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).

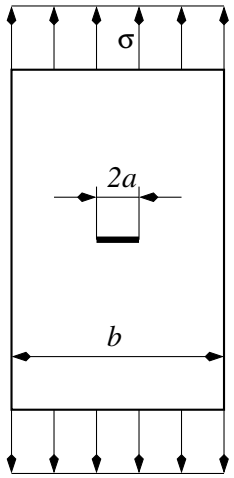


Figure 1: Through-thickness crack.

Note that for this problem, tabulated solutions for the mode-I SIF $K_{\rm I}$ are available in the literature. For example, an analytical solution given by W.D. Pilkey (*Formulas for Stress, Strain, and Structural Matrices*) is

$$K_{\rm I} = C\,\sigma\sqrt{\pi a}\ ,$$

where

$$\begin{aligned} C &= (1 - 0.1\,\eta^2 + 0.96\,\eta^4)\sqrt{1/\cos(\pi\eta)}\ , \\ \eta &= \frac{a}{b}\ . \end{aligned}$$

Use of this solution yields $K_{\rm I} = 25.680$ MPa$\cdot\sqrt{\rm m}$.

2 Assumptions and Approach

2.1 Assumptions

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

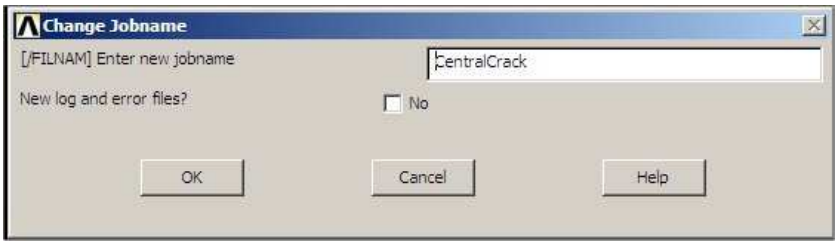
2.2 Approach

- 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 the symmetry of the problem, only a quarter model is analyzed.
- The crack-tip region is meshed using quarter-point (singular) 8-node quadrilateral elements (PLANE82).

3 Preprocessing

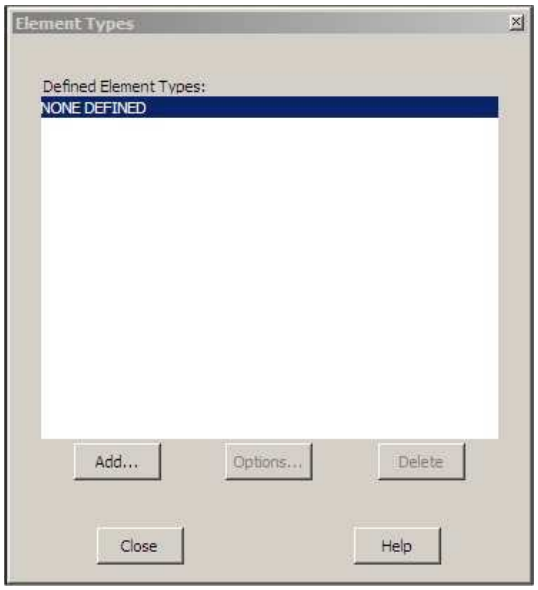
1. Give the Job a Name
   Utility Menu > File > Change Jobname ...

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

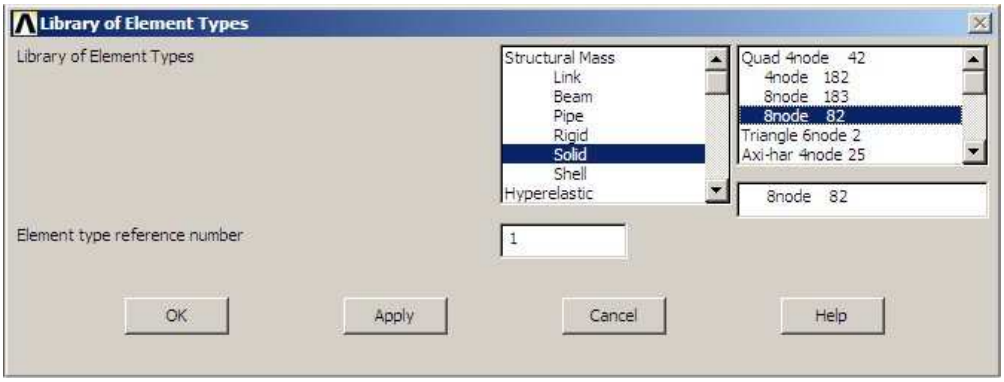


2. Define Element Type
   Main Menu > Preprocessor > Element Type > Add/Edit/Delete

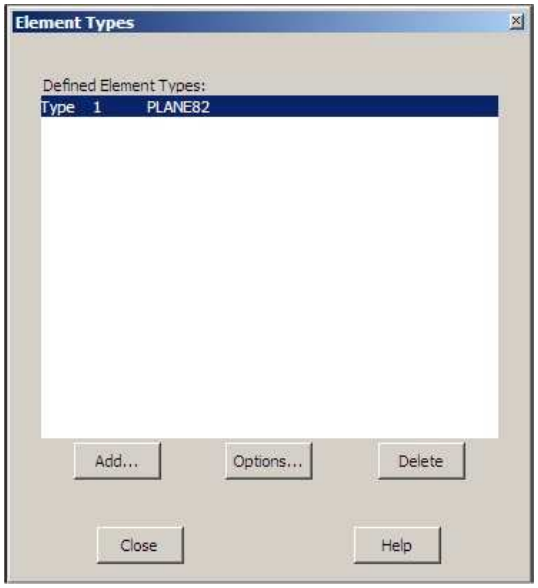
   - This brings up the 'Element Types' window. Click on the **Add...** button.



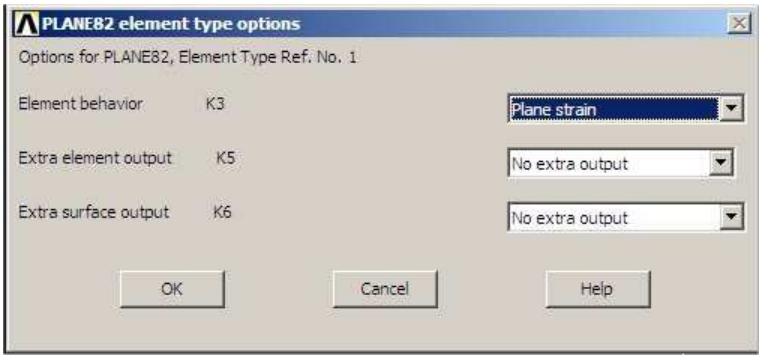
   - The 'Library of Element Types' window appears. Highlight '`Solid`', and '`8node 82`', as shown. Click on **OK**.



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



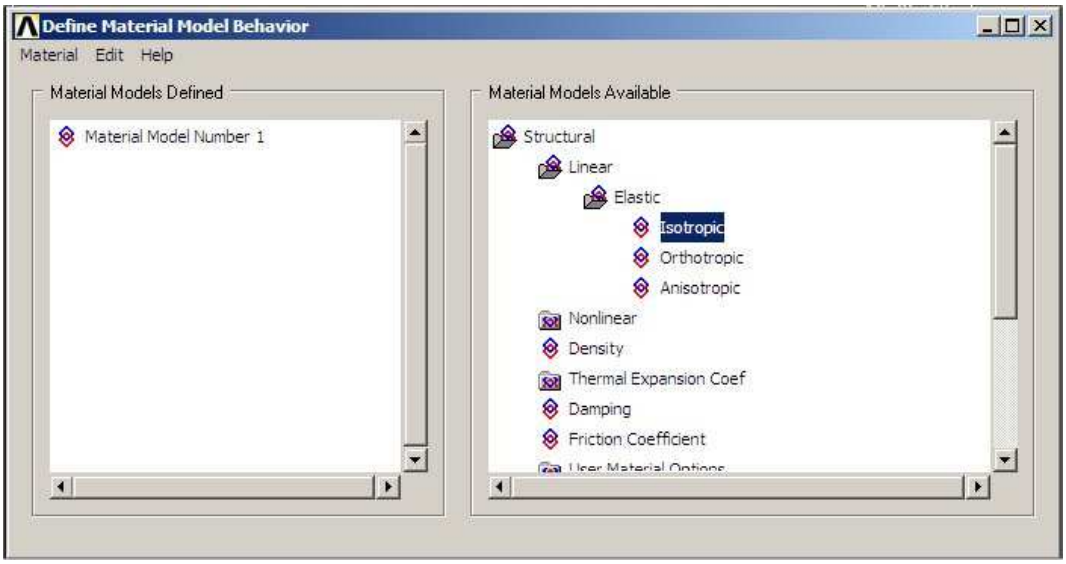
- Click on the **Options...** button in the above window. The below window comes up. Select '`Plane strain`' for '`Element behavior K3`' and click **OK**.



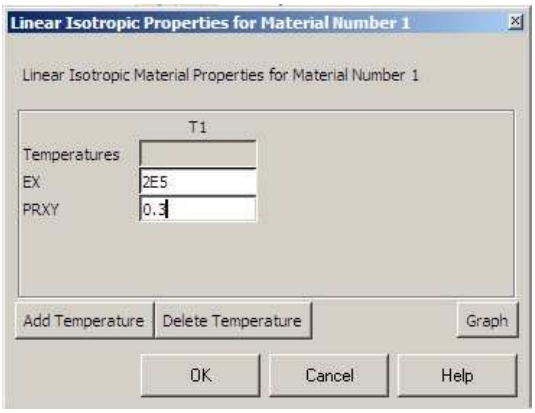
- Click on the **Close** button in the 'Element Types' window.

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.



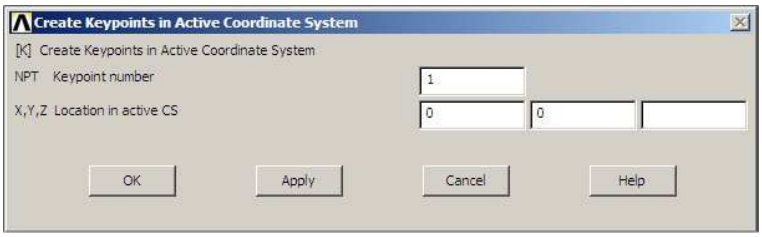
- Click **OK**, then close the 'Define Material Model Behavior' window.

4. Define Keypoints
   Main Menu > Preprocessor > Modeling > Create > Keypoints > In Active CS

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

Keypoint #	X	Y
1	0	0
2	0.02	0
3	0.1	0
4	0.1	0.1
5	0	0.1

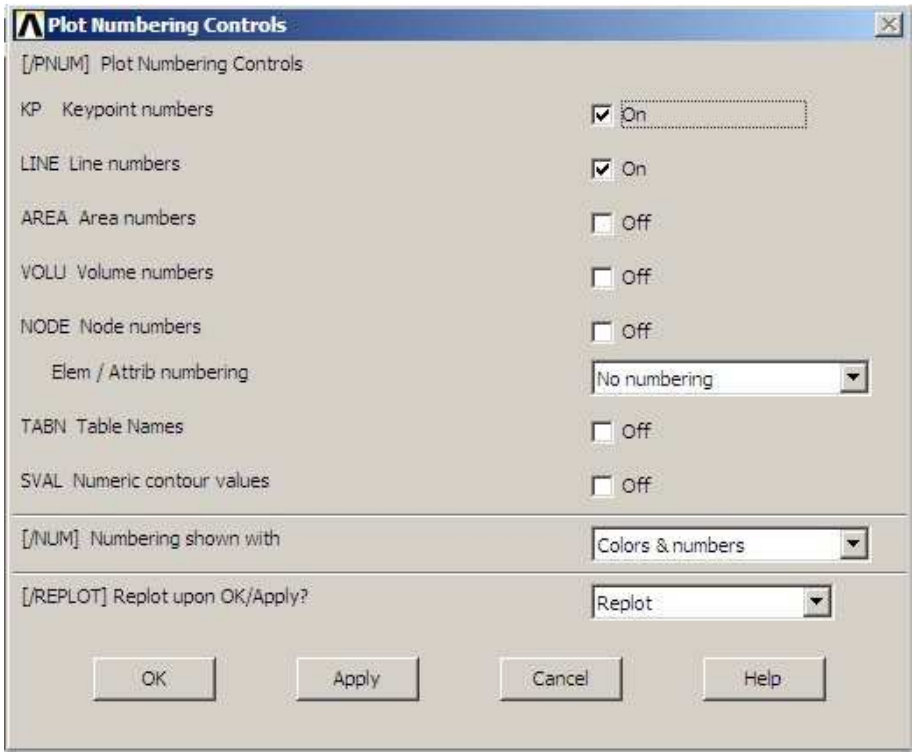
- To create keypoint #1, enter '`1`' as keypoint number, and '`0`' and '`0`' as the X and Y coordinates in the following window. Click on **Apply**.



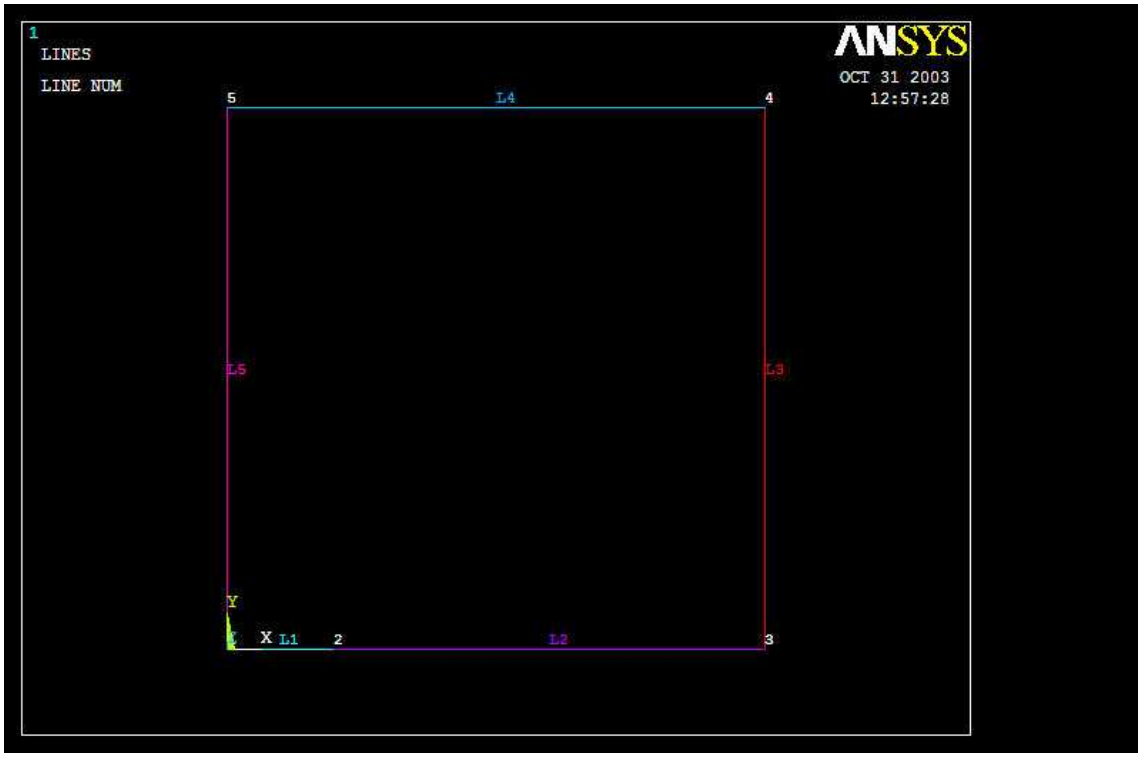
- Repeat the above step for keypoints #2 through #5. Note that you must click on **OK** instead of **Apply** after entering data of the final keypoint.

5. Define Line Segments
   Main Menu > Preprocessor > Modeling > Create > Lines > Lines > In Active Coord

   - Pick keypoint #1 then keypoint #2 to create a line connecting them (line #1).
   - Repeat the previous step to create lines connecting keypoints #2 and #3 (line #2), keypoints #3 and #4 (line #3), keypoints #4 and #5 (line #4), and keypoints #5 and #1 (line #5).
   - Click on **OK** to close the 'Lines in Active Coord' window (picking window).
   - Turn on the numbering by selecting **Utility Menu > PlotCtrls > Numbering ...**. The below window appears. Check the boxes for '`Keypoint numbers`' and '`Line numbers`' as shown, then click on **OK**.

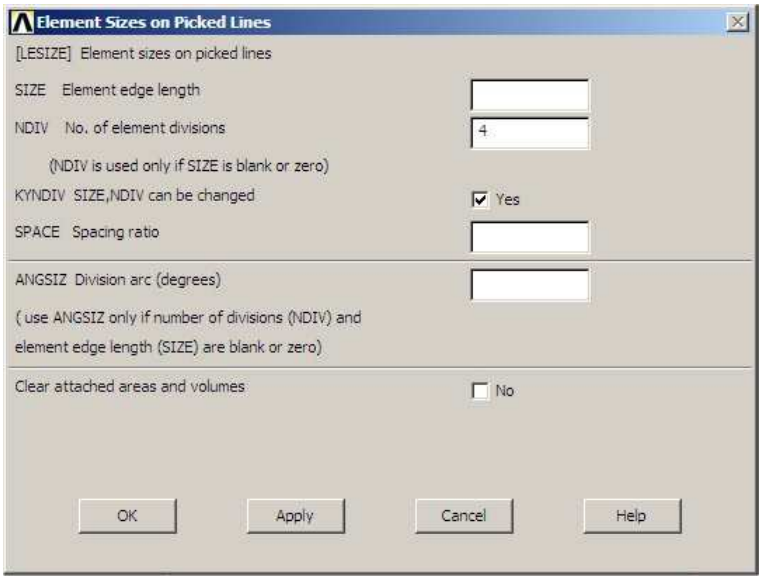


- Select **Utility Menu > Plot > Lines**. Your graphics window should look like this,

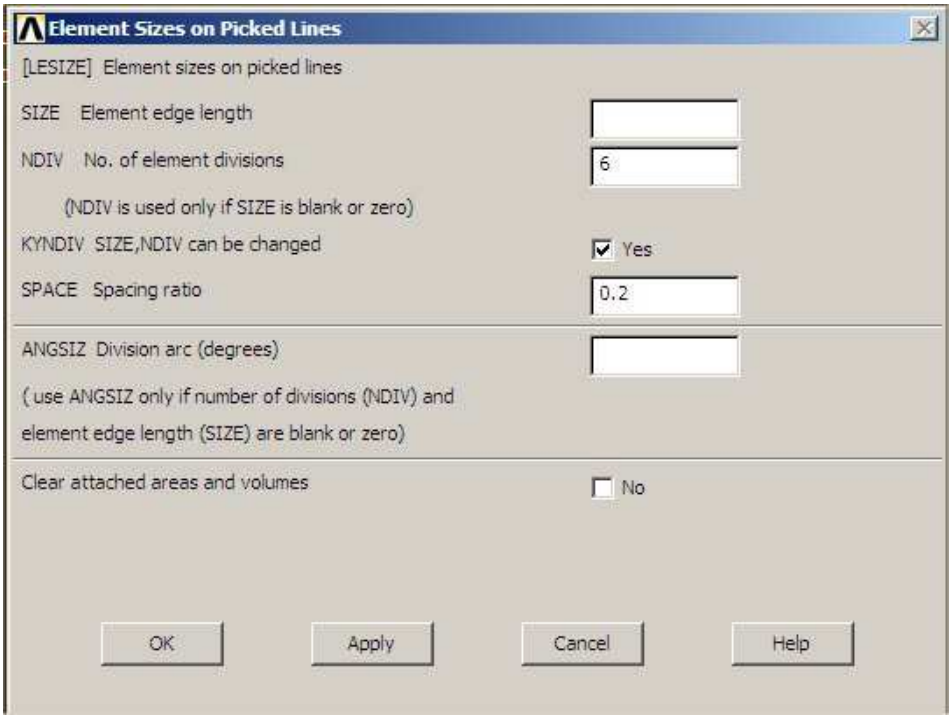


6. Discretize Lines L3, L4 and L5
   Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > Picked Lines

   - Pick lines #3 and #4. Click on the **OK** button in the picking window.
   - The below window opens. Enter '4' for '`No. of element divisions`', then click **Apply**.

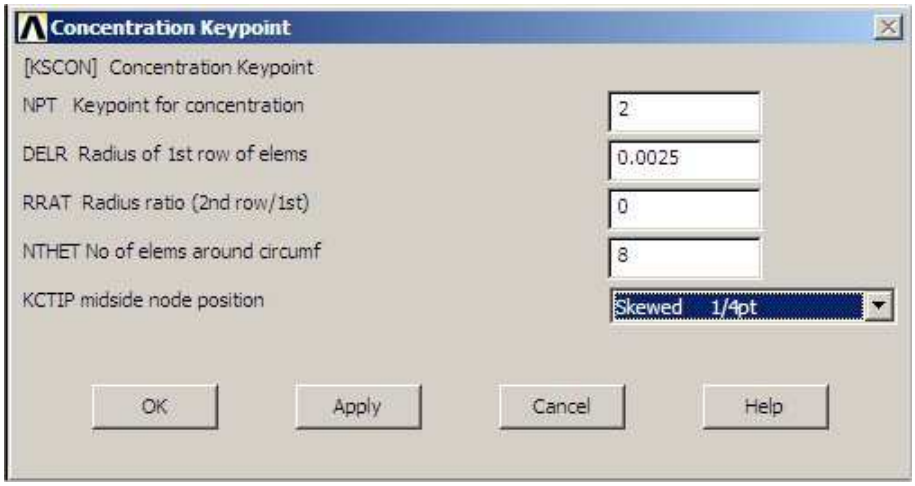


- Pick line #5, then click **OK** in the picking window.
- In the below window that comes up again, enter '6' for '`No. of element divisions`', and '`0.2`' for '`Spacing ratio`', then click **OK**.



7. Create the Concentration Keypoint (Crack Tip)
   Main Menu > Preprocessor > Meshing > Size Cntrls > Concentrat KPs > Create

   - Pick keypoint #2, then click **OK** in the picking window.
   - In the below window that appears, you should see '2' as '`Keypoint for concentration`'. Enter '`0.0025`' ($= a/8$) for '`Radius of 1st row of elems`', input '8' 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

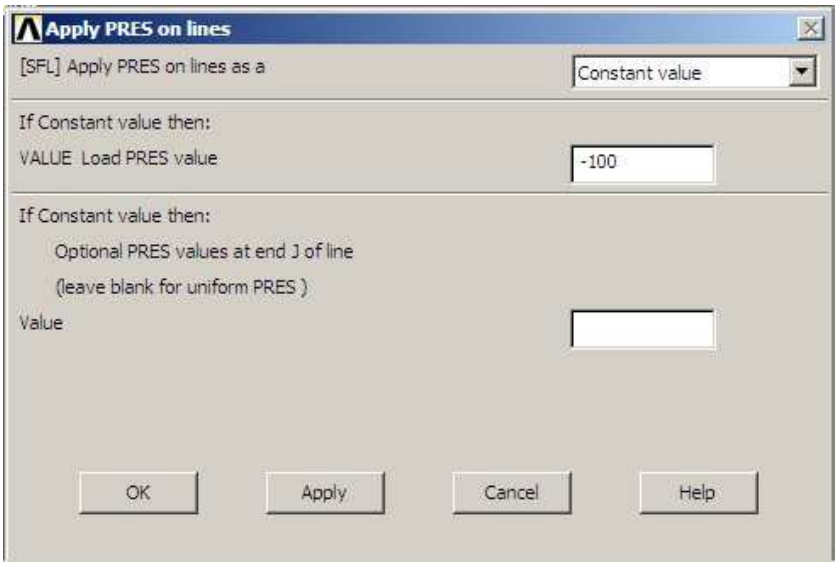
   - Pick all five lines (L1 through L5). Click **OK** in the picking window.

9. Apply Boundary Conditions
Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > Symmetry B.C. > ...with Area

   - Pick line #2. Click **Apply** (in the picking window). Pick the area. Click **Apply**.
   - Pick line #5. Click **Apply**. Pick the area. Click **OK**.

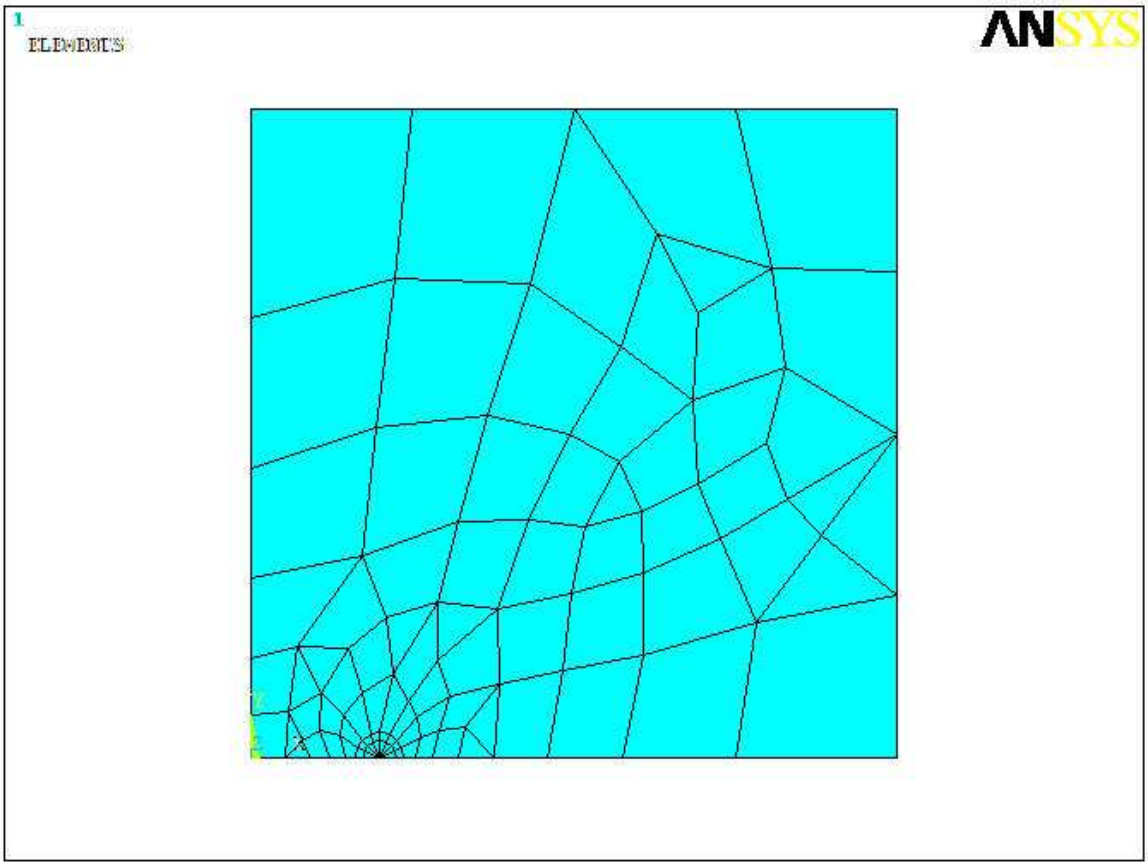
10. Apply Loads
Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Pressure > On Lines

    - Pick line #4. Click **OK** in the picking window.
    - In the below window that comes up, select '`Constant value`' for '`Apply PRES on lines as a`', enter '`-100`' for '`Load PRES value`', then click **OK**.



11. Mesh the Model
Main Menu > Preprocessor > Meshing > Mesh > Areas > Free

    - Pick the area. Click **OK** in the picking window.
    - Close the 'Warning' window. In your ANSYS window, a mesh as shown at the top of next page should appear.



4 Processing (Solving)

Main Menu > Solution > Analysis Type > New Analysis

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

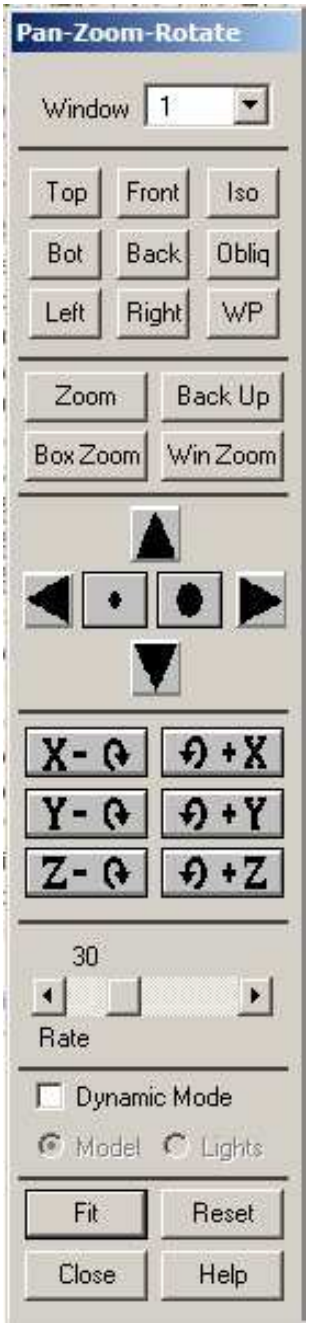
Main Menu > Solution > Solve > Current LS

- 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.

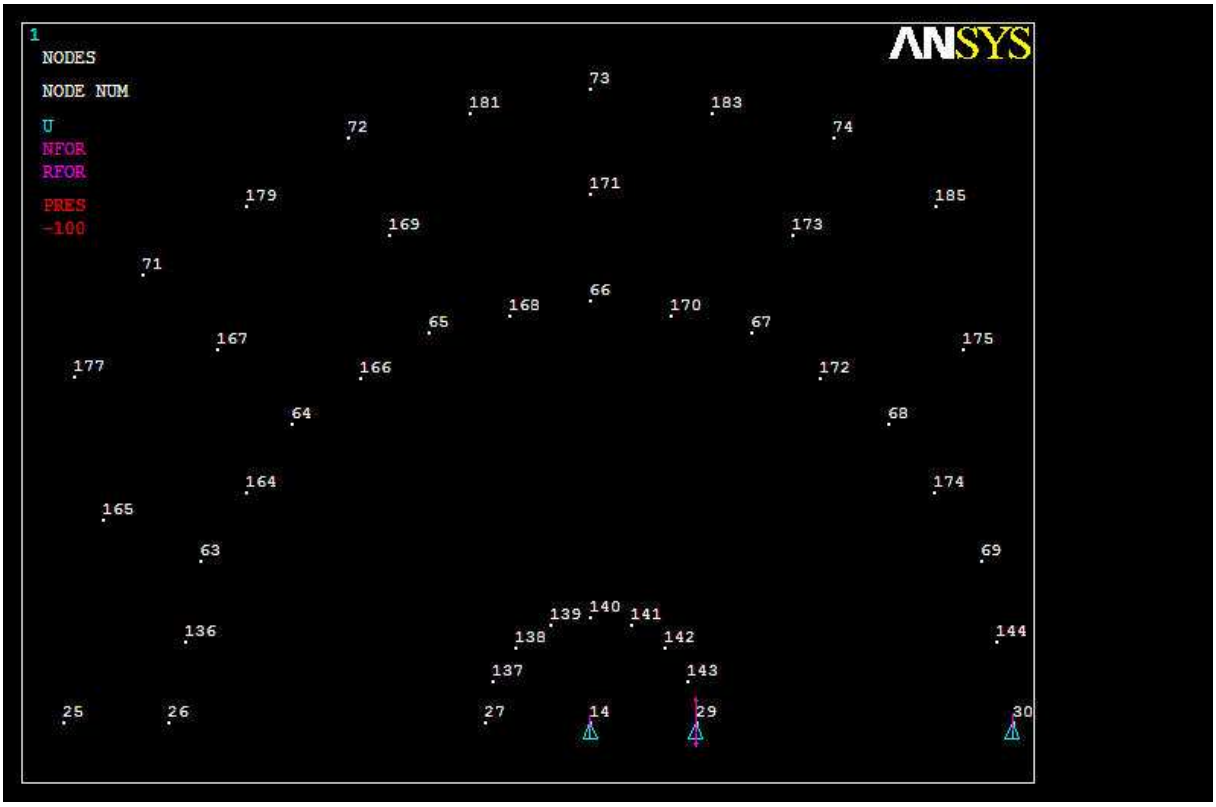
5 Postprocessing

1. Zoom the Crack-Tip Region
   Utility Menu > PlotCtrls > Pan Zoom Rotate ...

   This brings up the following window:

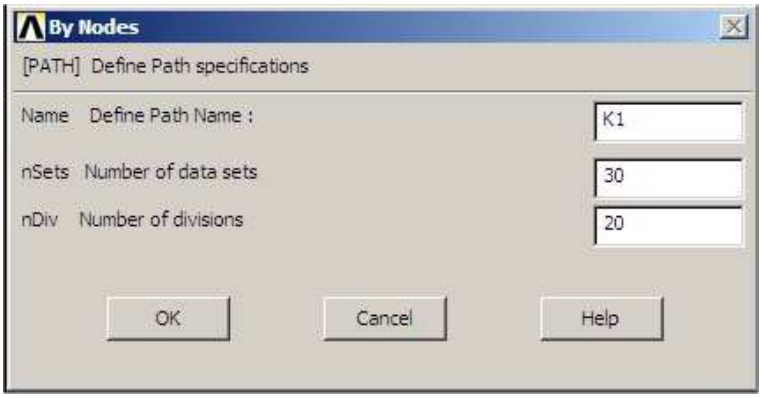


- In the above window, click on the **Win Zoom** button and zoom the crack-tip region, then click on the **Close** button to close the window.
- Plot the nodes by selecting **Utility Menu > Plot > Nodes**.
- Turn on the node numbering by selecting **Utility Menu > PlotCtrls > Numbering ...**, then check 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

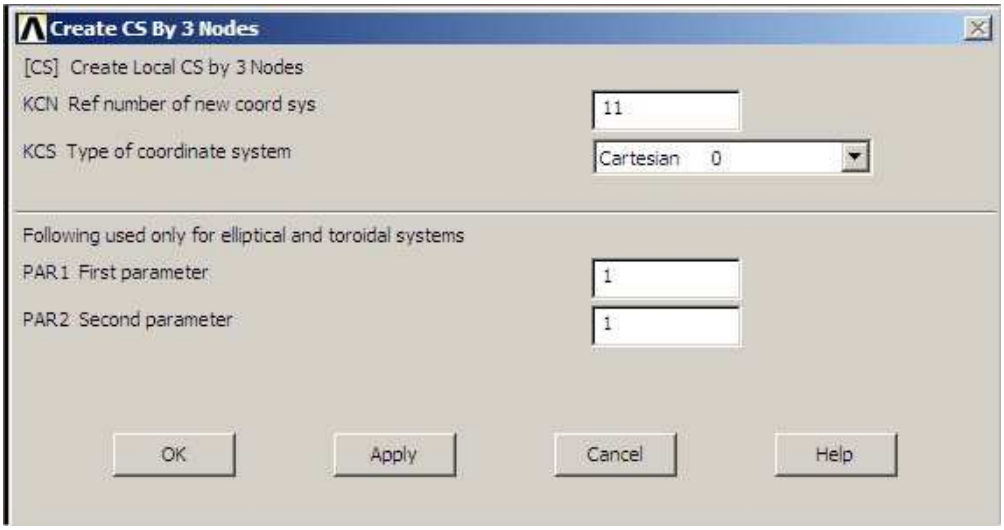
   - Pick the crack-tip node (node #14), then the quarter-point node (node #27), and finally the third node (node #26) on the crack face. Click **OK**.
   - In the below window that appears, enter '`K1`' for '`Define Path Name:`', then click **OK**.



   - 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 #14 (the crack-tip node), then node #29, and finally node #140. This brings up the following window:

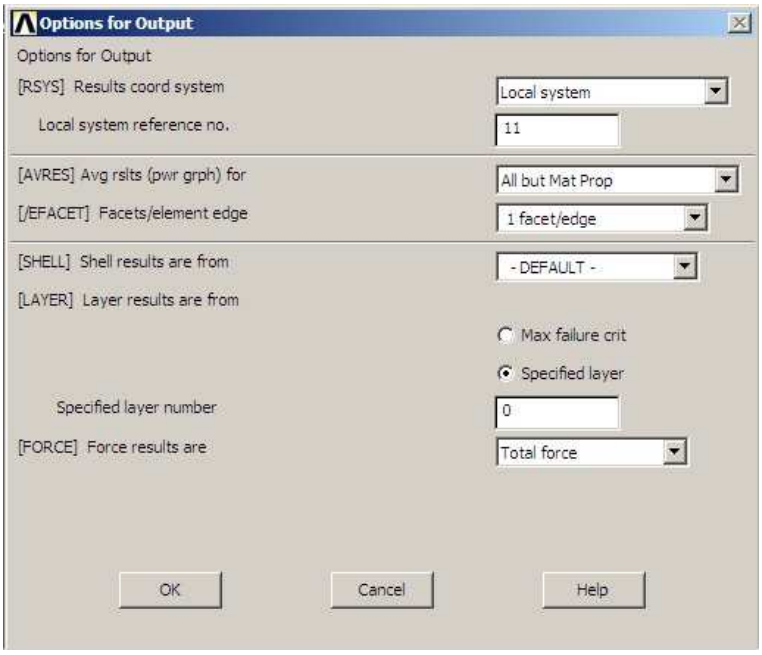


- Note from the above window that the reference number of the crack-tip coordinate system is 11. Click on the **OK** button.

4. Activate the Local Crack-Tip Coordinate System
   Utility Menu > WorkPlane > Change Active CS to > Specified Coord Sys ...

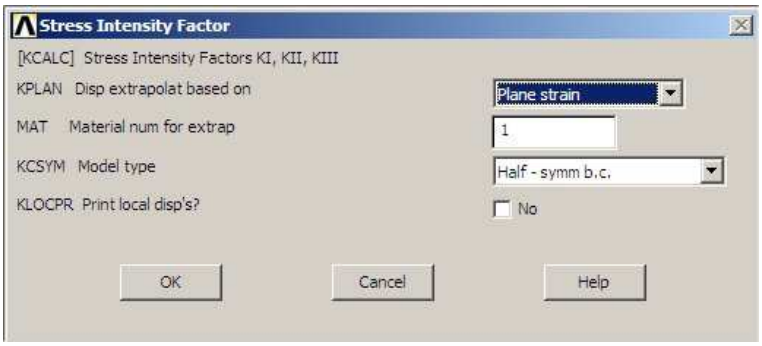
   - In the below window that comes up, enter '`11`' for '`Coordinate system number`', then click **OK**.

   - 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 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 '`Half-symm b.c.`' for '`Model Type`'.



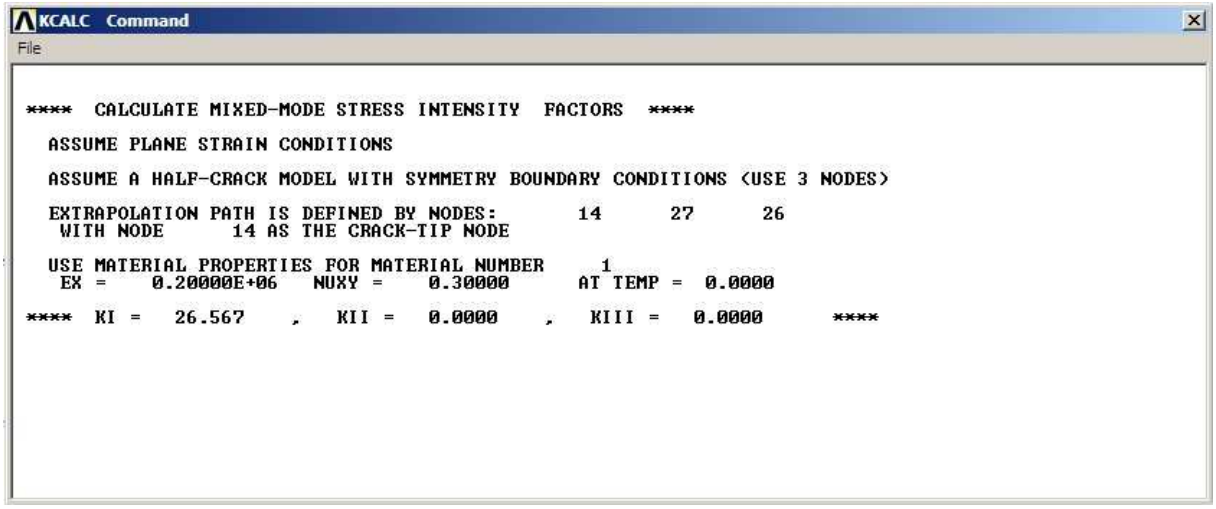
   - Click on **OK**. The window shown at the top of next page appears and it shows that the SIFs at the crack tip (node #4) are

$$K_{\rm I} = 26.567 \; ; \quad K_{\rm II} = 0 \; ; \quad K_{\rm III} = 0$$

   Note that the results $K_{\rm II} = 0$ and $K_{\rm III} = 0$ are obvious for this problem. The ANSYS solution for $K_{\rm I}$ (25.567 MPa$\cdot\sqrt{\rm m}$) is in very good agreement with that obtained from W.D. Pilkey (25.680 MPa$\cdot\sqrt{\rm m}$). The discrepancy is

$$\epsilon = \frac{K_{\rm I}^{\rm Pilkey} - K_{\rm I}^{\rm ANSYS}}{K_{\rm I}^{\rm Pilkey}} = \frac{25.680 - 25.567}{25.680}$$

$$= 0.44\ \% \qquad (1)$$



- Close the 'KCALC Command' window.
- You may want to recover the whole meshed model by
  - selecting **Utility Menu > PlotCtrls > Pan Zoom Rotate ...**, then click on the **Fit** button and close the 'Pan-Zoom-Rotate' window;
  - selecting **Utility Menu > Plot > Elements**.

6. Exit ANSYS, Saving All Data
   Utility Menu > File > Exit ...

   In the window that opens, select '`Save Everything`' and click on **OK**.