

# ANSYS TUTORIAL – 2-D Fracture Analysis

ANSYS Release 7.0

Dr. A.-V. Phan, University of South Alabama

## 1 Problem Description

Consider a finite plate in tension with a central crack as shown in Fig. 1. The plate is made of steel with Young's modulus  $E = 200$  GPa and Poisson's ratio  $\nu = 0.3$ . Let  $b = 0.2$  m,  $a = 0.02$  m,  $\sigma = 100$  MPa. Determine the stress intensity factors (SIFs).

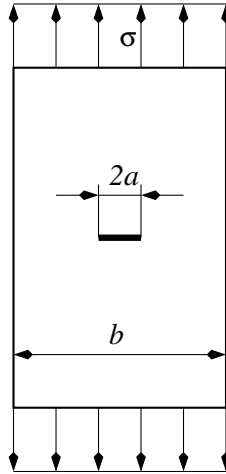


Figure 1: Through-thickness crack

An analytical solution given by W.D. Pilkey (*Formulas for Stress, Strain, and Structural Matrices*) is

$$K_I = C \sigma \sqrt{\pi a} ,$$

where

$$C = (1 - 0.1 \eta^2 + 0.96 \eta^4) \sqrt{1 / \cos(\pi \eta)} ,$$

$$\eta = \frac{a}{b} .$$

Use of this solution yields  $K_I = 25.680$  MPa $\cdot\sqrt{\text{m}}$ .

## 2 Assumptions and Approach

### 2.1 Assumptions

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

## 2.2 Approach

- Since the LEFM assumption is used, the SIFs at a crack tip may be computed using the ANSYS's **KCALC** command. The analysis used a fit of the nodal displacements in the vicinity of the crack tip (see the **ANSYS, Inc. Theory Reference**).
- Due to symmetry of the problem, a quarter model can be used as in the first fracture tutorial. However, to illustrate a way to model both upper and lower faces of a crack, the right-half model shown in Fig. 2 is considered in this tutorial.

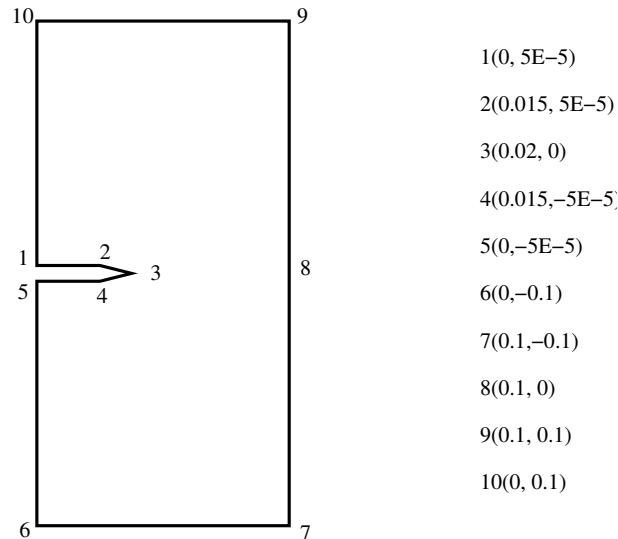


Figure 2: The right-half model: keypoints and their coordinates

To facilitate the modeling of two coincident faces, a very small opening of the crack needs to be created. A recommended geometry of the opening is shown in Fig. 3.

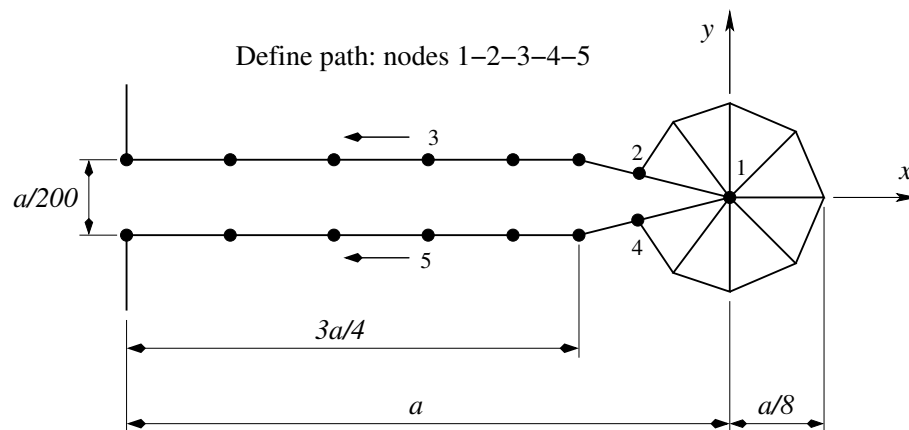


Figure 3: A small crack opening

- The crack-tip region is meshed using quarter-point (singular) 8-node quadrilateral elements (**PLANE82**).

### 3 Preprocessing

1. Give the Job a Name

**Utility Menu > File > Change Jobname ...**

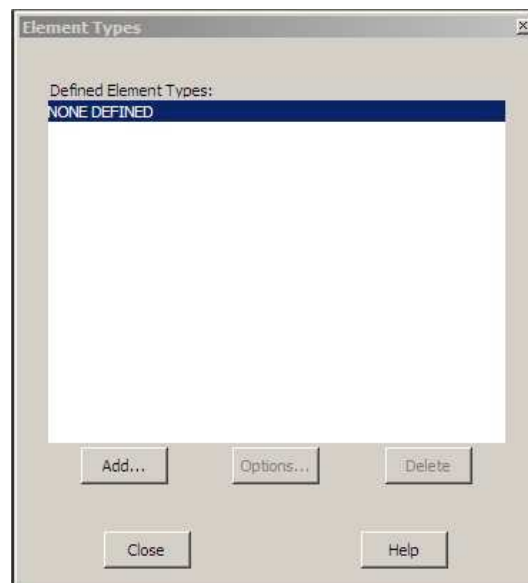
The following window comes up. Enter a name, for example 'CentralCrack', and click on **OK**.



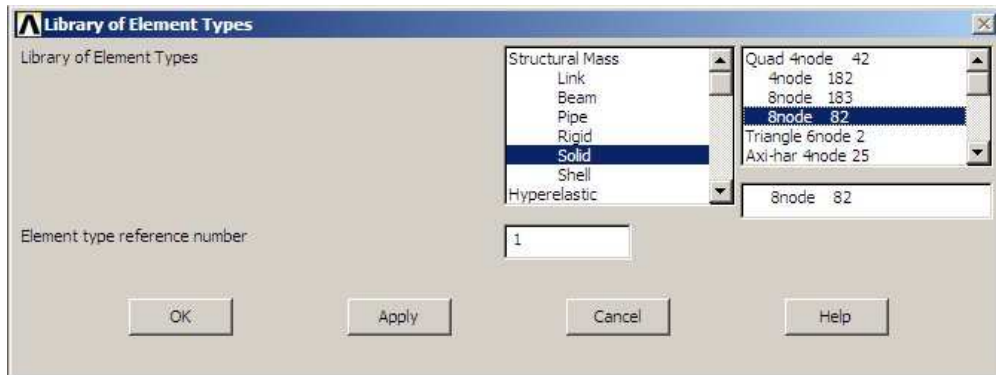
2. Define Element Type

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

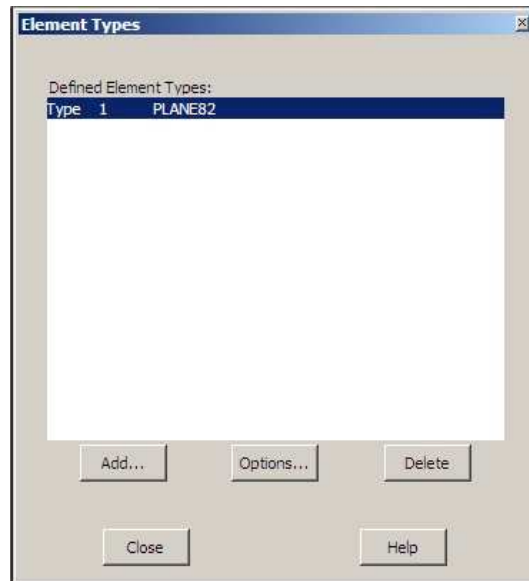
- This brings up the 'Element Types' window. Click on the **Add...** button.



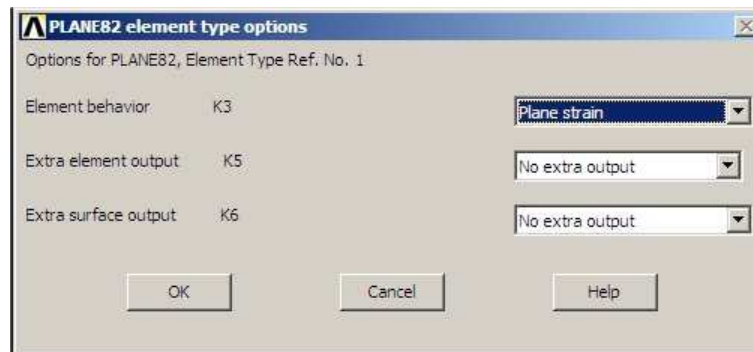
- The 'Library of Element Types' window appears. Highlight 'Solid', and '8node 82', as shown. Click on **OK**.



- You should see 'Type 1 PLANE82' in the 'Element Types' window as follows:



- Click on the **Options...** button in the above window. The below window comes up. Select 'Plane strain' for 'Element behavior K3' and click **OK**.

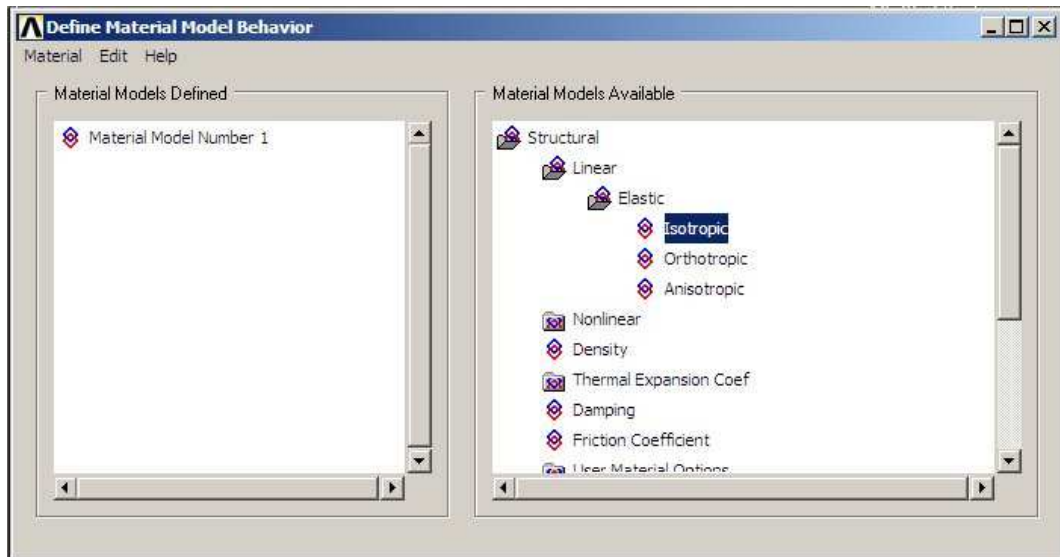


- Click on the **Close** button in the 'Element Types' window.

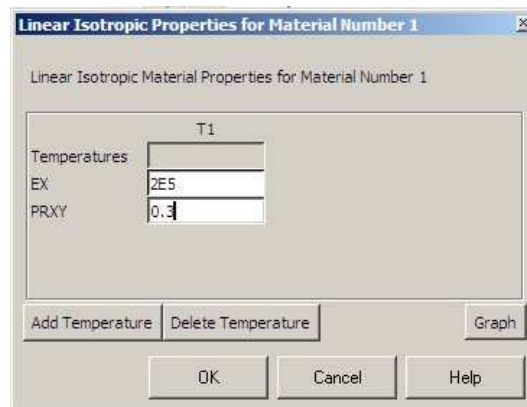
### 3. Define Material Properties

**Main Menu > Preprocessor > Material Props > Material Models**

- In the right side of the 'Define Material Model Behavior' window that opens, double click on 'Structural', then 'Linear', then 'Elastic', then finally 'Isotropic'.



- The following window comes up. Enter in values for the Young's modulus ( $EX = 2E5$ ) and Poisson's ratio ( $PRXY = 0.3$ ) of the plate material.



- Click **OK**, then close the 'Define Material Model Behavior' window.

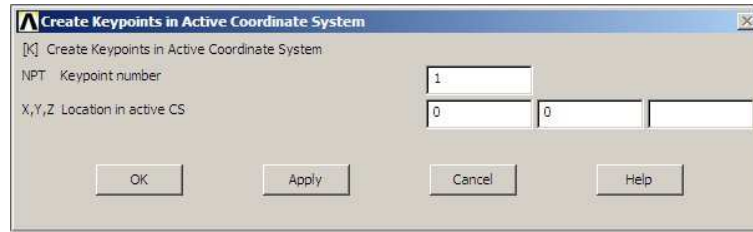
#### 4. Define Keypoints

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

We are going to create 10 keypoints given in the following table:

Keypoint #	X	Y	Keypoint #	X	Y
1	0	5E-5	6	0	-0.1
2	0.015	5E-5	7	0.1	-0.1
3	0.02	0	8	0.1	0
4	0.015	-5E-5	9	0.1	0.1
5	0	-5E-5	10	0	0.1

- To create keypoint #1, enter '1' as keypoint number, and '0' and '0' as the X and Y coordinates in the following window. Click on **Apply**.



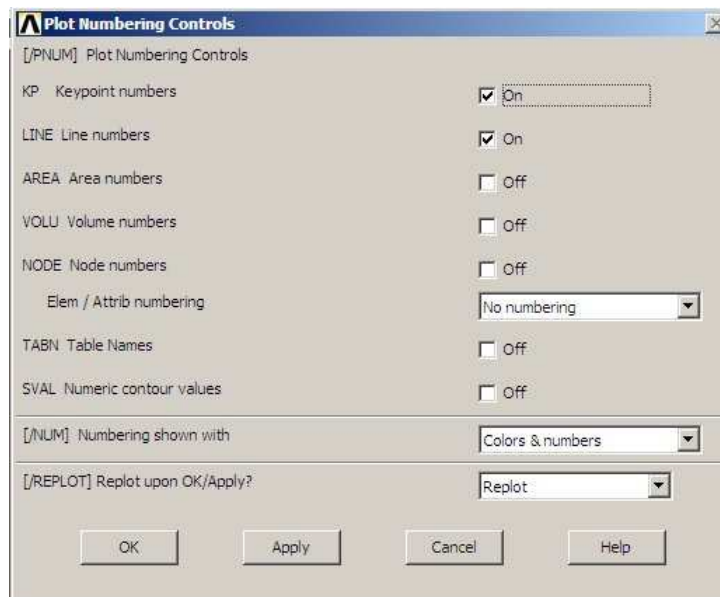
- Repeat the above step for keypoints #2 through #10. Note that you must click on **OK** instead of **Apply** after entering data of the final keypoint.

### 5. Define Line Segments

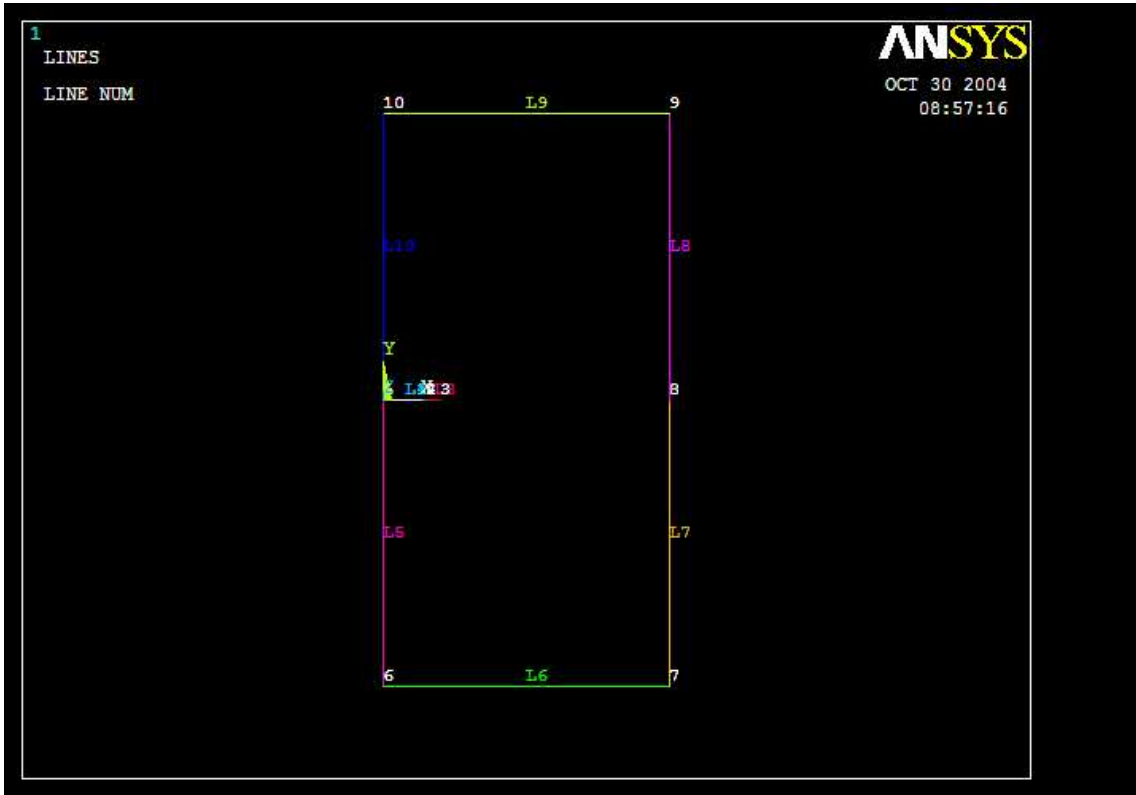
- We are going to create the following 10 line segments that define the boundary of the right-half model (see Fig. 2):

Line #	Starting keypoint	Ending keypoint	Line #	Starting keypoint	Ending keypoint
1	1	2	6	6	7
2	2	3	7	7	8
3	3	4	8	8	9
4	4	5	9	9	10
5	<b>6</b>	<b>5</b>	10	<b>10</b>	<b>1</b>

- The best way to create these lines is to enter 'L, (Starting KP), (Ending KP)' followed by the 'Enter' key in the prompting window. In this tutorial, it is important to respect the keypoint order of lines #5 and #10 as shown in the above table.
- Turn on the numbering by selecting **Utility Menu > PlotCtrls > Numbering ...** and complete the window that appears as shown. Click on **OK**.



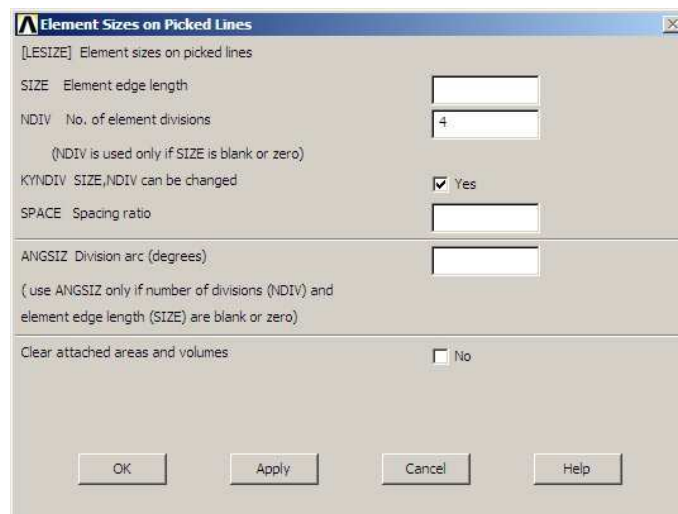
- Select **Utility Menu > Plot > Lines**. Your graphics window should look like this,



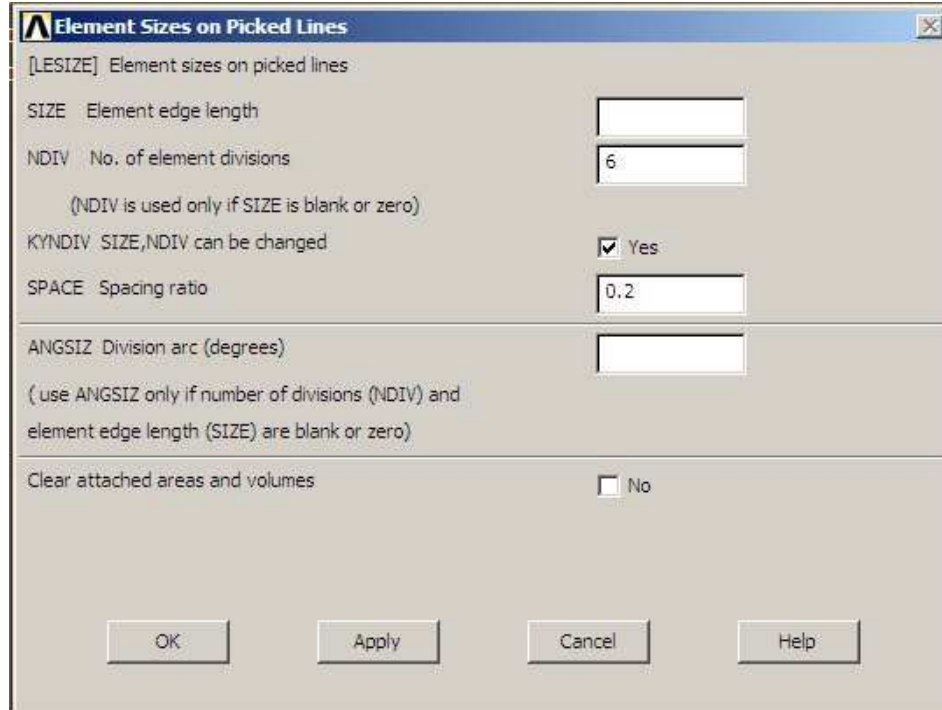
6. Discretize Lines L6, L7, L8, L9, L5 and L10

**Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > Picked Lines**

- Pick lines #6, #7, #8 and #9. Click on the **OK** button in the picking window.
- The below window opens. Enter '4' for 'No. of element divisions', then click **Apply**.



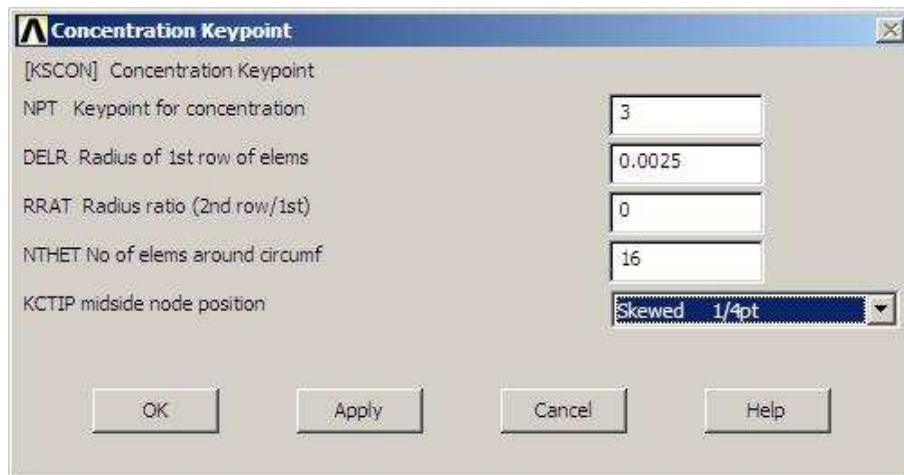
- Pick lines #5 and #10, then click **OK** in the picking window.
- In the below window that comes up again, enter '6' for 'No. of element divisions', and '0.2' for 'Spacing ratio', then click **OK**.



7. Create the Concentration Keypoint (Crack Tip)

**Main Menu > Preprocessor > Meshing > Size Cntrls > Concentrat KPs > Create**

- Pick keypoint #3, then click **OK** in the picking window.
- In the below window that appears, you should see '3' as 'Keypoint for concentration'. Enter '0.0025' (=  $a/8$ ) for 'Radius of 1st row of elems', input '16' for 'No of elems around circumf', and select 'Skewed 1/4pt' for 'midside node position'. Click **OK**.



8. Create the Area

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

- Pick all the lines (L1 through L10) by selecting 'Loop' in the picking window, then selecting any one of those lines. Click on **OK**.

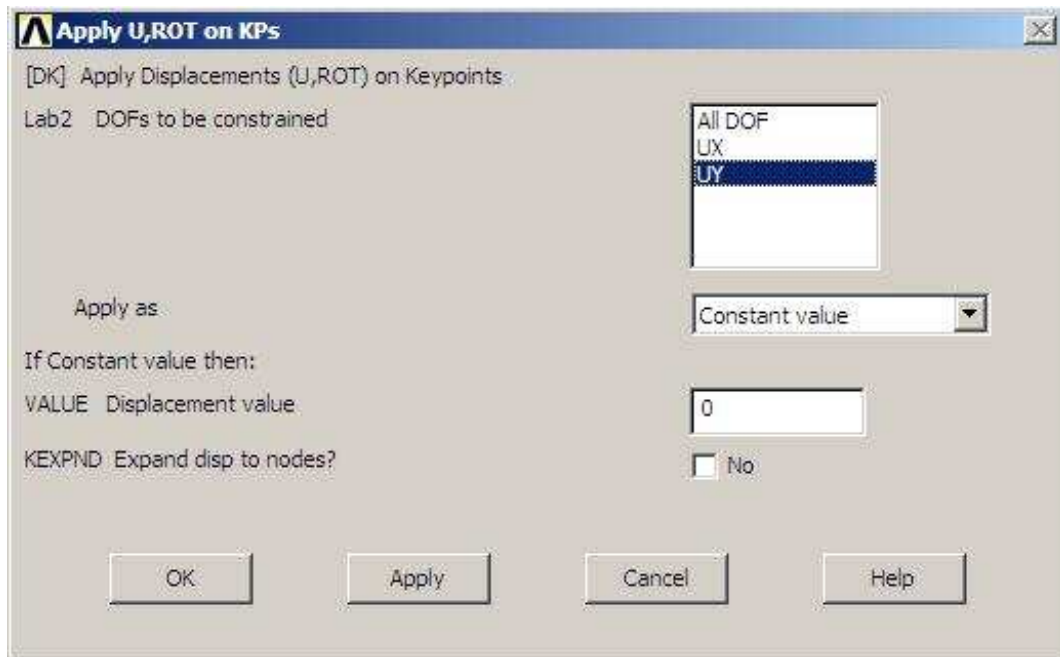
9. Apply Boundary Conditions

**Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > Symmetry B.C. > ...with Area**

- Pick line #5. Click **Apply** (in the picking window). Pick the area. Click **Apply**.
- Pick line #10. Click **Apply**. Pick the area. Click **OK**.

**Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Keypoints**

- Pick keypoint #8. Click on **OK** in the picking window.
- In the following window that pops up, select 'UY' and enter '0' for 'Displacement value', then click on **OK**.



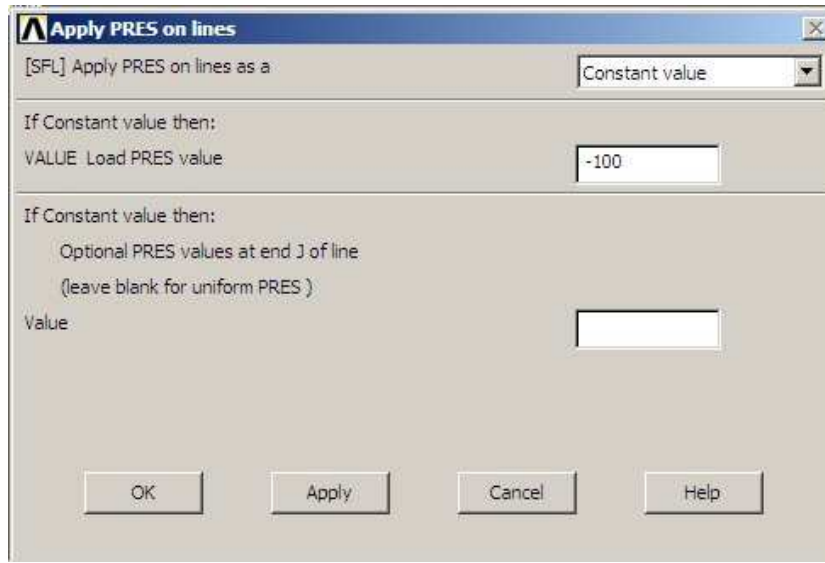
10. Apply Loads

**Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Pressure > On Lines**

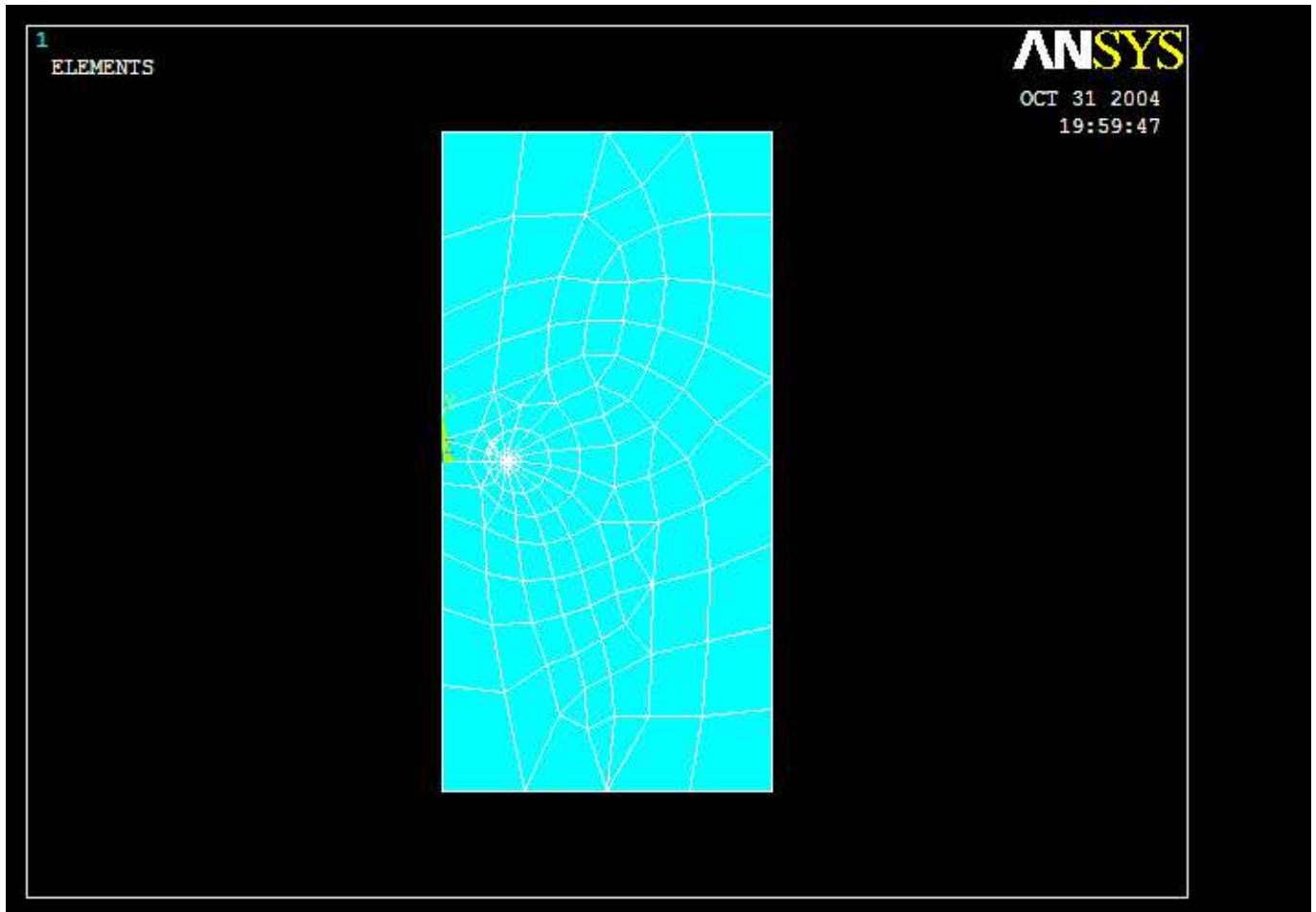
- Pick lines #6 and #9. Click **OK** in the picking window.
- In the following window that comes up, select 'Constant value' for 'Apply PRES on lines as a', enter '-100' for 'Load PRES value', then click on **OK**.

11. Mesh the Model

**Main Menu > Preprocessor > Meshing > Mesh > Areas > Free**



- Pick the area and click on **OK**.
- Close the 'Warning' window. Your ANSYS window should look like this,



## 4 Processing (Solving)

**Main Menu > Solution > Analysis Type > New Analysis**

- Make sure that 'Static' is selected. Click **OK**.

**Main Menu > Solution > Solve > Current LS**

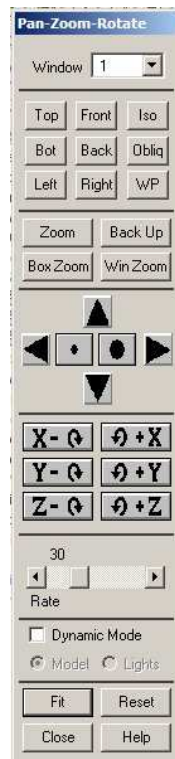
- Check your solution options listed in the '/STATUS Command' window.
- Click the **OK** button in the 'Solve Current Load Step' window.
- Click the **Yes** button in the 'Verify' window.
- You should see the message 'Solution is done!' in the 'Note' window that comes up. Close the 'Note' and '/STATUS Command' windows.

## 5 Postprocessing

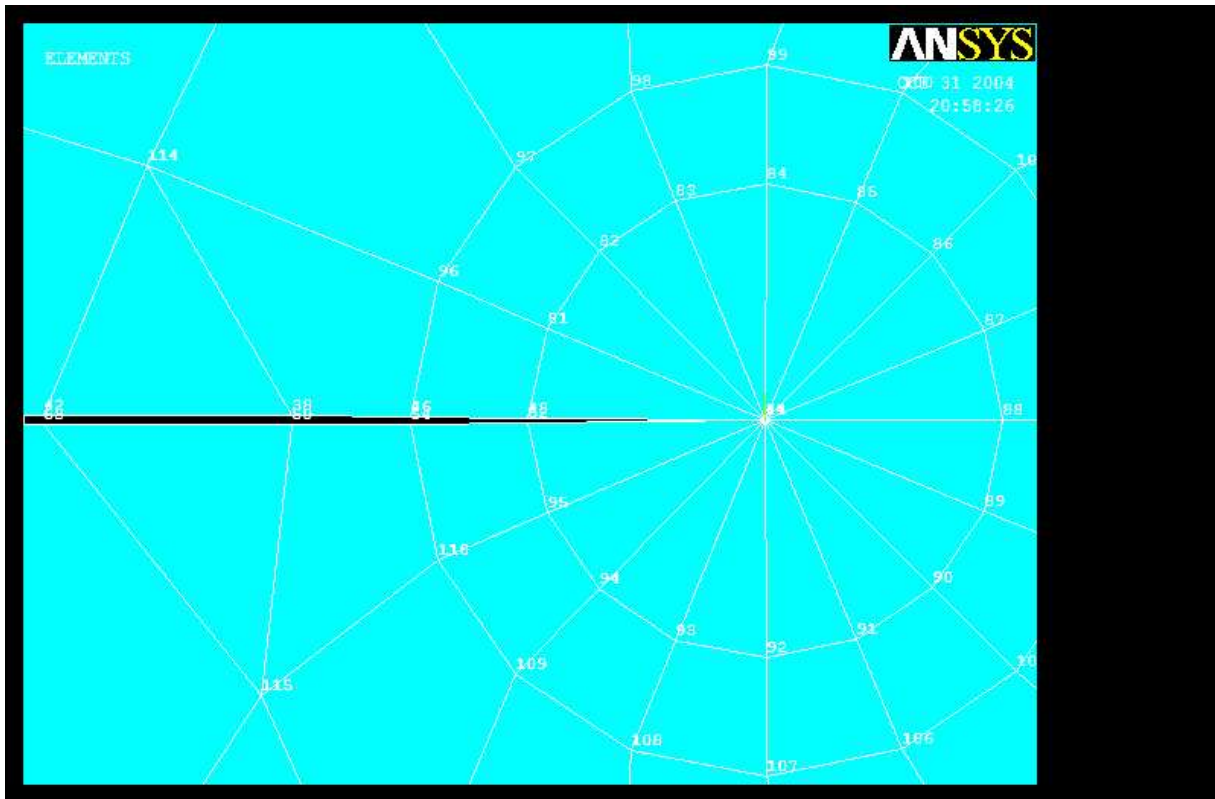
1. Zoom the Crack-Tip Region

**Utility Menu > PlotCtrls > Pan Zoom Rotate ...**

This brings up the following window:



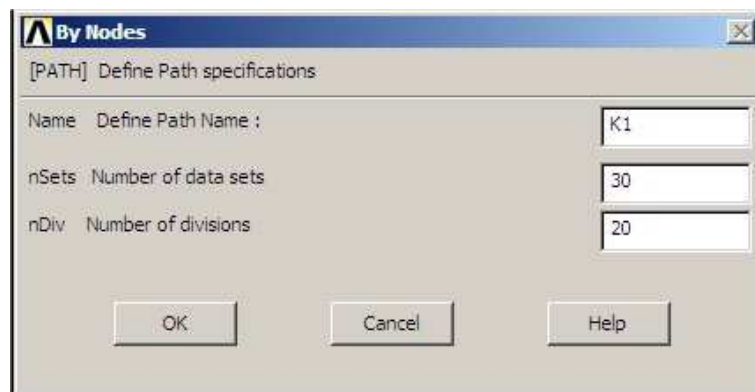
- In the above window, click on the **Box Zoom** button and zoom the crack-tip region. You may want to leave the 'Pan-Zoom-Rotate' window open for further use.
- Turn on the node numbering by selecting **Utility Menu > PlotCtrls > Numbering ...**, then check the box for 'Node numbers', then finally click on **OK**. Your ANSYS Graphics windows should be similar to the following:



## 2. Define Crack-Face Path

**Main Menu > General Postproc > Path Operations > Define Path > By Nodes**

- Pick the crack-tip node (node #44), then nodes #48, #42, #52 and #58 on the crack faces (see Fig. 3). Click **OK**.
- In the below window that appears, enter 'K1' for 'Define Path Name:', then click **OK**.

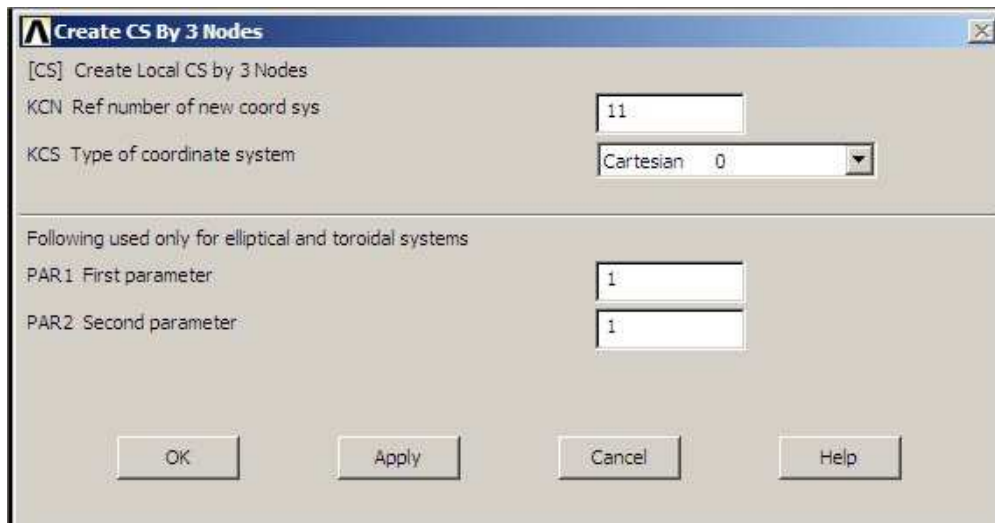


- Close the 'PATH Command' window.

## 3. Define Local Crack-Tip Coordinate System

**Utility Menu > WorkPlane > Local Coordinate Systems > Create Local CS > By 3 Nodes**

- Pick node #44 (the crack-tip node), then node #88, and finally node #84. This brings up the following window:



- Note from the above window that the reference number of the crack-tip coordinate system is 11. Click on the **OK** button.

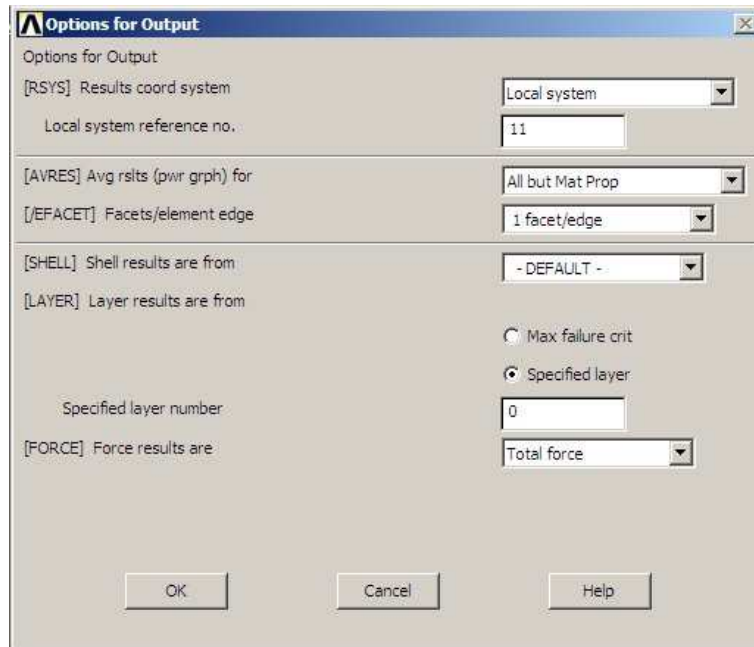
4. Activate the Local Crack-Tip Coordinate System

**Utility Menu > WorkPlane > Change Active CS to > Specified Coord Sys ...**

- In the below window that comes up, enter '11' for 'Coordinate system number', then click **OK**.



- To activate the crack-tip coordinate system as results coordinate system, select **Main Menu > General Postproc > Options for Outp.** In the window that appears (as shown at the top of the next page), select 'Local system' for 'Results coord system' and enter '11' for 'Local system reference no.'. Click **OK** in this window.



5. Determine the Mode-I Stress Intensity Factor using KCALC  
**Main Menu > General Postproc > Nodal Calcs > Stress Int Factr**

- In the below window that opens, select 'Plain strain' for 'Disp extrapolat based on' and 'Full-crack model' for 'Model Type'. Note that the 'Full-crack model' must be selected as both the crack faces are included in the model.

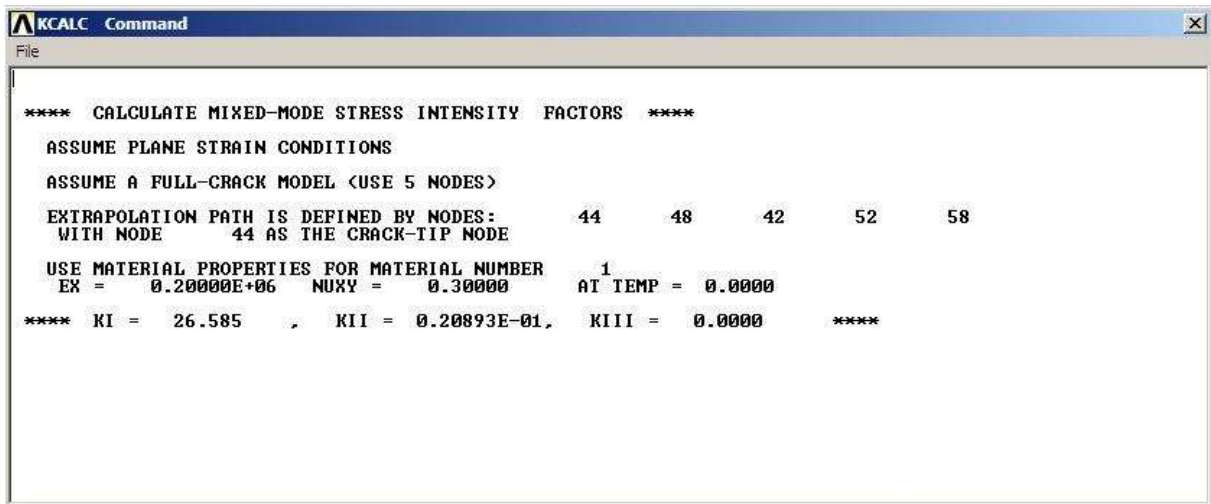


- Click on **OK**. The window shown at the top of the next page appears and it shows that the SIFs at the crack tip (node #4) are

$$K_I = 26.585 ; \quad K_{II} = 0.020893 ; \quad K_{III} = 0$$

Note that the numerical results for both  $K_I$  and  $K_{II}$  are not as accurate as in the case of a quarter model presented in the first fracture tutorial (due to the use of an artificial crack-opening and the mesh is not perfectly symmetric about the X-axis). Comparing with the Pilkey's solution ( $K_I = 25.680 \text{ MPa}\cdot\sqrt{\text{m}}$ ), the percentage error of  $K_I$  is

$$\epsilon = \frac{K_I^{\text{ANSYS}} - K_I^{\text{Pilkey}}}{K_I^{\text{Pilkey}}} = \frac{26.585 - 25.680}{25.680} = 3.52 \%$$



```
**** CALCULATE MIXED-MODE STRESS INTENSITY FACTORS ****
ASSUME PLANE STRAIN CONDITIONS
ASSUME A FULL-CRACK MODEL (USE 5 NODES)
EXTRAPOLATION PATH IS DEFINED BY NODES:      44      48      42      52      58
WITH NODE      44 AS THE CRACK-TIP NODE

USE MATERIAL PROPERTIES FOR MATERIAL NUMBER      1
EX =      0.200000E+06      NUXY =      0.300000      AT TEMP =      0.0000
**** KI =      26.585      ,      KII =      0.20893E-01,      KIII =      0.0000      ****
```

- Close the 'KCALC Command' window.
  - Close the 'Pan-Zoom-Rotate' window.
6. Exit ANSYS, Saving All Data  
Utility Menu > File > Exit ...
- In the window that opens, select 'Save Everything' and click on **OK**.