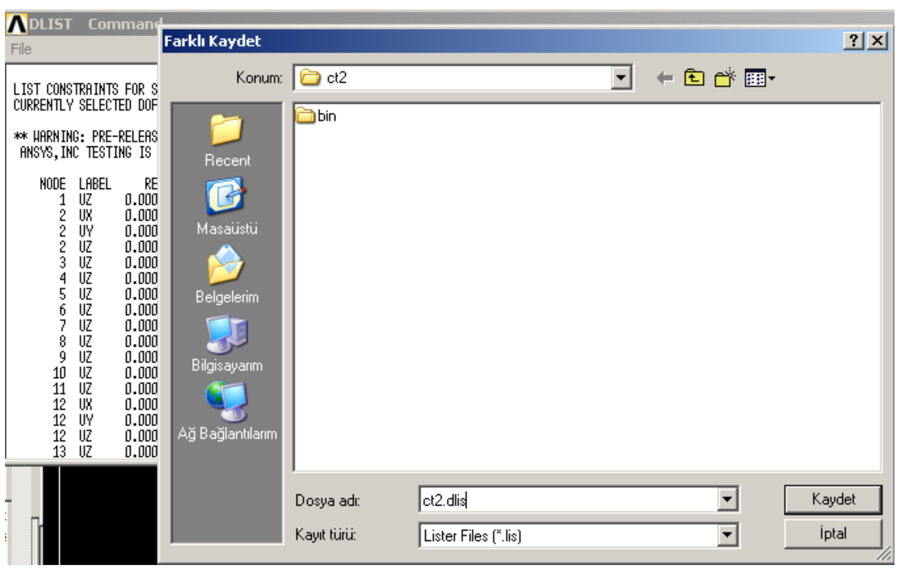
Also using **nwrite** all nodes are saved as ***ct2.node*** automatically in current working directory.

SBCTRAN is used to transfer solid model loads and boundary conditions to the FE model. Loads and boundary conditions on unselected keypoints, lines, areas, and volumes are not transferred.

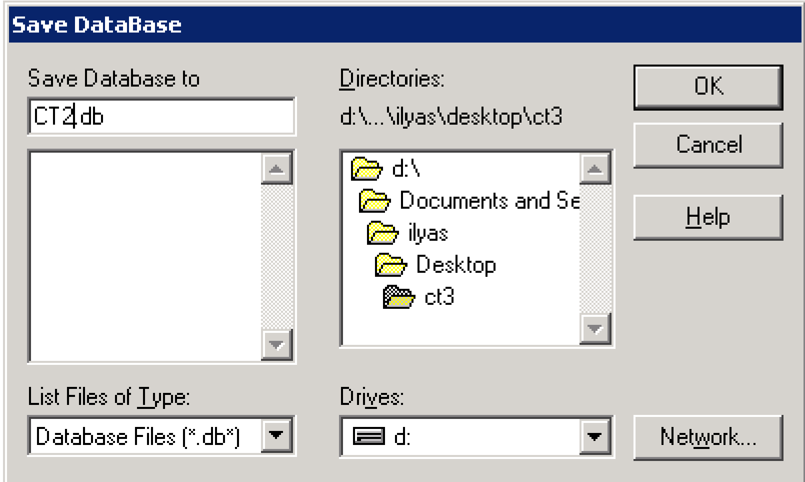
sbct



Using **dlist** displacement BC's are saved as ***ct2.dlis***



And using **flist** the nodal loads are saved as ***ct2.flis***



Now, we completed all modeling steps in ANSYS™. Now, we can save the model and close ANSYS™. We are ready to convert all the model information into FRAC3D format using the converter program.

T.2.3. Using converter codes for FRAC3D (Generation of *ct2.geo* File)

FRAC3D requires its model information in a specific format. To convert ANSYS™ model files into FRAC3D format, we can use the *convert_ansys_frac3d.exe* program. The converter program can be run by typing, its path in MSDOS prompt or from the "Geo File" tab from FCPAS. Both methods are shown respectively.

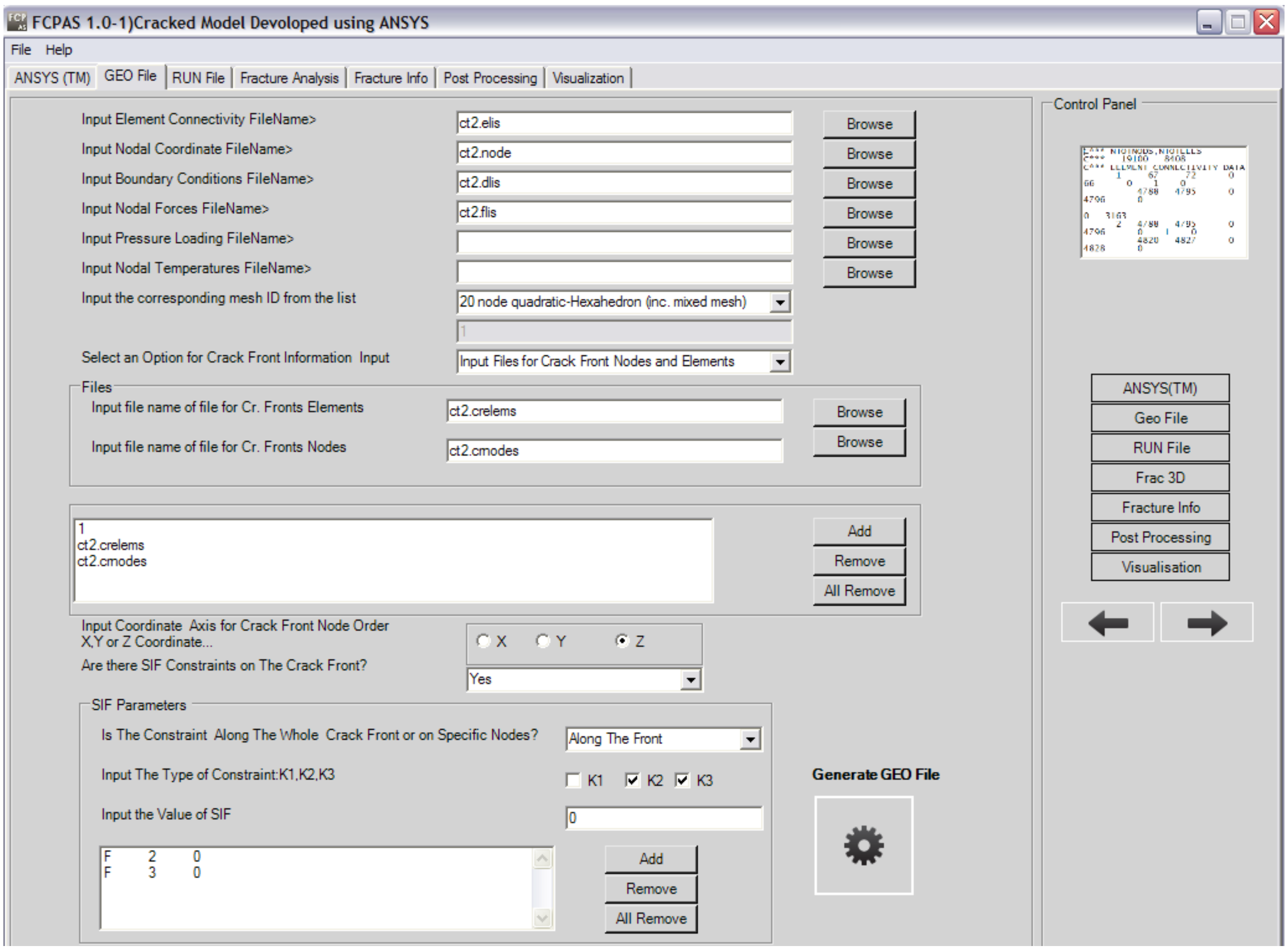
T.2.3.1. Using *convert_ansys_frac3d.exe*

Run *convert_ansys_frac3d.exe.* Using this exe file, we can obtain *ct2.geo* file, which contains element connectivity, nodal coordinates, boundary conditions, loads, and crack information. The following table shows the steps and input for the current problem.

Input Element Connectivity FileName >	*ct2.elis*
Input Nodal Coordinate FileName >	*ct2.node*
Input Boundary Conditions FileName >	*ct2.dlis*
Input Nodal Forces FileName >	*ct2.flis*
Input Pressure Loaging FileName >	
Input Nodal Temperatures FileName >	If there is any temperature file, it's name is entered, otherwise hit return.
Input the corresponding mesh ID from the below list *1. 20 node quadratic- Hexahedorn (incl. mixed mesh)* *2. 15 node quadratic Pentahedron* *3. 10 node quadratic Tetrahedron* *4. 8 node linear Hexahedron* *5. 6 node linear Pentahedron* *6. 4 node linear Tetrahedron*	*1*
How many cracks do you have? <Input 0 if no crack>	*1*
Select an Option for Crack Front Information Input Input Files for Crack Front Nodes and Elements: 1 Input Crack Front Nodes and Elements Interactively: 2	*1*
Input file name of file for Cr. Fronts Elements	*ct2.crelems*
Input file name of file for Cr. Fronts Nodes	*ct2.crnodes*
Input Coordinate Axis for Crack Front Node Order 1 for X, 2 for Y or 3 for Z Coordinate...	*3*
Are There SIF Constraints on The Crack Front? (Def: n)	*y*
Is The Constraint Along The Whole Crack Front or on Specific Nodes? Along The Front: F, On Nodes: N	*f*
Input The Type of Constraint/K1: 1, K2: 2, K3: 3	*2*
Input the Value of SIF	*0*
Do you have a more SIF constrains? <Def:n>	*Y* *Bu ifade mevcut değil*
Is The Constraint Along The Whole Crack Front or on Specific Nodes?	*f*

Along The Front: F, On Nodes: N	
Input The Type of Constraint/K1: 1, K2: 2, K3: 3	*3*
Input the Value of SIF	*0*
Do you have a more SIF constrains? <Def:n>	*n*
Generating The FRAC3D .geo File, Please Wait ...	*Finalization Message*

T.2.3.2. Using FCPAS



Input file names can be selected by “Browse” buttons. “Generate Geo file” creates *ct2.elis_3d.geo* file. To go to “Run File” preparation, press “Next Step”.

T.2.4. Generation of *.run FILE

Now, we need to create a run file which is also required for FRAC3D. We use *writerun_frac3d.exe* or FCPAS to generate *.run file (*ct2.run* file). The **.run* file contains analysis type, material properties, solver type tolerances, body forces and local coordinate data.

T.2.4.1 Using *writerun_frac3d.exe*

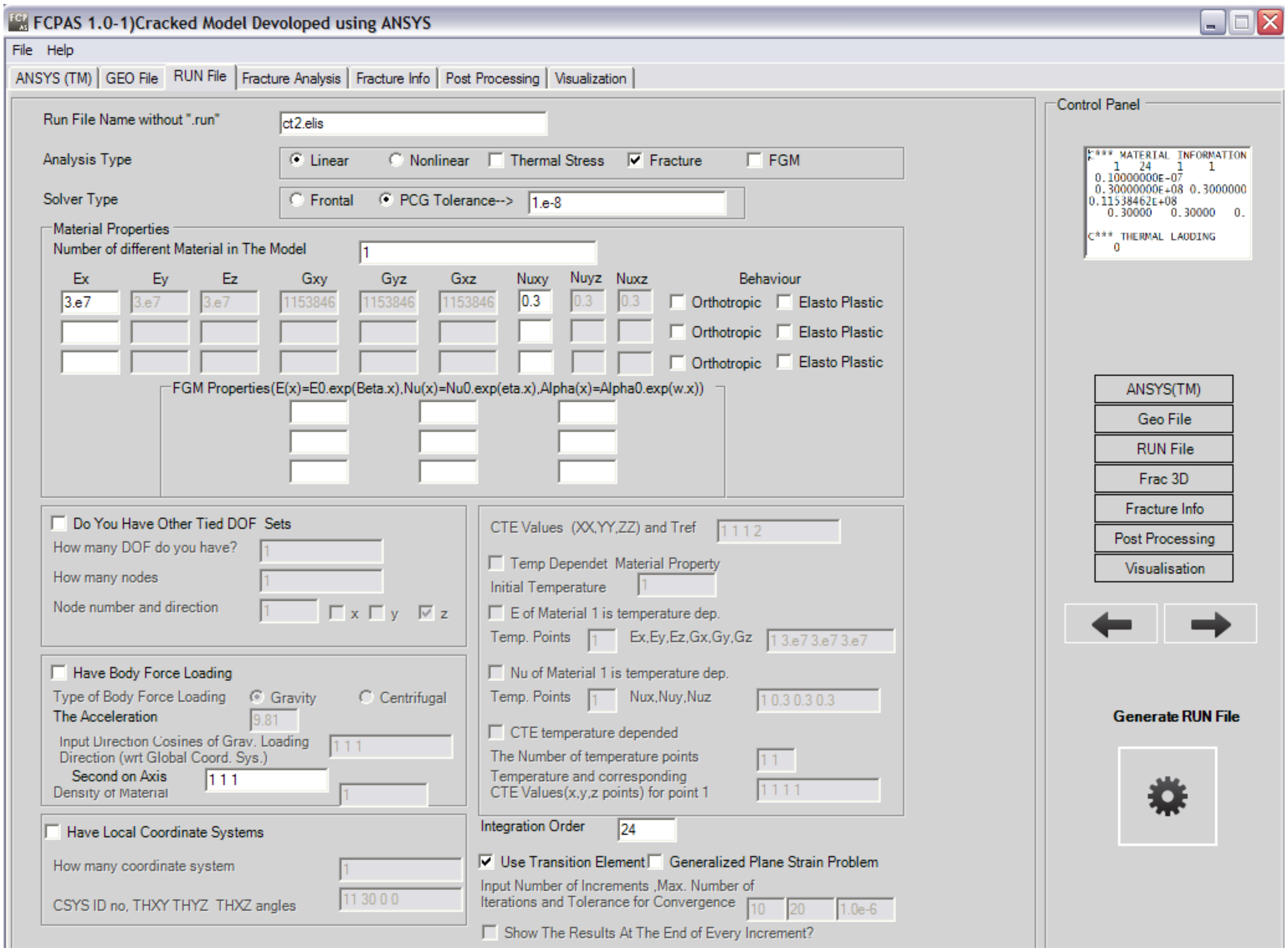
The following table shows the steps and input for this specific problem.

Input Run File Name without ".run" (Include "_3d")	ct2.elis_3d
Is This A Non-linear Analysis (y,n)?, (Default:n)	N
Is This A Thermal Stress Analysis (y,n)?, (Default:n)	N
Do you have temperature dependent material properties (y,n)?, (Default:n)	
Is This A Fracture Analysis? (y,n)?, (Default:n)	y
Please Choose a Solver Type ... Input "0" for Frontal Solver, "1" for PCG Solver'	1
Input the Tolerance for PCG Solver (1.E-8 Recommended)'	1.E-8
Input Number of Different Materials in The Model, (Default:1)	1
Input Ex, Ey, Ez, Gxy, Gyz and Gxz for Mat.	Hit Enter
Input Nuxy, Nuyz, Nuxz for Mat.#	Hit Enter
Enter CTE Values (XX,YY,ZZ) and Tref for Mat.#	
Input Number of Different Materials in the Model (Default:1	
Input Integration Order for Enriched Elements	24
Do You Want to Use Transition Elements? (Default: n)	Y
Is This A Generalized Plane Strain Problem? (y,n) (Default: n)	n
Do You Have Other Tied DOF Sets? (y,n) (Default: n)	N
How Many Sets Do You Have?	
How Many Nodes to be Tied in Set#	
Input Node Numbers and Tieing Direction (x:1, y:2, z:3)	
How Many Nodes to be Tied in Set#	
Input Number of Increments, Max. Number of Iterations and Tolerance for Convergence(Default: 10, 20, 1.0E-6)	
Does Material#,NM, Exhibit Elasto-Plastic Behavior (y,n)? (Default:n)	
Input Initial Yield Stress and Number of Break-Points/Including The Initial Yield Stress	
Do You Want to Output The Results At The End of Every Increment? (y,n) (Default:y)	
Input Number of Increments for Which The Results To Be Printed	
Input The Initial Temperature	
Is Elastic Modulus of Material # Temperature Dependent? (y,n)	
Is Poisson',"'",'s Ratio of Material # Temperature Dependent? (y,n)	

Input Temperature and The Corresponding Poiss. Ratio Value for Point #	
Is CTE of Material # Temperature Dependent? (y,n)	
Input Number of Temperature Points and Tref	
Input Temperature and The Corresponding CTE Values (x,y,z) for Point #	
Is Yield Stress of Material # Temperature Dependent? (y,n)	
Input Number of Temperature Points	
Input Temperature Value for Set #	
Input The Corresponding Yield Stress and Plastic Strain Values for Temp. Set #', i3,' Stress Point #	
Do You Have Body Force Loading ? (y,n), (Default:n)	n
Input Type of Body Force Loading ...'1: Gravity, 2: Centrifugal	
Input The Acceleration	
Input Direction Cosines of Grav. Loading Direction	
Input Density for Material #	
Input The Angular Velocity in Rd/Sec	
Input The x,y,z Coord.s of The 1st Point on The Rot. Axis	
Input The x,y,z Coord.s of The 2nd Point on The Rot. Axis	
Do You Have Local Coordinate Systems To Be Included In The Analysis (y,n)?, (Default:n)	n
How Many Coordinate Systems Will Be Defined?	
Do You Really Want to Exit ? (y,n)	
Input The Initial Temperature'	
Is Elastic Modulus of Material # Temperature Dependent? (y,n)	
Input Temperature and Elastic Moduli (Ex,Ey,Ez,Gxy,Gyz and Gxz) for Point #	
Is Poisson',"'",'s Ratio of Material # Temperature Dependent? (y,n)	
Input Temperature and The Corresponding Poiss. Ratio Values Nuxy, Nuyz, Nuxz for Point #	
Input Temperature and The Corresponding CTE Values (x,y,z) for Point #	

T.2. 4.2 Using FCPAS for **.run* FILE

Parameters can be selected by clicking the objects in the tab. “Generate Run file” creates *ct2.elis_3d.run* file.



To go to "Frac3D" tab, press "Next Step".

T.2.5 RUNNING FRAC3D

T.2.5.1 Using *frac3d.exe*

To run the FRAC3D, three kinds of input files are required;

- *.run (compulsory)
- *.geo (compulsory)
- *.tem (optional)

FRAC3D gives the results in the following output files;

- *.out
- *.str
- *.stn
- *.crk

Now, we are ready to run FRAC3D. To do this we can use *frac3d.exe*. When running FRAC3D, geo and run files names have to be entered. The following table shows the steps and input for this specific problem.

Input Run File Name without ".run"	ct2.elis_3d
Input geo File Name without ".geo"	ct2.elis_3d
Input ter File Name without ".ter"	Hit Enter

As a result, ***.crk** file is created like this:

```
            FRACTURE MECHANICS INFORMATION

  ct2.elis_3d.crk
 24 X  24 X  24 INTEGRATION IS USED FOR ENRICHED CRACK TIP ELEMENTS
         TRANSITION ELEMENTS ARE INCLUDED IN THE ANALYSIS


                 CRACK NO:   1
  CRACK TIP NODES:
    18   2200   2201   2202   2203   2204   2205   2206   2207   2208
   2209   2210   2211   2212   2213   2214   2215   2216   2217   2218
   5715   5716   5717   5718   5719   5720   5721   5722   5723   5724
   5725   5726   5727   5728   5729   5730   5731   5732   5733   5734
   2199

  CRACK IN AN ORTHOTROPIC MATERIAL


        K1            K2            K3

  18    0.1643408E+04   0.0000000E+00   0.0000000E+00
 2200   0.1789624E+04   0.0000000E+00   0.0000000E+00
 2201   0.1939526E+04   0.0000000E+00   0.0000000E+00
 2202   0.1971490E+04   0.0000000E+00   0.0000000E+00
 2203   0.2004858E+04   0.0000000E+00   0.0000000E+00
 2204   0.2033197E+04   0.0000000E+00   0.0000000E+00
 2205   0.2061087E+04   0.0000000E+00   0.0000000E+00
 2206   0.2078518E+04   0.0000000E+00   0.0000000E+00
 2207   0.2095062E+04   0.0000000E+00   0.0000000E+00
 2208   0.2108352E+04   0.0000000E+00   0.0000000E+00
 2209   0.2120820E+04   0.0000000E+00   0.0000000E+00
 2210   0.2130562E+04   0.0000000E+00   0.0000000E+00
 2211   0.2139531E+04   0.0000000E+00   0.0000000E+00
```

2212	0.2146679E+04	0.0000000E+00	0.0000000E+00
2213	0.2153085E+04	0.0000000E+00	0.0000000E+00
2214	0.2158029E+04	0.0000000E+00	0.0000000E+00
2215	0.2162254E+04	0.0000000E+00	0.0000000E+00
2216	0.2165264E+04	0.0000000E+00	0.0000000E+00
2217	0.2167572E+04	0.0000000E+00	0.0000000E+00
2218	0.2168789E+04	0.0000000E+00	0.0000000E+00
5715	0.2169318E+04	0.0000000E+00	0.0000000E+00
5716	0.2168789E+04	0.0000000E+00	0.0000000E+00
5717	0.2167584E+04	0.0000000E+00	0.0000000E+00
5718	0.2165264E+04	0.0000000E+00	0.0000000E+00
5719	0.2162278E+04	0.0000000E+00	0.0000000E+00
5720	0.2158029E+04	0.0000000E+00	0.0000000E+00
5721	0.2153123E+04	0.0000000E+00	0.0000000E+00
5722	0.2146679E+04	0.0000000E+00	0.0000000E+00
5723	0.2139585E+04	0.0000000E+00	0.0000000E+00
5724	0.2130564E+04	0.0000000E+00	0.0000000E+00
5725	0.2120897E+04	0.0000000E+00	0.0000000E+00
5726	0.2108353E+04	0.0000000E+00	0.0000000E+00
5727	0.2095165E+04	0.0000000E+00	0.0000000E+00
5728	0.2078529E+04	0.0000000E+00	0.0000000E+00
5729	0.2061253E+04	0.0000000E+00	0.0000000E+00
5730	0.2033191E+04	0.0000000E+00	0.0000000E+00
5731	0.2004985E+04	0.0000000E+00	0.0000000E+00
5732	0.1971658E+04	0.0000000E+00	0.0000000E+00
5733	0.1939746E+04	0.0000000E+00	0.0000000E+00
5734	0.1789692E+04	0.0000000E+00	0.0000000E+00
2199	0.1640899E+04	0.0000000E+00	0.0000000E+00

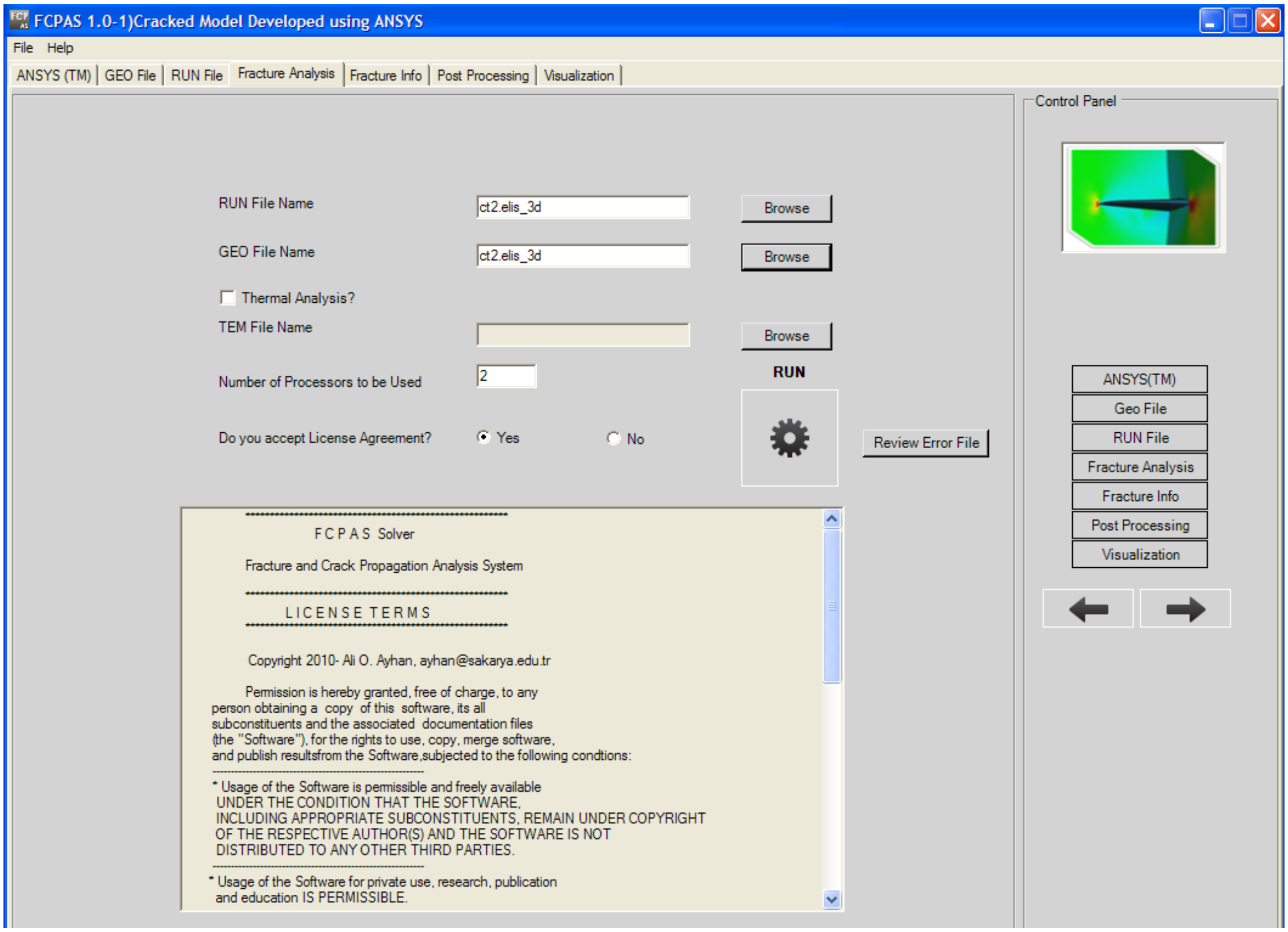
G1 G2 G3 GTOT

18	0.3858269E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2200	0.4575365E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2201	0.5373944E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2202	0.5552534E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2203	0.5742077E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2204	0.5905559E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2205	0.6068684E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2206	0.6171768E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2207	0.6270406E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2208	0.6350212E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2209	0.6425540E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2210	0.6484707E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2211	0.6539416E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2212	0.6583186E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00

2213	0.6622536E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2214	0.6652986E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2215	0.6679059E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2216	0.6697667E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2217	0.6711955E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2218	0.6719491E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5715	0.6722773E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5716	0.6719491E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5717	0.6712029E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5718	0.6697667E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5719	0.6679210E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5720	0.6652987E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5721	0.6622771E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5722	0.6583189E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5723	0.6539746E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5724	0.6484716E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5725	0.6426004E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5726	0.6350218E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5727	0.6271023E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5728	0.6171833E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5729	0.6069664E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5730	0.5905523E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5731	0.5742809E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5732	0.5553480E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5733	0.5375165E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
5734	0.4575709E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00
2199	0.3846499E-04	0.0000000E+00	0.0000000E+00	0.0000000E+00

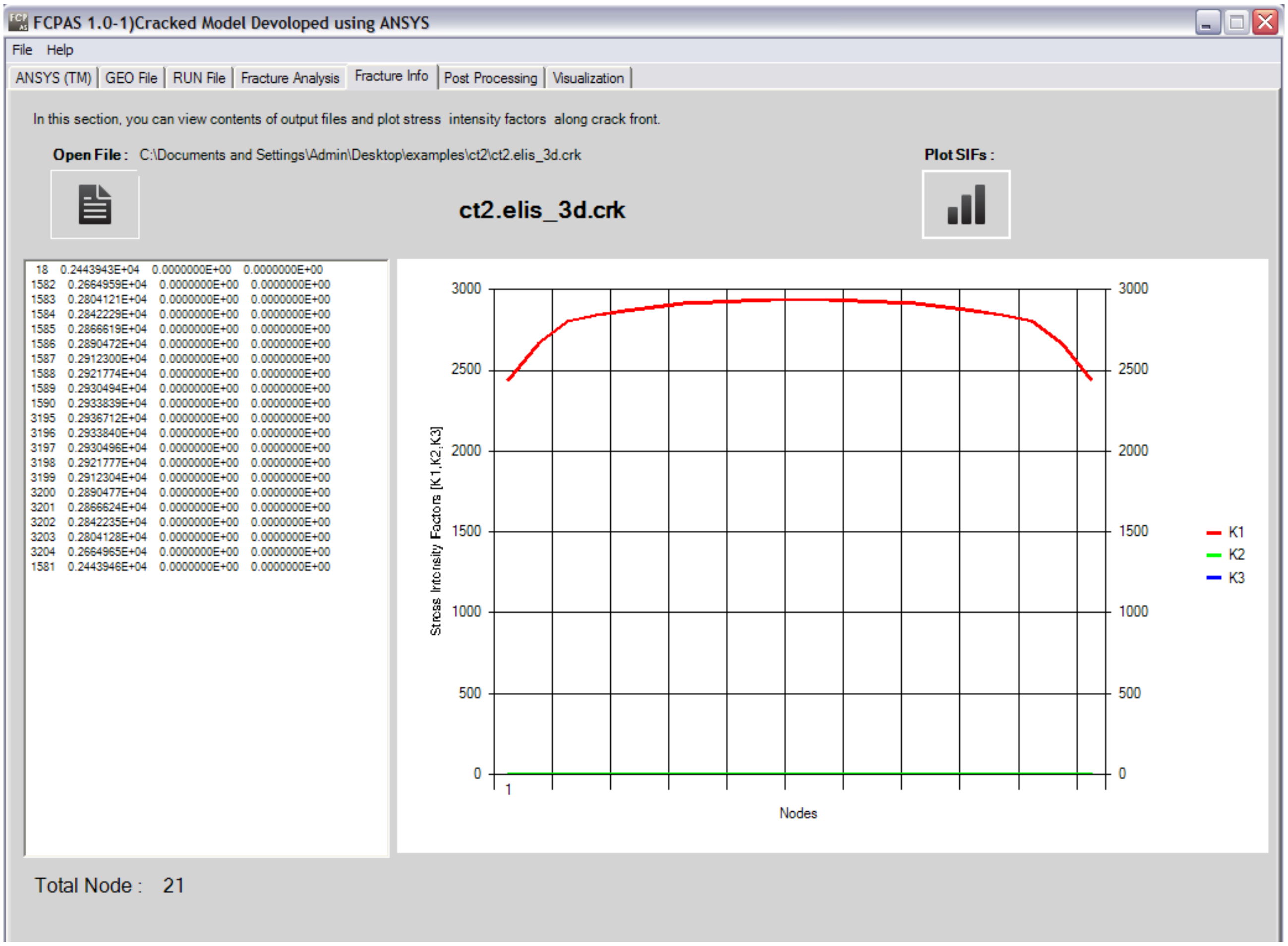
T.2.5.2 Using FCPAS to run FRAC3D

Select the *ct2.elis_3d.geo, ct2.elis_3d.run,* and *ct2.elis_3d.tem* (if required) files by browsing and press run button to run the Frac3D.exe in the background.



After FRAC 3D run ends, output files can be viewed in the "Fracture Info" tab. In the "Fracture Info" tab, you can browse anyfile to see its content and plot the K_1, K_2 and K_3 data in an x-y plot.

To plot the K_1, K_2 and K_3 data, just press "Plot SIF's" button.



K_I value is Mode –I crack stress intensity factor along the crack front and depends on both load and crack geometry as follow (plane strain conditions) [4];

$$K_I = \frac{P_Q}{B\sqrt{W}} f\left(\frac{a}{W}\right)$$

where

$$f\left(\frac{a}{W}\right) = \frac{\left(2+\frac{a}{W}\right)\left(0{,}886+4{,}64\frac{a}{W}-13{,}32\left(\frac{a}{W}\right)^2+14{,}72\left(\frac{a}{W}\right)^3-5{,}6\left(\frac{a}{W}\right)^4\right)}{\sqrt{\left(1-\frac{a}{W}\right)^3}}$$

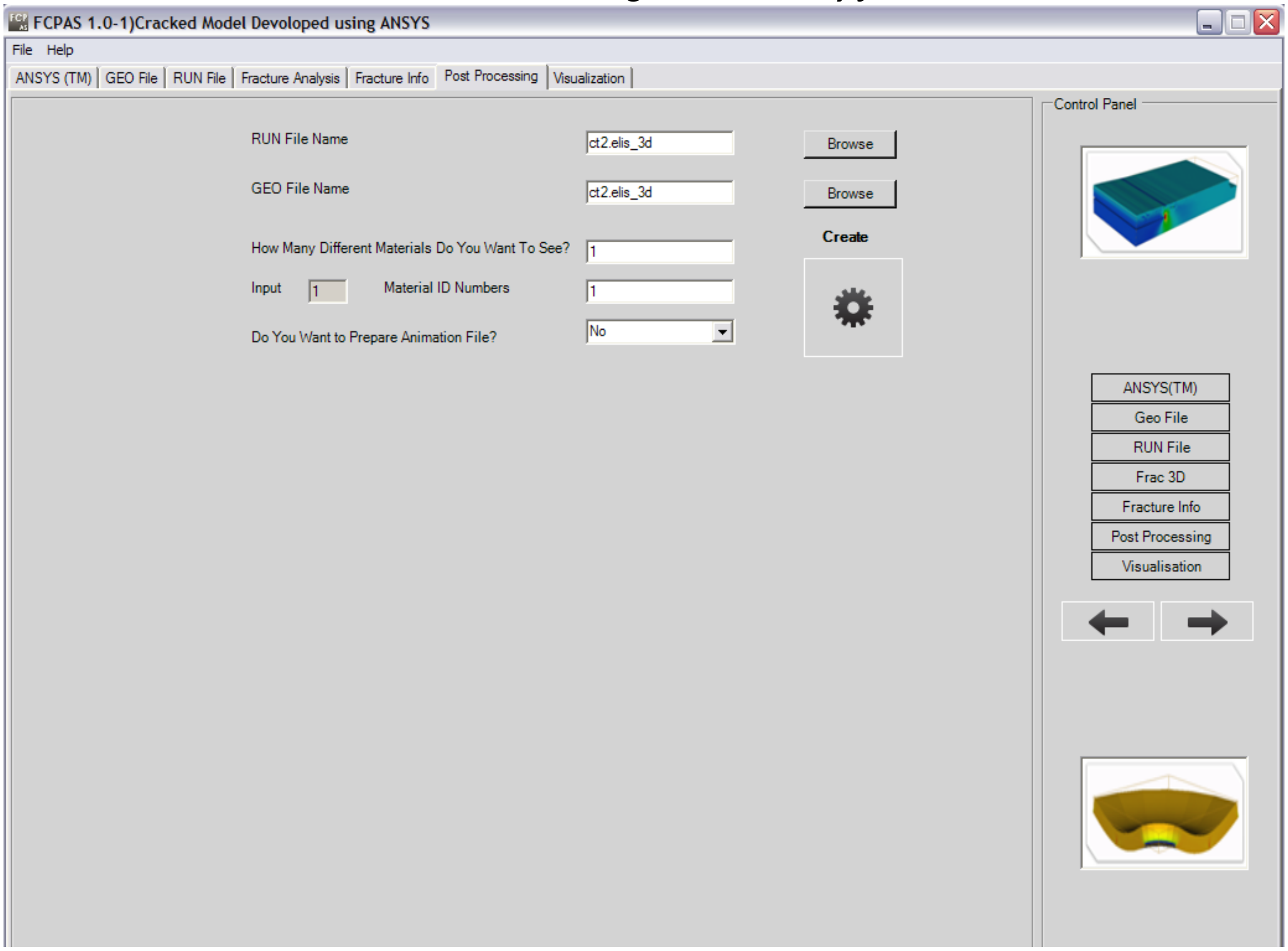
P_Q=Load as determined in *P-v* diagram, *B*=Specimen thickness, *W*=Specimen width, *a*=crack length.

a [m]	W [m]	a/W	f(a/W)	B [m]	Pq [MN]	K1 [MPa√m]
2.75E-02	5.00E-02	0.55	11.36428629	0.025	1.00E+00	2032.90533

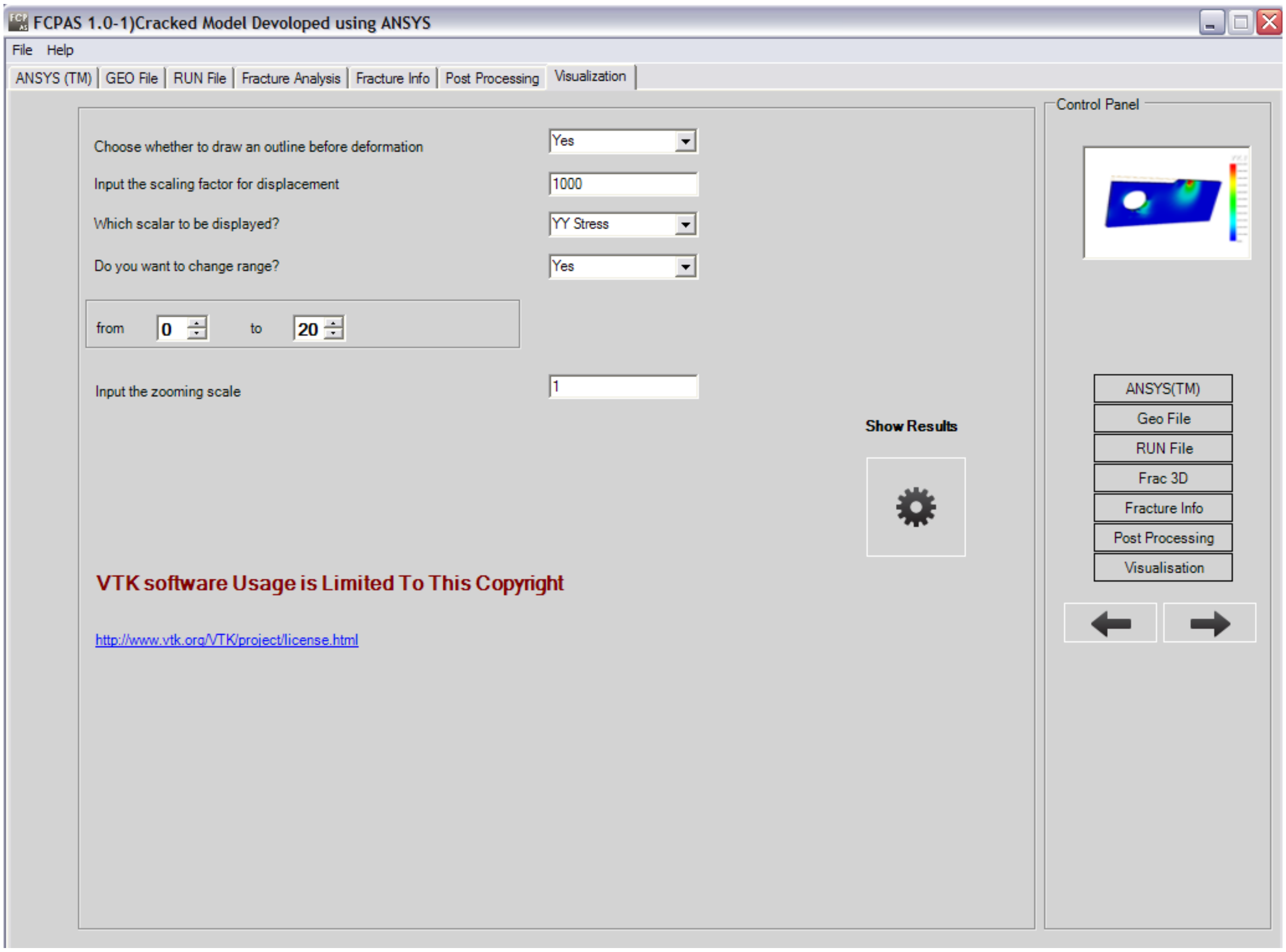
Frac3D gives K_1=2147 MPa√m. We can use the K_1 value at the mid thickness location. Difference is

$$\frac{2147-2032.90533}{2032.90533}=0.056\cong\%5.6$$

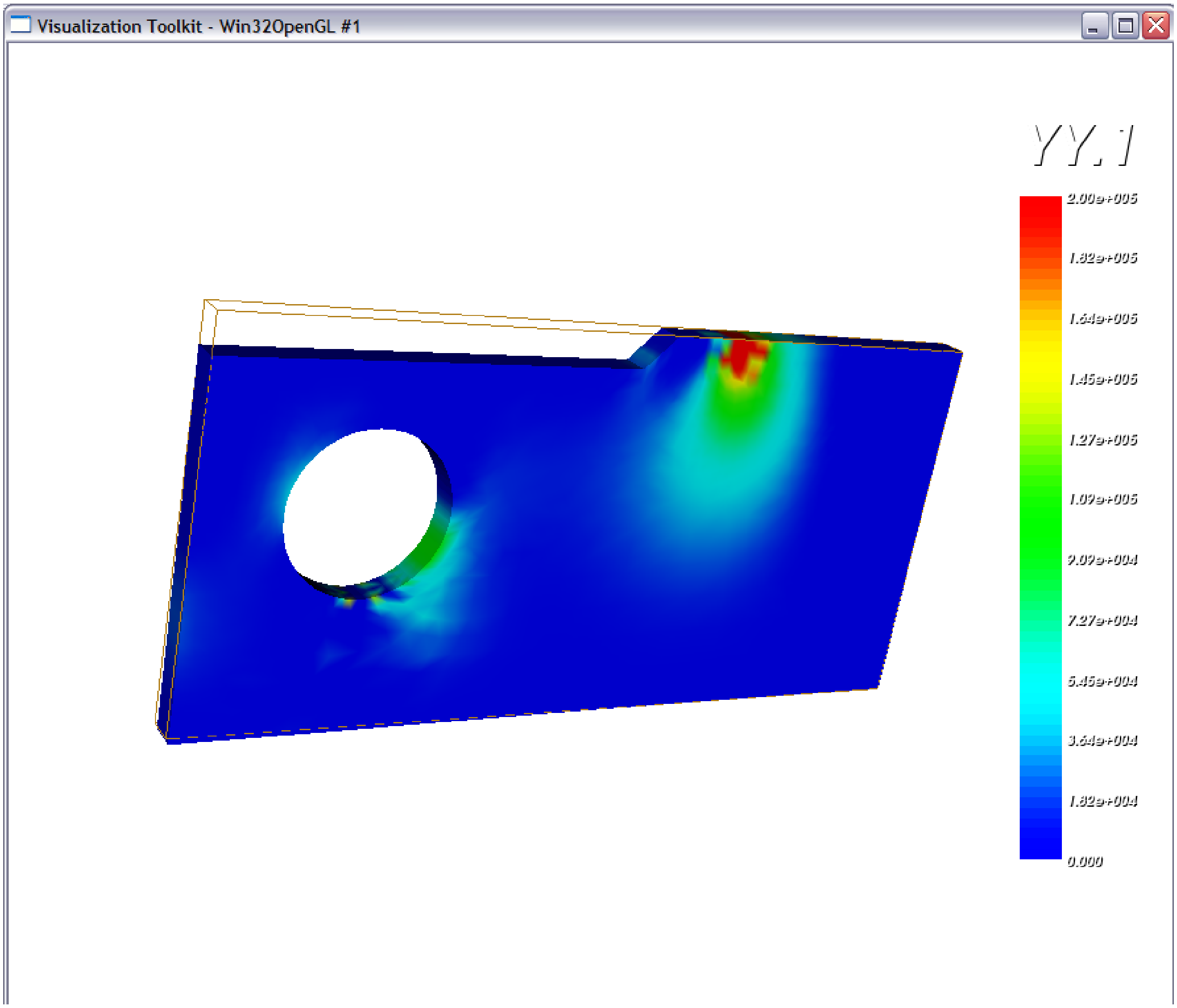
T.2.6 POSTPROCESSING of FRAC3D Results Using *movieassembly.f*



T.2.7 Visualization of FRAC3D Results



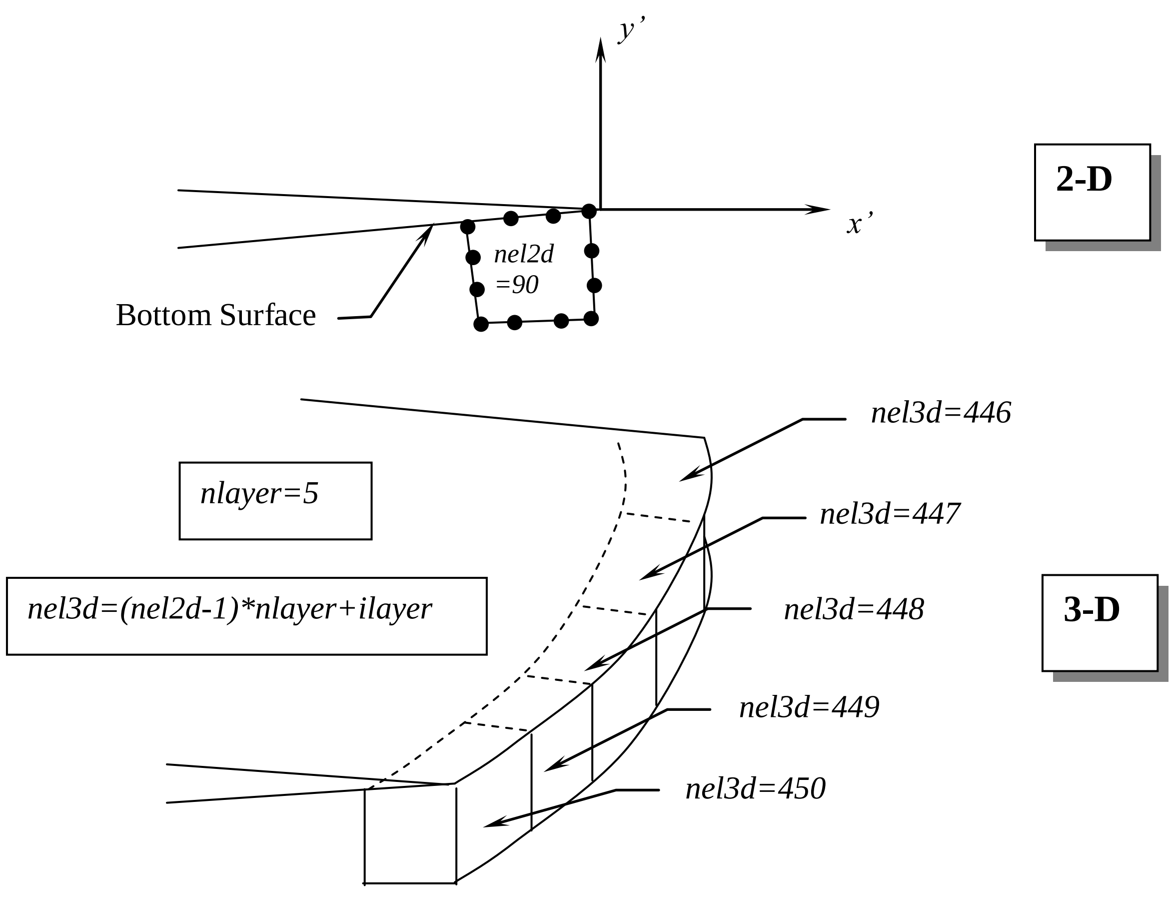
To see the Cracked Model results, choose the parameter you would like to contour plot and press "Show Results" button.


Visualization Toolkit - Win32OpenGL #1
YY, 1
2.00e+005
1.82e+005
1.64e+005
1.45e+005
1.27e+005
1.09e+005
9.09e+004
7.27e+004
5.45e+004
3.64e+004
1.82e+004
0.000

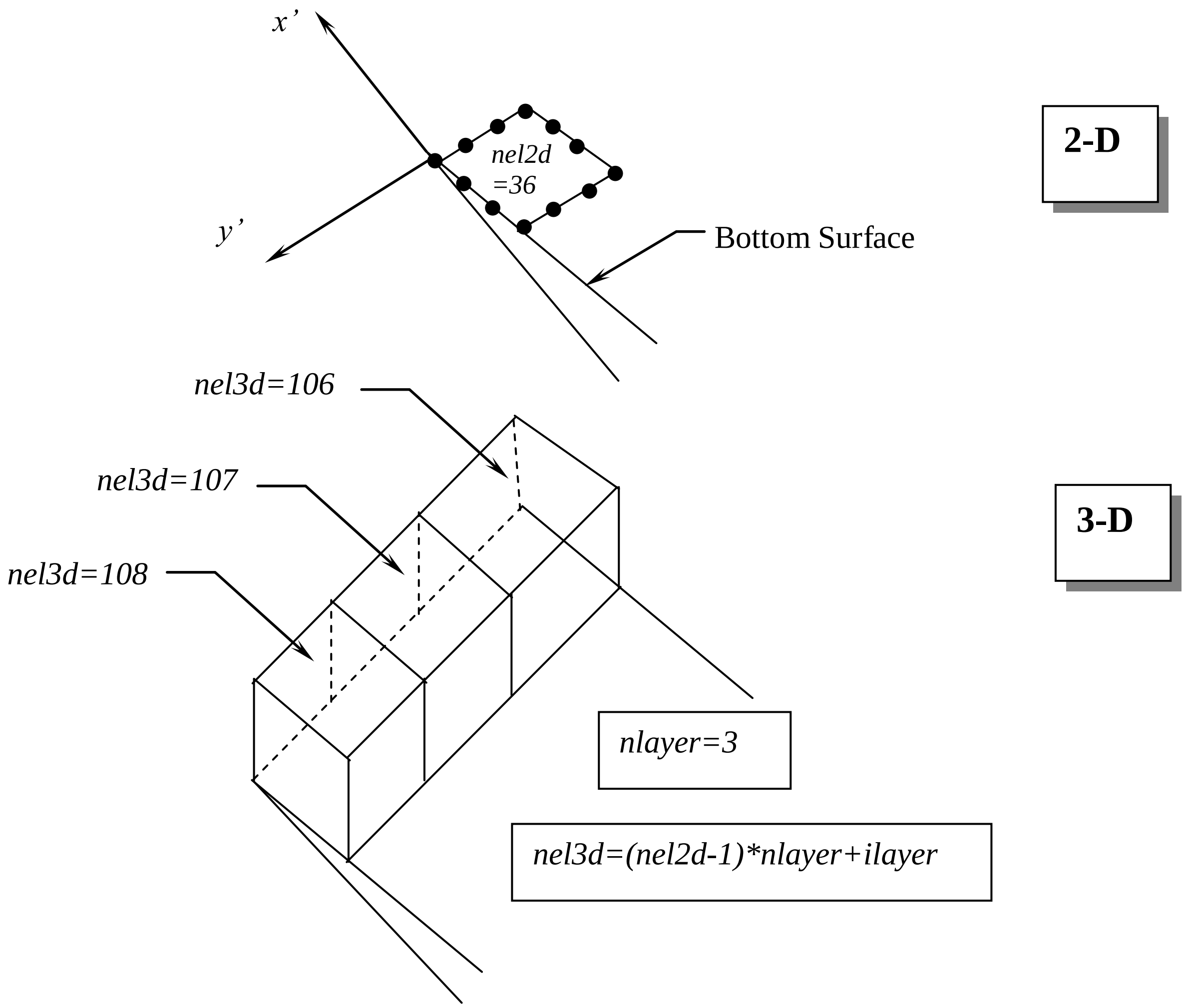
Appendix A Definition of Crack in FRAC3D

In fracture analysis of solid structures, FRAC3D uses special 3-D enriched crack tip elements. The enriched elements are defined as the finite elements that have common border with the crack front. In FRAC3D, the crack is defined by the nodes along the crack front and the enriched elements on the bottom crack surface (with respect to the local orientation of the crack tip). The current version of the program interacts with FRAC2D and the converter program determines the crack front nodes automatically. On the other hand, the enriched element numbers as reference elements along the bottom crack surface is needed. These element numbers must be added to next line after the crack tip node numbers (at the end of the *_3d.geo file) in the order that the crack tip node numbers are listed, i.e., from back face of the model to front face. Examples 1 and 2 provided below explain the procedure. Alternatively, if 3-D *.geo file is prepared by translating the external list files from ANSYS (section 2.1.2 in this report), then users should prepare the crack tip node files and reference element files in ANSYS, the ANSYS-to-FRAC3D program would automatically add these fracture information into the *.geo file. Example 3 illustrated the definition of a curved crack tip.

EXAMPLE 1



EXAMPLE 2



For the latter case, for example, the element number information shown in the rectangle must be added by the user to the "*_3d.geo" file as shown below.

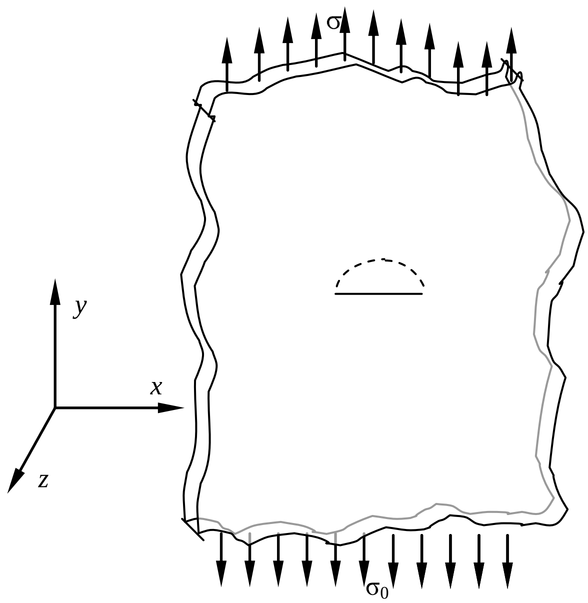
```
C*** FRACTURE MECHANICS DATA
   1
  10    120.00000000
 431 2829 2830 1607 3455 3456 3101 4201 4202 3800
 106  107  108
```

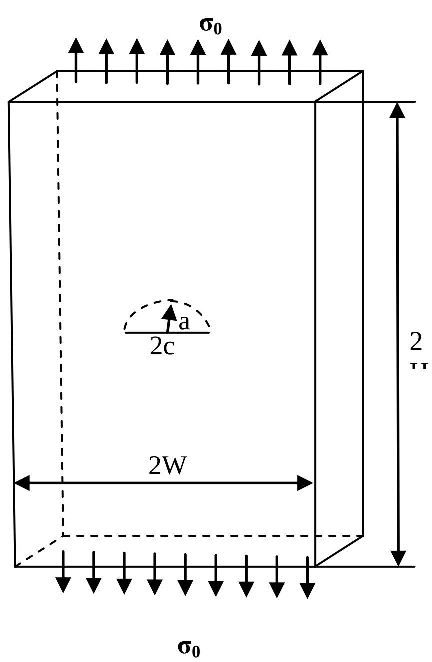
C*** J-INTEGRAL PATHS

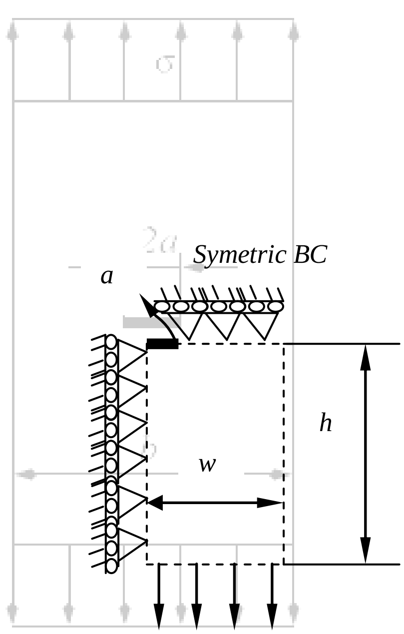
EXAMPLE.3.Two-Dimensional Mode-I Central Elliptical Crack in a Large Isotropic Medium

T.3.1 Problem Description

Develop a new problem case in the tutorial with the following data: A three-dimensional elliptical surface crack (a/c=0.3) in a finite-thickness plate under uniform tension with 2H x 2W x t (height x width x thickness), where H = W = 5c and t = 2a. compare your results from FRAC3D/FCPAS with those of Newman and Raju's surface crack formula.







In the ANSYS^{TM} tab of the FCPAS, we browse “"C:\Program Files\ANSYS Inc\v120\ansys\bin\intel\launcher120.exe”.

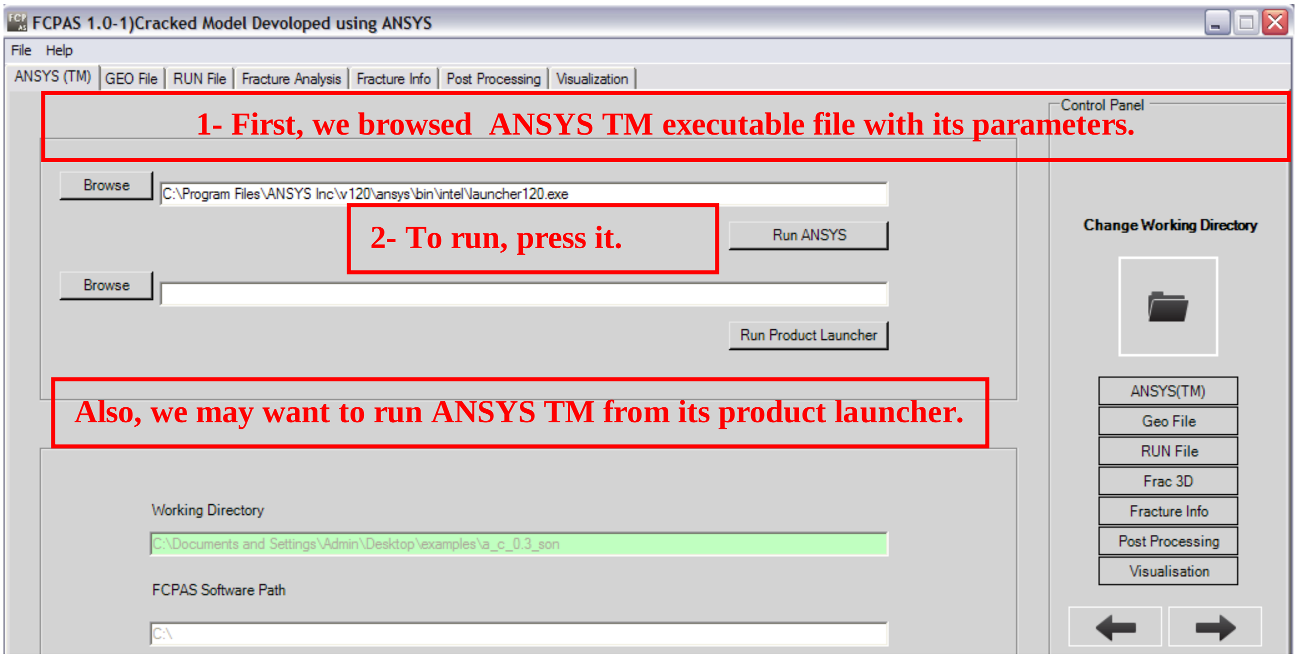


Figure T. 7 ANSYS^{TM} tab of the FCPAS

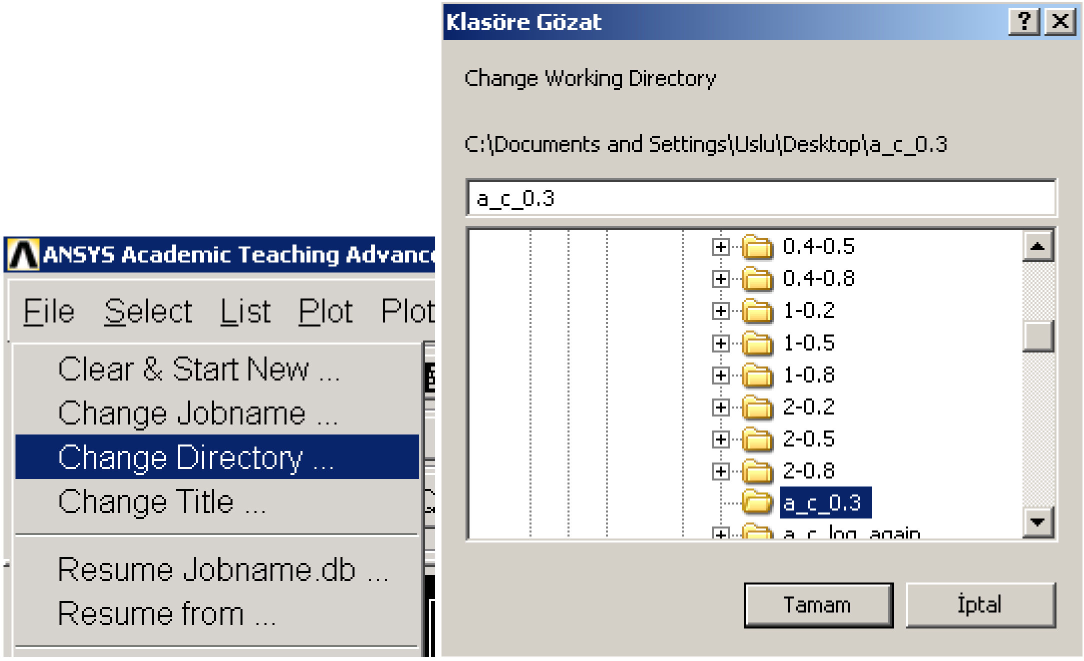
T.3.2 Generation of the Finite Element Model within the ANSYS^{TM} Preprocessor

First of all, we must take into account the problem type, which is plane strain. Also, due to the symmetry of the problem, only analysis of a quarter model is needed. We will model this two dimensional problem using one-layer (in the out-of-plane direction) three-dimensional elements. To do this, we will first mesh the back face of the domain with area (2D) elements and extrude the mesh into the third direction. To do this, we will use PLANE82, SHELL 281 and SOLID95 elements from the ANSYS^{TM} element library [3]. Note that ANSYS^{TM} Help is very useful tool to identify and select the suitable elements for the problem of interest (Figure D.6).

Preprocessing

Change Directory

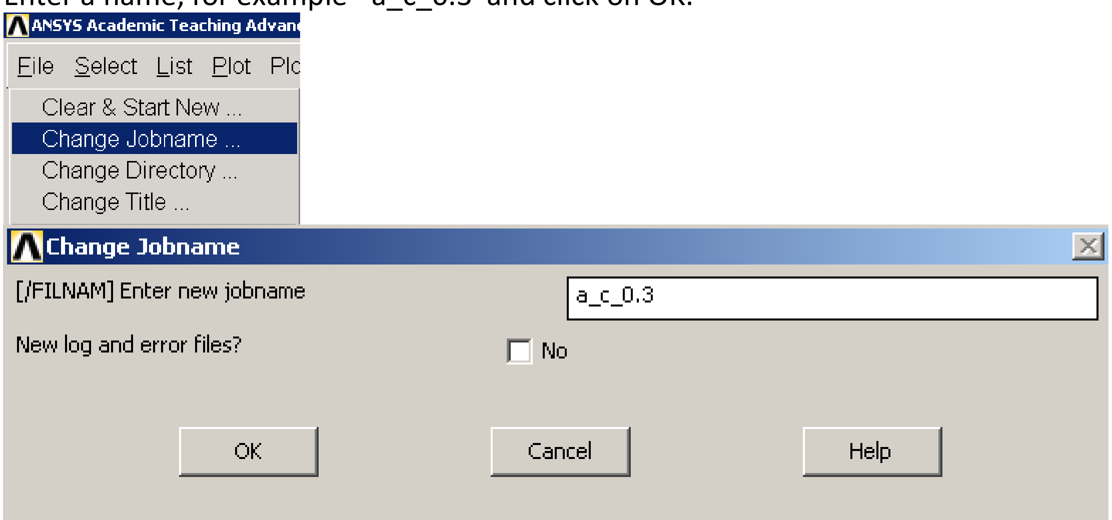
Before starting the model, create a folder in which you would like to work&change directory to this folder.



Give the Job a Name

Utility Menu>File>Change Jobname...

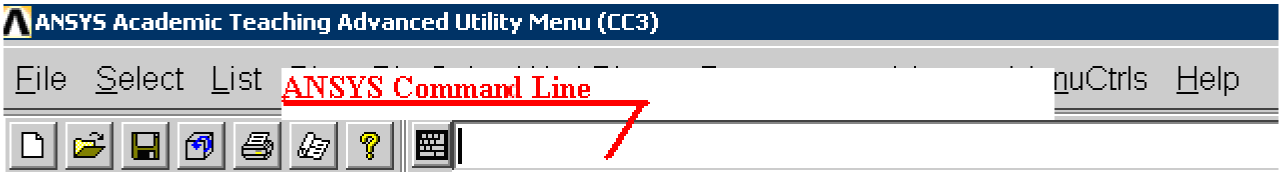
Enter a name, for example ` a_c_0.3' and click on OK.



Define Element Type

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

This brings up the 'Element Types' window. Click on the Add... button. The 'Library of Element Types' window appears. Highlight "PLANE82-8 node 95" Shell 281 and "SOLID95-20 node 93 95". Click on OK or in command line, use **(ET,1,95)**, **(ET,2,93).**

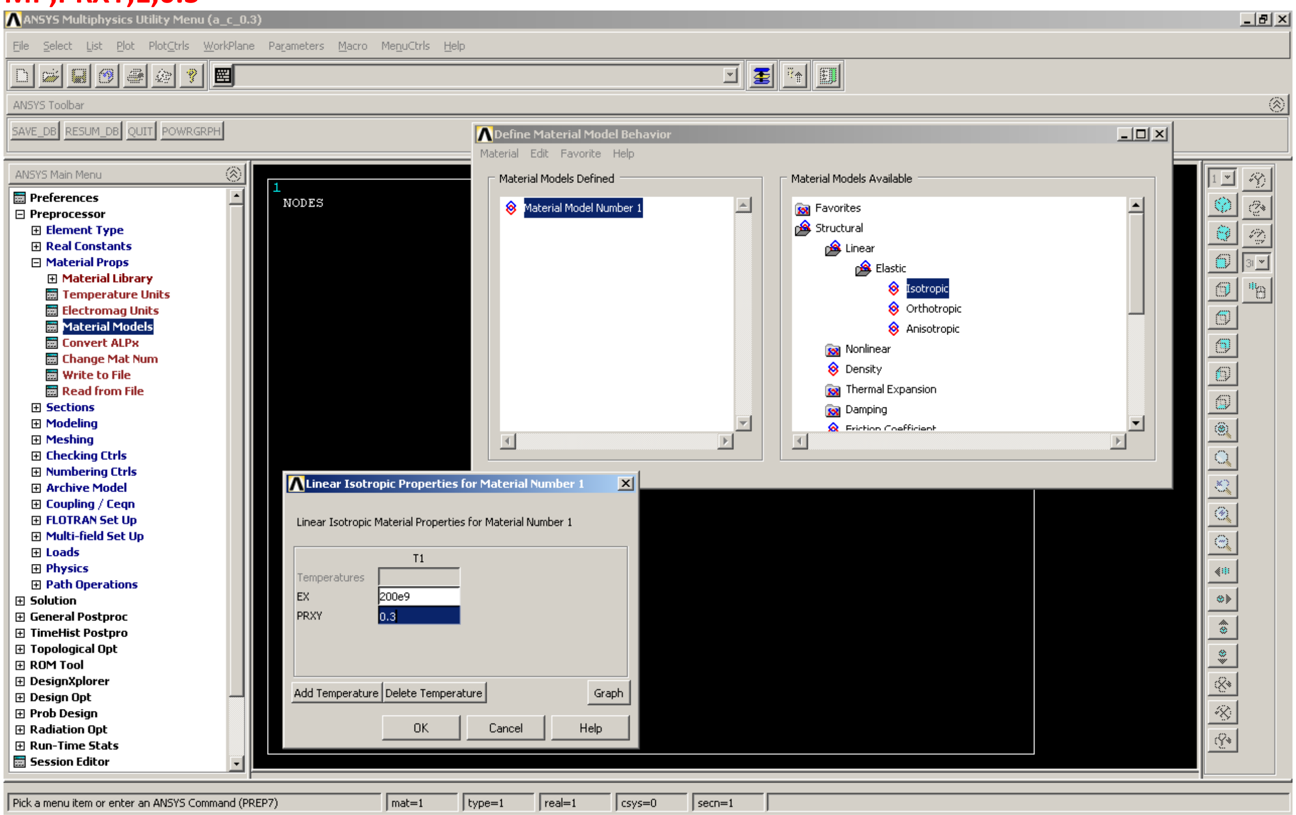


Define Material Properties

Main Menu>Preprocessor>Material Props>Material Models

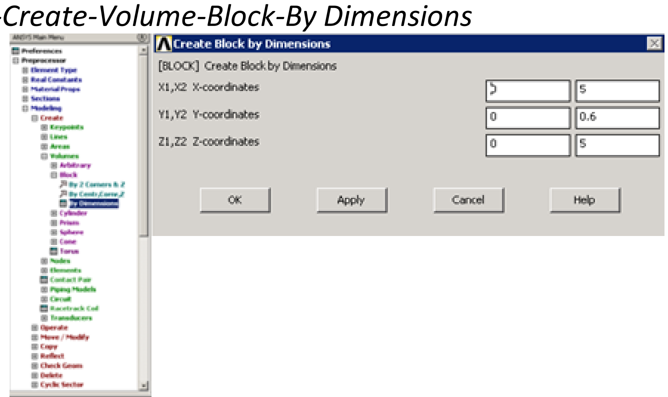
On the right side of the `Define Material Model Behavior' window that opens, double click on `Structural', then `Linear', then `Elastic', finally `Isotropic'. Enter in values for the Young's modulus (EX = 200E9) and Poisson's ratio (PRXY = 0.3) of the plate material.

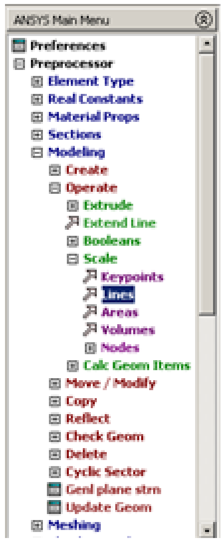
MP,EX,1,200e9
MP,PRXY,1,0.3

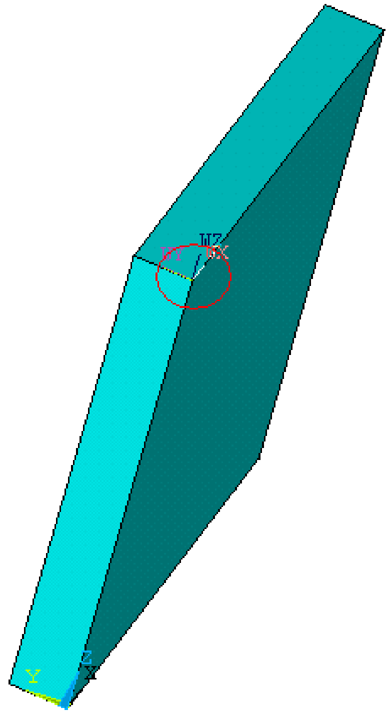


Modeling

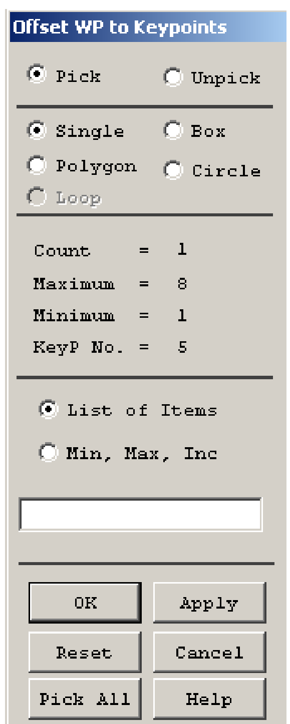
Preprocessor-Modeling-Create-Volume-Block-By Dimensions

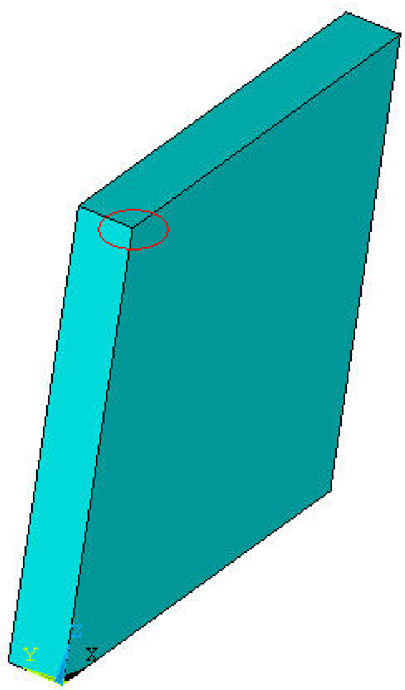






We must be choose the ***Workplane-offset WP with – keypoints on*** upper X Y surface



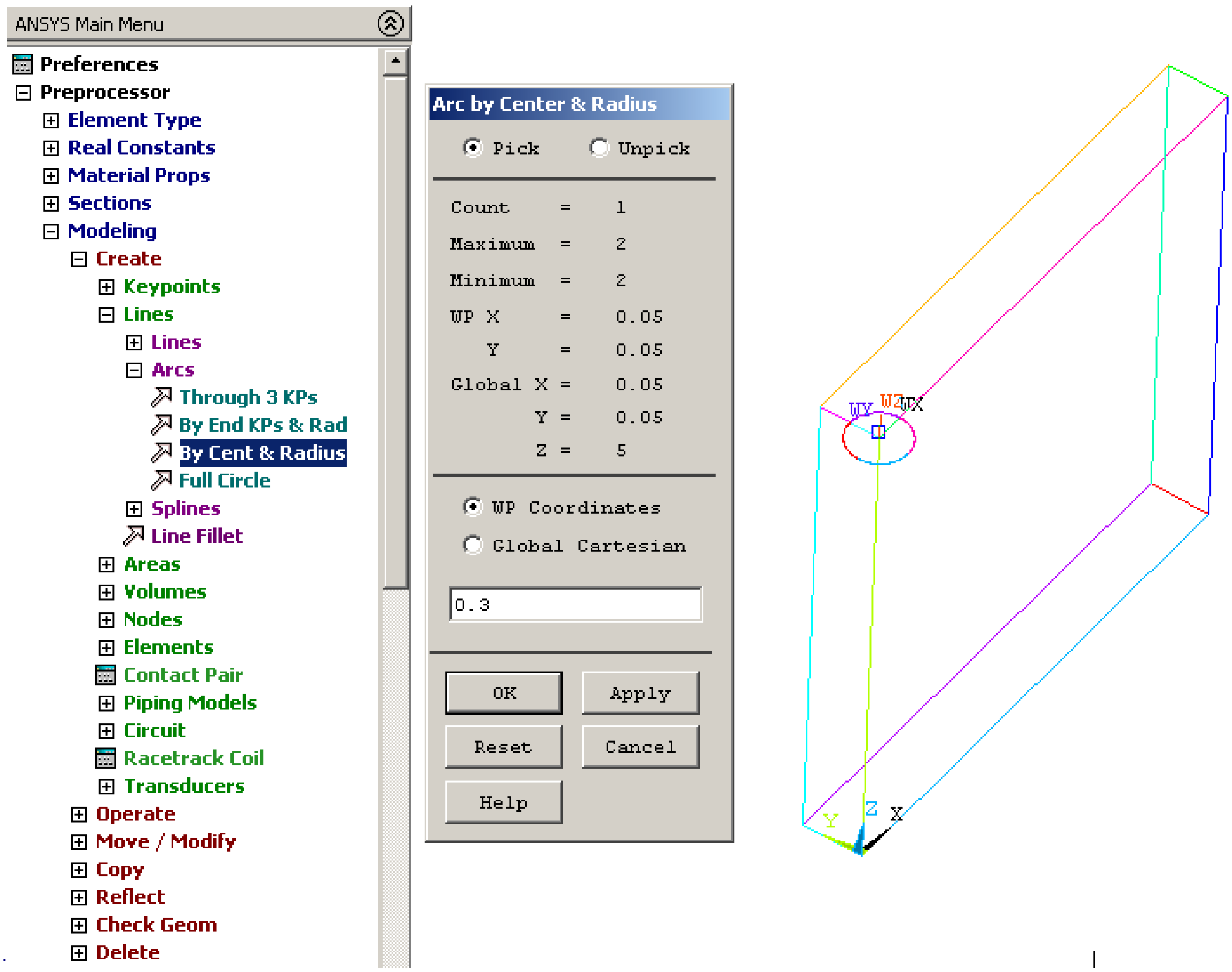


Ansys Command Prompt (lplot)

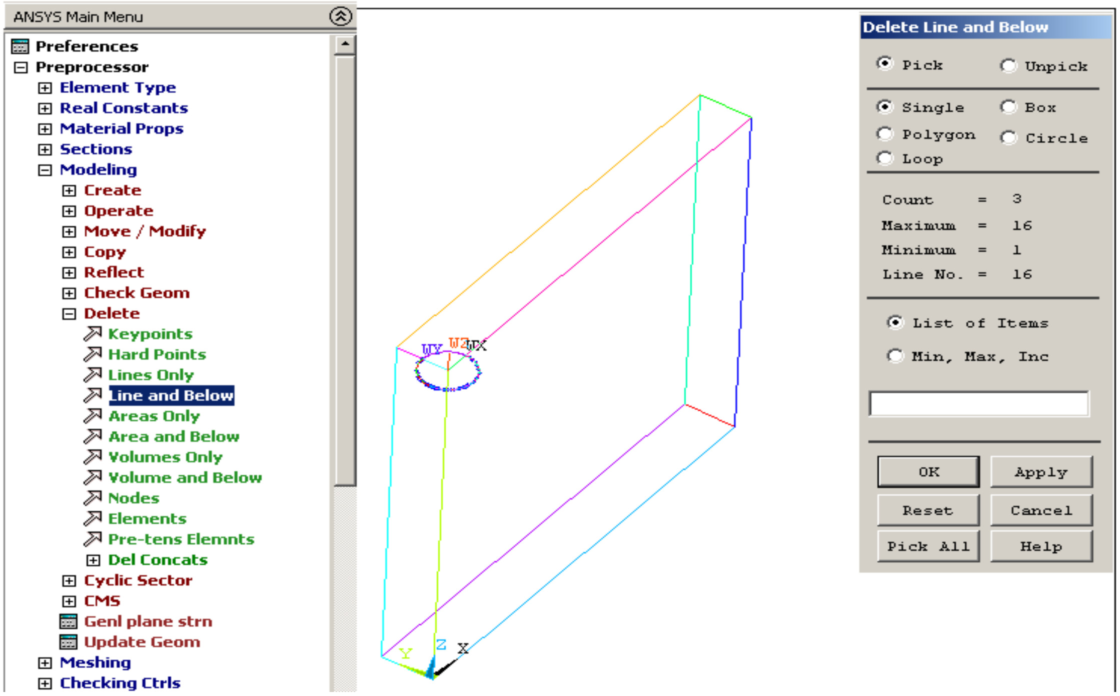
Creating elliptic Crack

Preprocessor-Modeling-Create-Lines-Arcs-By Cent & Radius

First, we must create a circle r = 0.3

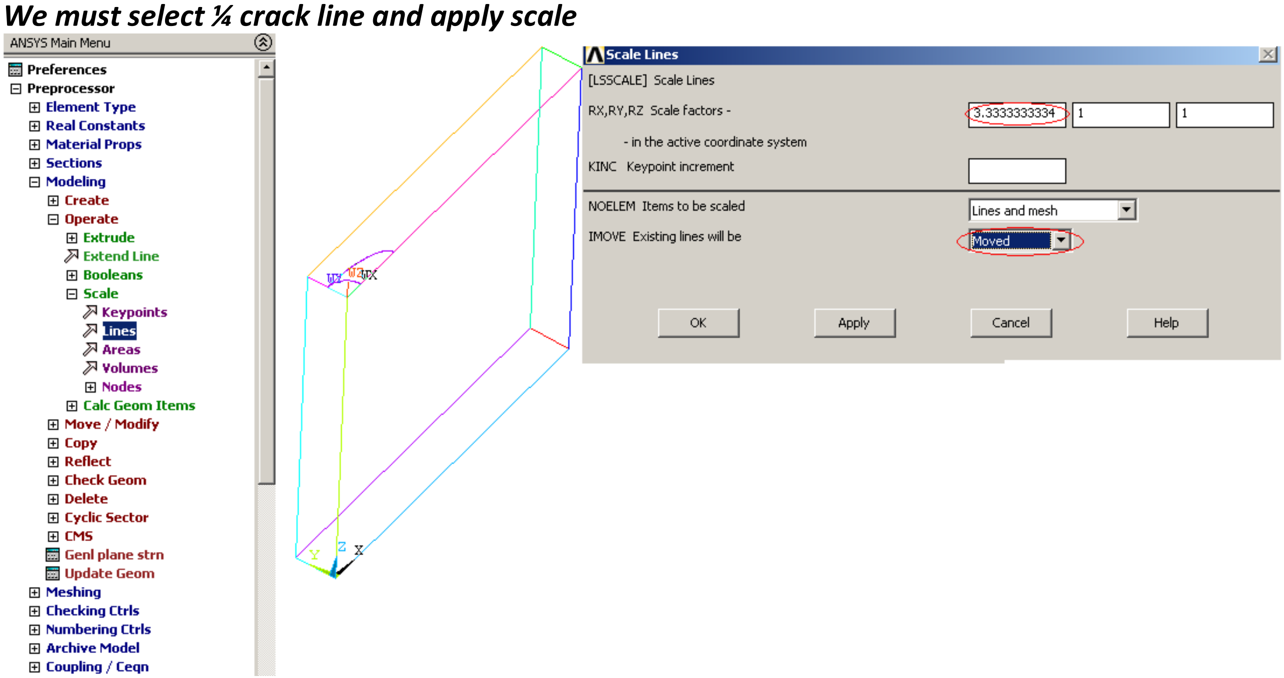


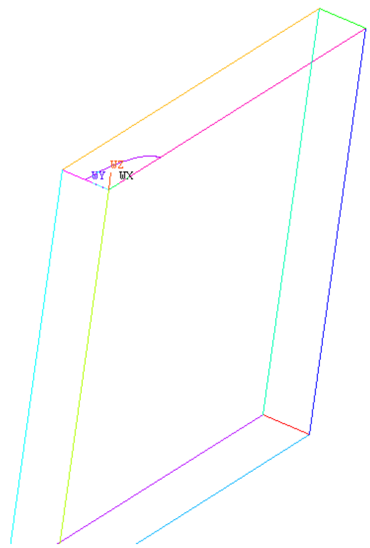
Then we delete the lines by ***<u>Delete-lines and below…</u>***



Preprocessor-Modeling-Operate-Scale-Lines

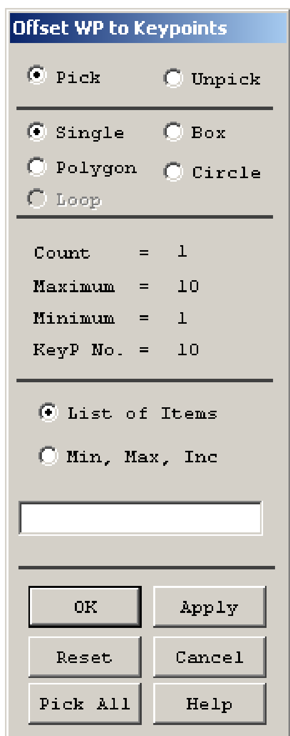
We must select ¼ crack line and apply scale

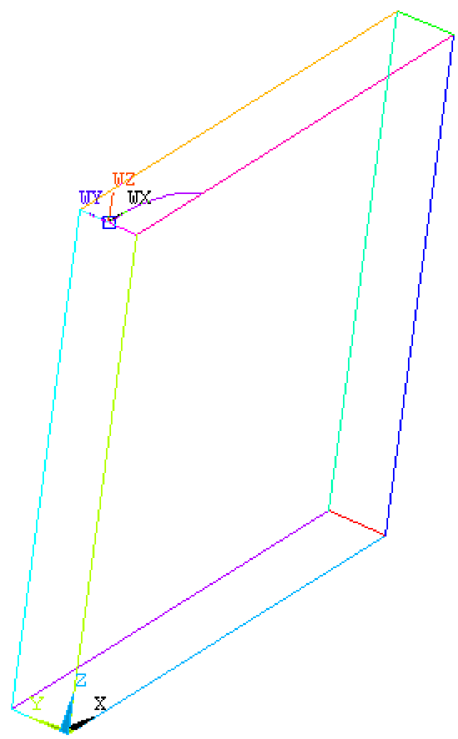




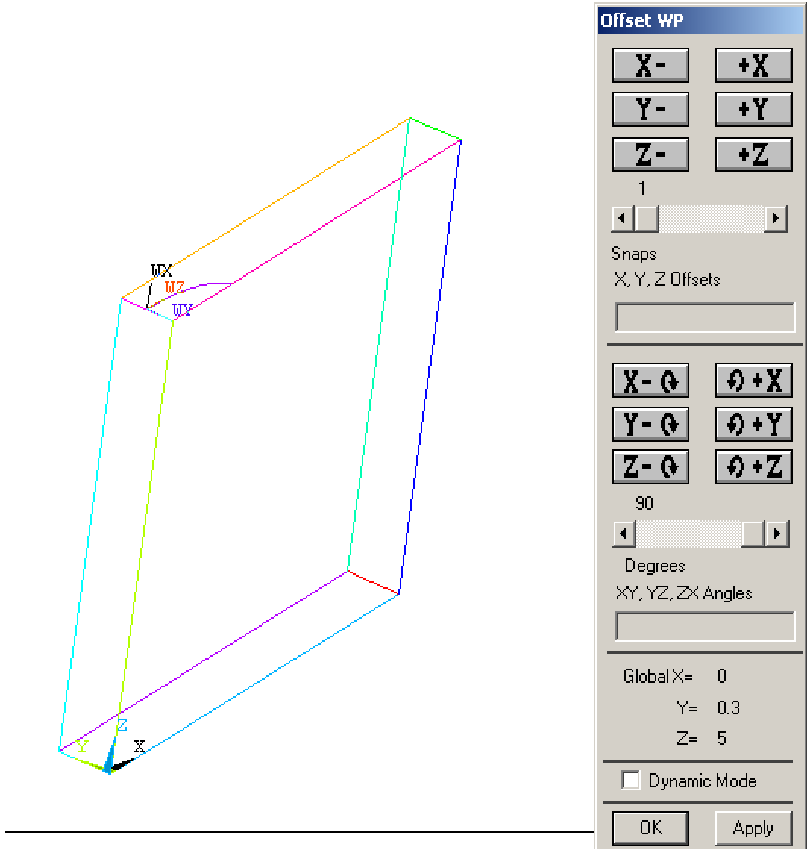
We must be delete the quarter part of the circle **ANSYS Command Line** **/repl.** We offset to the workplane to the centure of the model

And we use ***Workplane-Offset WP with – keypoints***





And we use ***Offset workplane by increments***

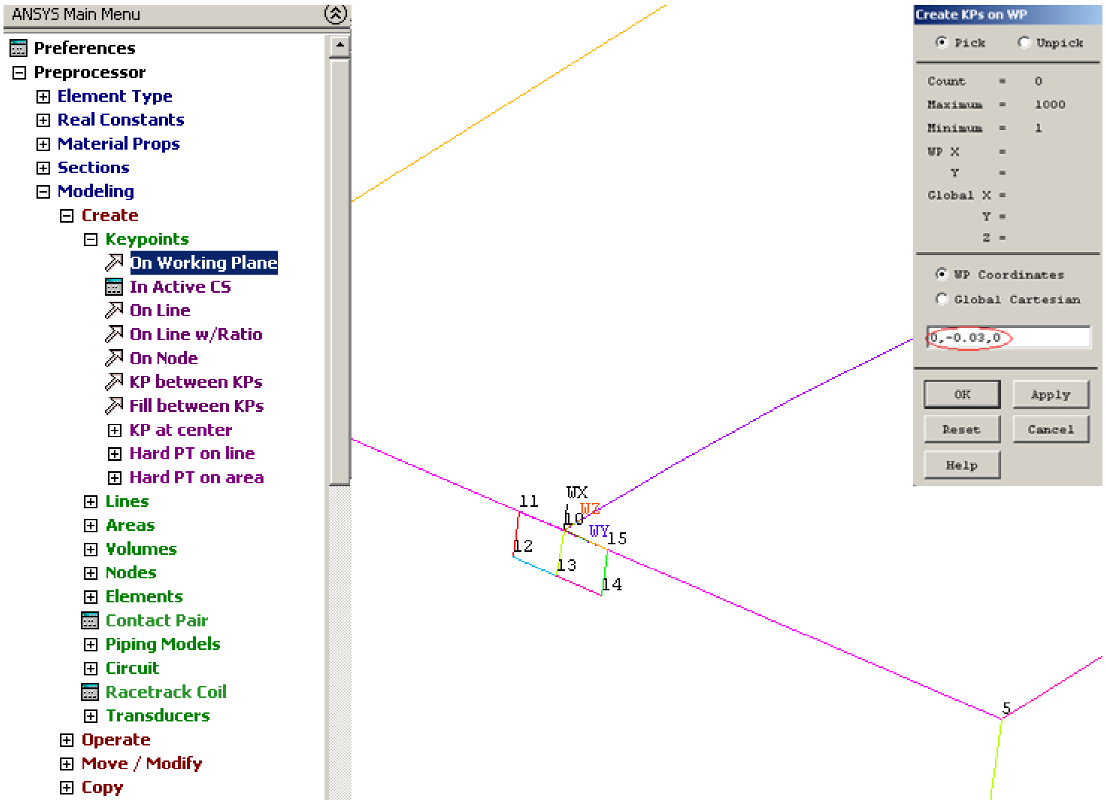


Create Keypoints on Wp

Preprocessor-Create-Keypoints-On Working Plane...

We are going to create 5 key points given in the following table:

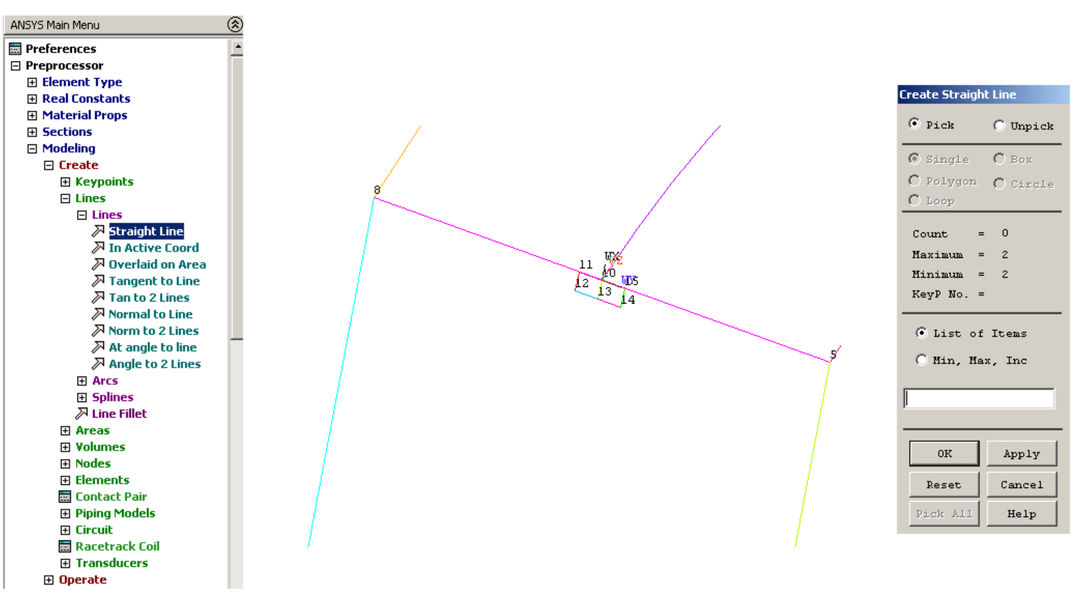
Keypoints	X(m)	Y(m)	Z(m)
11	0	-0.03	0
12	-0.03	-0.03	0
13	-0.03	0	0
14	-0.03	0.03	0
15	0	0.03	0



Create Lines

Preprocessor-Modeling-Creat-Lines-Straight Lines

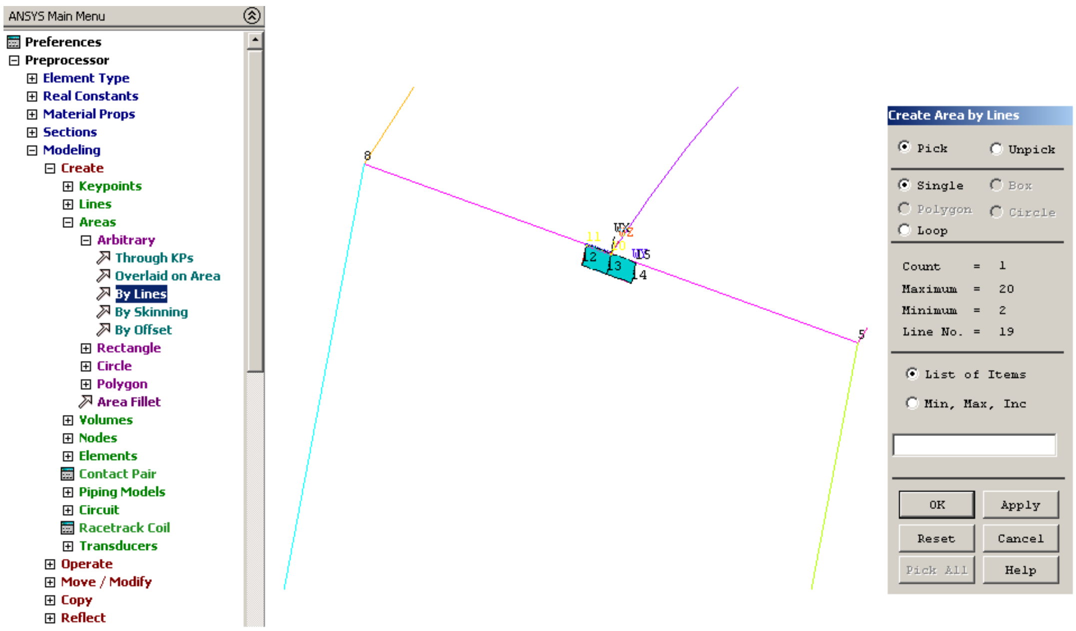
This is required to create the models boundary lines, successively like first 11 to 12, 12 to 13, 13 to 14, 15 to 10, 10to 11 and finally 10 to 13..



Create Lines

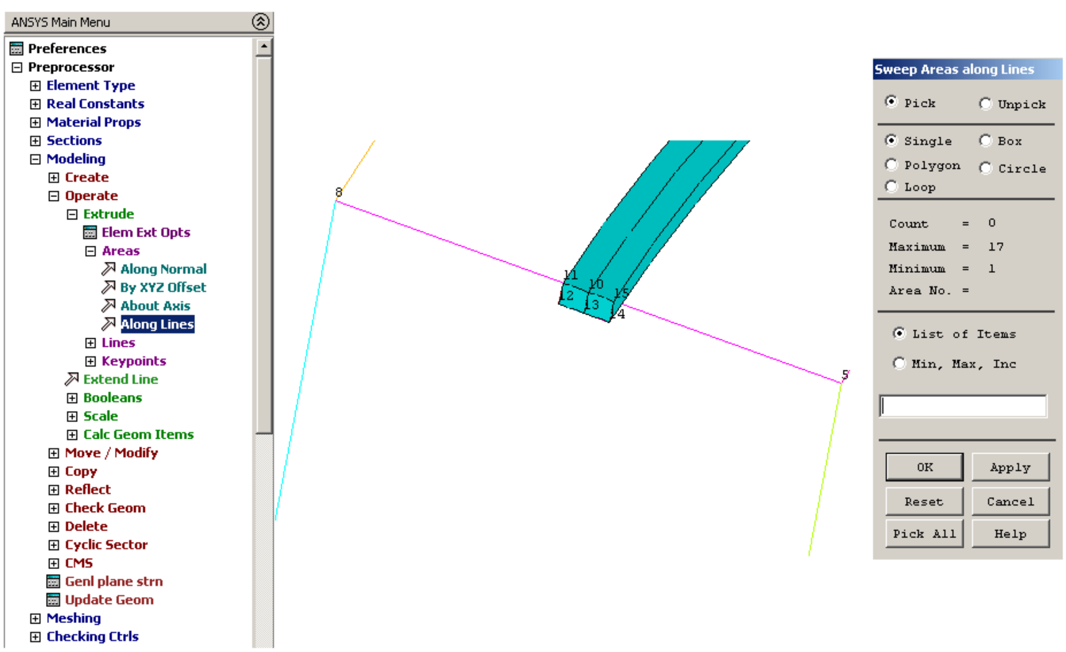
Preprocessor-Modeling-Creat-Areas-Arbitrary-By Lines

Pick all lines (Click OK in the picking window).



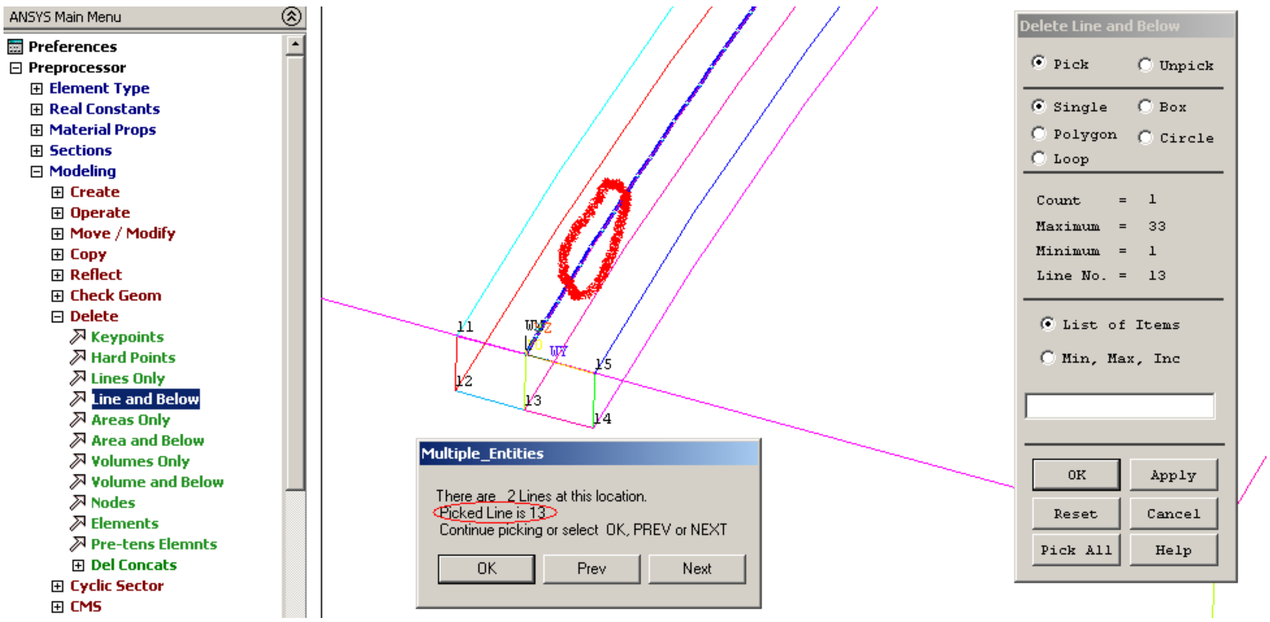
Preprocessor-Modeling-Operate-Extrude-Areas-Along lines

First, we choose small areas and click Apply Button. After choosing line click Ok.



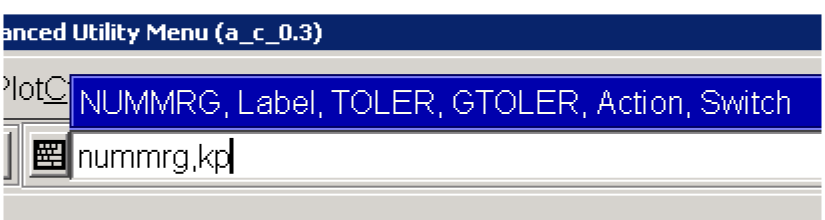
Ansys Command Prompt (lplot)

and than Delete-Line and below choose the line

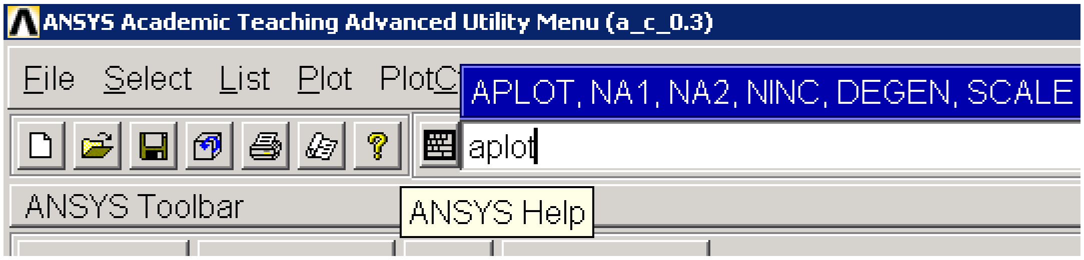


Ansys Command Prompt (allsel)

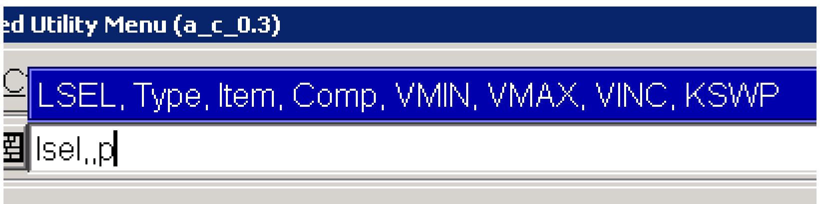
Ansys Command Prompt (nummrg,kp)



Ansys Command Prompt (aplot)



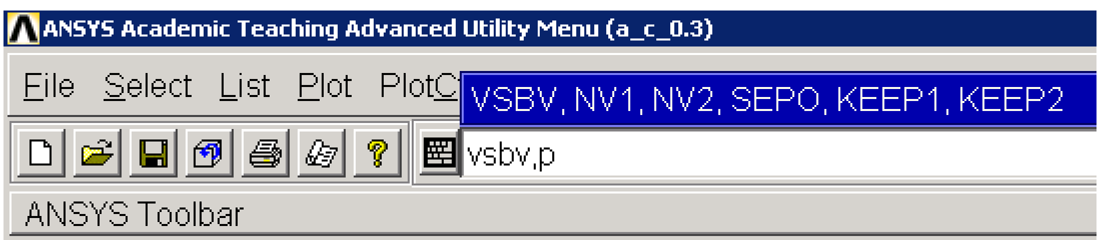
Ansys Command Prompt (lsel,,p) **Select line ın cevabı yok Komut ve resmi silinmeli**

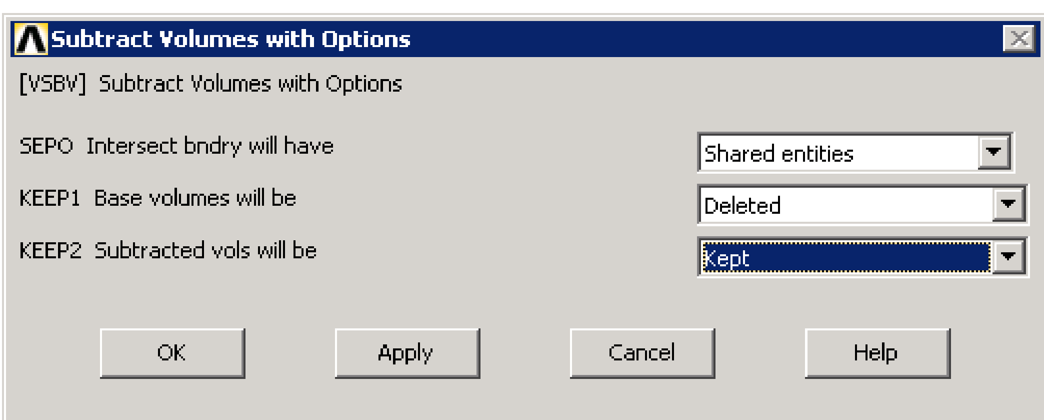


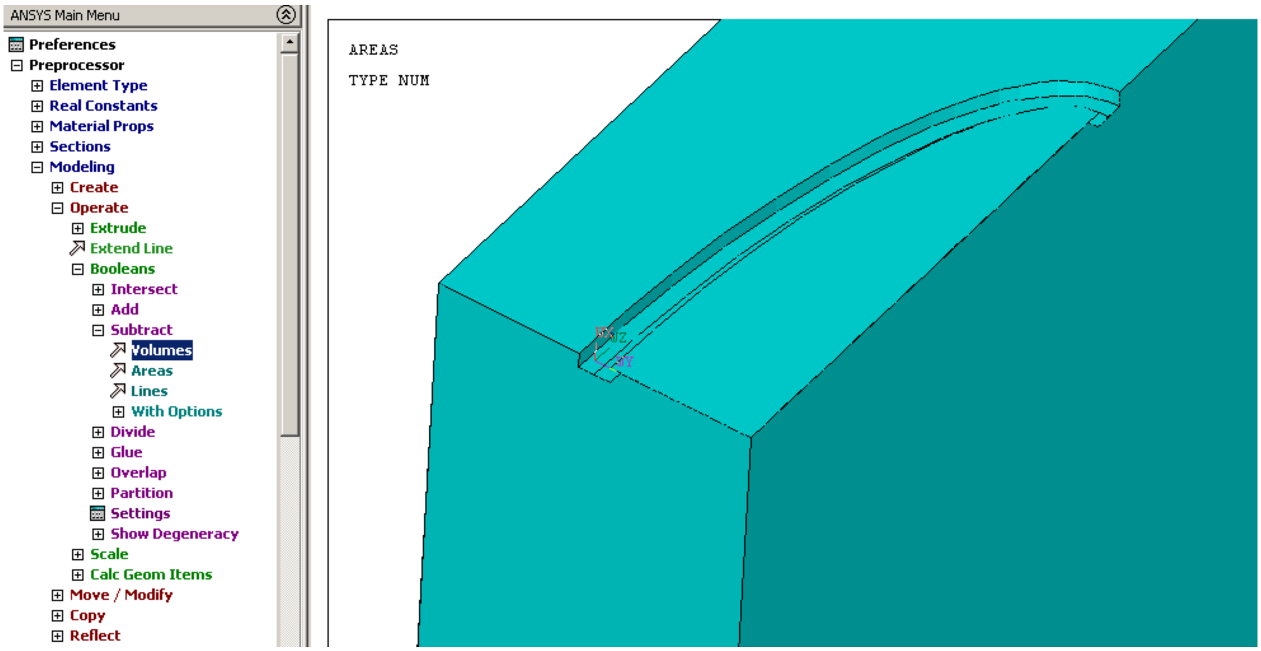
Preprocessor-Modeling-Operate-Booleans-Substract-Volumes

First we choose the all volume to pick apply and than select crack area to delete

Ansys Command Prompt (vsbv,p)

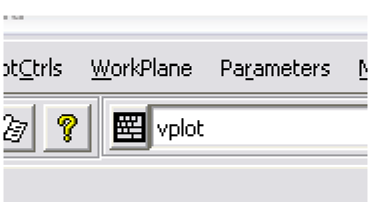


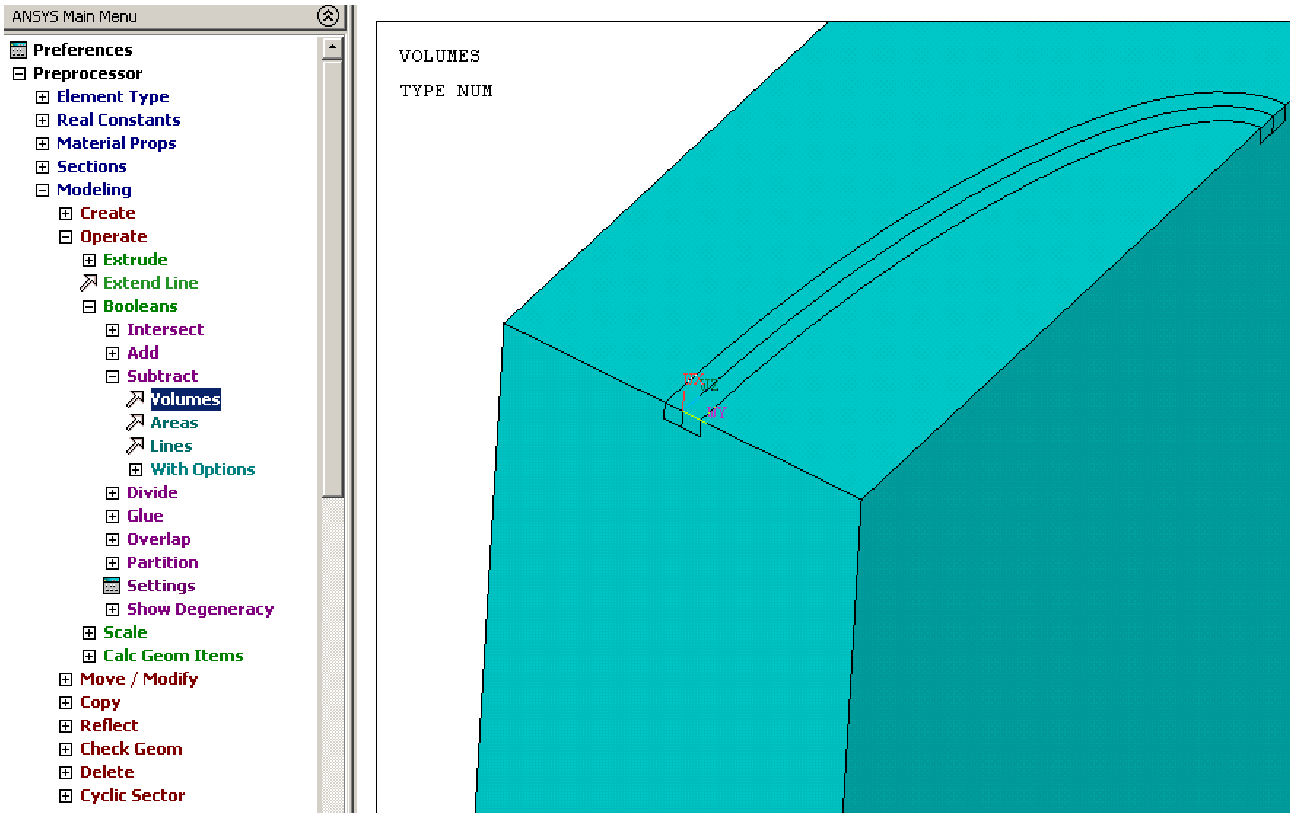




Ansys Command Prompt (allsel)

Ansys Command Prompt (vplot)



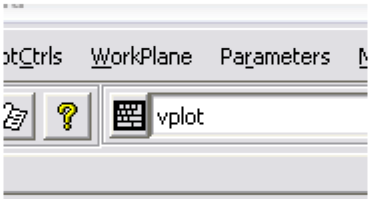


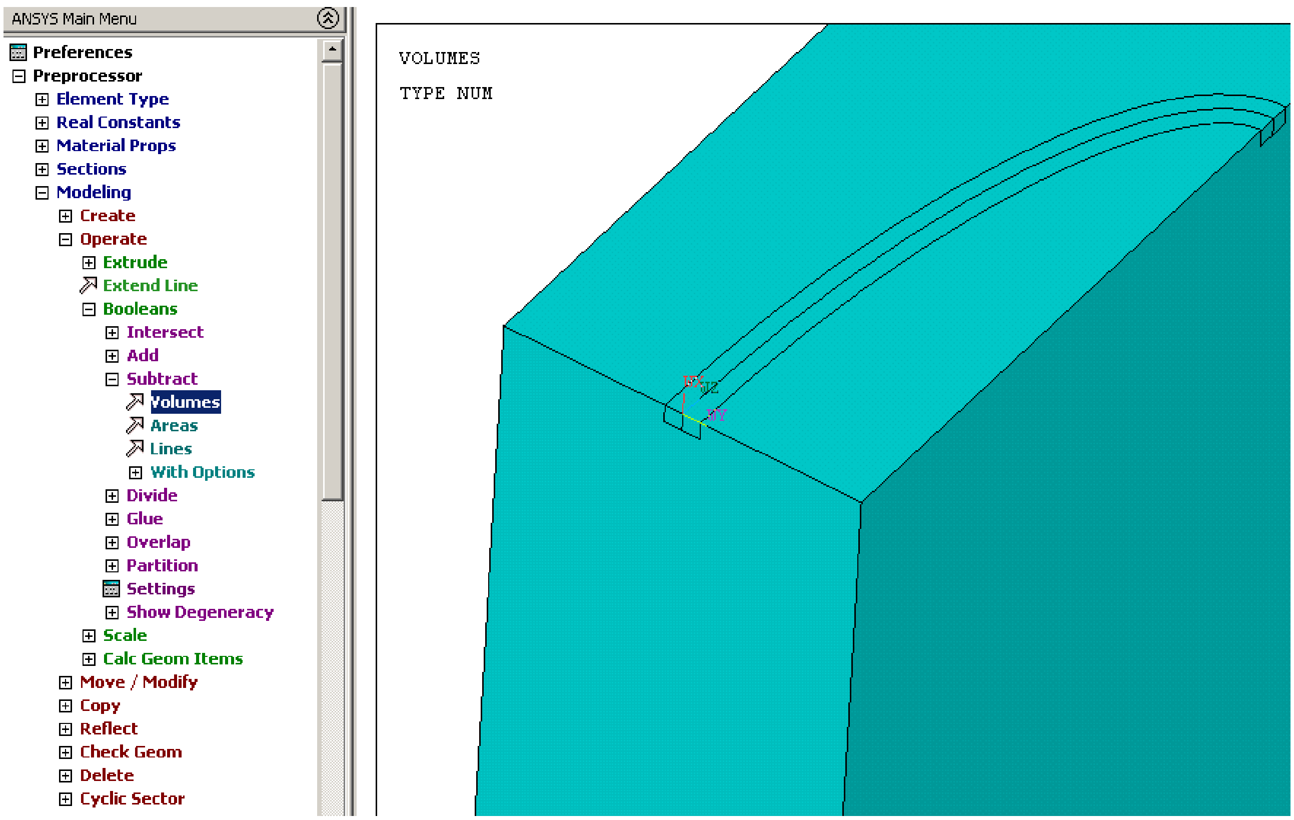
Ansys Command Prompt (lplot)

Ansys Command Prompt (allsel)

Ansys Command Prompt (nummrg,kp) (no keypoint were merged)

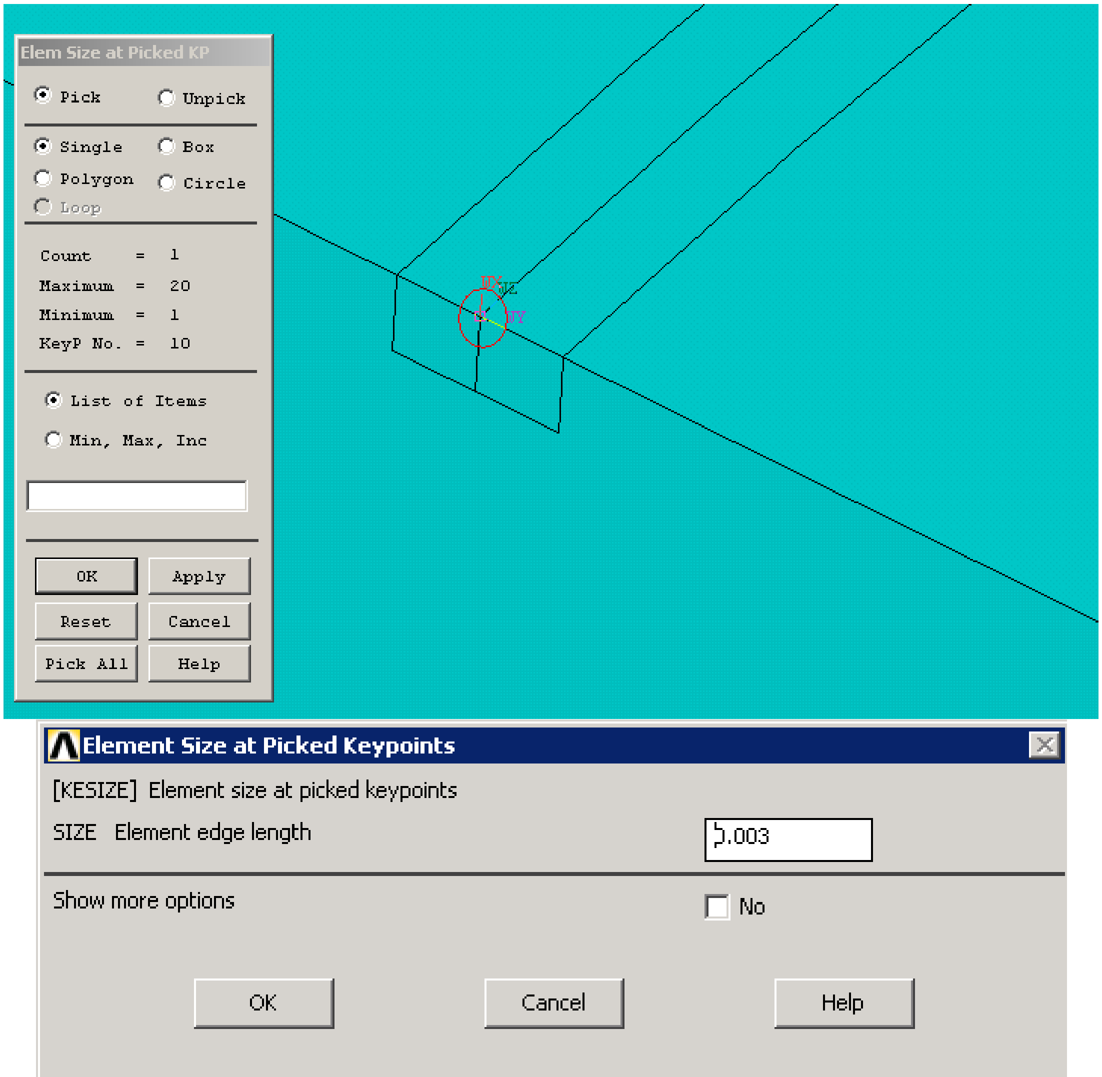
Ansys Command Prompt (vplot)



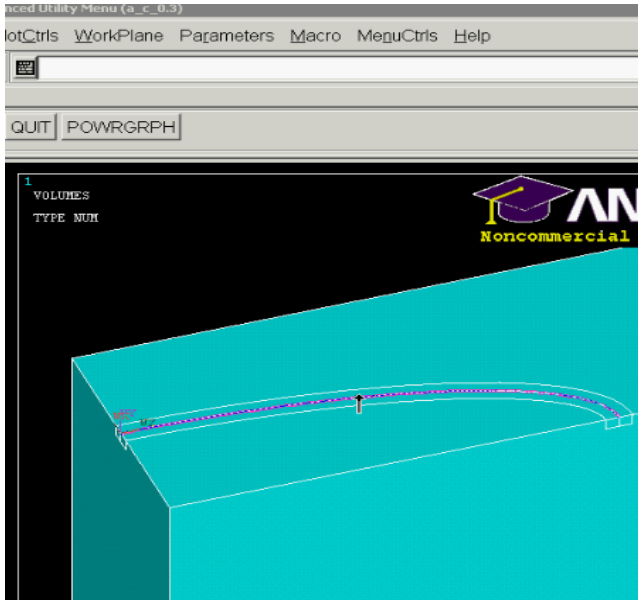


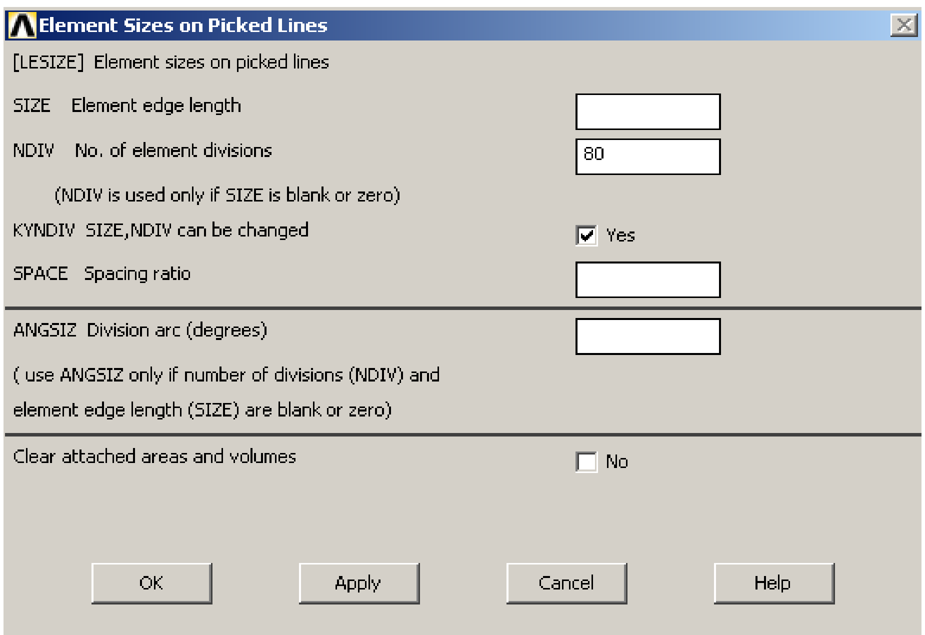
Ansys Command Prompt (kesize,p) and we choose to WP

To obtain accurate fracture solution, we need to generate fine mesh near the crack tip. For this, we can use the **KESIZE** command to specify element size at the crack tip keypoint. First zoom into the crack tip region.



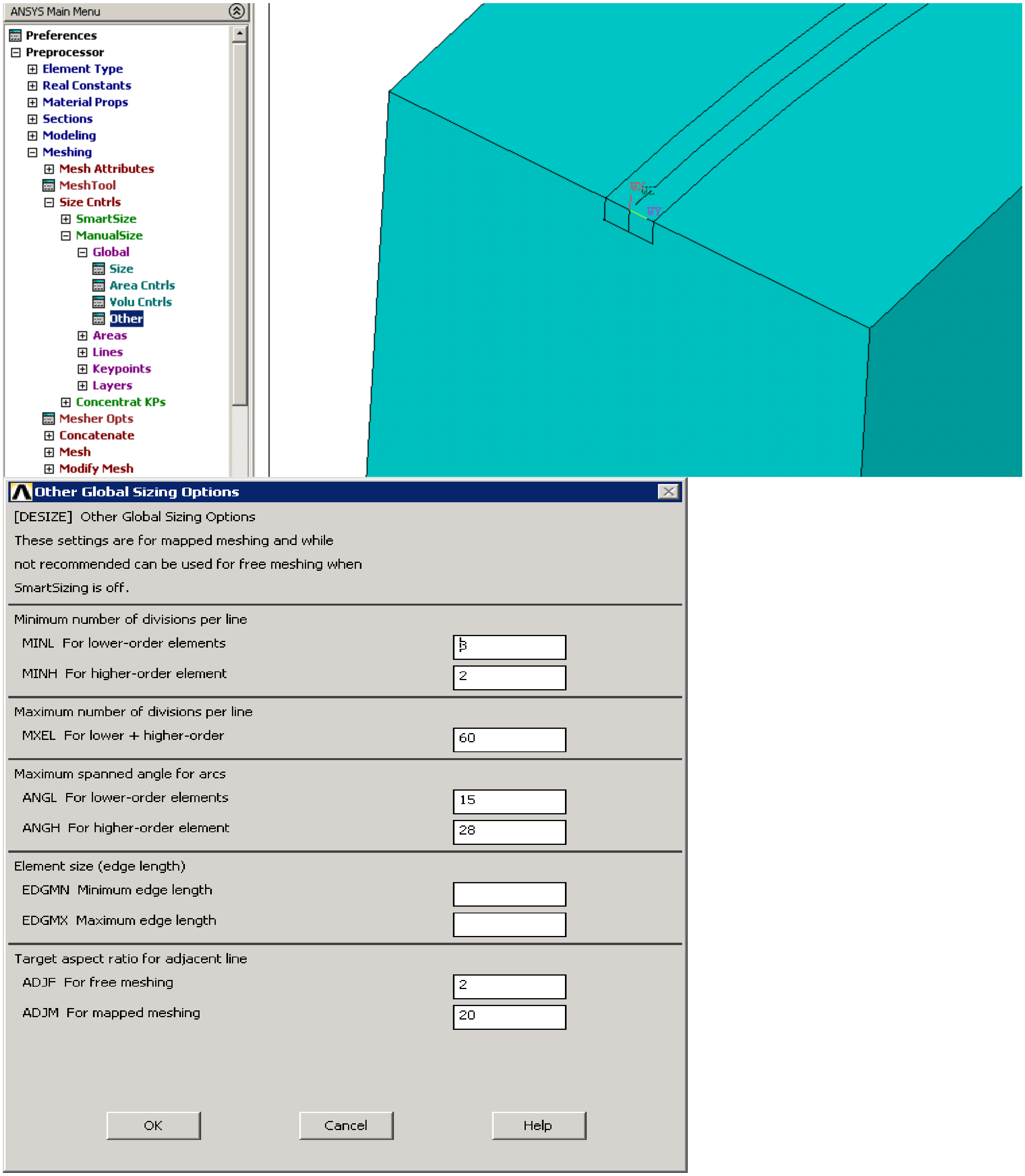
Ansys Command Prompt (lesize,p)





Size Controls

Preprocessor-Meshing-Size cntrls-Manual Size-Global-Others

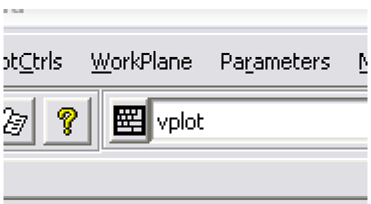

ANSYS Main Menu
Preferences
Preprocessor
Element Type
Real Constants
Material Props
Sections
Modeling
Meshing
Mesh Attributes
MeshTool
Size Cntrls
SmartSize
ManualSize
Global
Size
Area Cntrls
Volu Cntrls
Other
Areas
Lines
Keypoints
Layers
Concentrat KPs
Mesher Opts
Concatenate
Mesh
Modify Mesh
Other Global Sizing Options
[DESIZE] Other Global Sizing Options
These settings are for mapped meshing and while
not recommended can be used for free meshing when
SmartSizing is off.
Minimum number of divisions per line
MINL For lower-order elements
3
MINH For higher-order element
2
Maximum number of divisions per line
MXEL For lower + higher-order
60
Maximum spanned angle for arcs
ANGL For lower-order elements
15
ANGH For higher-order element
28
Element size (edge length)
EDGMN Minimum edge length
EDGMX Maximum edge length
Target aspect ratio for adjacent line
ADJF For free meshing
2
ADJM For mapped meshing
20
OK
Cancel
Help

Ansys Command Prompt (allsell)

Ansys Command Prompt (lplot)

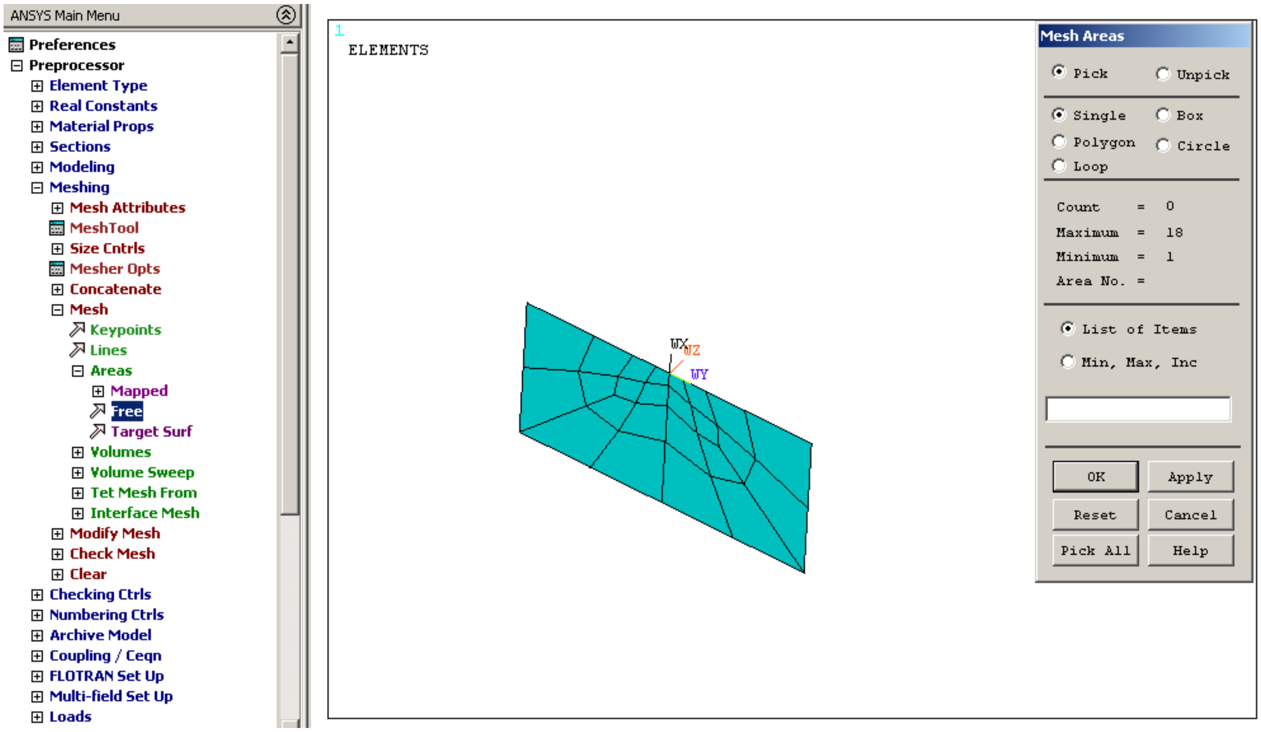
Ansys Command Prompt (aplot)

Ansys Command Prompt (vplot)



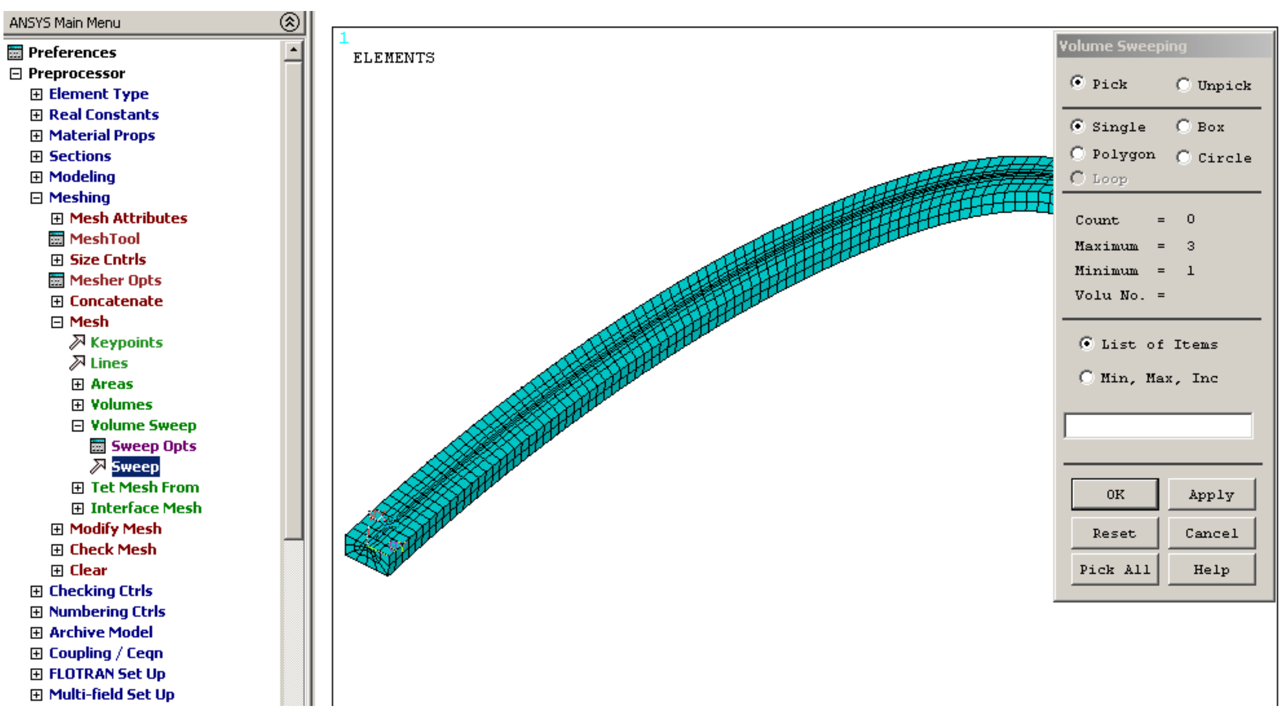
Meshing The Areas

Preprocessor-Meshing-Mesh-Areas-Free

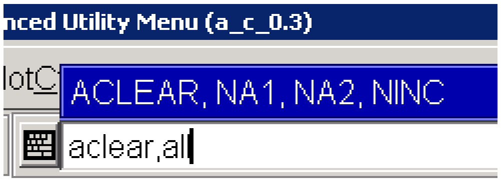


Areas Sweep For Creating Crack Volume

Preprocessor-Meshing-Mesh-Volume Sweep-Sweep

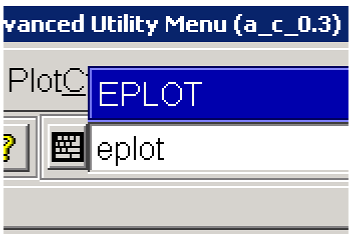


Ansys Command Prompt (aclear,all)

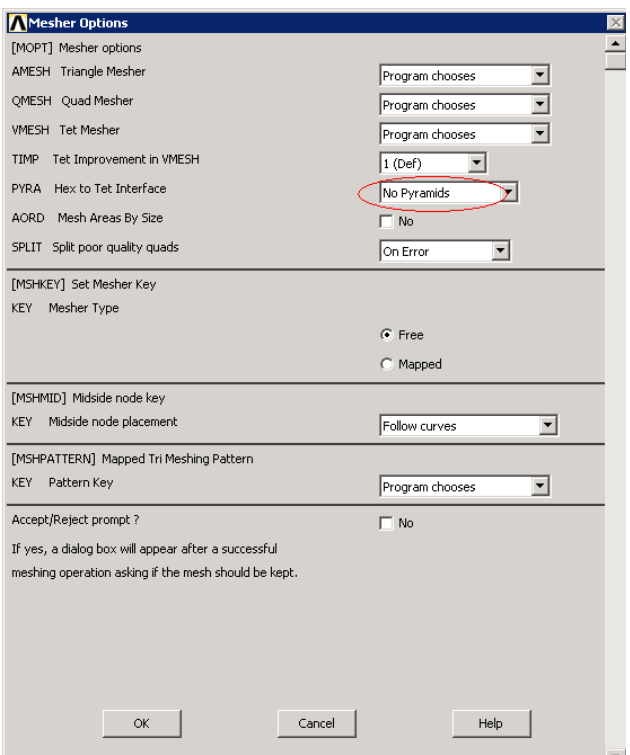


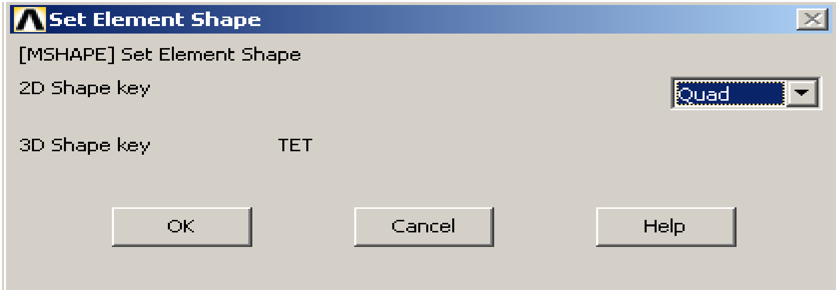
We use **Aclear, all** ' Area mesh is deleted. Because of this, a gap occurs in the sequence of element numbers. To remove the gap we use, **Numcmp, elem**

Ansys Command Prompt (eplot)

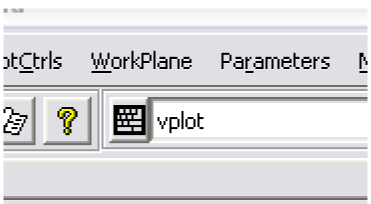


Preprocessor-Meshing-Mesher Opts

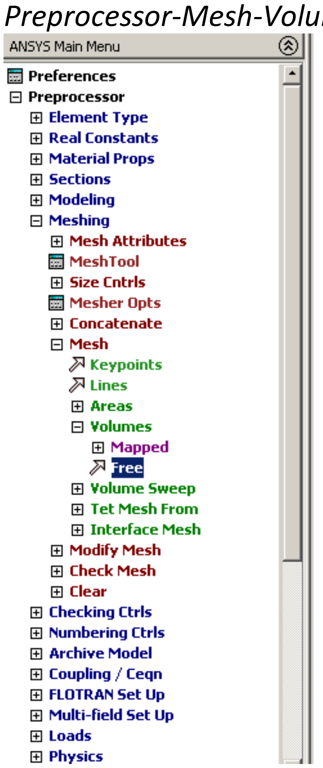


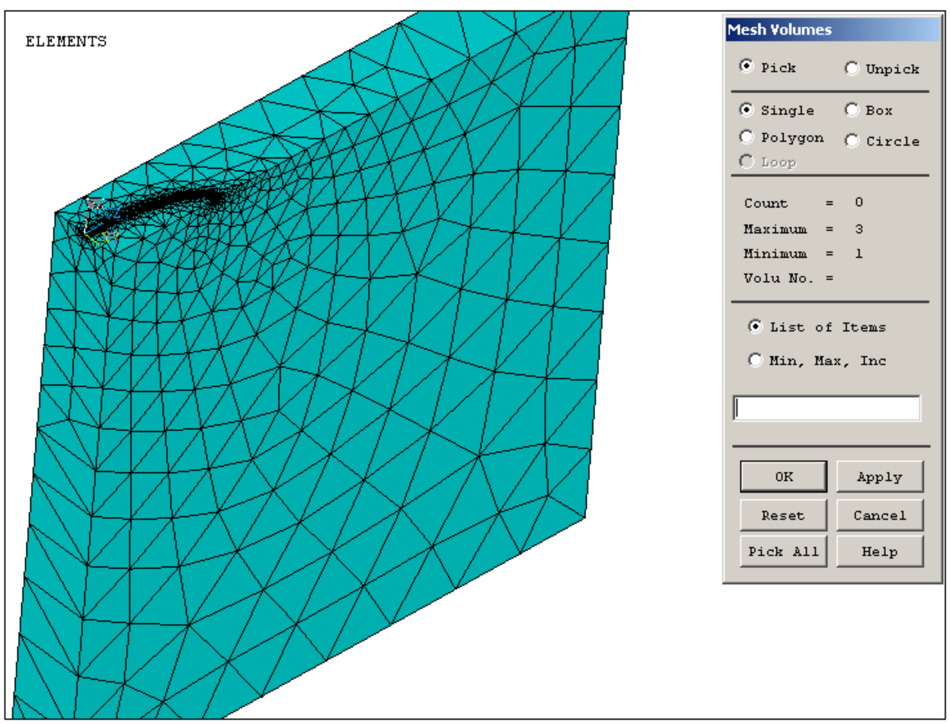


Ansys Command Prompt (vplot)



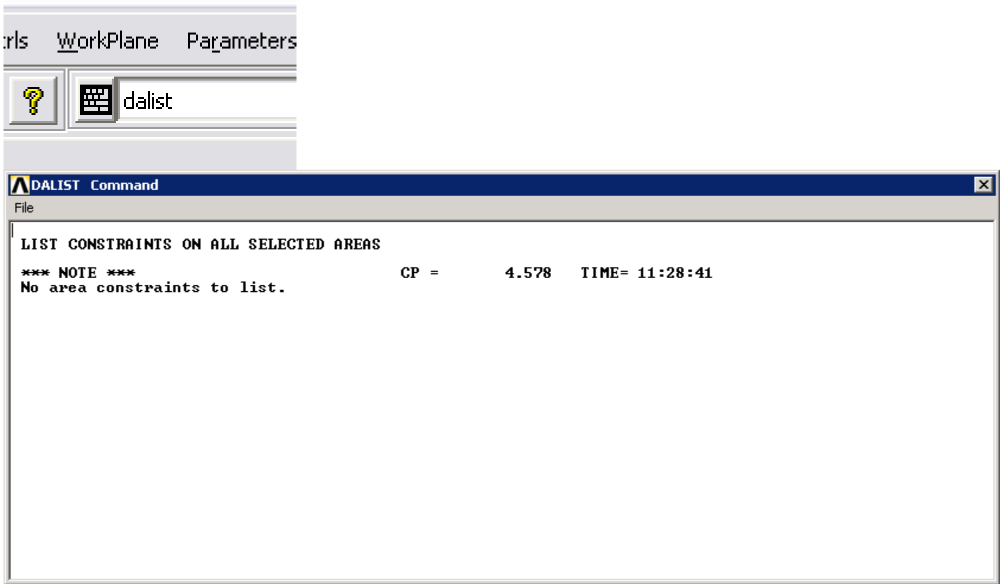
Preprocessor-Mesh-Volume-Free





Ansys Command Prompt (dalist)

To check the applied boundary conditions on areas, **DALIST** is used in the command lineFme



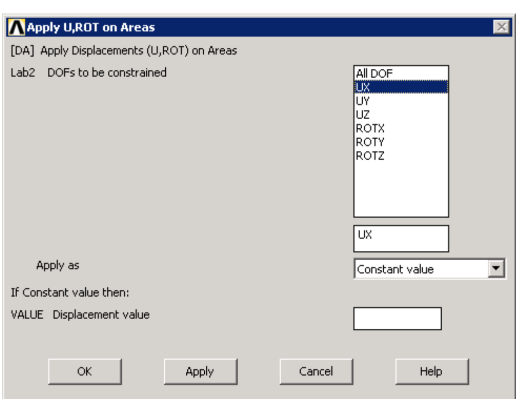
Ansys Command Prompt (da,p)

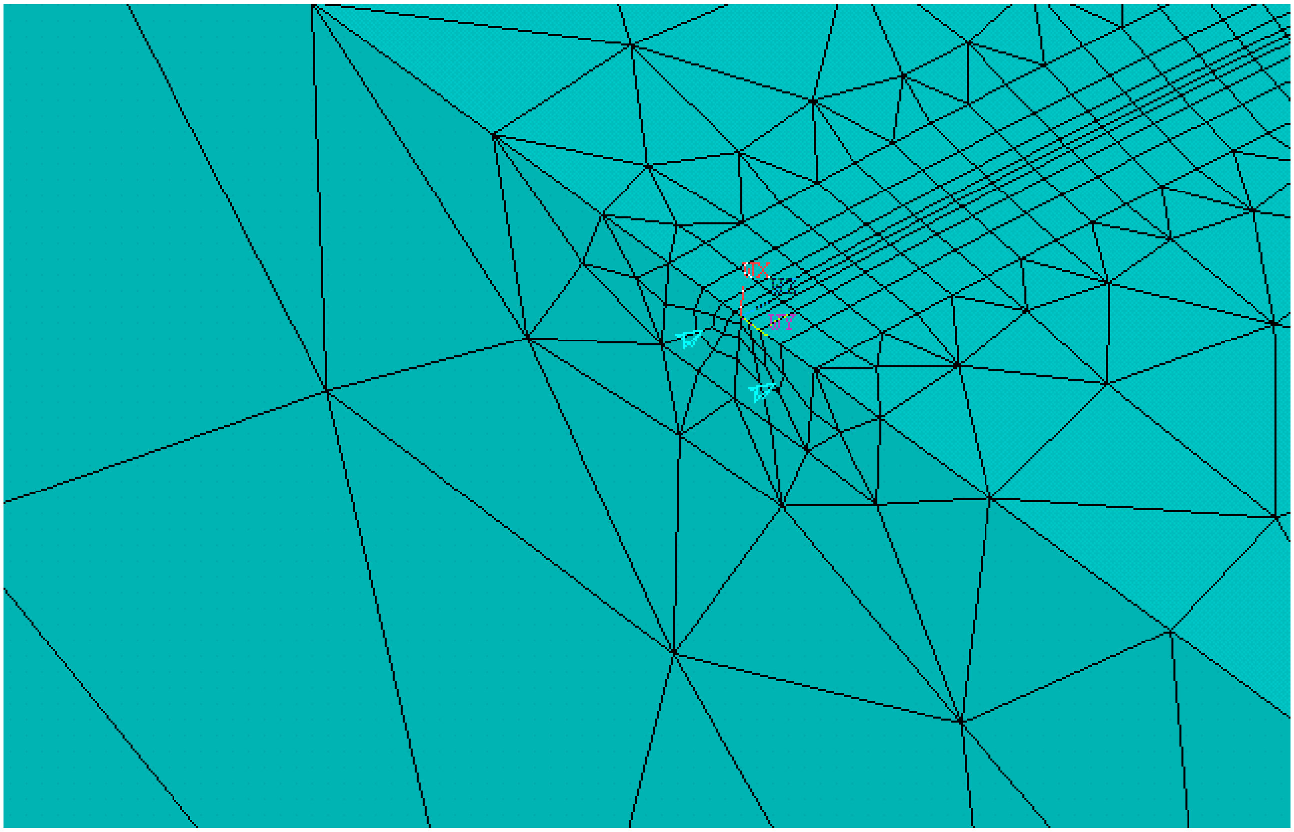
Apply the displacement constrains using

Apply Boundary Conditions

Because of the symmetry, our system has the following BC's:

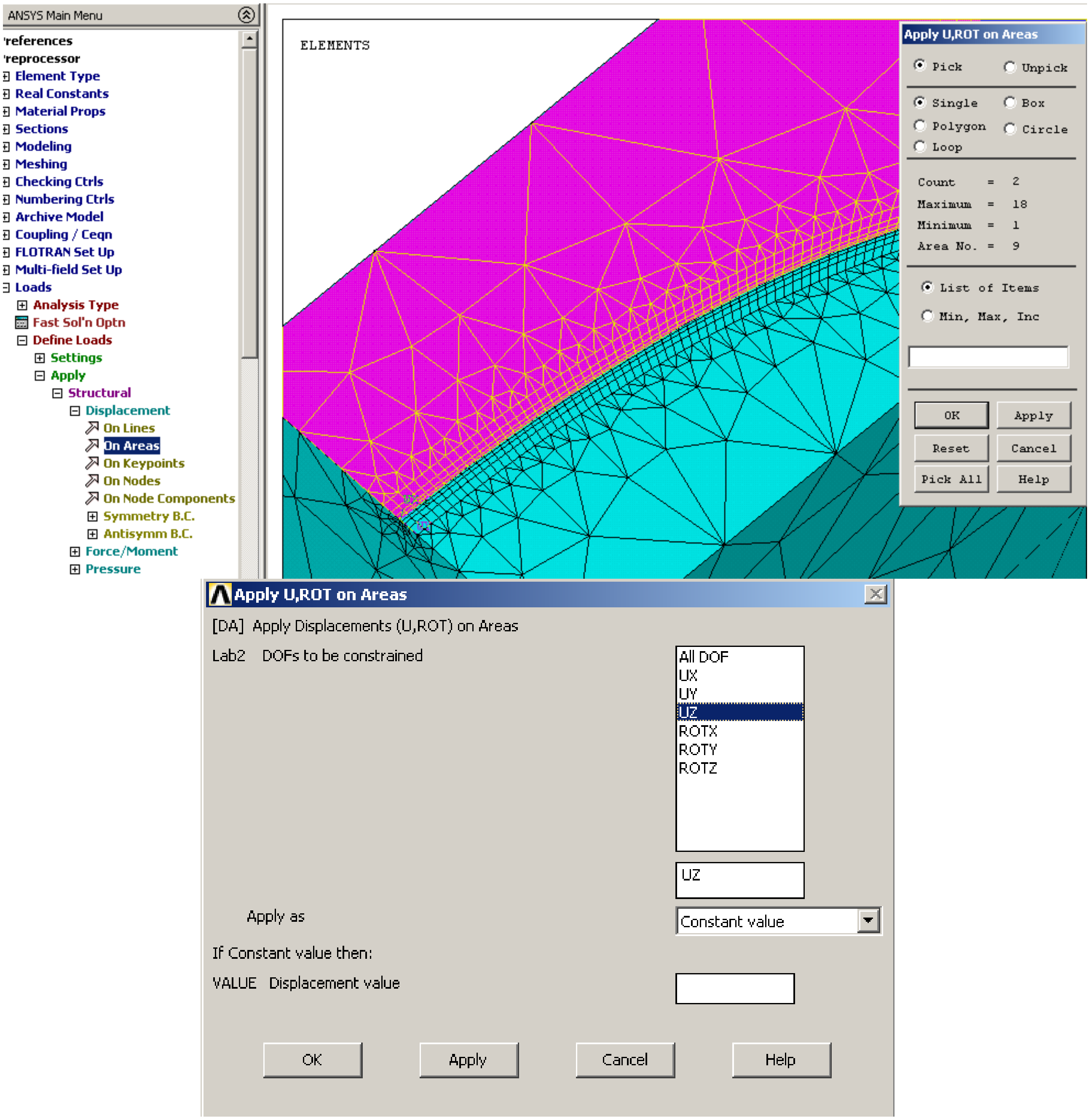
Select the all left X Y surface... (With small areas)



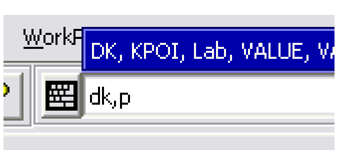


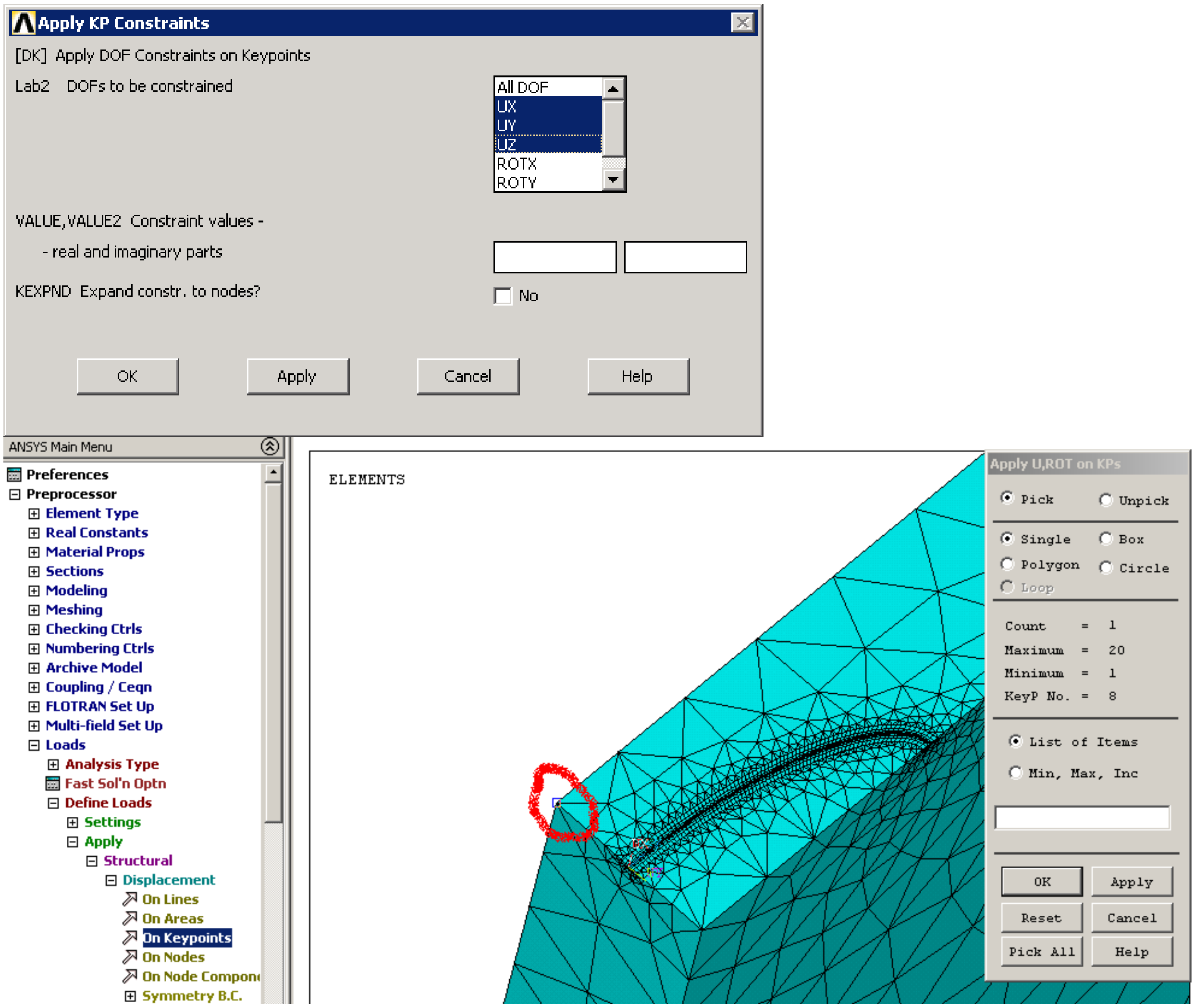
Ansys Command Prompt (da,p)

Select the Y Z up areas but big area and the crack one part...

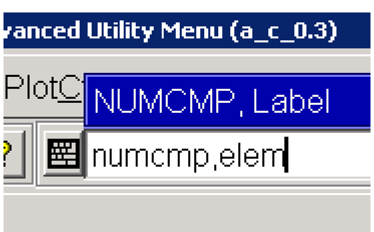


Ansys Command Prompt (dk,p)

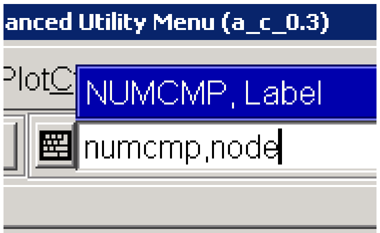




Ansys Command Prompt (numcmp,elem)

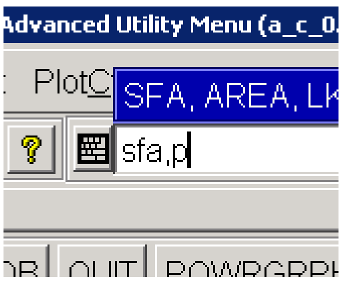


Ansys Command Prompt (numcmp,node)

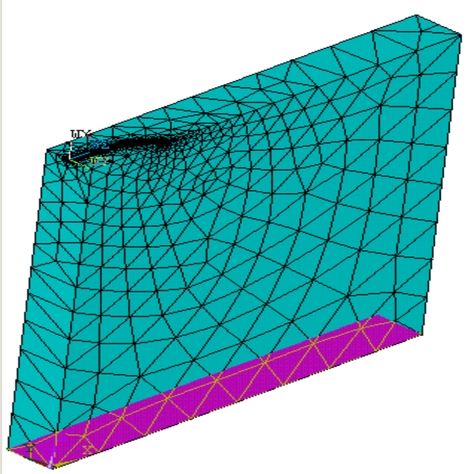


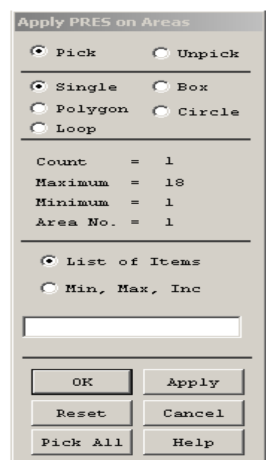
Ansys Command Prompt (sfa,p)

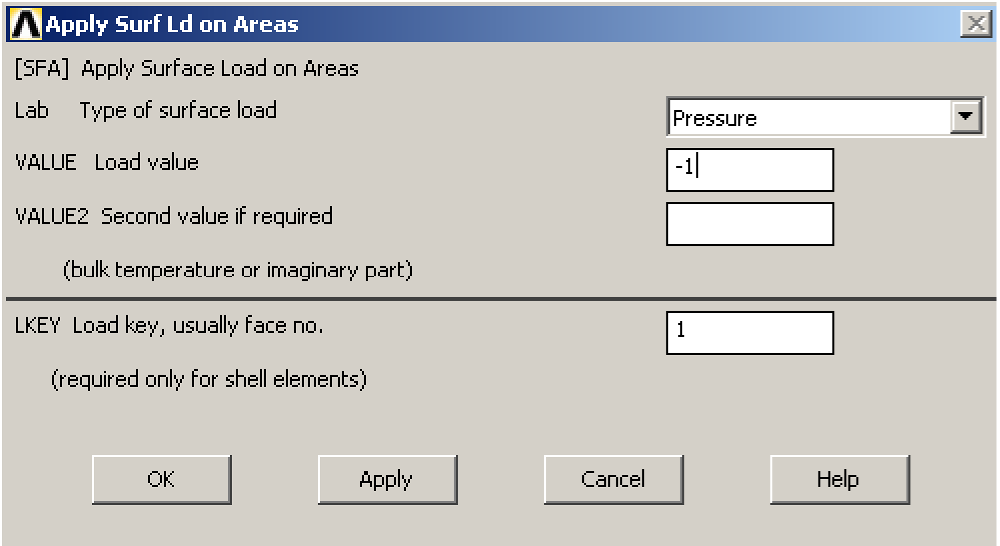
Now we will apply the distributed surface forces (pressure).



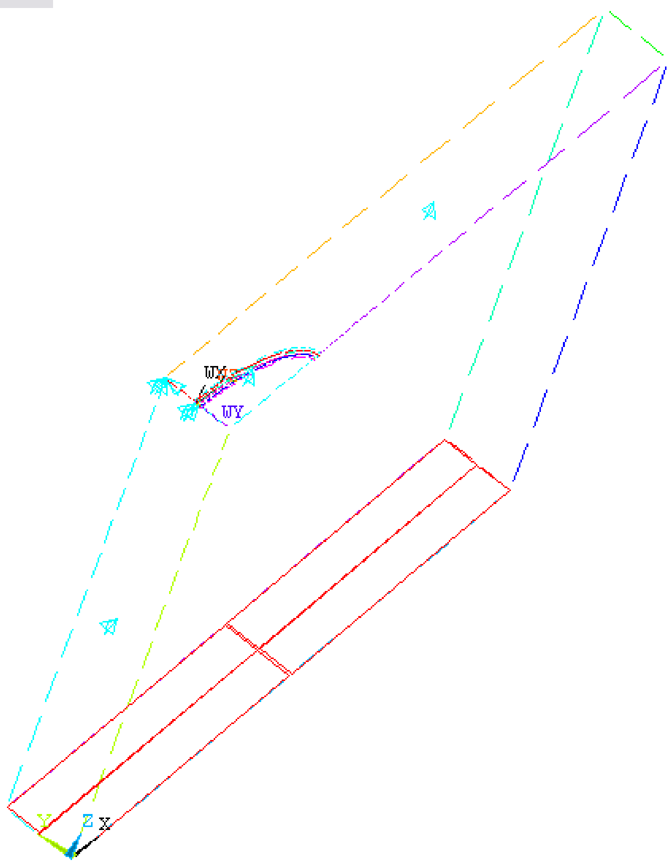
Select the Y Z area of down volume



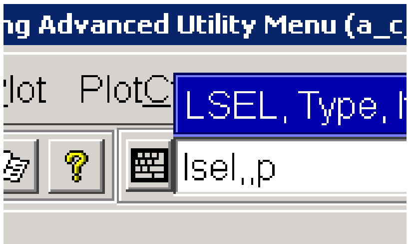


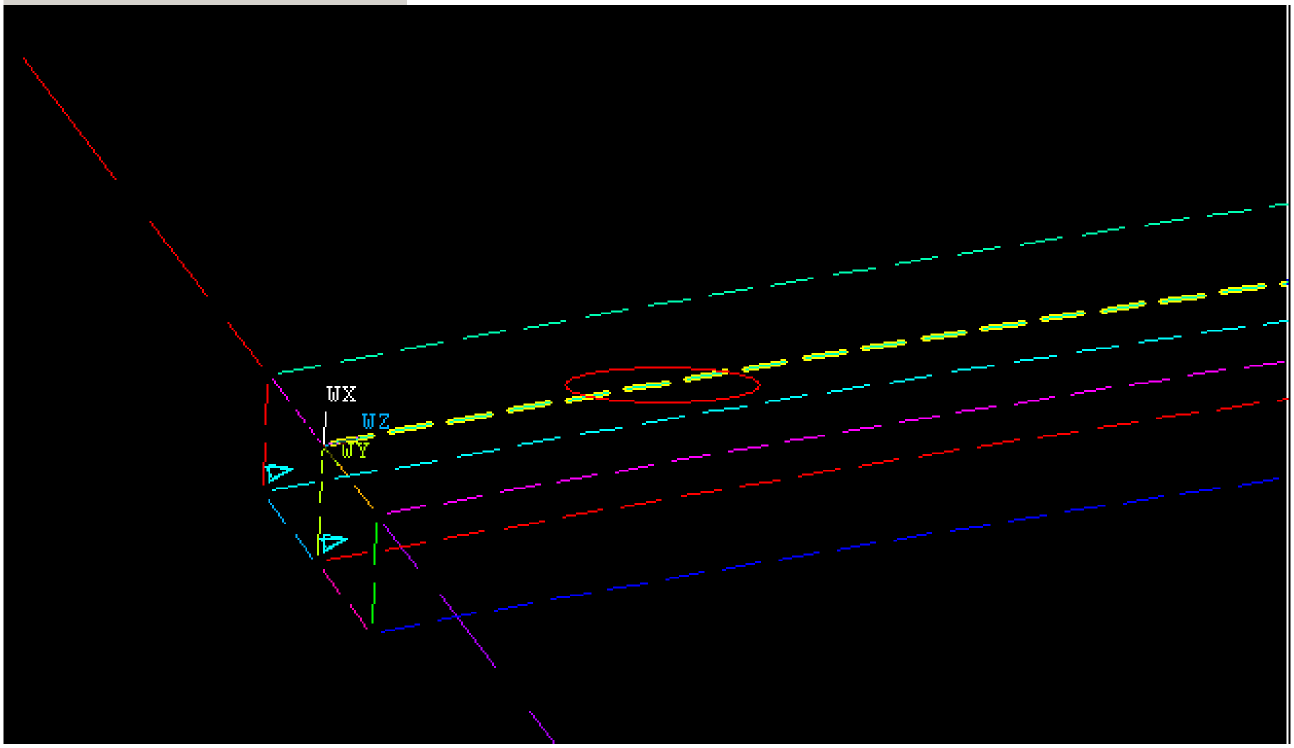


Ansys Command Prompt (lplot)

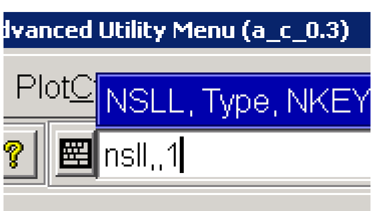


Ansys Command Prompt (lsel,,p)

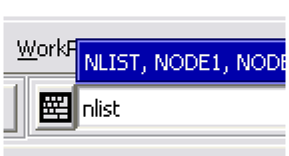




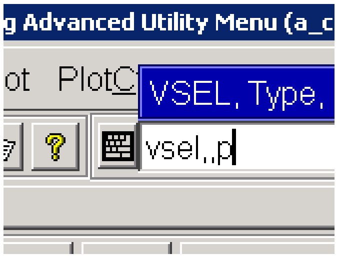
Ansys Command Prompt (nsll,,1)

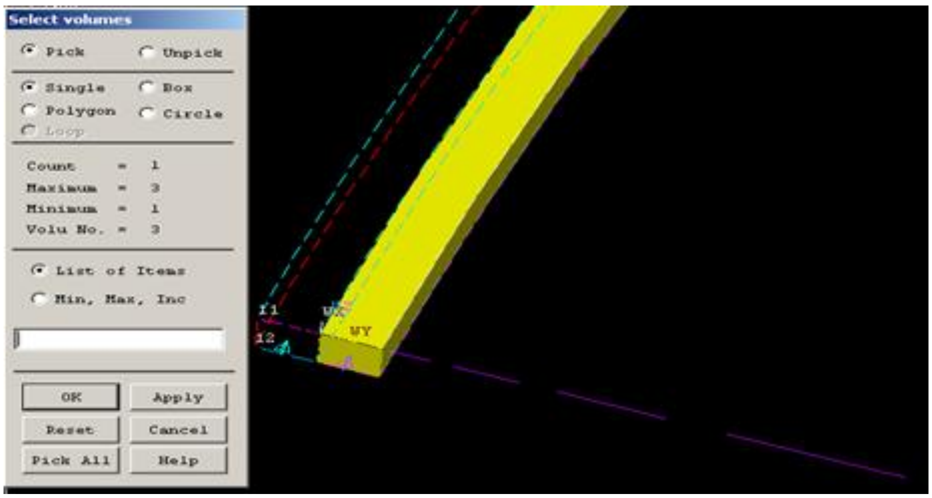


Ansys Command Prompt (nlist) a_c_0.3crnodes
The node information (for the crack tip nodes) is saved from the NLIST window as **a_c_0.3.crnodes**

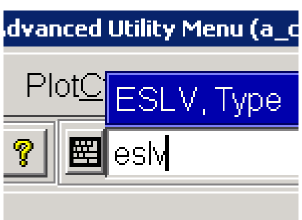


Ansys Command Prompt (vsel,,p)



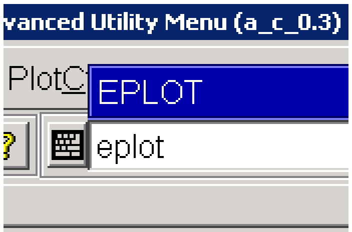


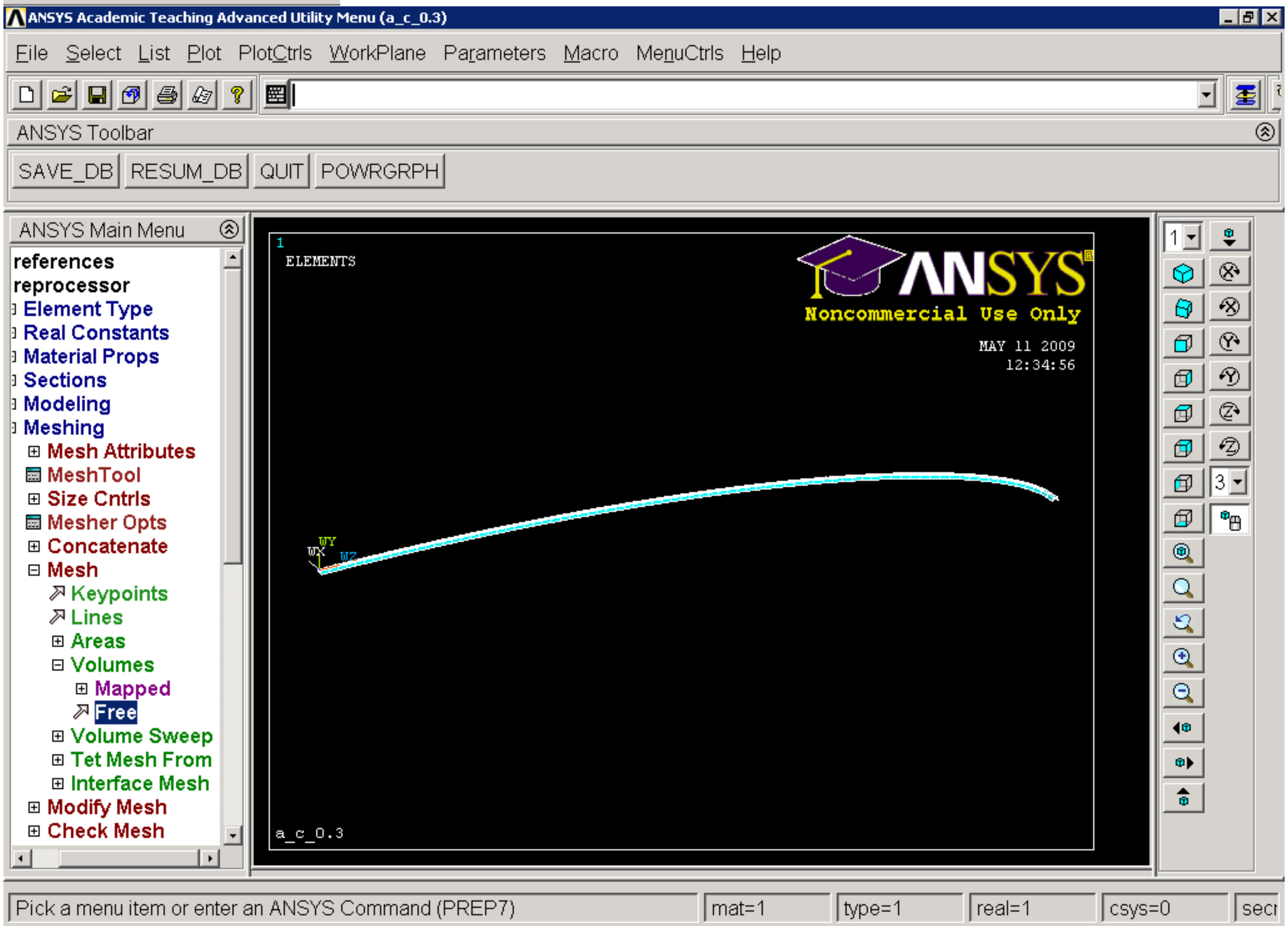
Ansys Command Prompt (eslv)



Ansys Command Prompt (esln,r)

Ansys Command Prompt (eplot)





Ansys Command Prompt (elist) a_c_0.3.crelems

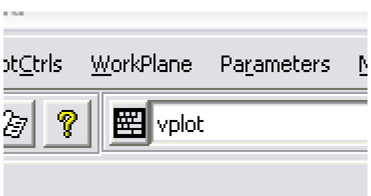
The element information (for the crack tip elements) is saved from the Elist window as **a_c_0.3.crelems**

Ansys Command Prompt (allsell)

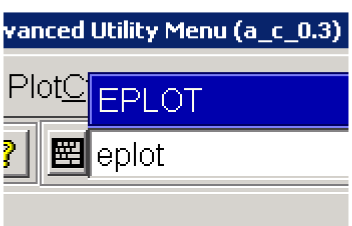
Ansys Command Prompt (lplot)

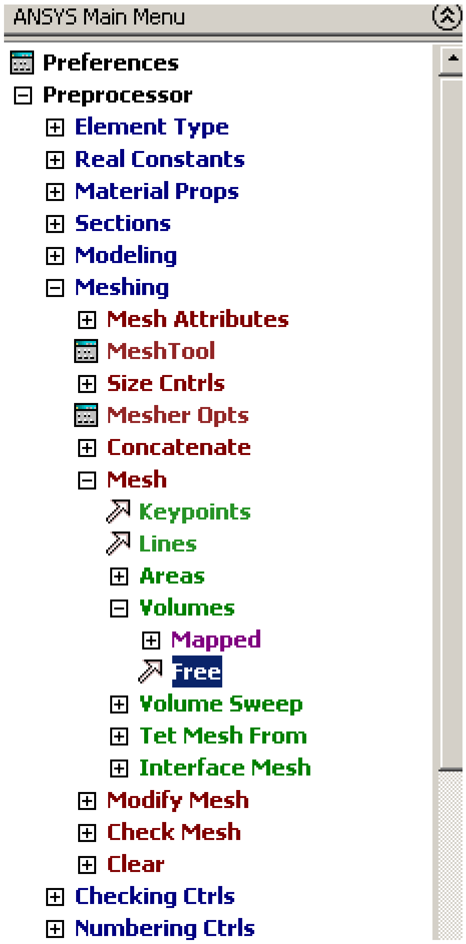
Ansys Command Prompt (aplot)

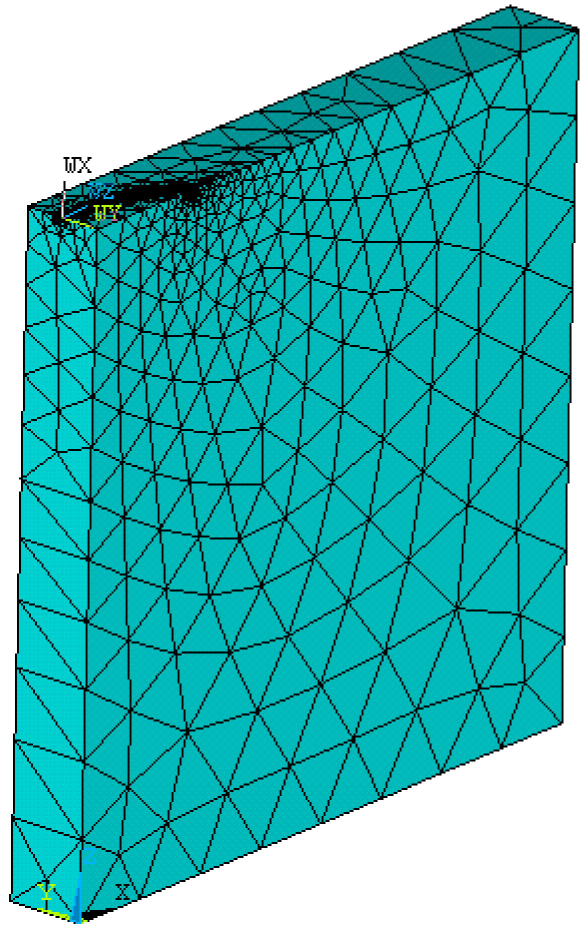
Ansys Command Prompt (vplot)



Ansys Command Prompt (eplot)

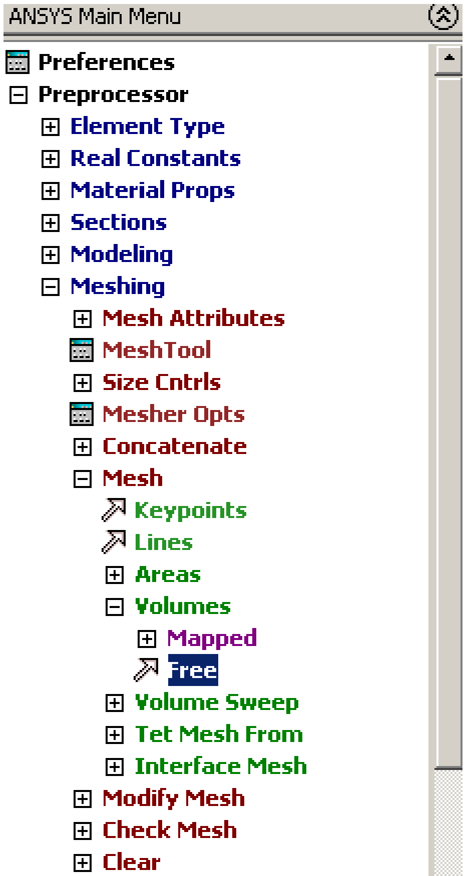


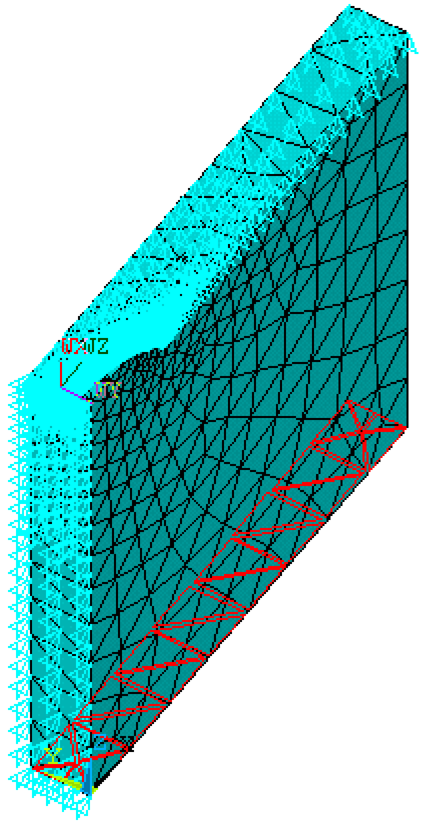




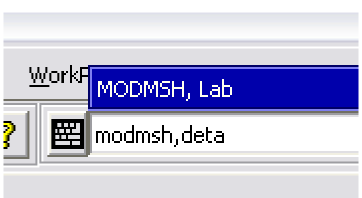
Ansys Command Prompt (sbct)

SBCTRAN is used to transfer solid model loads and boundary conditions to the FE model. Loadsand boundary conditions on unselected keypoints, lines, areas, and volumes are not transferred. **sbct**

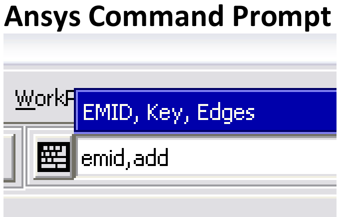


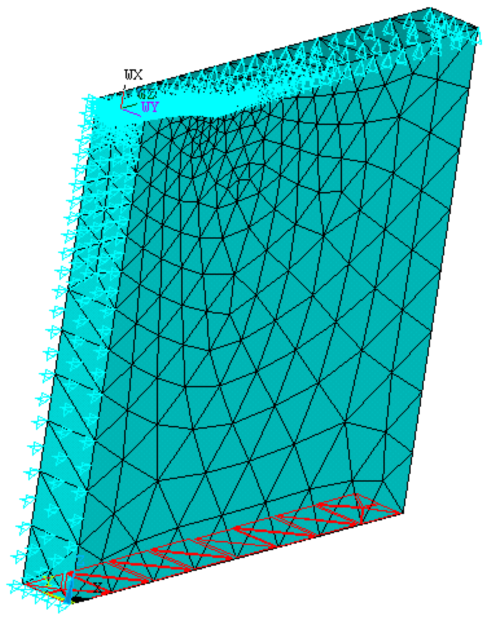


Ansys Command Prompt (modmsh,deta)

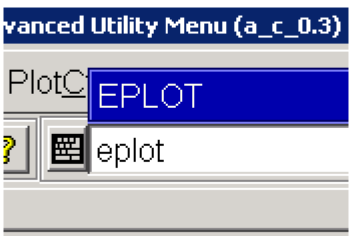


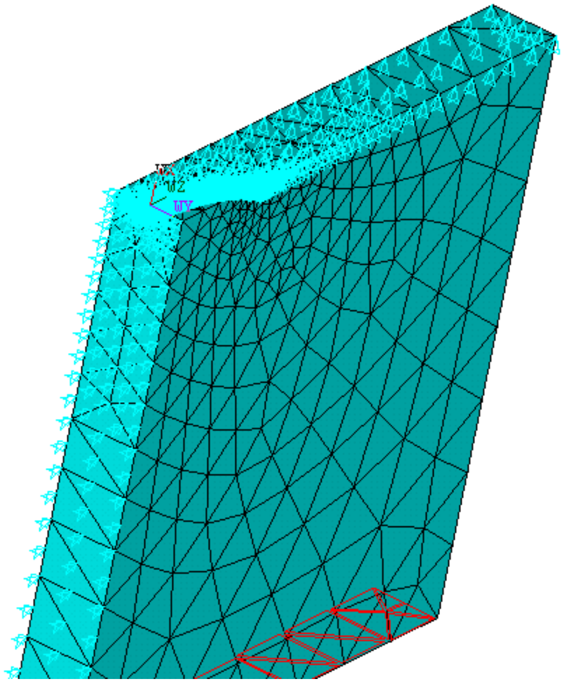
Ansys Command Prompt (emid,add)



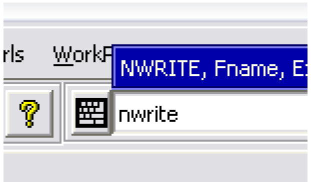


Ansys Command Prompt (eplot)





Ansys Command Prompt (nwrite)
Also using **nwrite** all nodes are saved as ***a_c_03.node*** automatically in current working directory.



Ansys Command Prompt (elis) a_c_0.3.elis
Also using **Select-Everything** the whole element list is saved as ***a_c_0.3.elis***

Ansys Command Prompt (dlis)
Using **dlist** displacement BC's are saved as ***a_c_0.3.dlis***

Ansys Command Prompt (sflis)
Using **sflist** pressure loads on elements are saved as ***a_c_0.3.sflis***

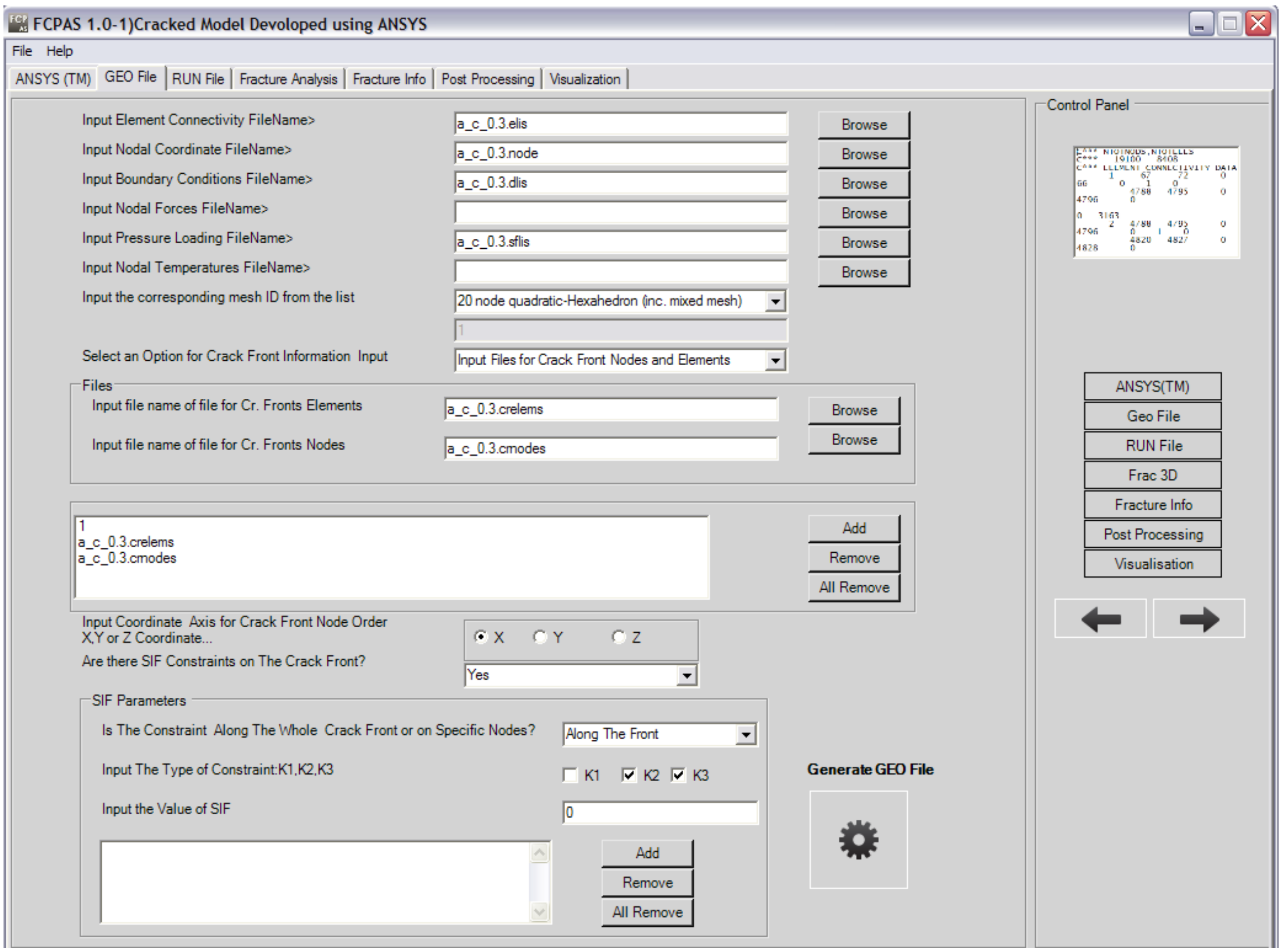
Now, we completed all modeling steps in ANSYS^{TM}. Now, we are ready to convert all the model information into FRAC3D format using the converter program.

T.3.3 Using Converter Codes for FRAC3D (Generation of *cc3.geo* File)

FRAC3D requires its model information in a specific format. To convert ANSYS^{TM} model files into FRAC3D format, we can use the *convert_ansys_frac3d.exe* program. The converter program can be run by typing, its path in MSDOS prompt or from the "Geo File" tab from FCPAS. Both methods are shown respectively.

T.3.3.1 Using *convert_ansys_frac3d.exe* in FCPAS

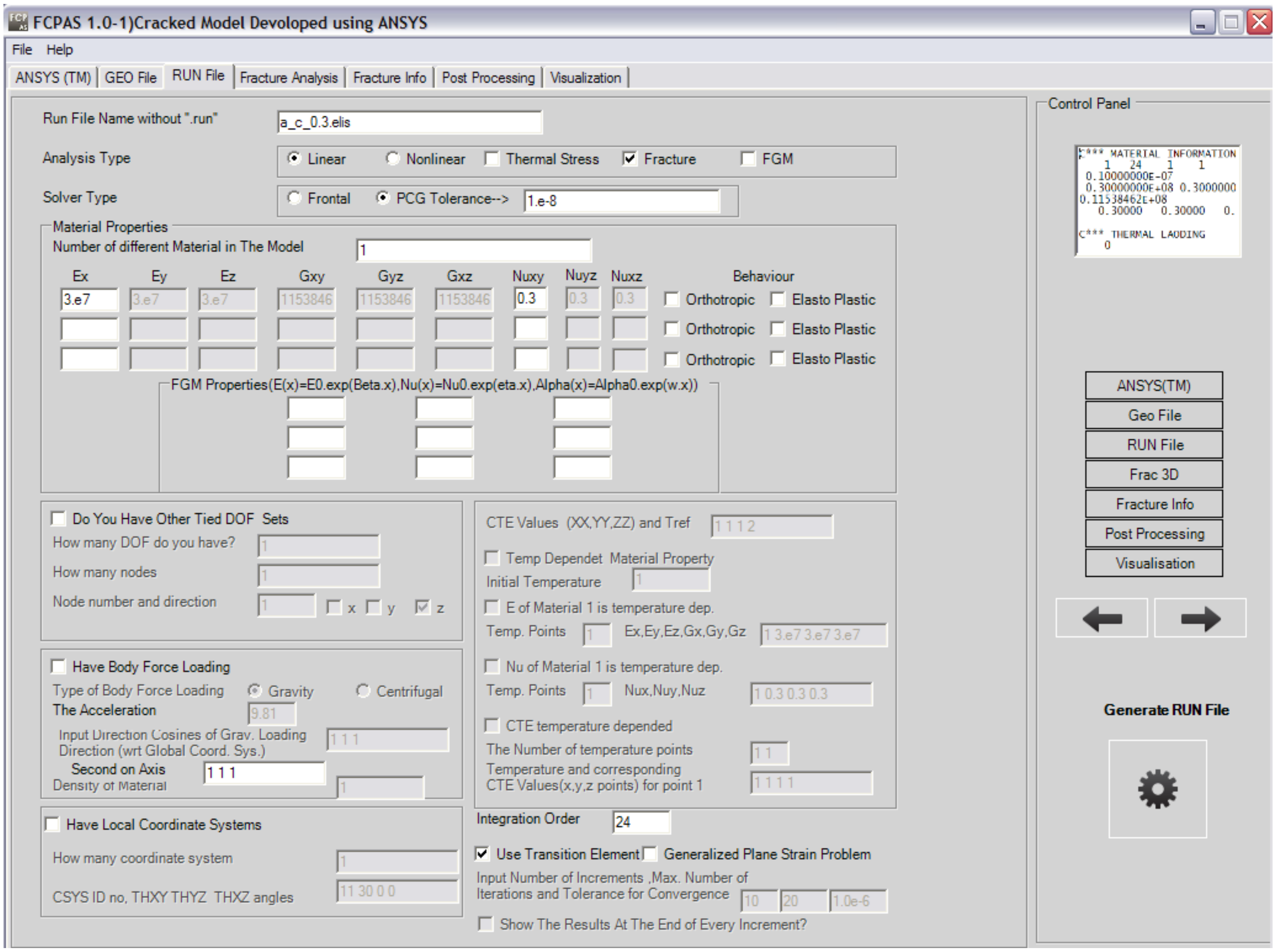
Run *convert_ansys_frac3d.exe.* Using this exe file, we can obtain *cc3.elis_3d.geo* file, which contains element connectivity, nodal coordinates, boundary conditions, loads, and crack information. The following table shows the steps and input for the current problem.



Input file names can be selected by "Browse" buttons. "Generate Geo file" creates *cc3.elis_3d.geo* file. To go to Run File preparation, press "Next Step".

T.3.3.2 Generation of *.run file (*writerun_frac3d.exe*) using FCPAS

Now, we need to create a run file which is also required for FRAC3D. We use *writerun_frac3d.exe* or FCPAS to generate *.run file (*cc3.elis_3d.run* file). The **.run* file contains analysis type, material properties, solver type tolerances, body forces and local coordinate systems data.



To pass "Frac3D" tab, press "Next Step".

T.3.4 Running FRAC3D

T.3.4.1 Using *frac3d.exe*

To run the FRAC3D, three kinds of input files are required;

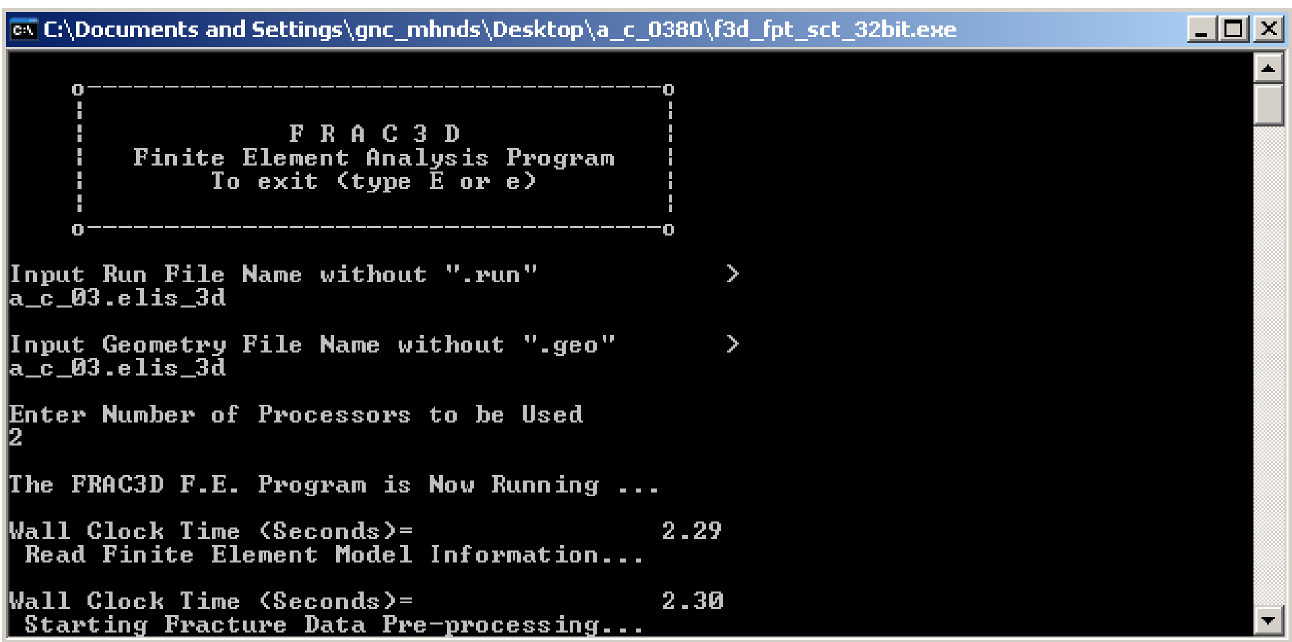
- *.run (compulsory)
- *.geo (compulsory)
- *.tem (optional)

FRAC3D gives the results in the following output files;

- *.out
- *.str
- *.stn
- *.crk

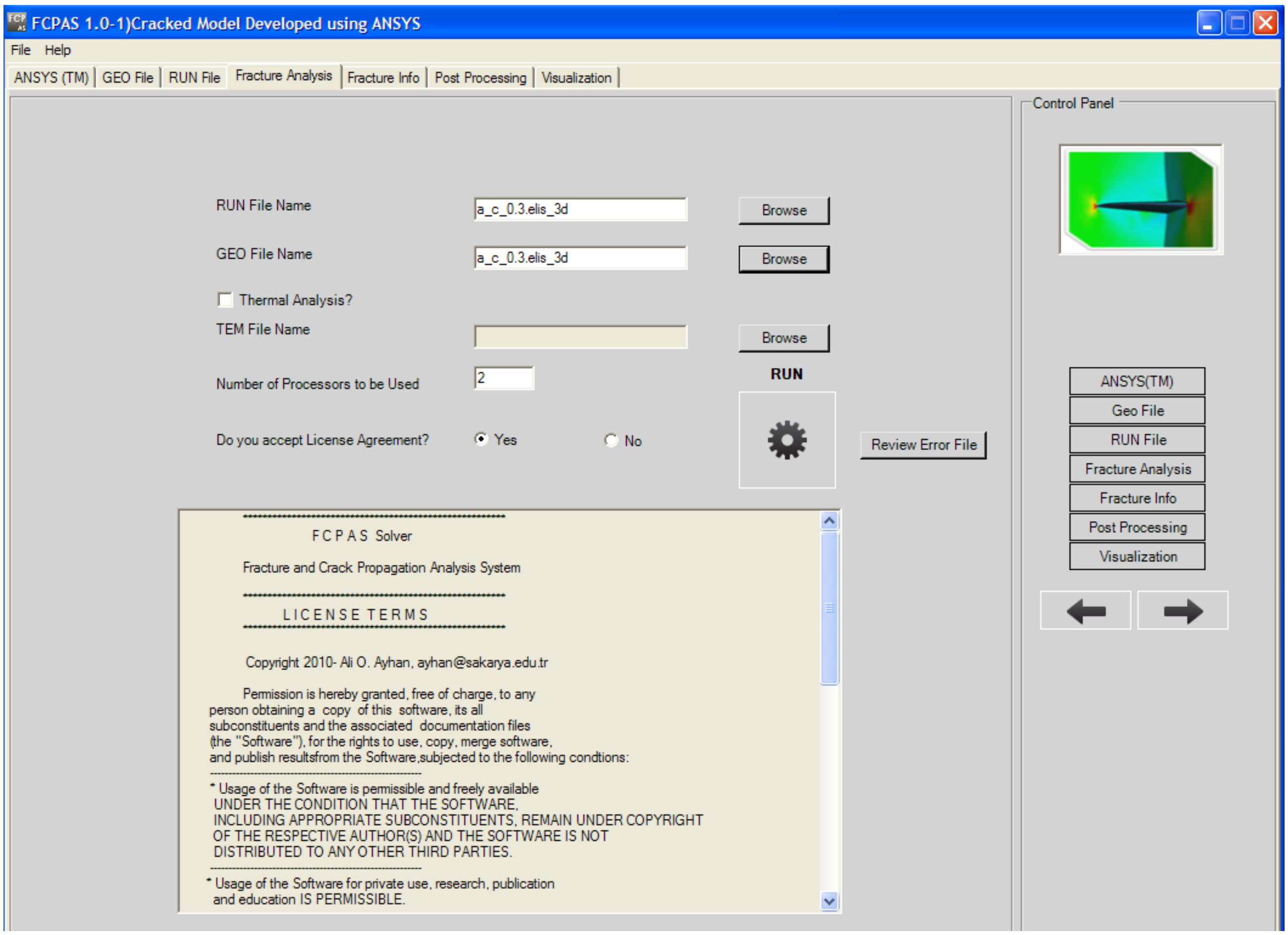
Now, we are ready to run FRAC3D. To do this we can use *frac3d.exe*. When running FRAC3D, geo and run files names have to be entered. The following table shows the steps and input for this specific problem.

Input Run File Name without ".run"	**a_c_0.3.elis_3d**
Input geo File Name without ".geo"	**a_c_0.3.elis_3d**
Input ter File Name without ".ter"	**Hit Enter**

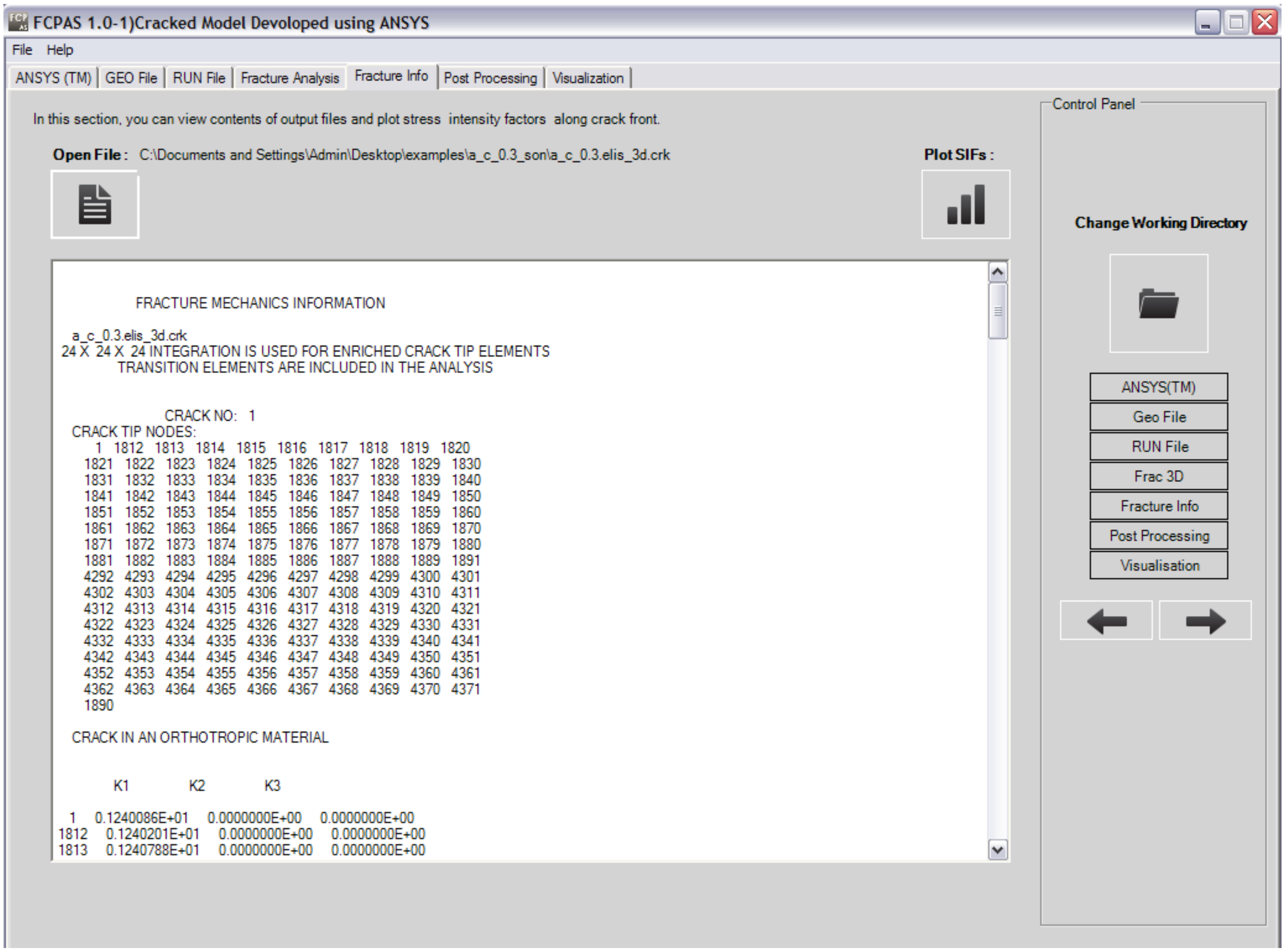


T.3.4.2 Using FCPAS to run FRAC3D

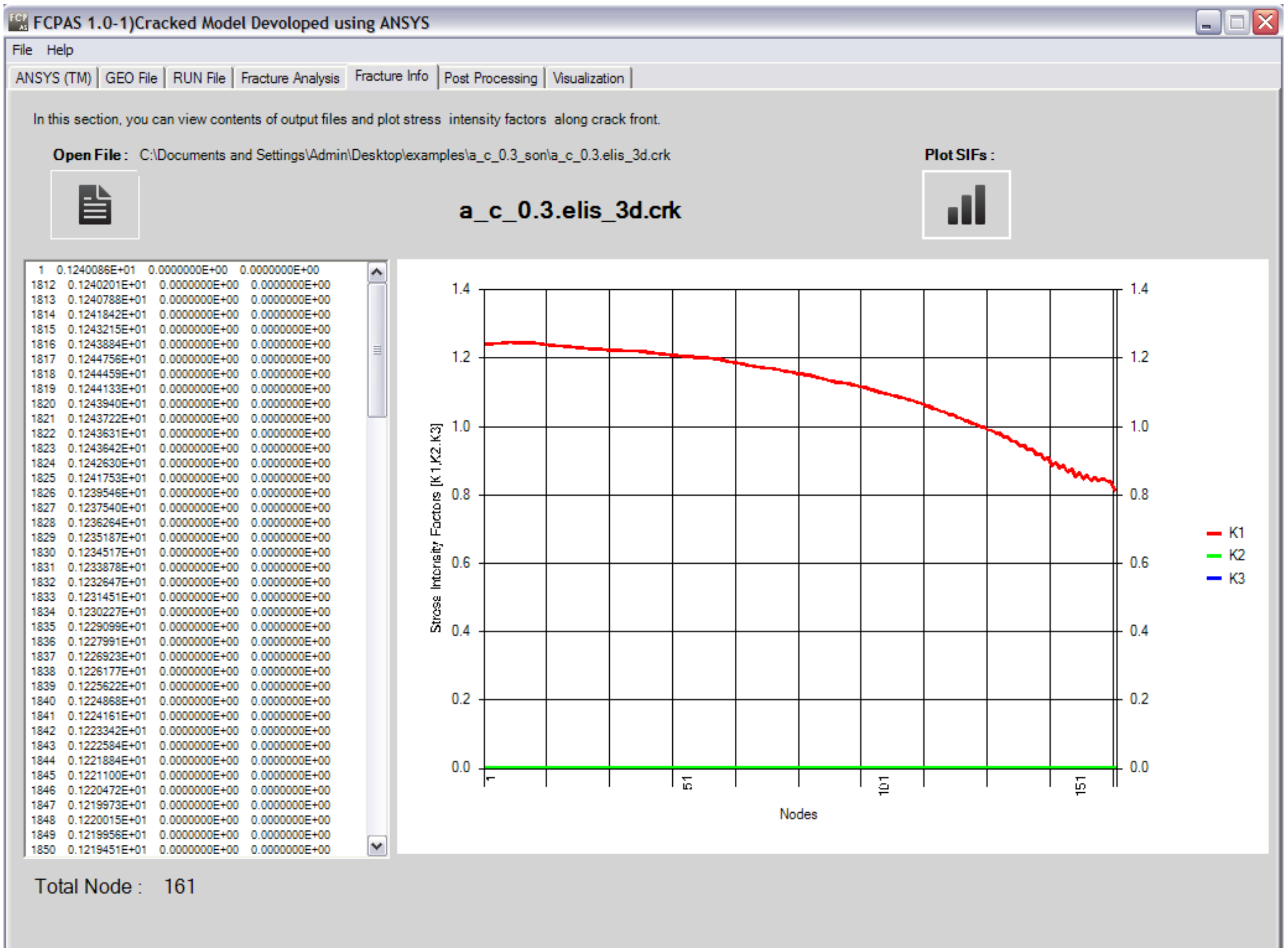
Select the *a_c_0.3.elis_3d.geo*, *a_c_0.3.elis_3d.run*, and *a_c_0.3.elis_3d.tem* (if required) files by browsing and press run button to run the *Frac3D.exe* in the background.



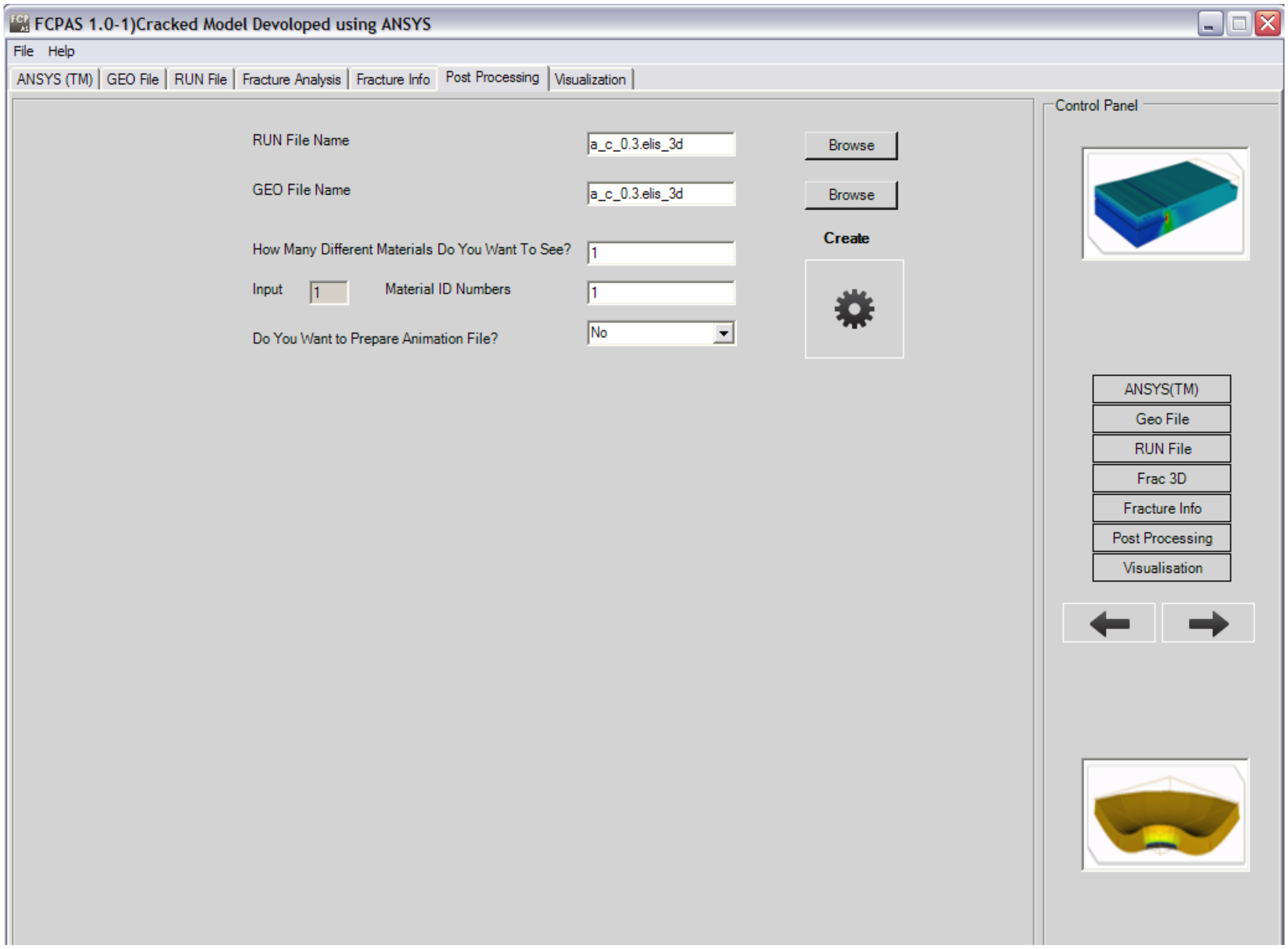
After FRAC 3D run ends, output files can be viewed in the “Fracture Info” tab. In the “Fracture Info” tab, you can browse any file to see its content and plot the K_1, K_2 and K_3 data in an x-y plot.



To plot the K_1, K_2 and K_3 data, just press “Plot SIF’s” button.

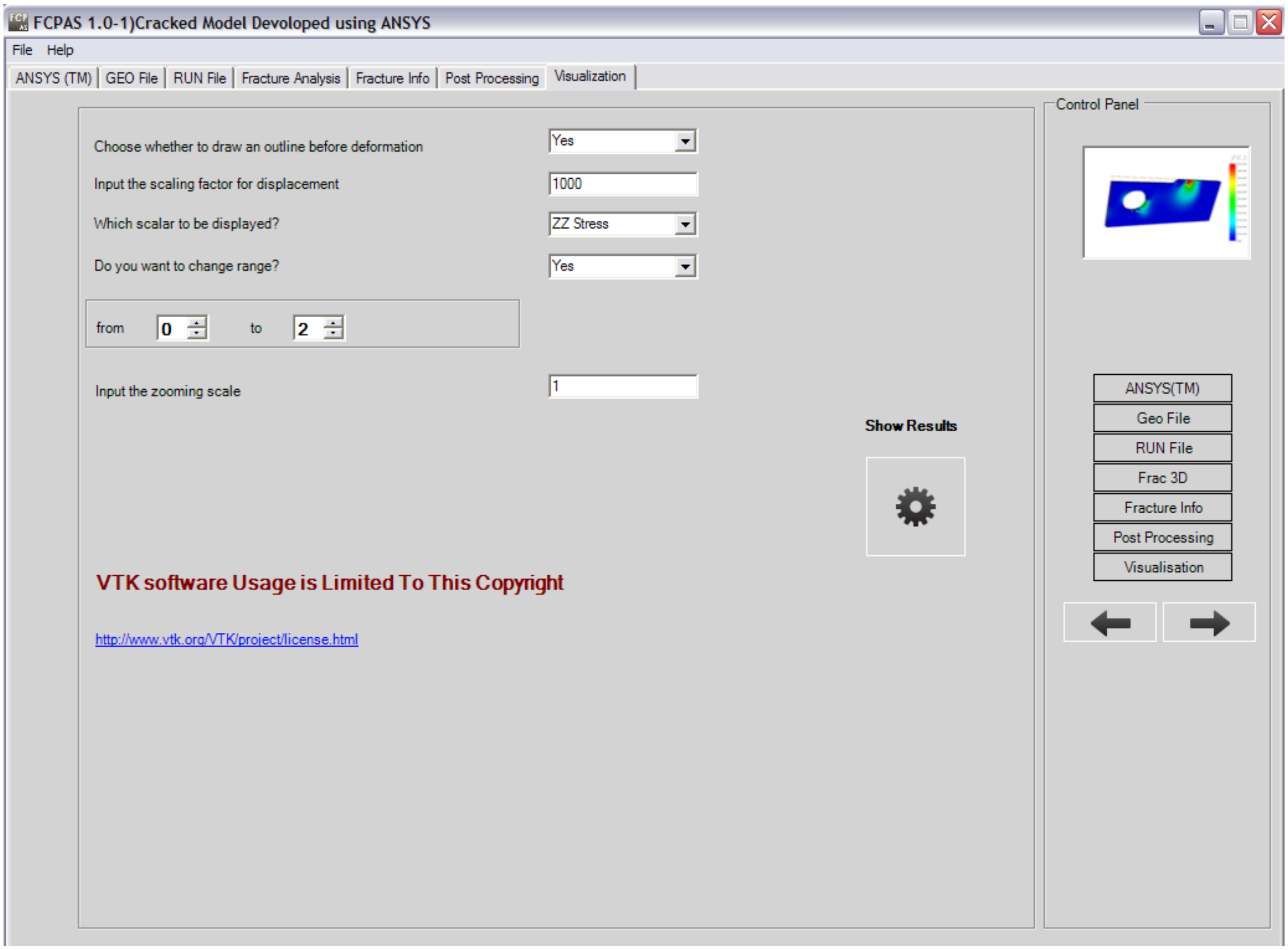


T.3.4.2 Post-processing of FRAC3D Results

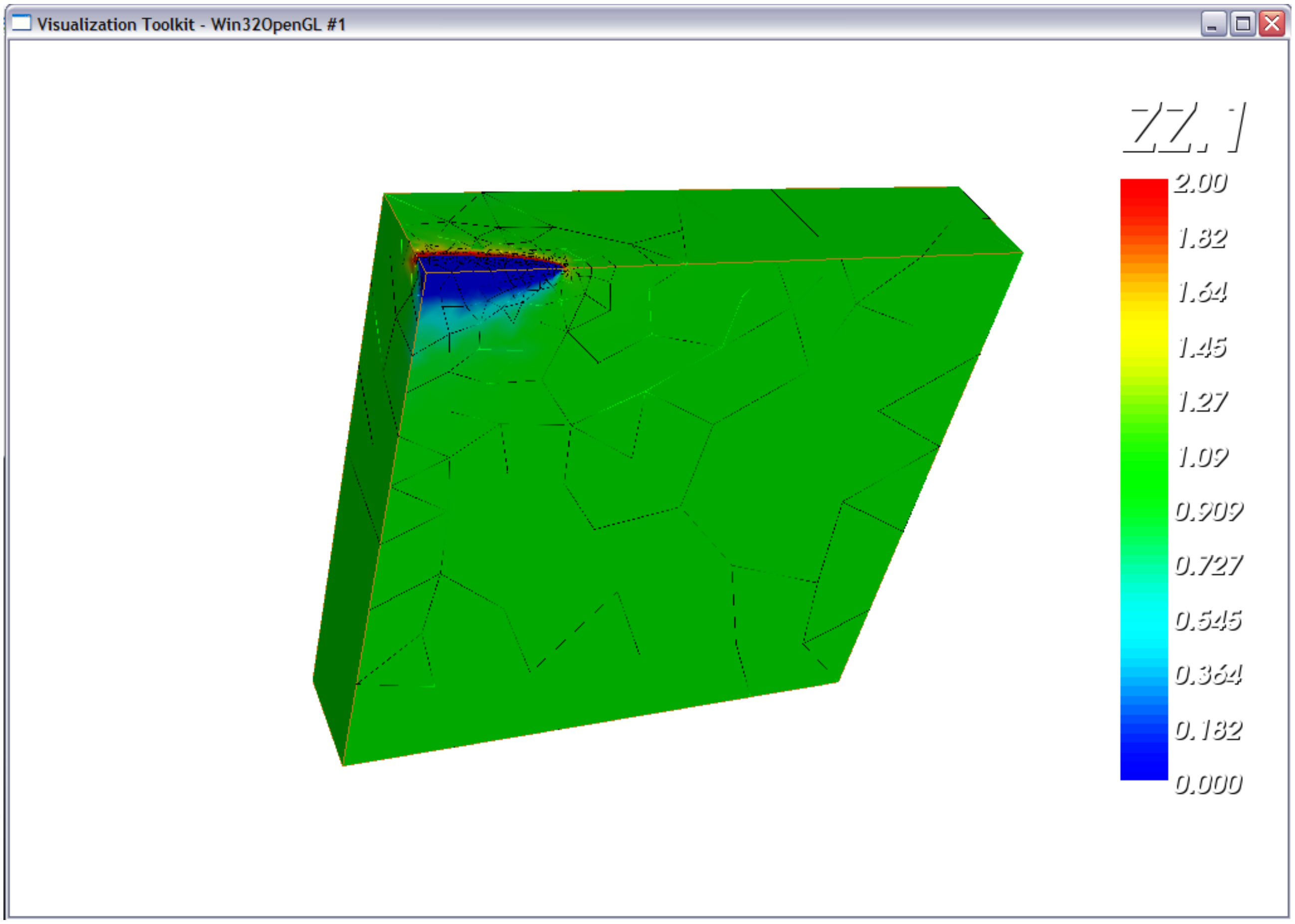


T.3.5 Visualization of FRAC3D Results

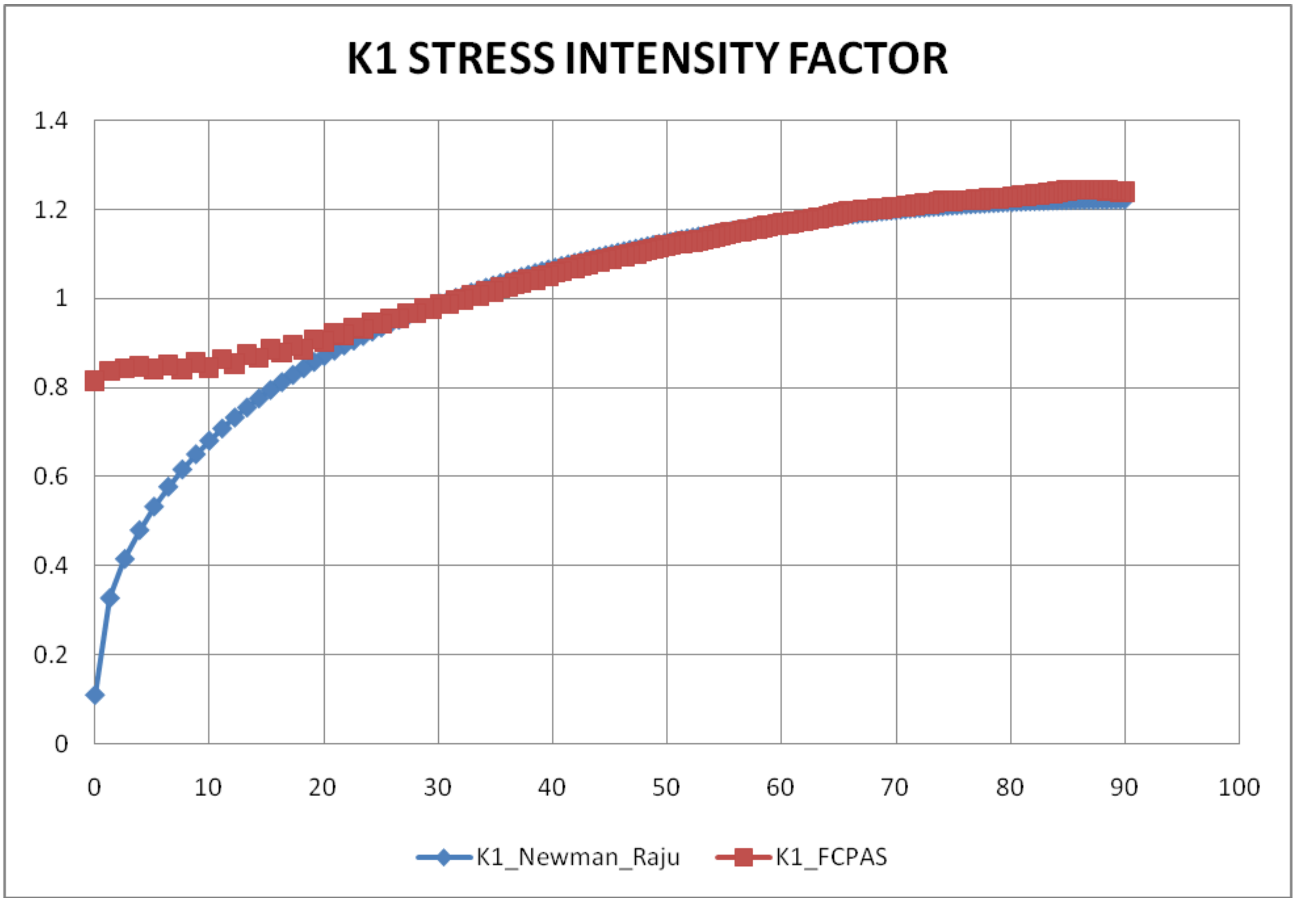
To see the Cracked Model results, choose the parameter you would like to contour plot and press "Show Results" button.



FCPAS 1.0-1)Cracked Model Devoloped using ANSYS
File
Help
ANSYS (TM)
GEO File
RUN File
Fracture Analysis
Fracture Info
Post Processing
Visualization
Choose whether to draw an outline before deformation
Yes
Input the scaling factor for displacement
1000
Which scalar to be displayed?
ZZ Stress
Do you want to change range?
Yes
from
0
to
2
Input the zooming scale
1
Show Results
VTK software Usage is Limited To This Copyright
http://www.vtk.org/VTK/project/license.html
Control Panel
ANSYS(TM)
Geo File
RUN File
Frac 3D
Fracture Info
Post Processing
Visualisation

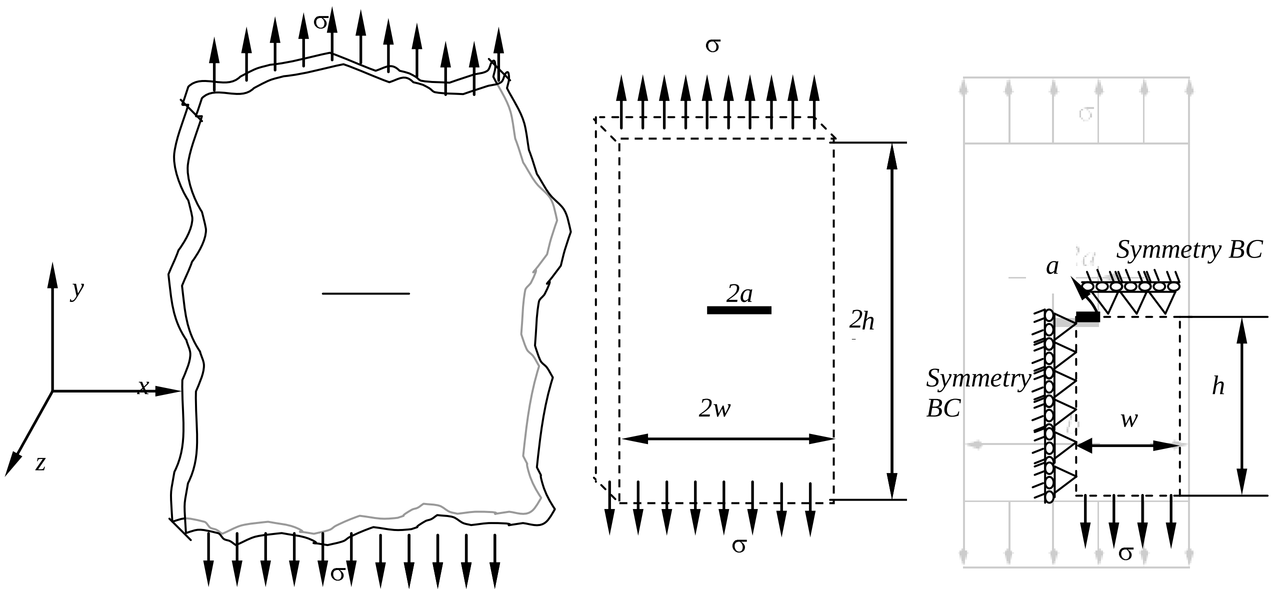

Visualization Toolkit - Win32OpenGL #1
ZZ. 1
2.00
1.82
1.64
1.45
1.27
1.09
0.909
0.727
0.545
0.364
0.182
0.000

K1 Comparison of Stress Intensity Factor: FCPAS and Newman_Raju

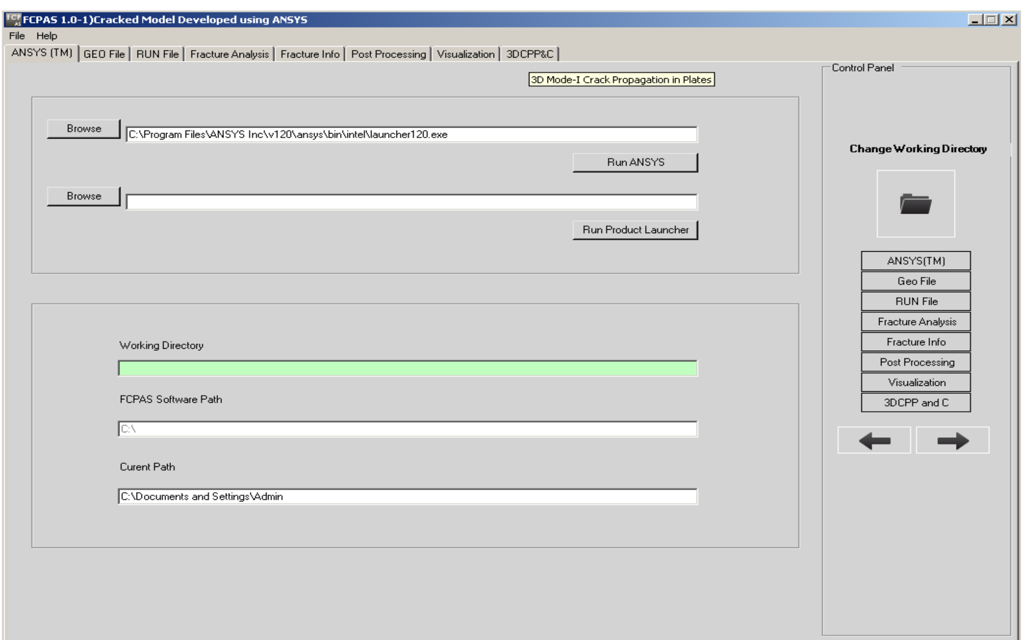


EXAMPLE.4.Crack Growth in Plate Using ANSYS Macro for Mode I

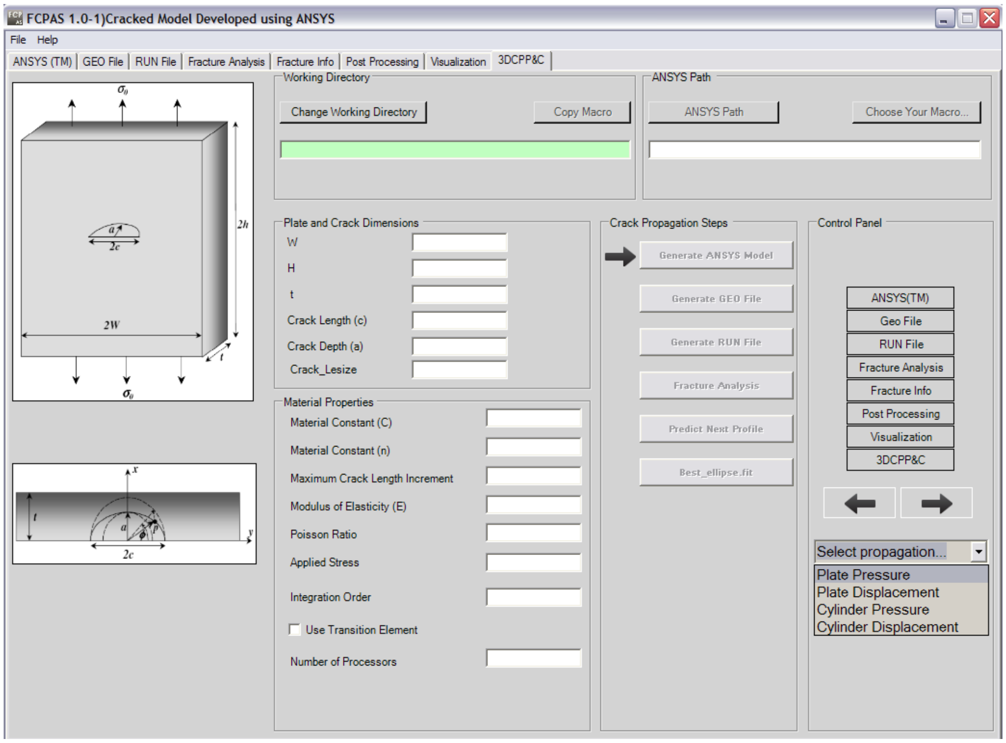
In this example the aim is to get crack growth profiles by using written macros. A three-dimensional elliptical surface crack in a finite-thickness plate under uniform tension with 2H x 2W x t (height x width x thickness). Plate dimensions are width: 0.175m, height: 0.295m, thickness: 0.03m and also initial crack dimensional are crack length (c): 0.0196m , crack depth (a): 0.0144m.



First we open FCPAS Cracked Model Development with ANSYS after click 3DCPP&C

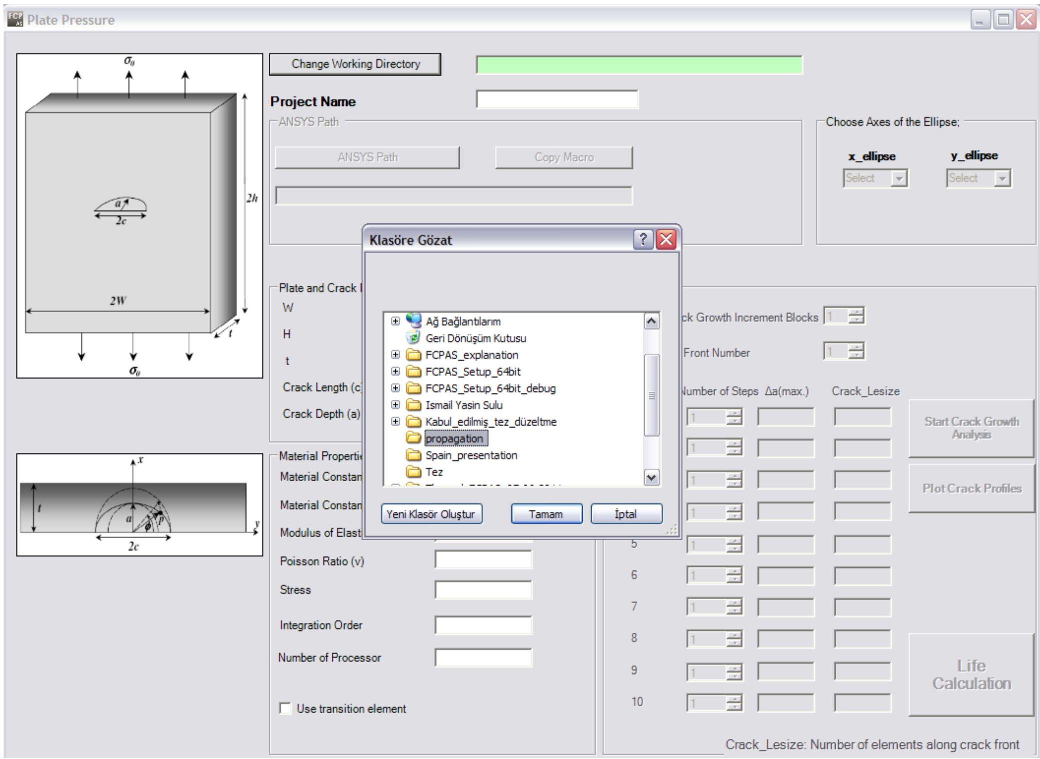


After that we can *'Select Propagation'*



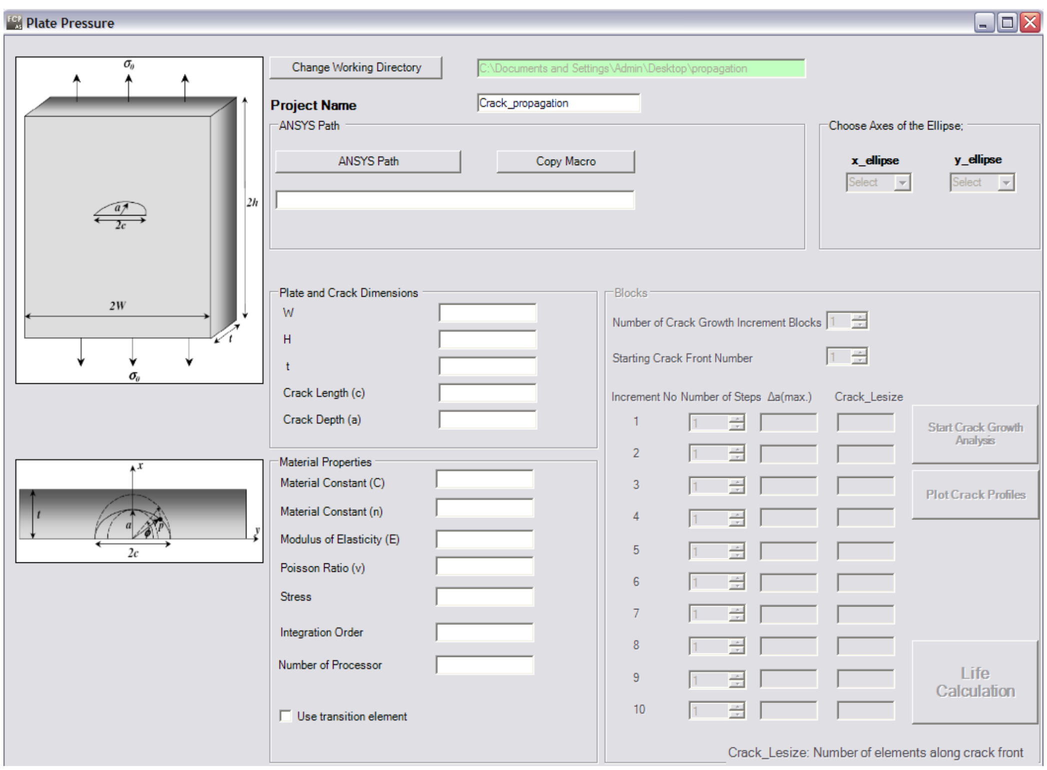
Change Working Directory

Before starting to crack growth analysis create a folder in which you would like to work&change directory to this folder.

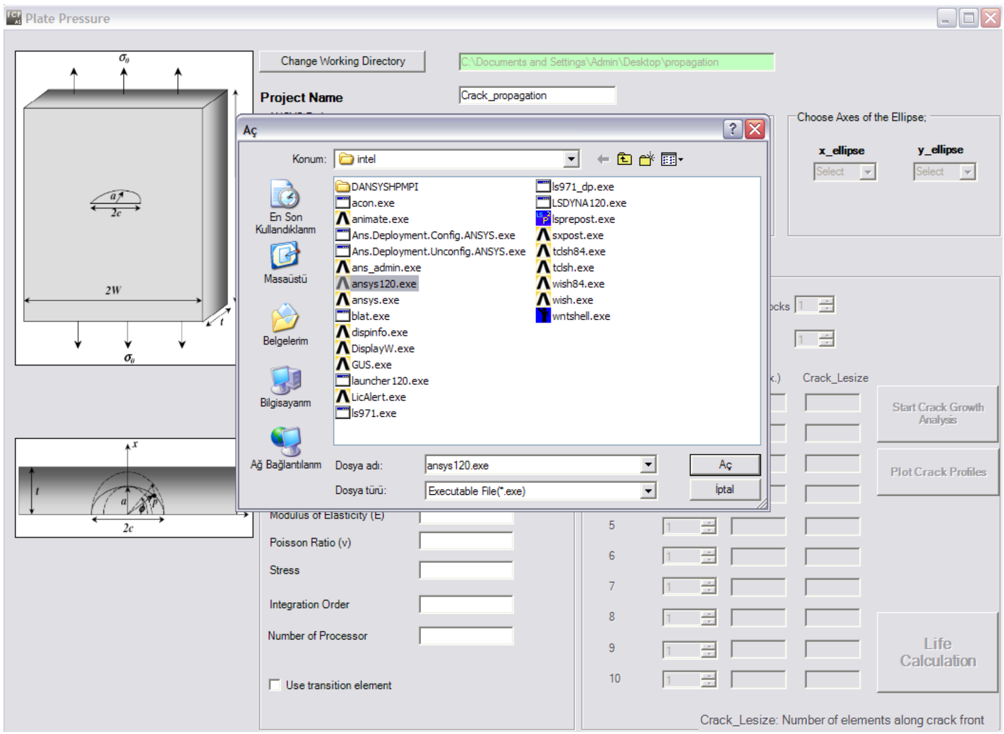


Give the Crack Propagation Project Name

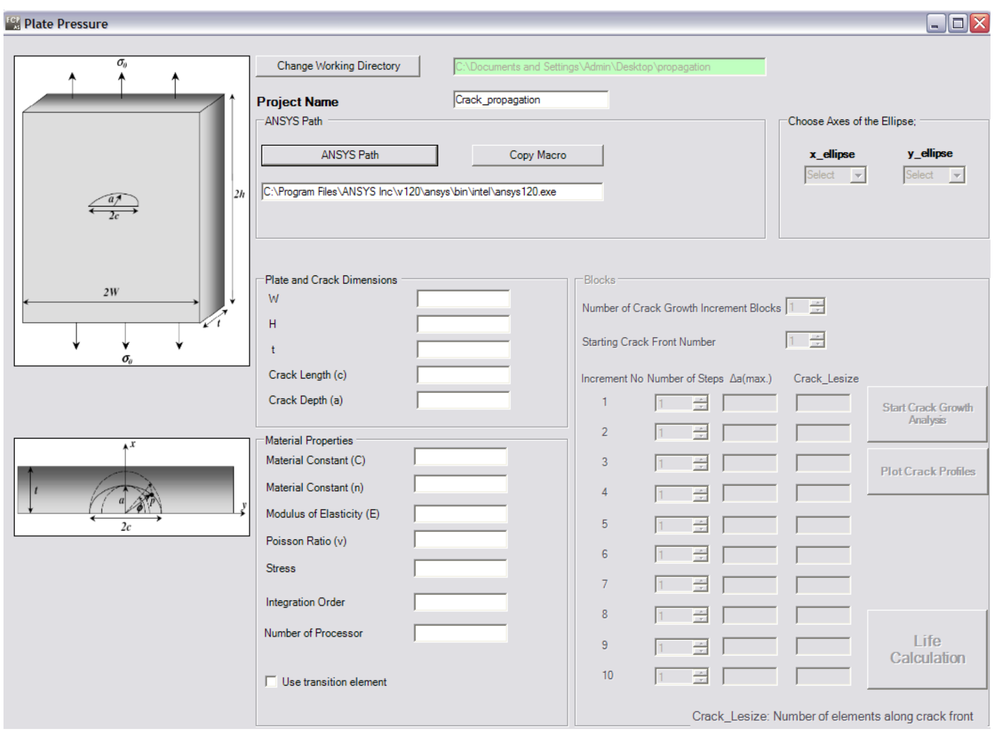
Enter a name, for example *'Crack_propagation'*



Choose ‘*Ansys Path*’, in this example '*ansys121.exe*' should be chosen.

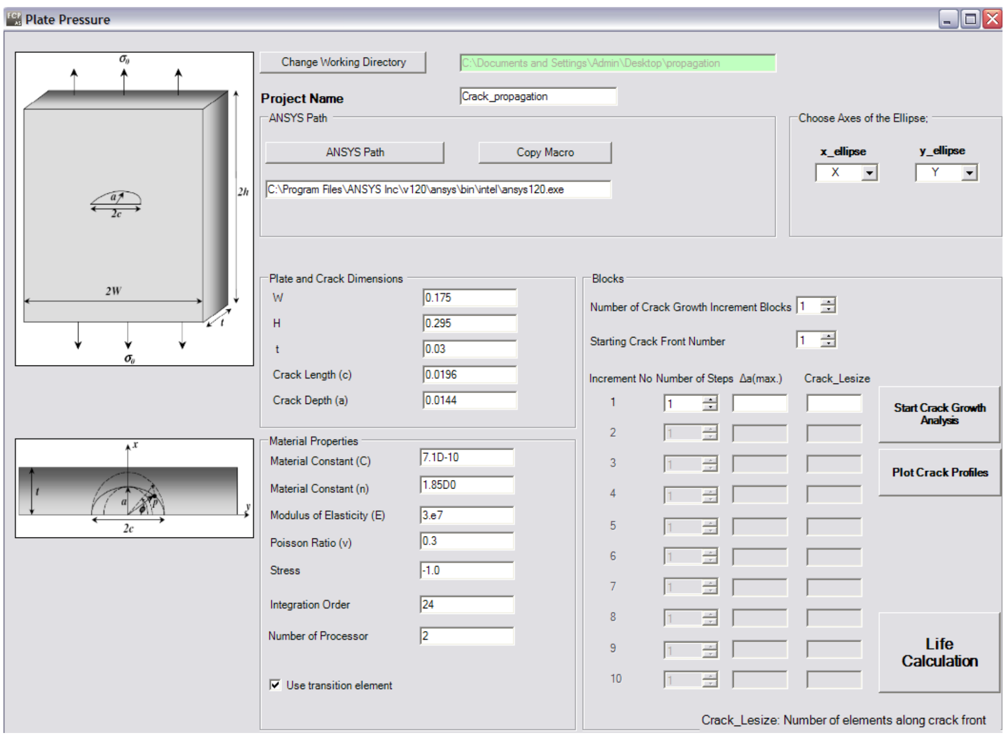


In the following picture is shown that, before picking ‘*Copy Macro*’ button, the scheme has got project name and selected working directory.

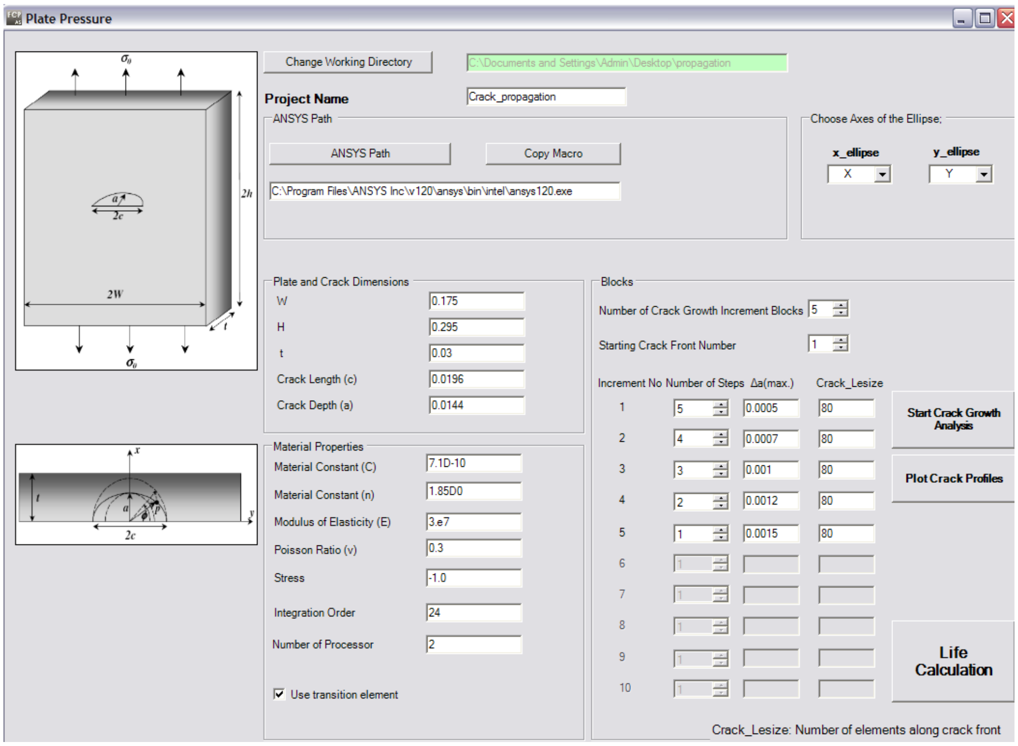


When we select copy macro, specific prepared macro is copied from bin directory to working direction folder. At the same time, *'Choose Areas of the Ellipse'* is enabled.

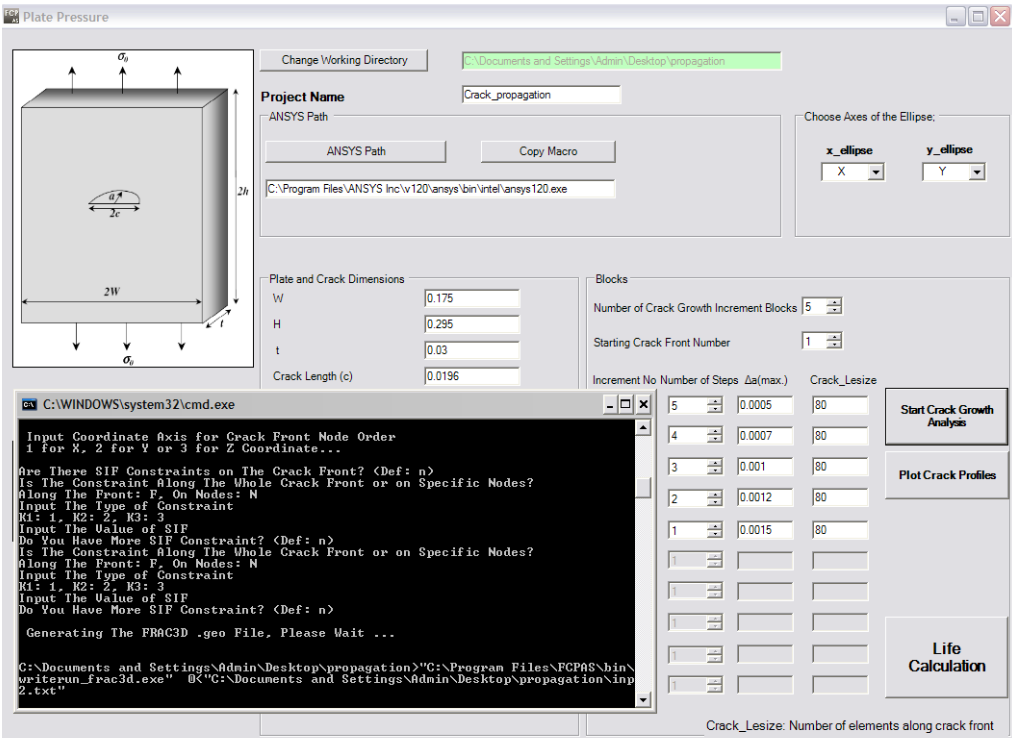
Then *'Plate And Crack Dimensions'* and *'Material Properties''* are written. If you use transition element we check *'Transition Element'* button.



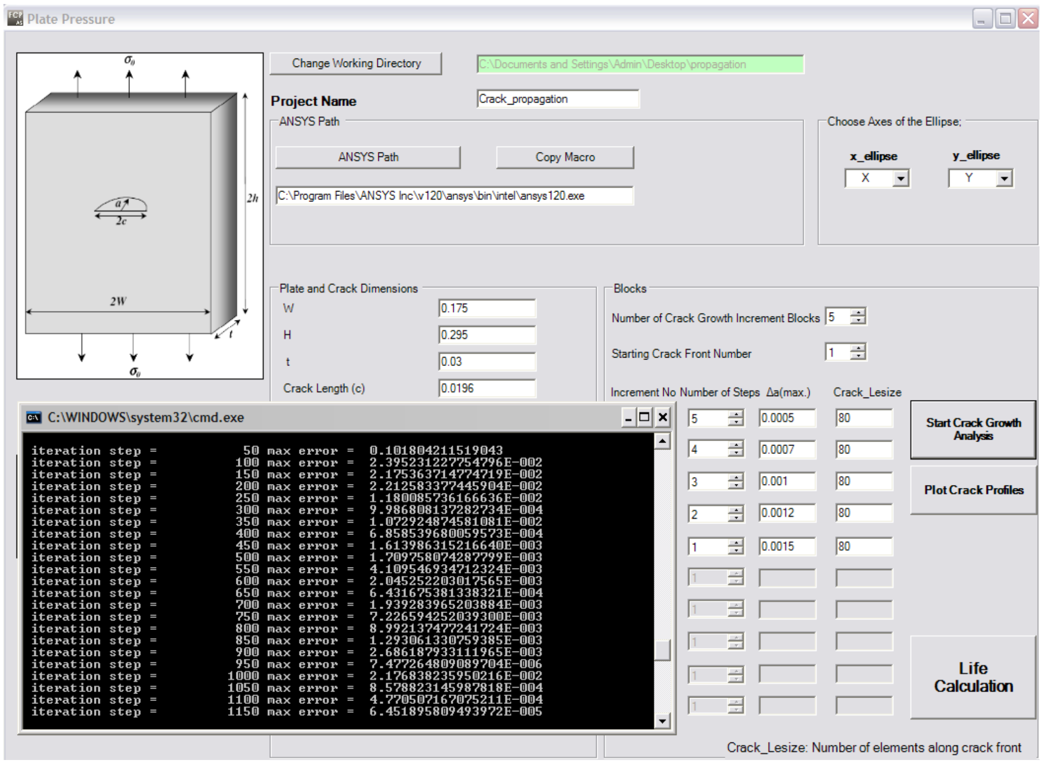
Terms of block is related to number of difference maximum crack advancement distance in a step along the crack front.($\Delta a_{\max}$)If you would like to get more crack profiles, you can also go on by changing *'Start Crack Front Number'* .Increment number is equal number of crack growth increment blocks. We write number of step is each value of ($\Delta a_{\max}$) to obtain order again. At that time '*Crack Lesize'* number of elements along crack front can be changed.



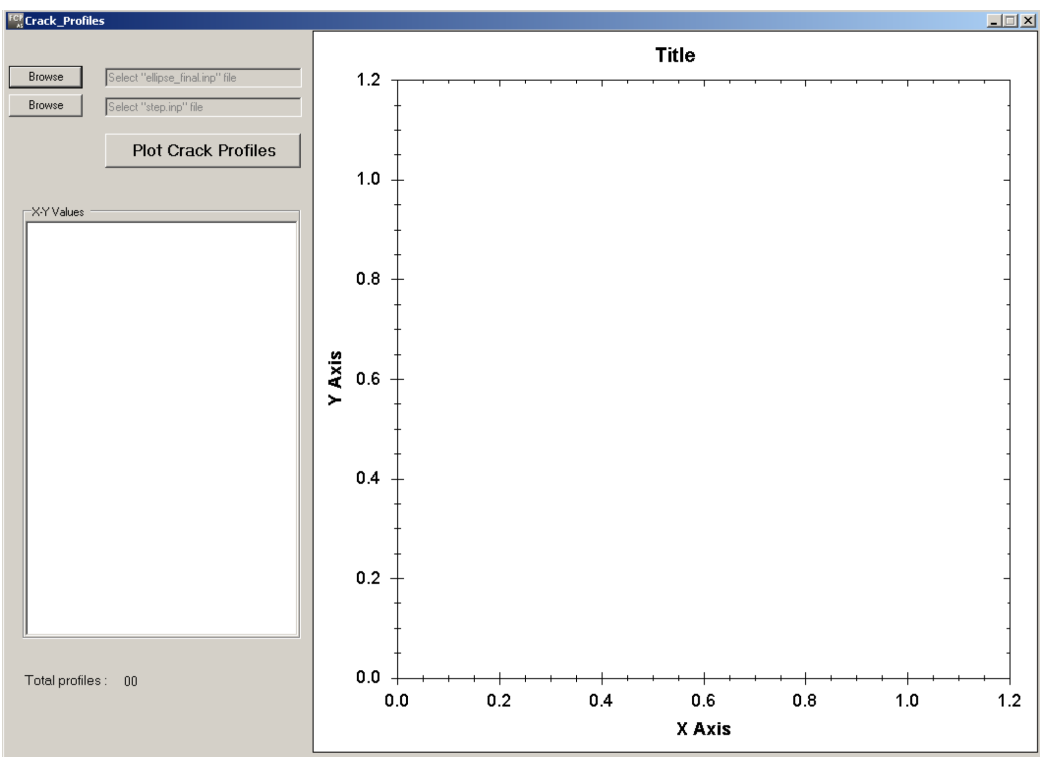
For the solve, click to *'Start Crack Growth Analysis'.*



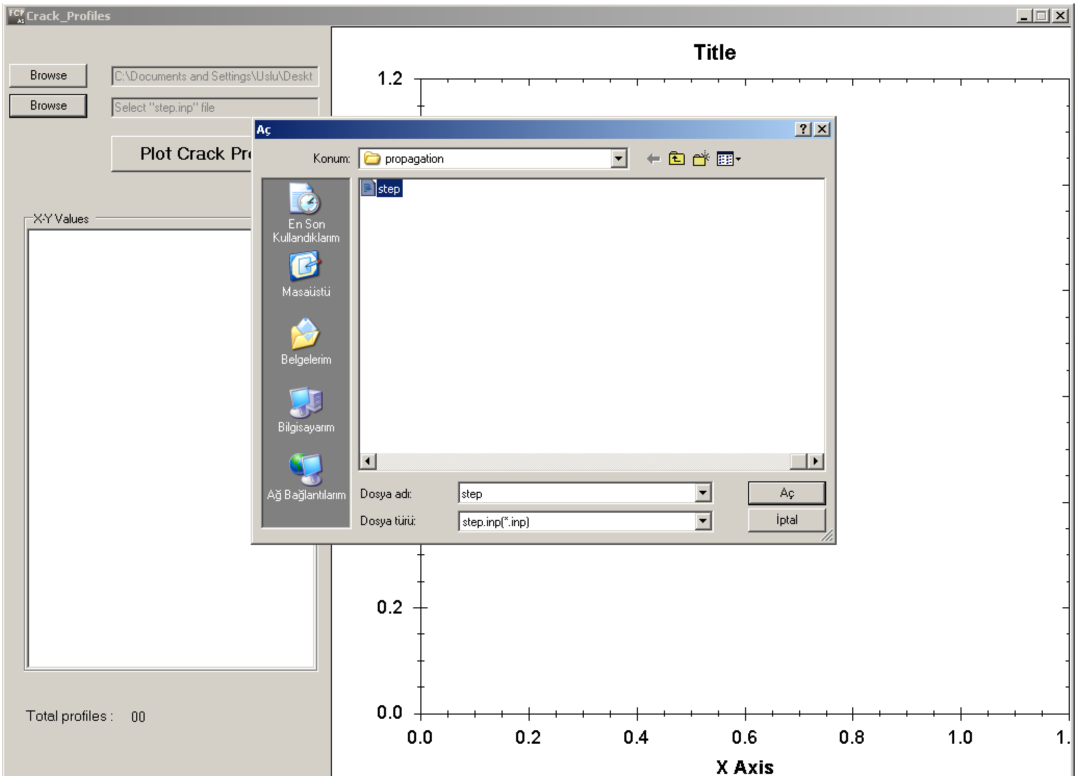
'Frac3d Solver'

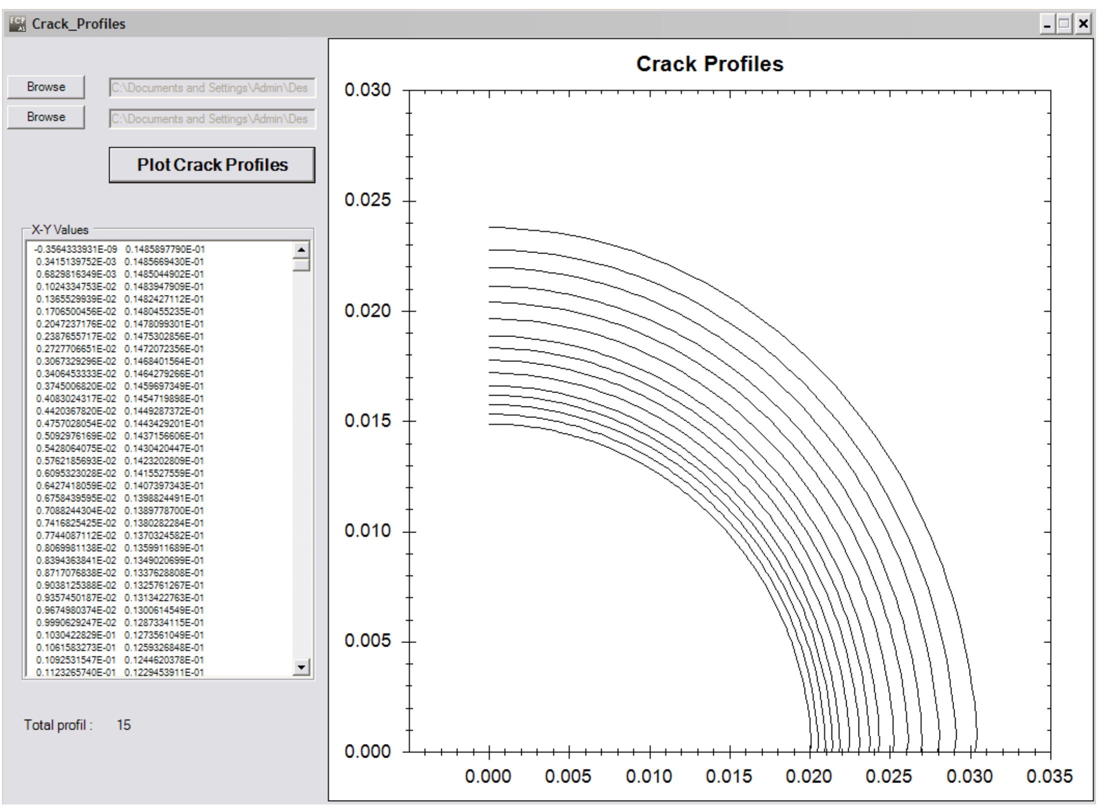


After the crack growth analysis solution you can plot crack profiles click *'Plot Crack Profiles'* button.



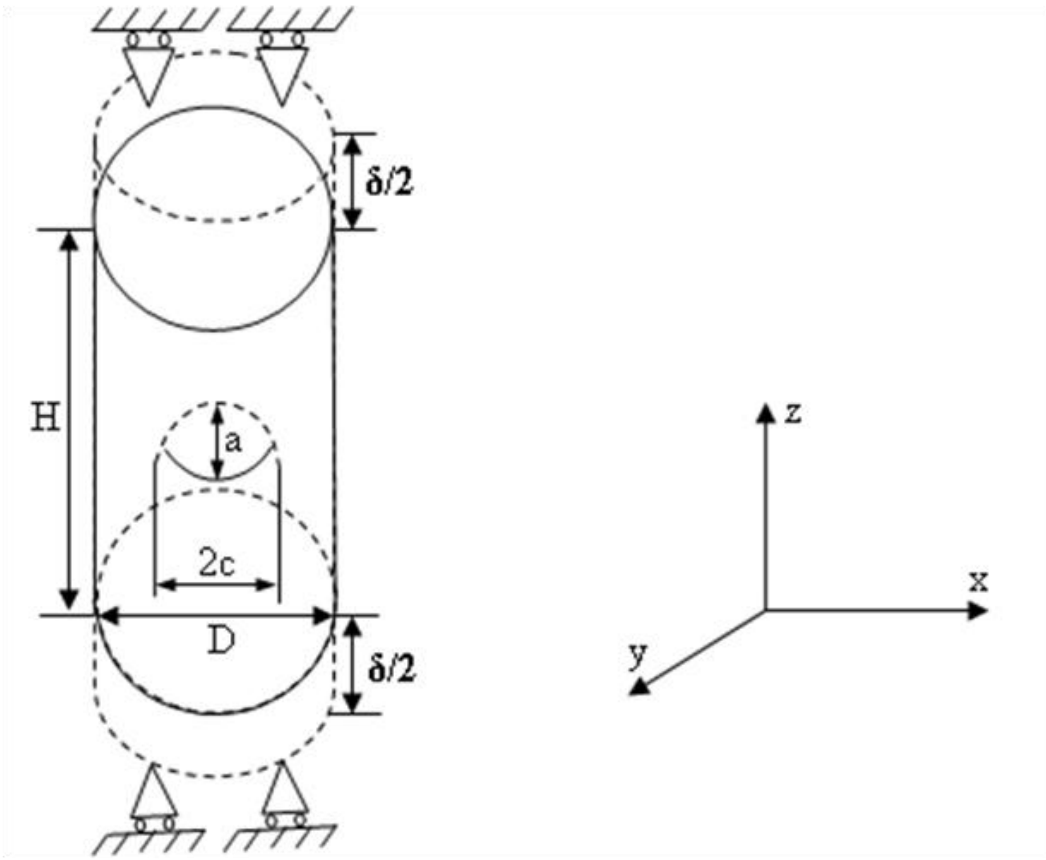
First we browse *'ellipse_final.inp'* after select *'step.inp'* to plot crack propagation profiles.





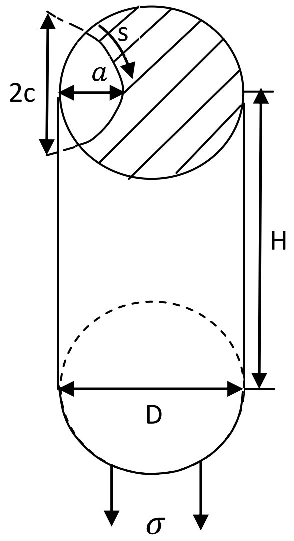
EXAMPLE.5. Crack Growth in Cylinder Using ANSYS Macro for Uniform Displacement Load (a/D=0.1, a/c=0.2)

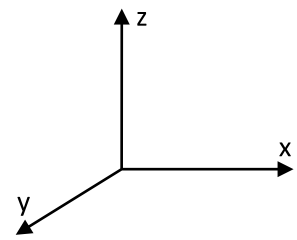
In this example, the aim is to get crack growth profiles by using written macros. A three-dimensional elliptical surface crack in a cylinder under uniform displacement load with D x H dimensions. Cylinder dimensions are D=1, H=5, and also initial crack dimensions are crack length c=0.5, crack depth a=0.1 unit.



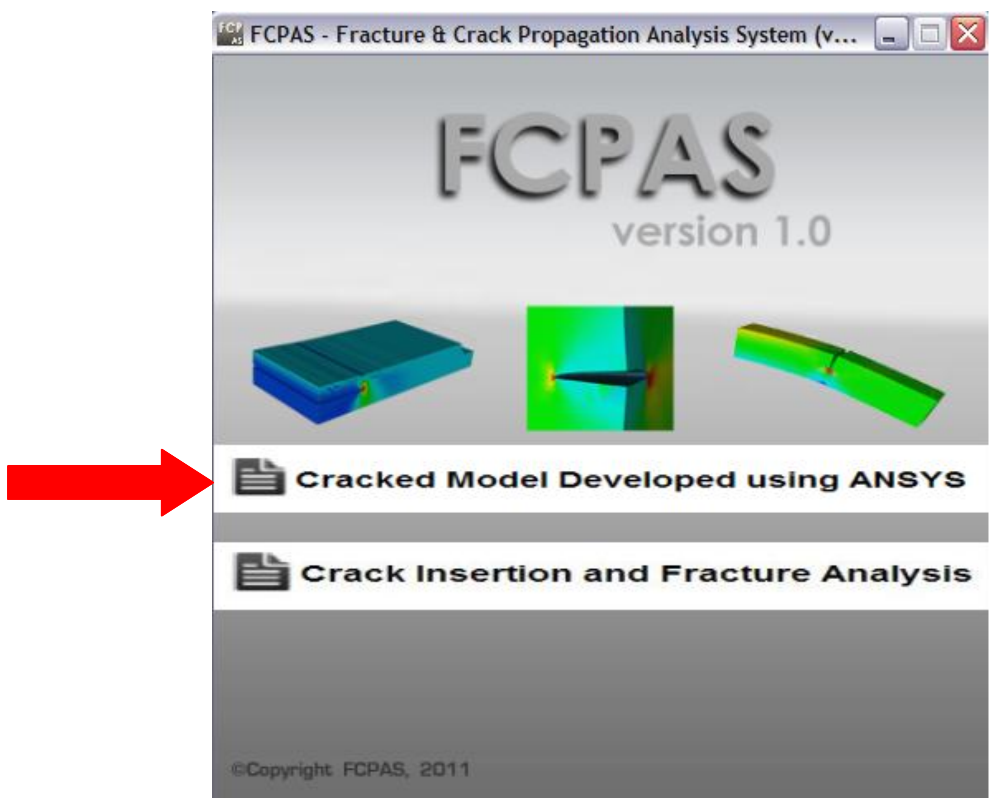
D=1
H=5
a=0.1
c=0.5
Δa_{max}=0.1, 0.2, 0.3, 0.4, 0.5

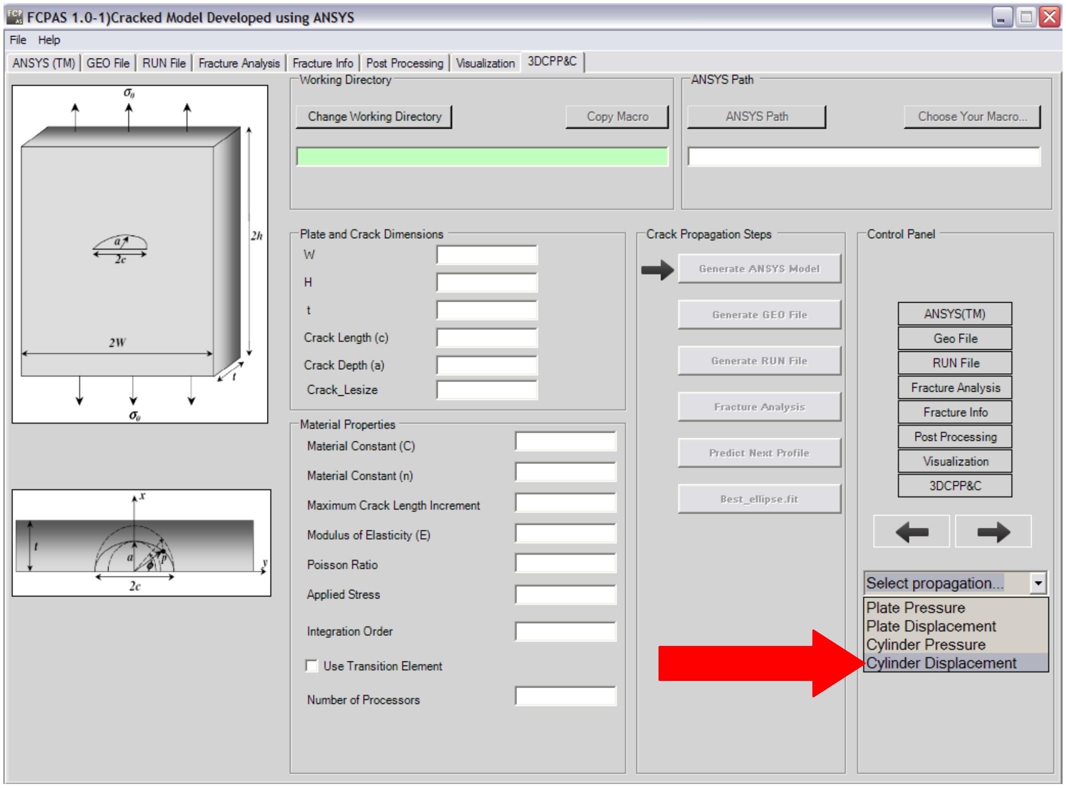
We use symmetry of the fracture model.



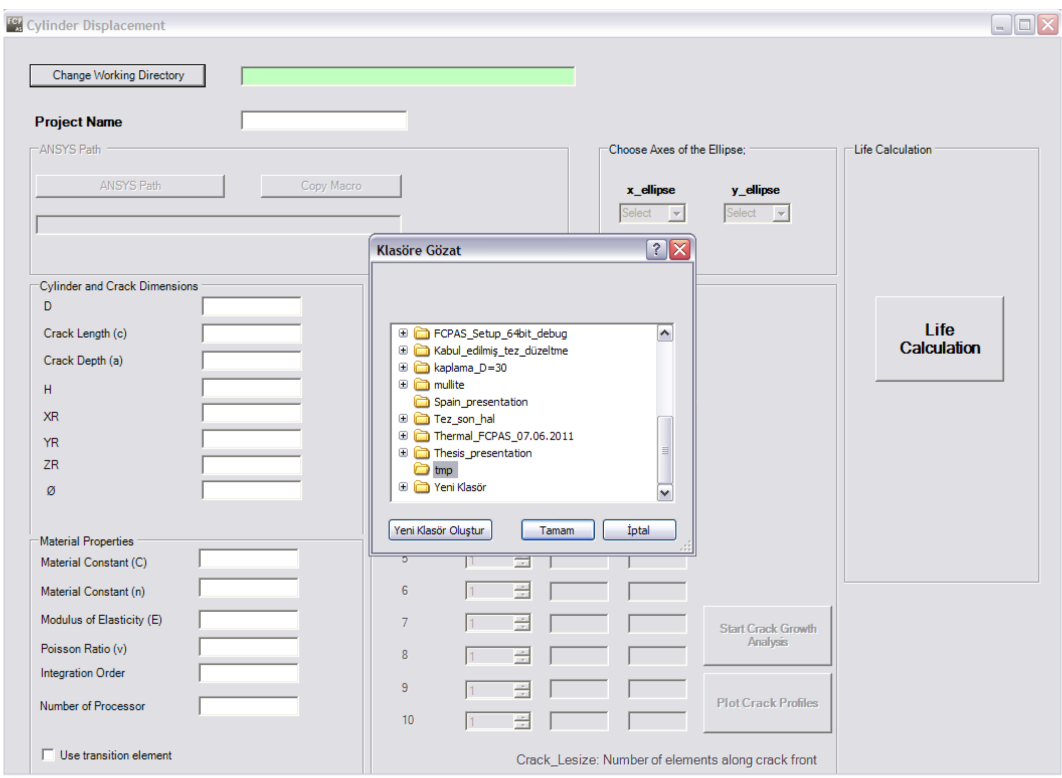


First, we select "Cracked Model Developed using ANSYS" button.

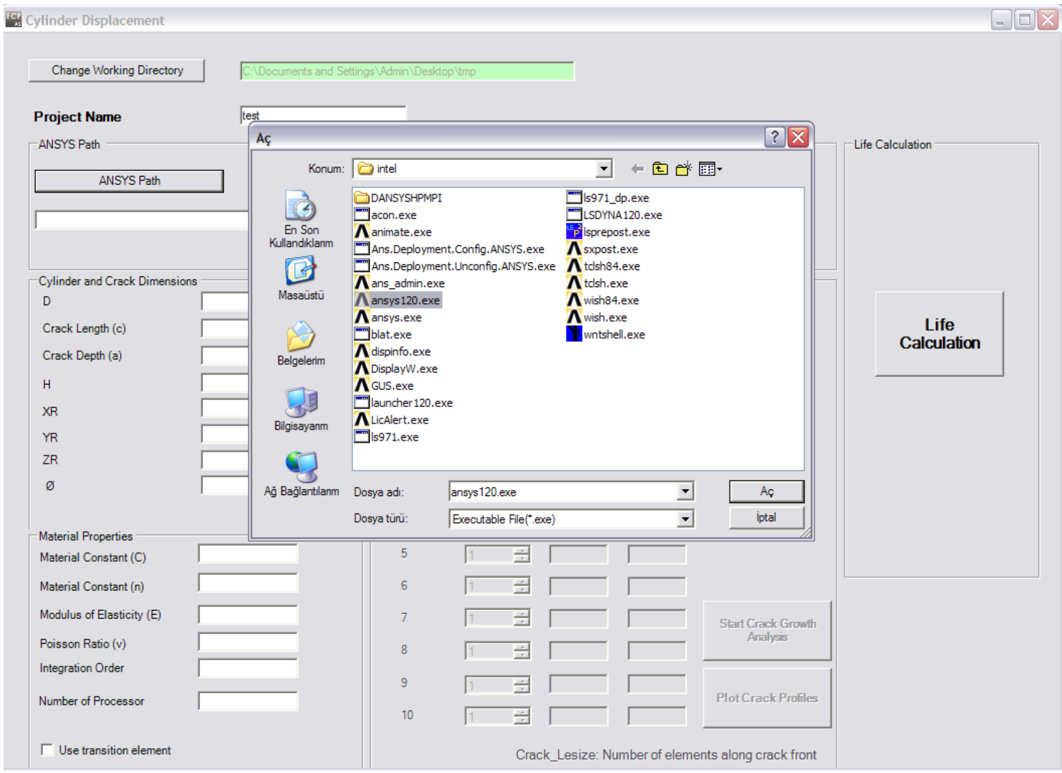


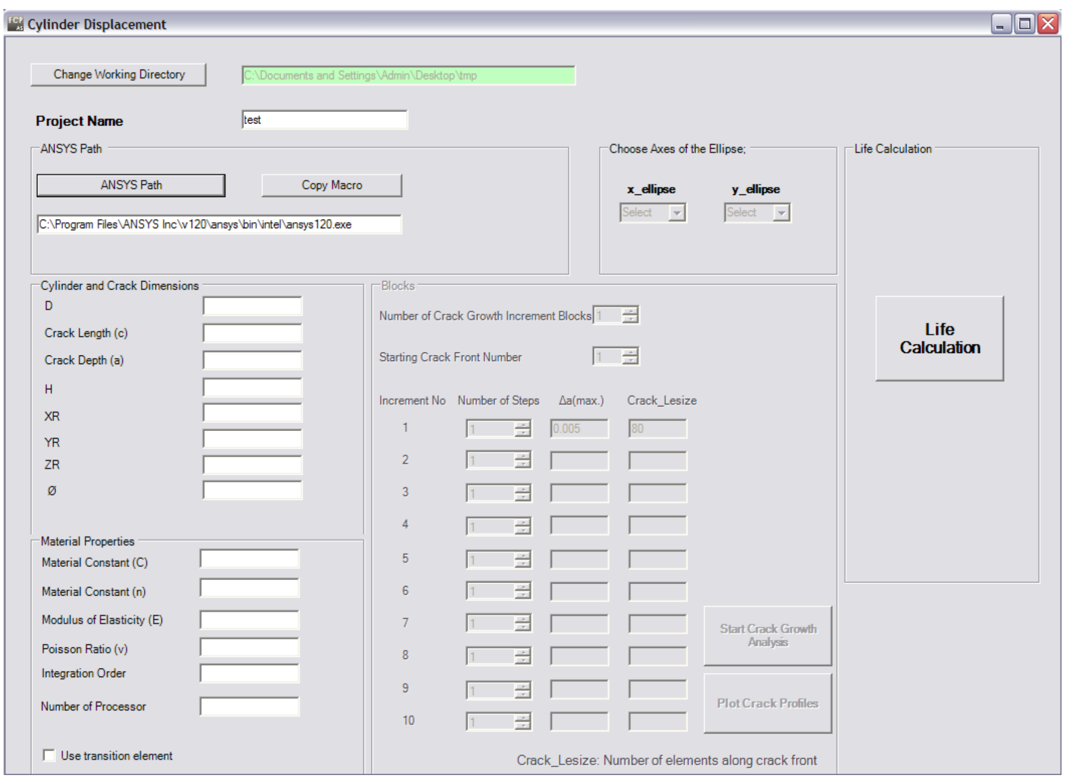


Change working directory and write “Project Name”.

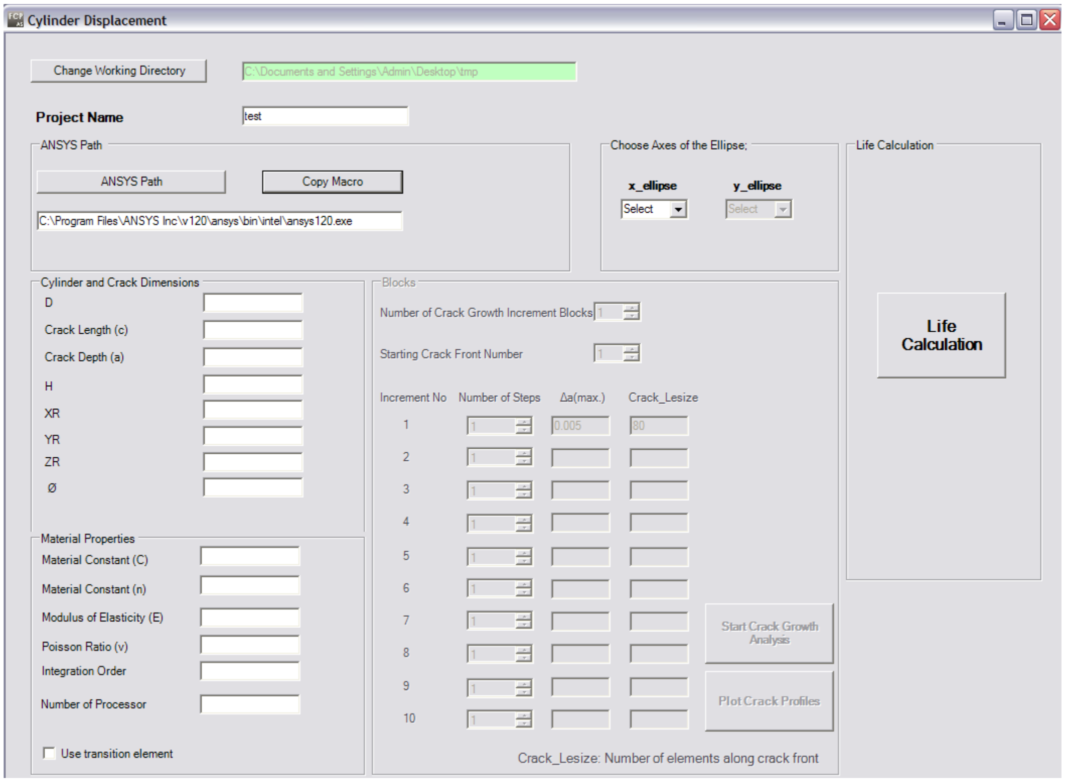


After we write "Project Name", we select ANSYS^{TM} path using "ANSYS PATH" button.

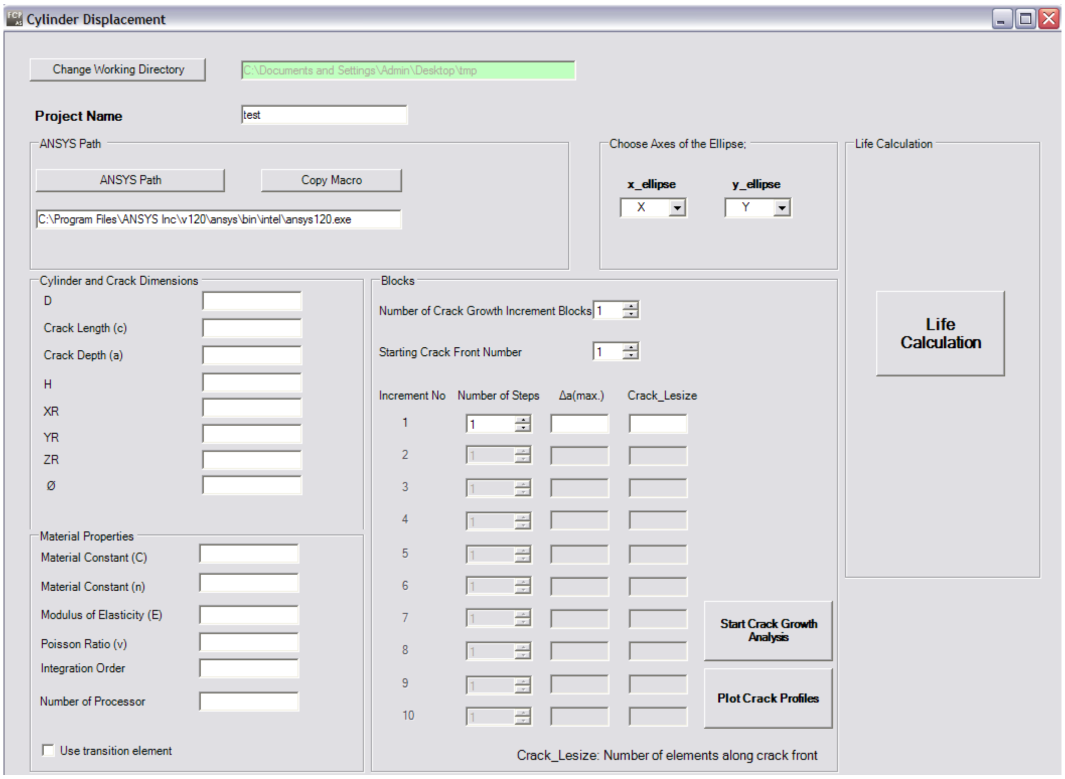




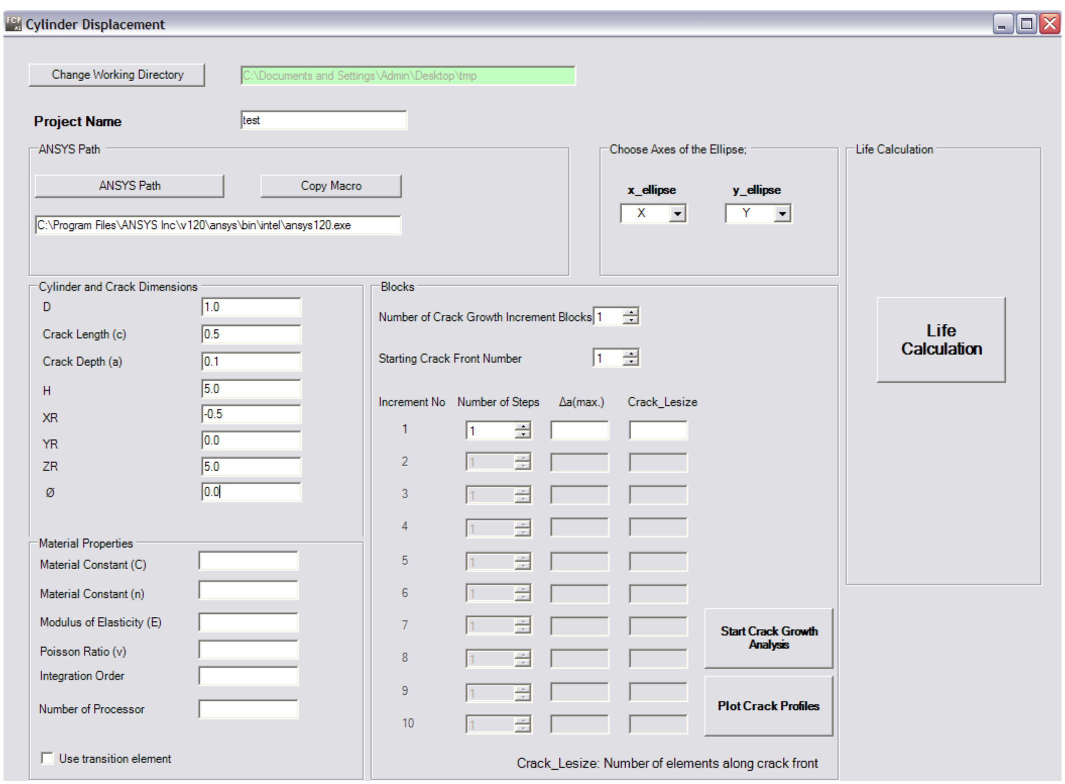
When we select "ANSYS Path", "Copy Macro" button is enabled. We click "Copy Macro" button and copy cylinder displacement macro into the working directory. At the same time, when we click "Copy Macro", "Choose x_ellipse" is enabled.



In this example, we select X and Y axes for ellipse fitting.

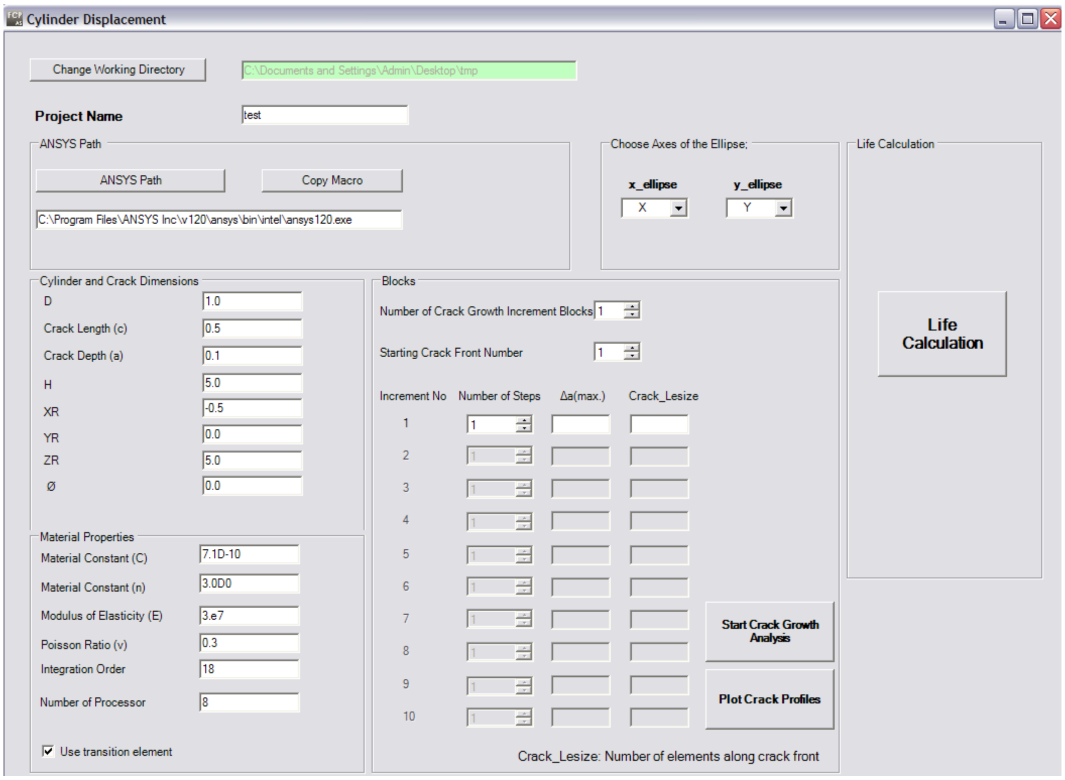


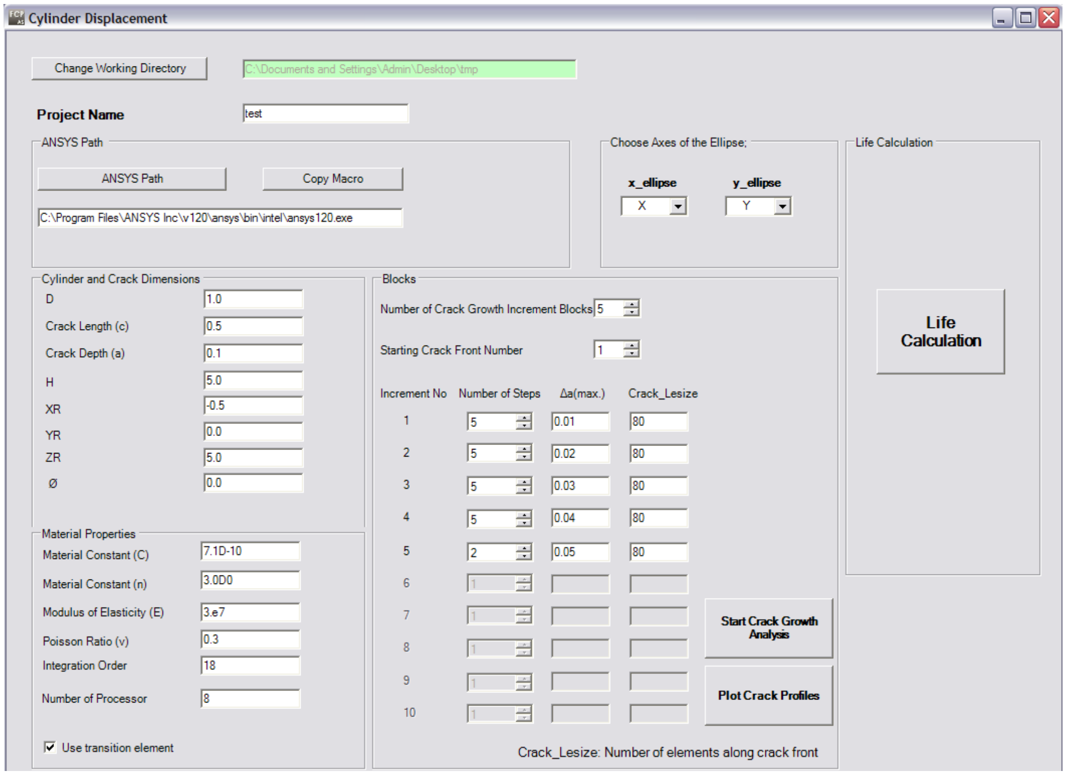
We write cylinder and initial crack dimensions.



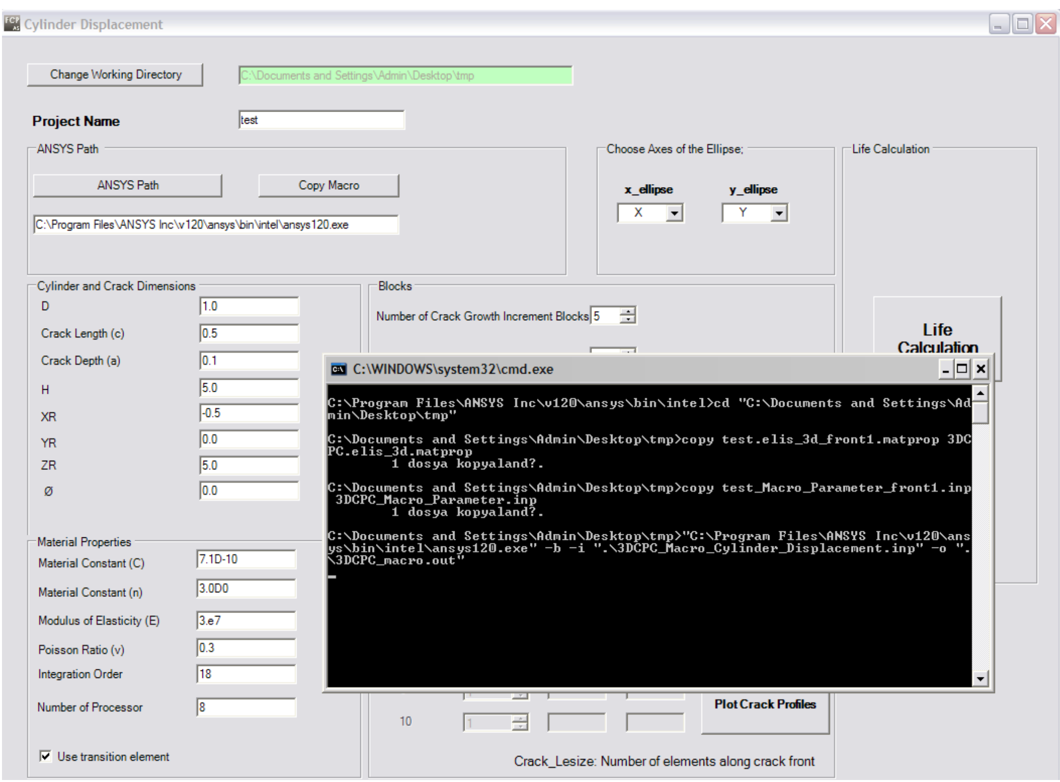
XR: Crack center coordinate.
YR: Crack center coordinate.
ZR: Crack center coordinate.
Ø: Angle with X axes.

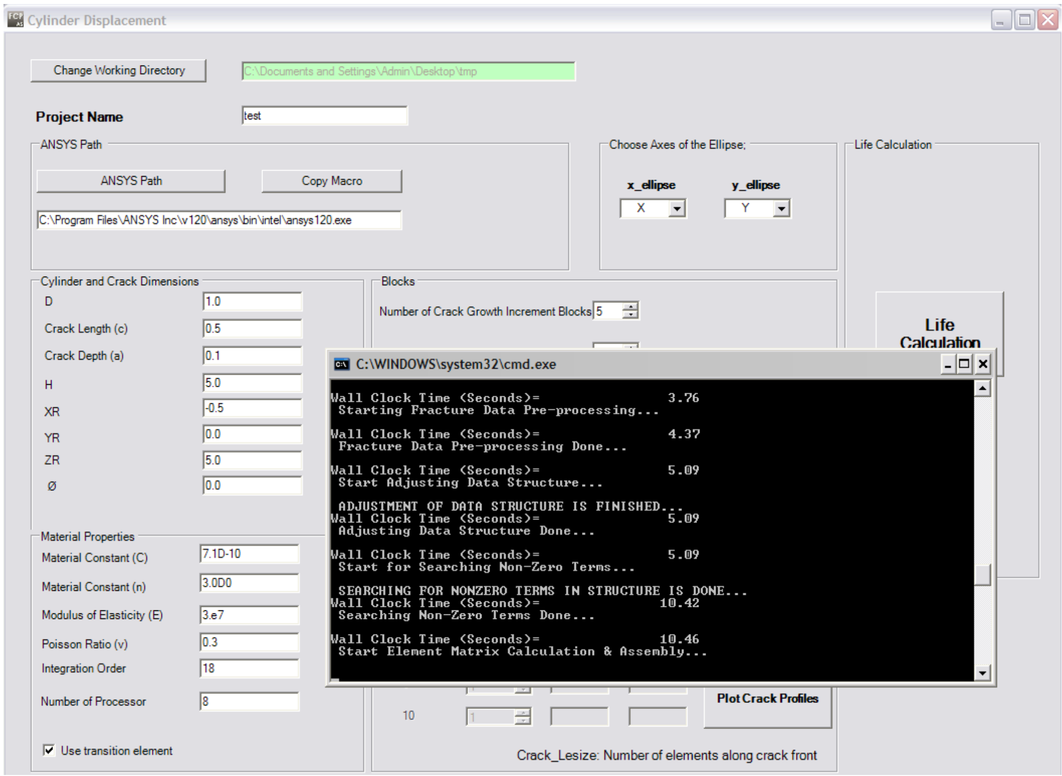
Write material properties.



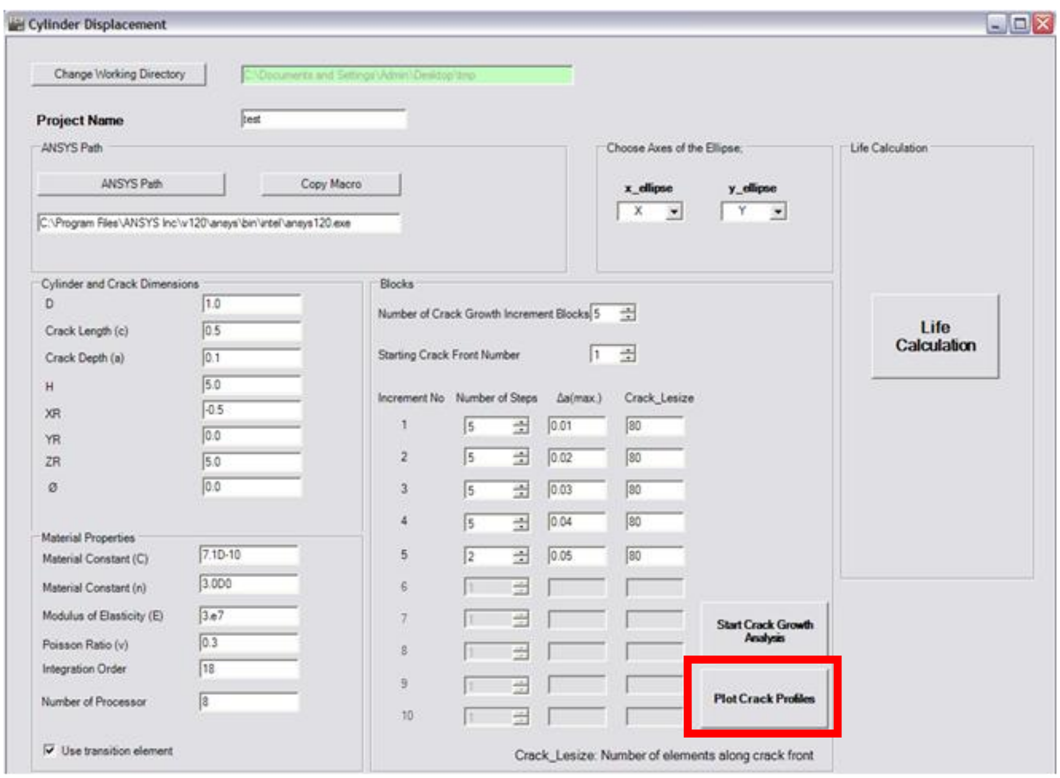


We click “Start Crack Growth Analysis” and crack growth analysis starts.
Frac3D Solver

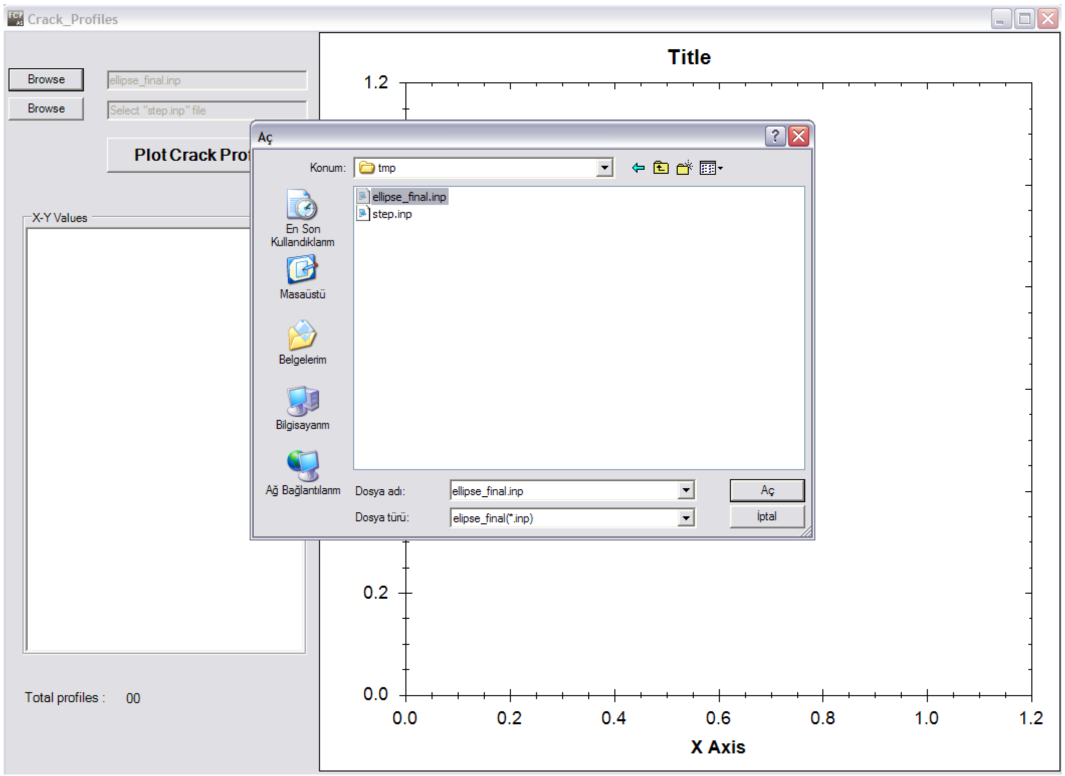




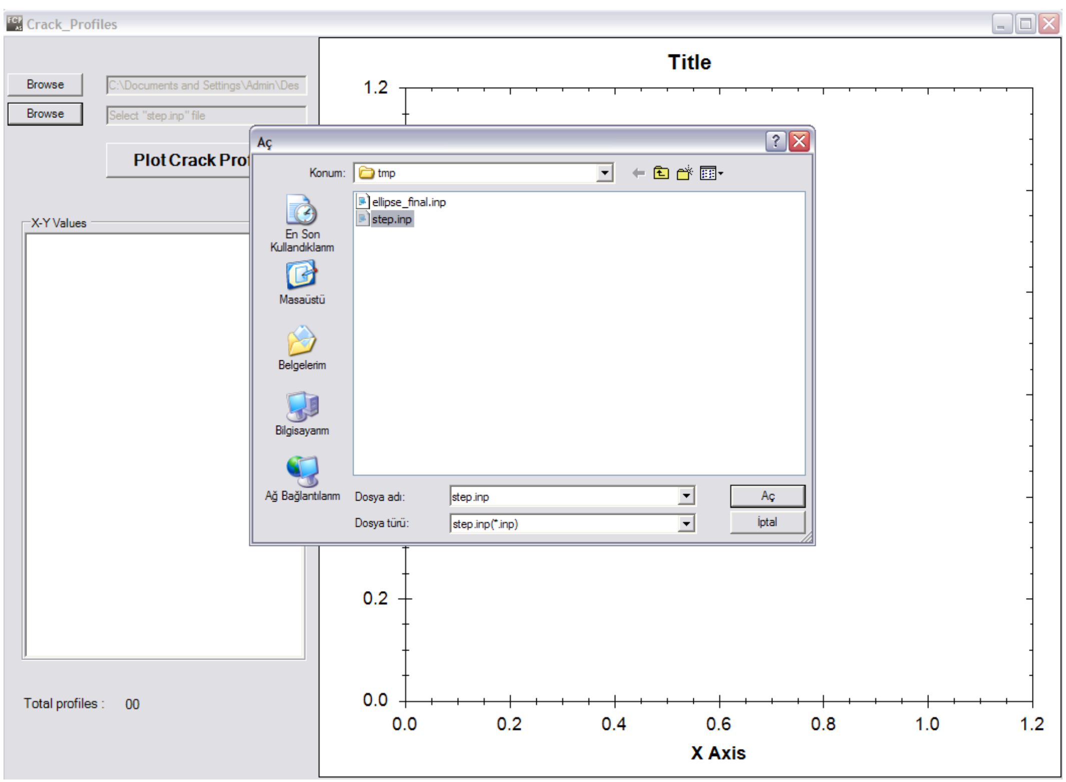
After solution, click "Plot Crack Profiles" button.



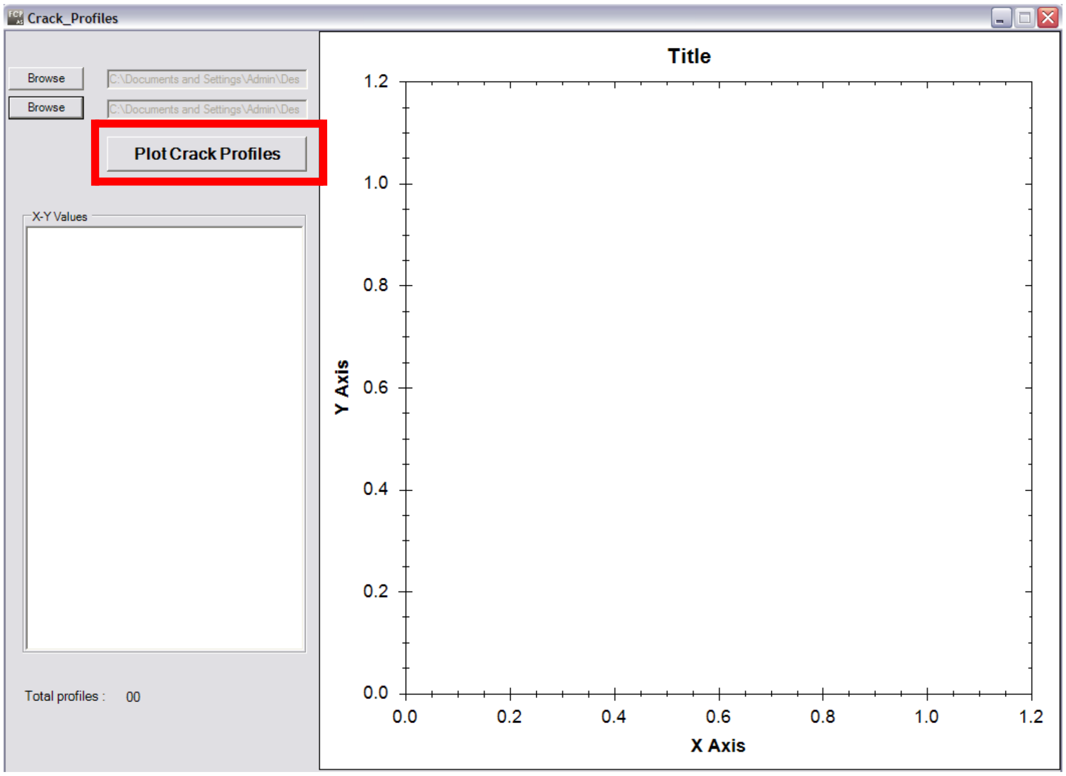
First, select ellipse_final.inp file from the working directory.



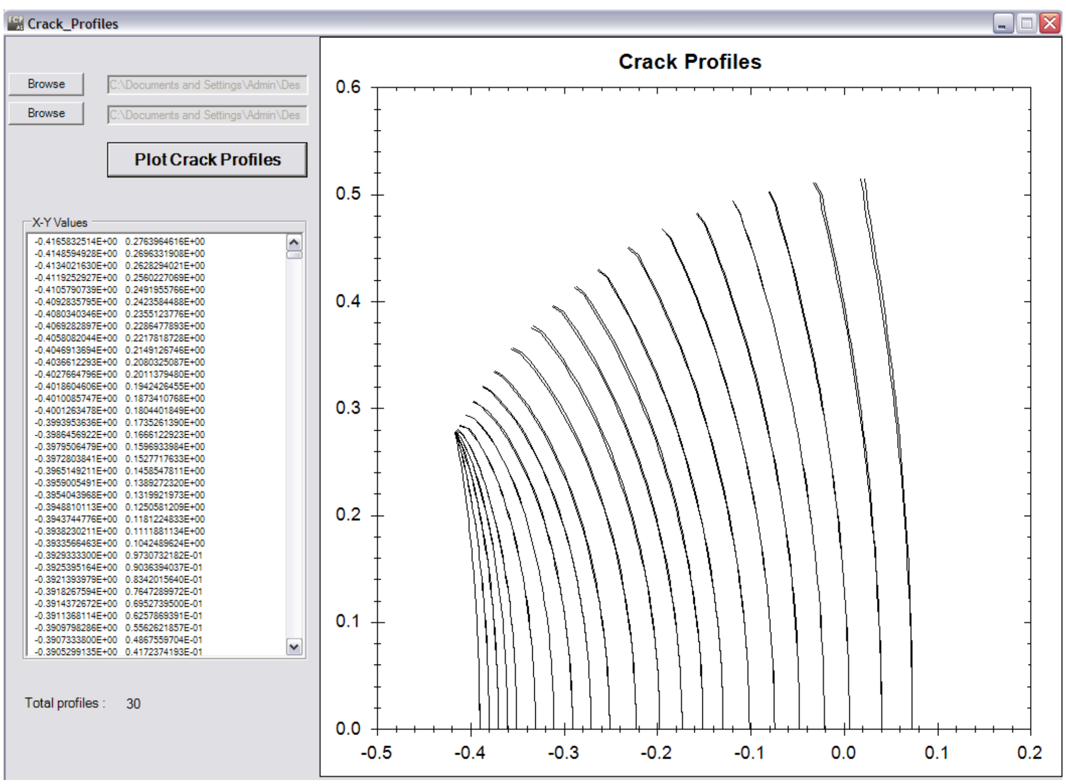
Select step.inp file from the working directory.



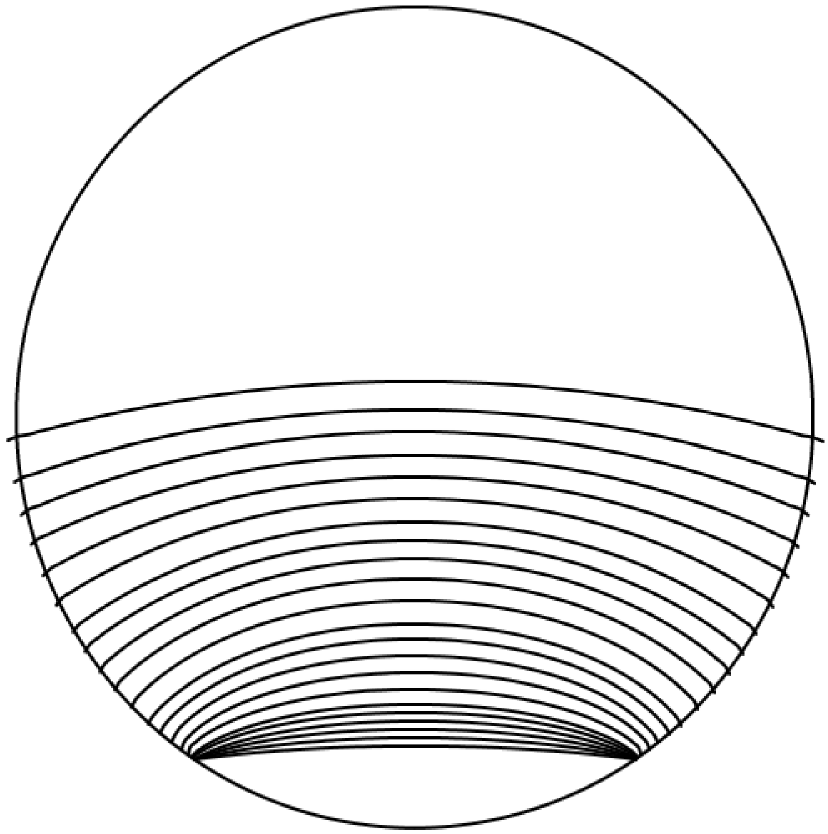
Click “Plot Crack Profiles” button.



You can see symmetry of the crack profiles.

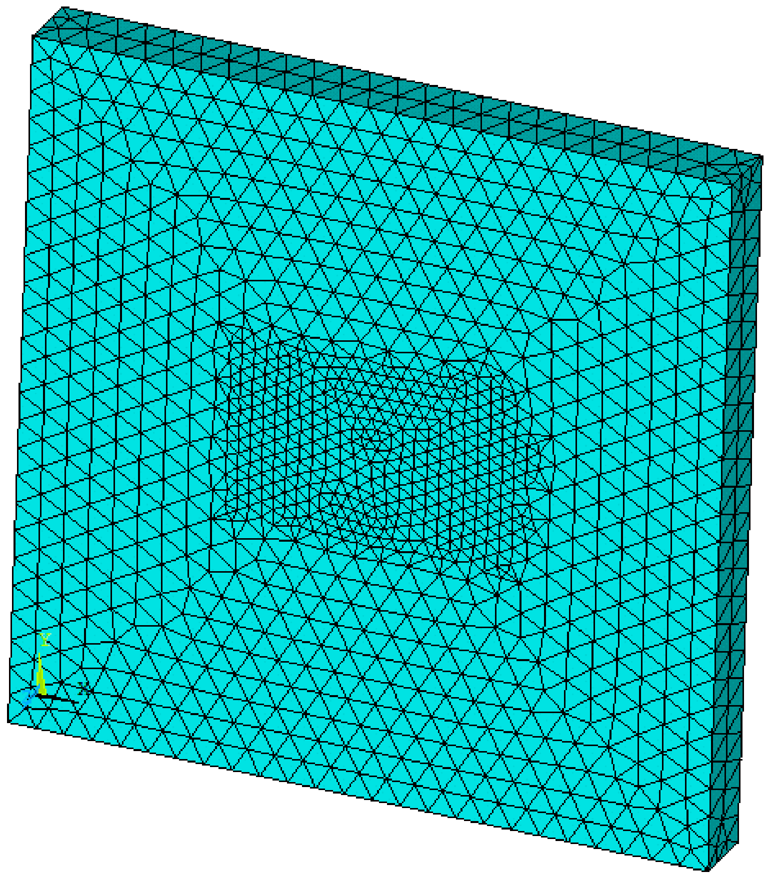


It is also possible to get crack profiles using Microsoft Excel.



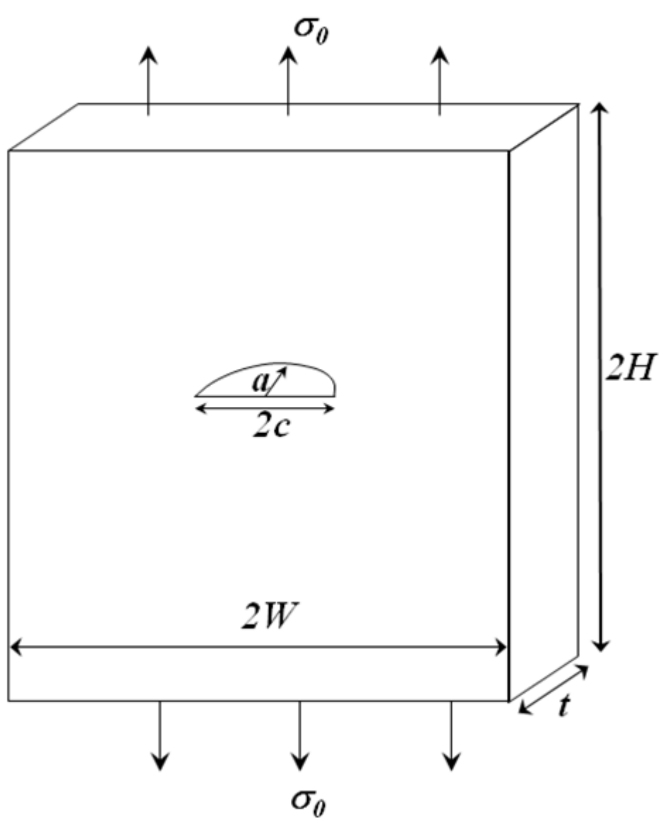
EXAMPLE.6. Plate Crack Insertion and Fracture Analysis

We generate finite element model without crack using ANSYS™ Dimensions of plate are 2W=50 mm, 2H=50mm, t=5 mm. Plate is subjected to uniform tension loading.

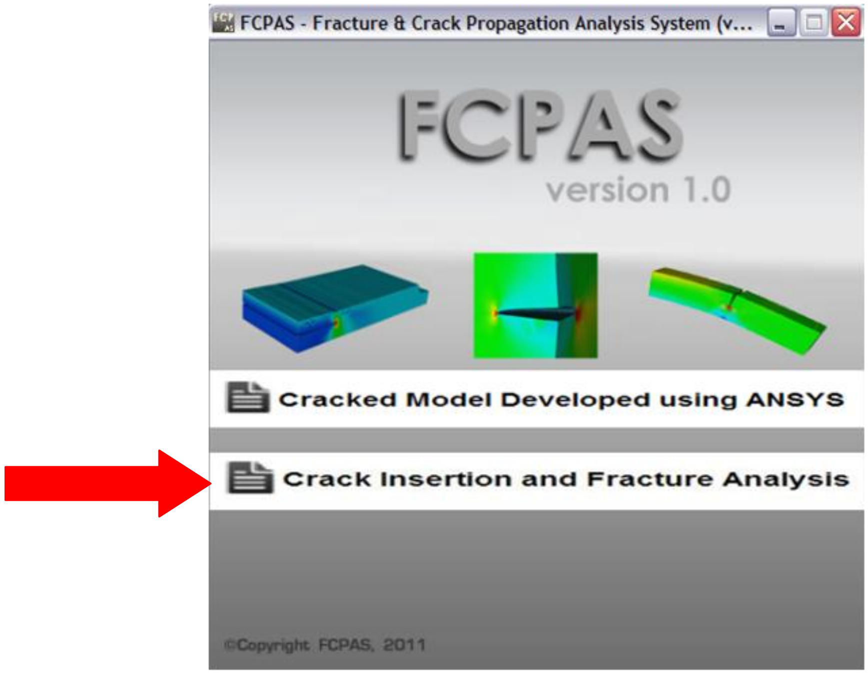


After we get finite ANSYS™ model without crack, we start to insert crack into the plate.In this example, a/t and a/c ratios are equal to 0.2. So, crack length (2c) is 5 mm. and also crack depth (a) is 1 mm.

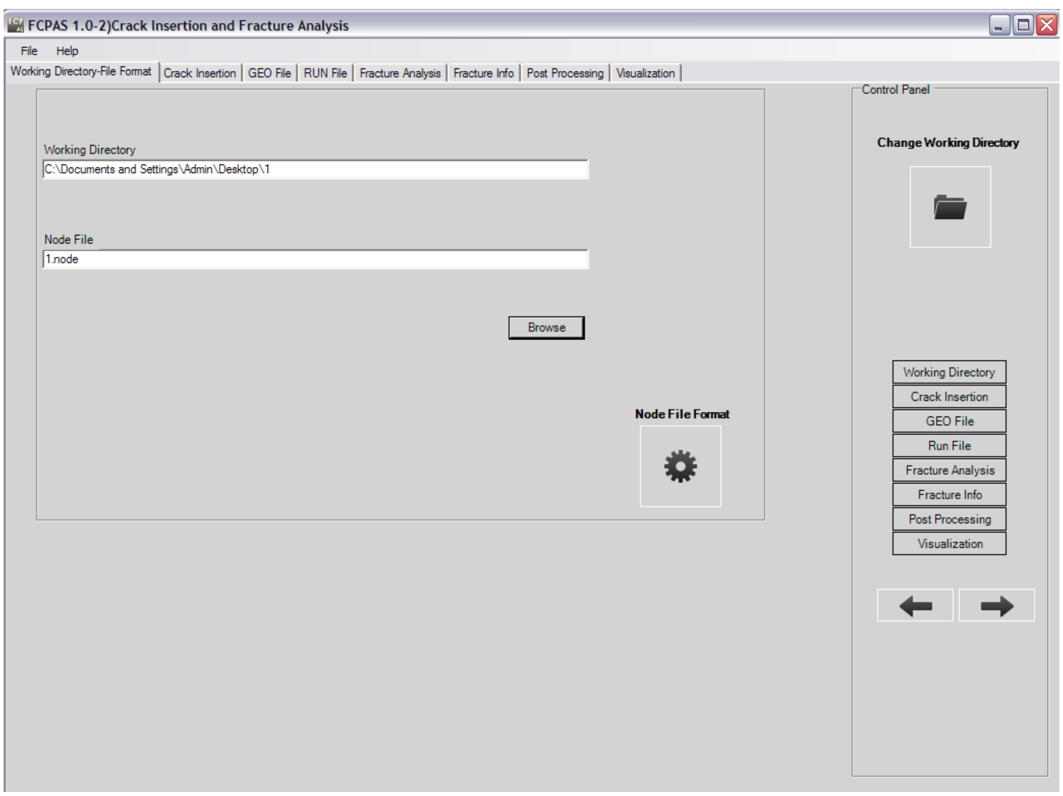
2W= 50 mm.
2H= 50 mm.
t= 5 mm.
2c=5 mm.
a= 1 mm.



First, we select "Crack Insertion and Fracture Analysis" button.

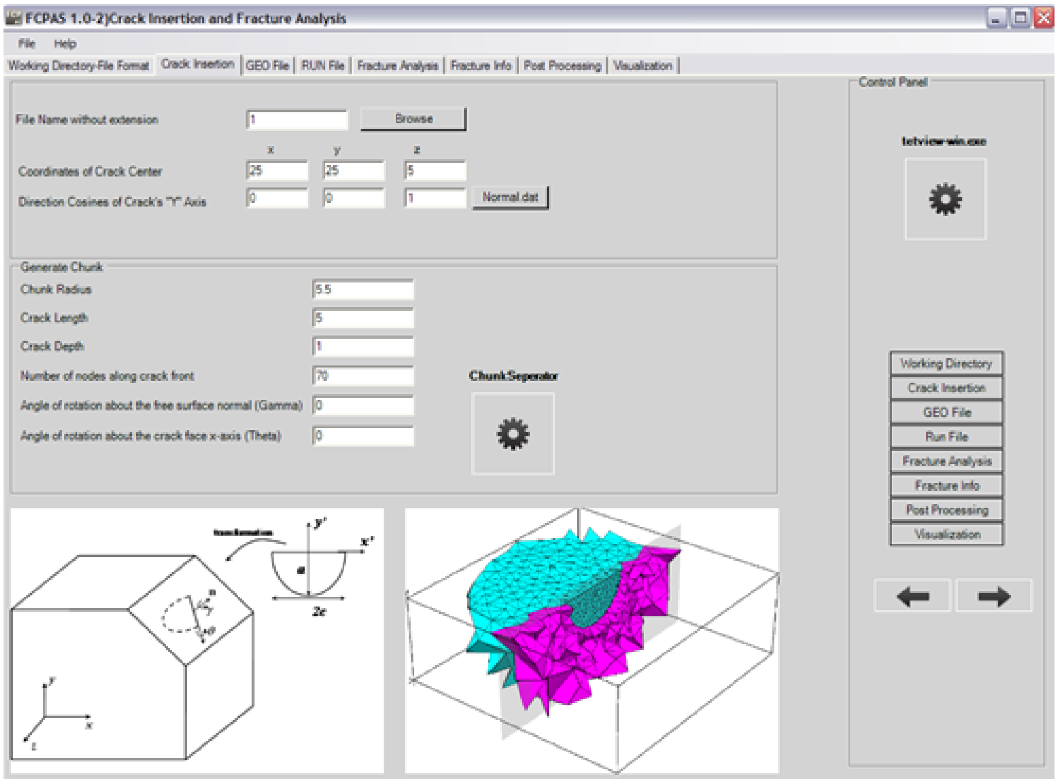


Select "working directory" and *.node extension file, comes from ANSYS™.



In this tab, we perform "Crack Insertion".

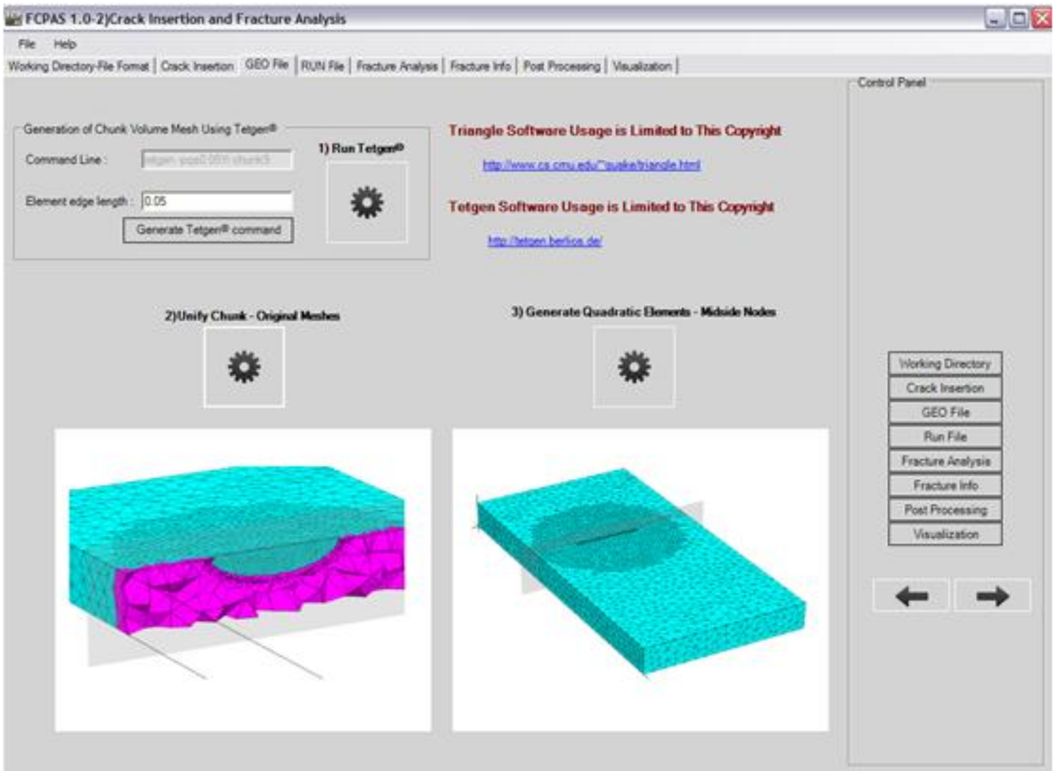
- Select *.node2 extension file using "Browse" button.
- Give coordinates of crack center.
- Chunk radius: Chunk is a volume that contains crack.
- Give Crack Length, Crack Depth, Number of nodes along crack front and angles.
- Click "ChunkSeperator" button and create chunk with crack.



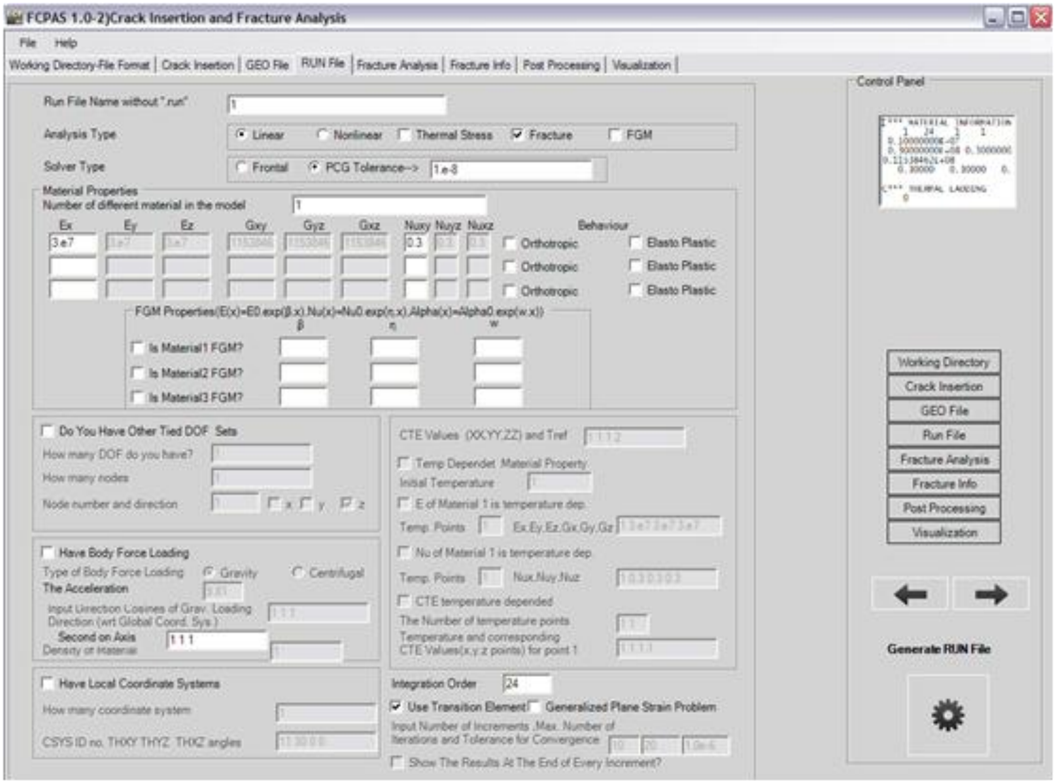
Here, we write "Element edge length".
Click;

- 1)Run Tetgen
- 2)Unify Chunk – Original Meshes
- 3)Generate Quadratic Elements – Midside Nodes.

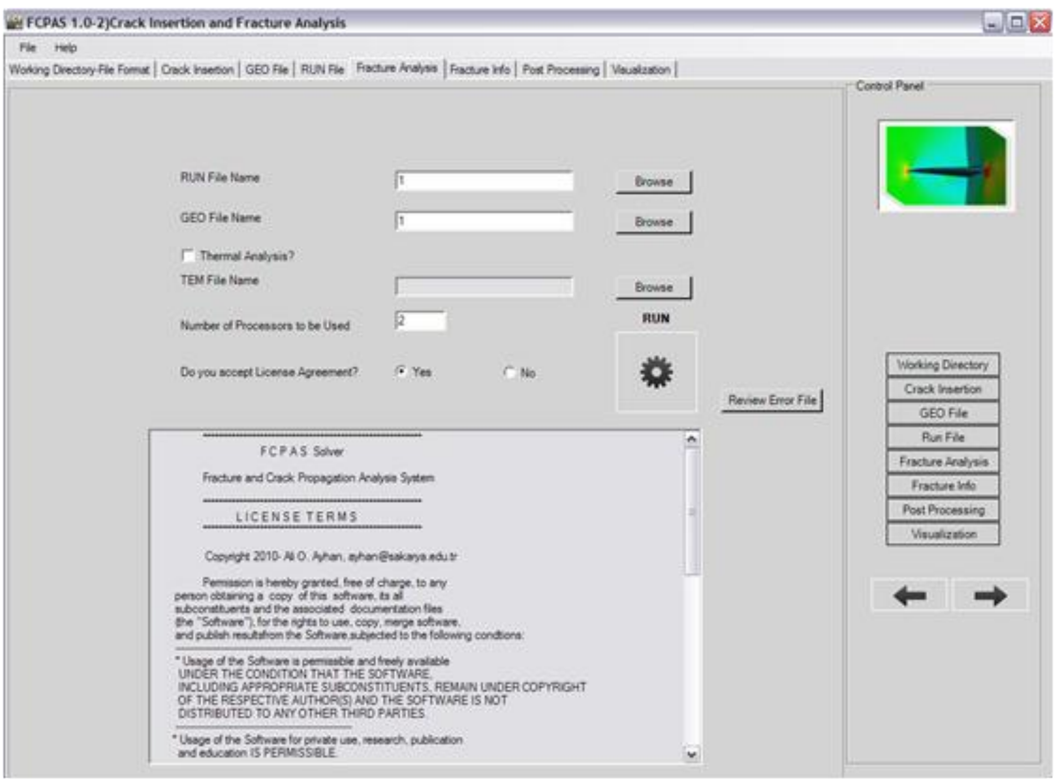
Now, cracked finite element is ready to fracture analysis.



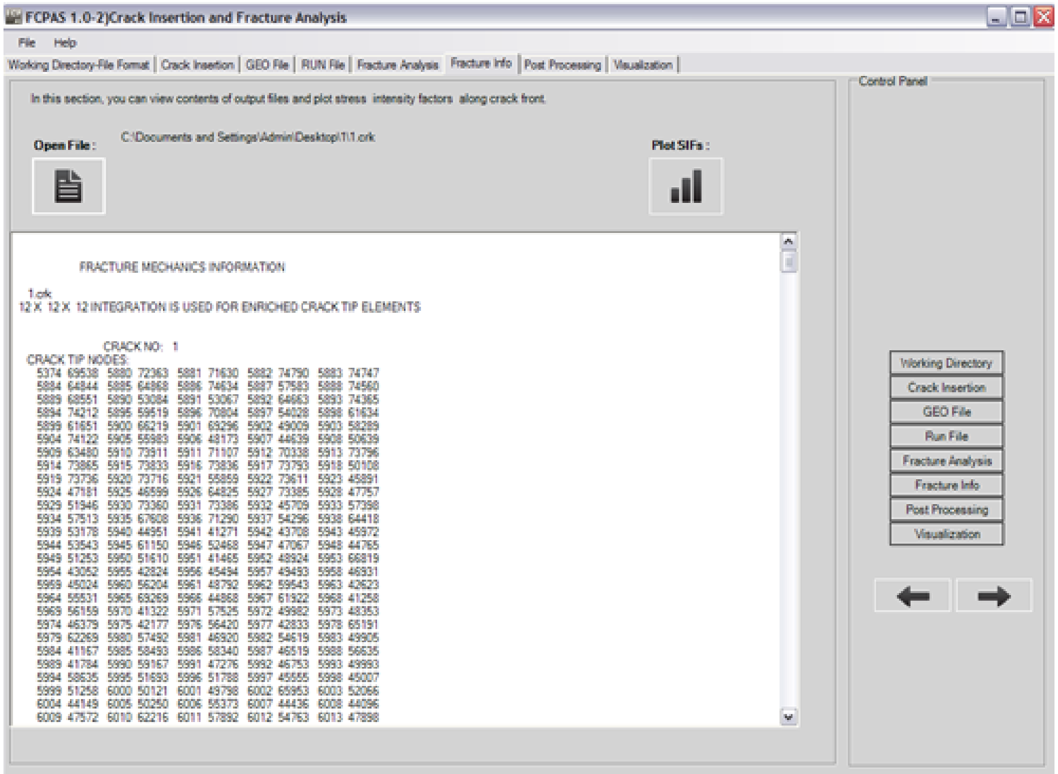
In "RUN File" tab, we select material properties.

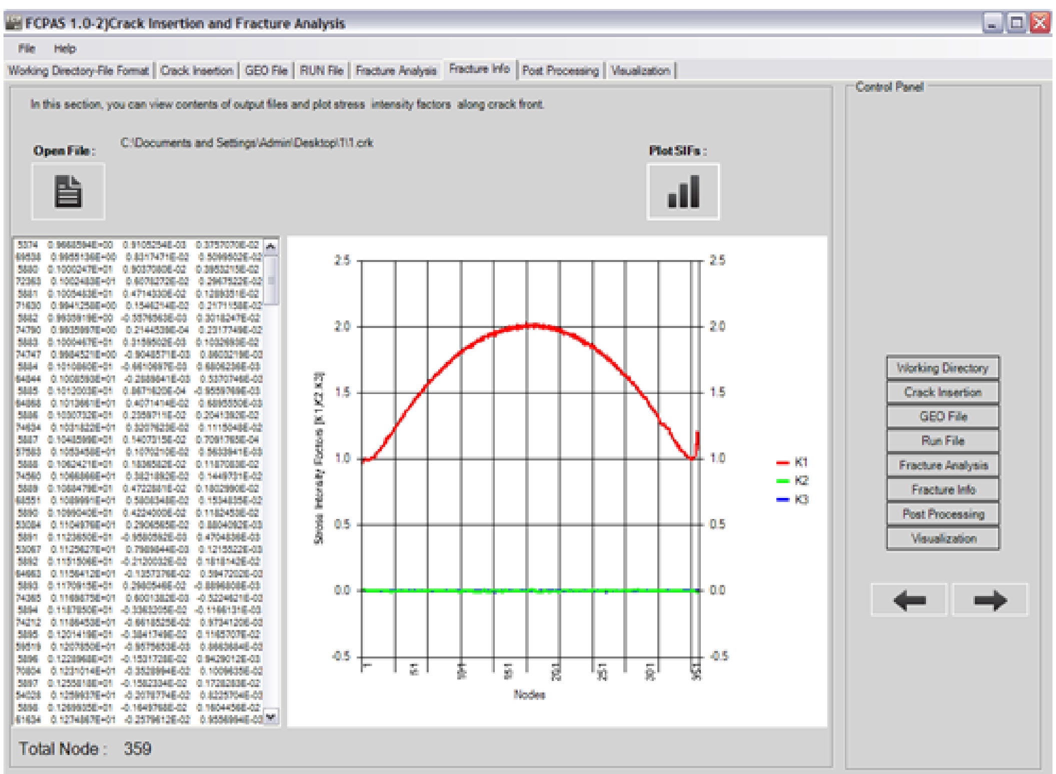


Fracture Analysis.
"Frac3D Solver".

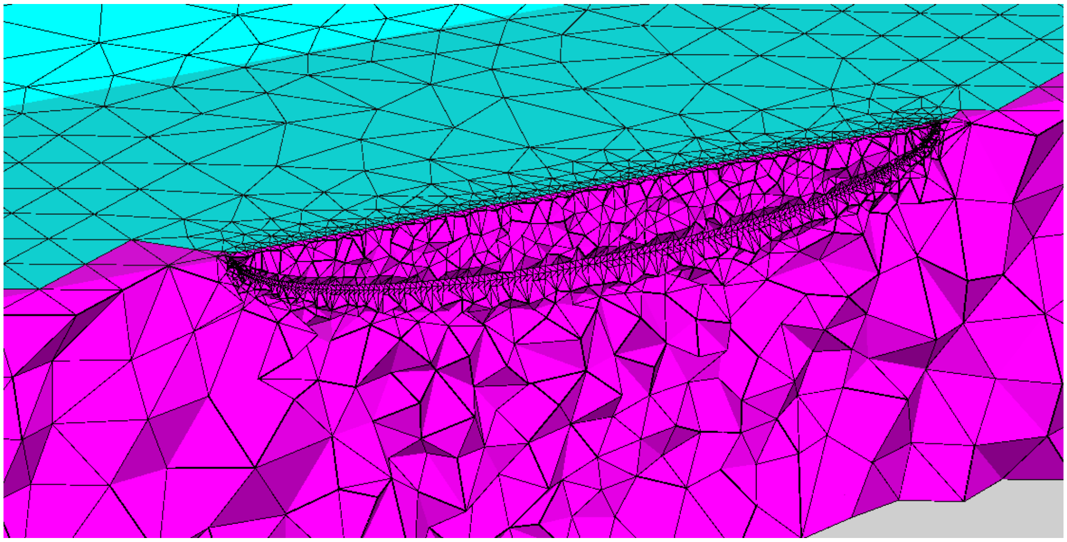


Fracture Info and SIFs graphic.





Cut-views of near-crack-surface meshes



References

[1] ANSYS 12.0 Academic Research Advanced Version, Canonsburg, PA, U.S.A.

Contributions/Applications By:

C. Kurtiş
M. Uslu
G. Atalı
İ. Y. Sülü
H. Pekel
İ. Kacar
A. R. Zaloğlu
E. Nart
A.O. Ayhan
H. F. Nied

Contact: Dr. Ali O. Ayhan, aoayhan@yildiz.edu.tr