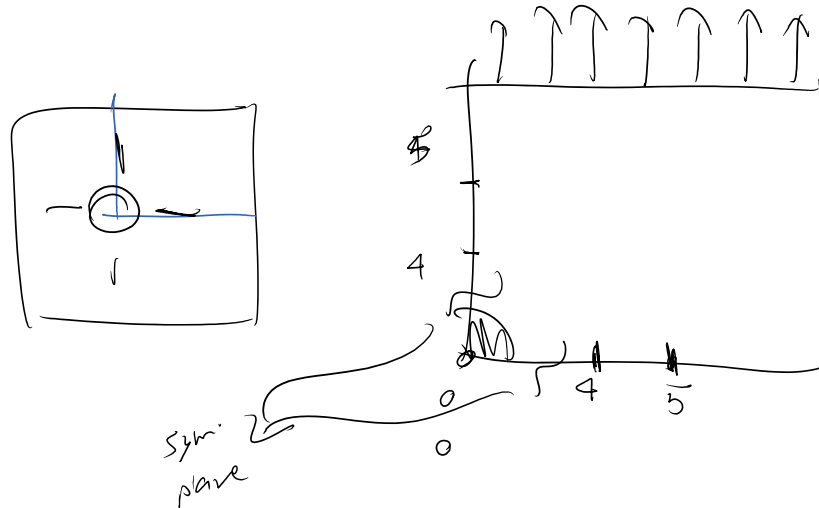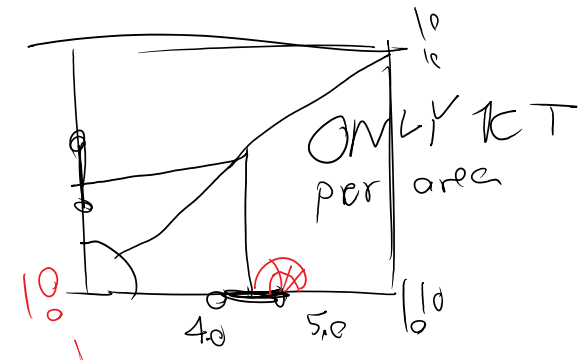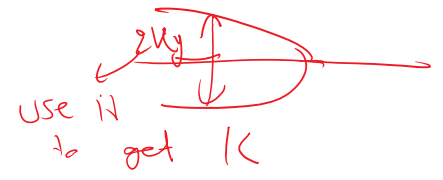Run Ansys by choosing:

- ANSYS 18.2
  - ANSYS Icepak 18.2
  - ANSYS Viewer 18.2
  - Mechanical APDL 18.2
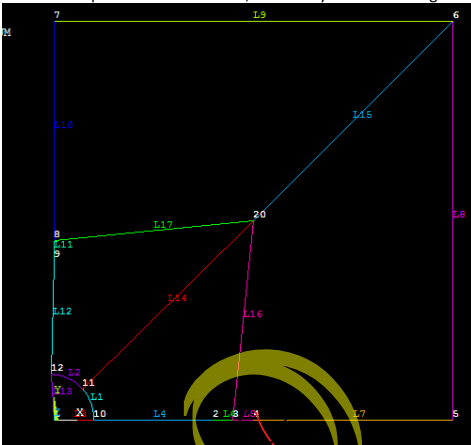 - **Mechanical APDL Product Launcher**
  - SCDM 18.2

Sym. plane

- Create geometry, material, element types
- Create crack-tip specific spider web
- Solve

Postprocessing

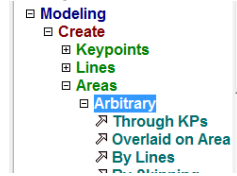→ K from displacement

→ J (EDI)

ONLY 1CT per area

use in to get K

Each crack tip must be in one area, that's why we broke the geometry as follows:
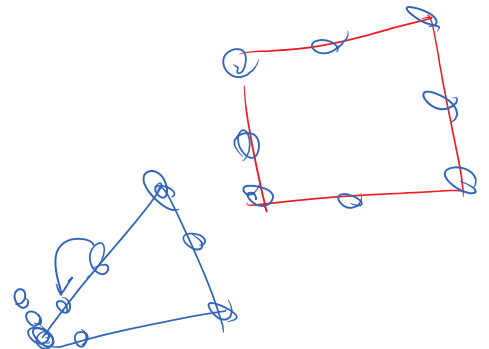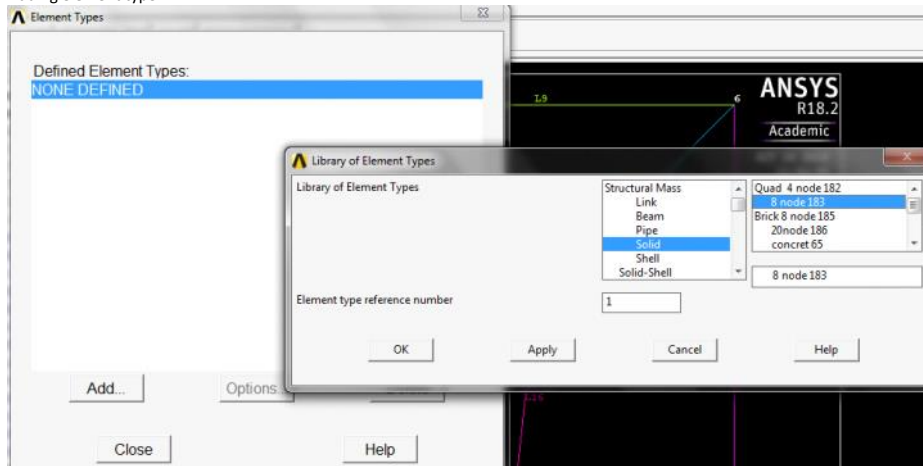

tell Ansys that this is a crack tip & want to mesh it as

tell ANJ—

& want to mesh it as ...

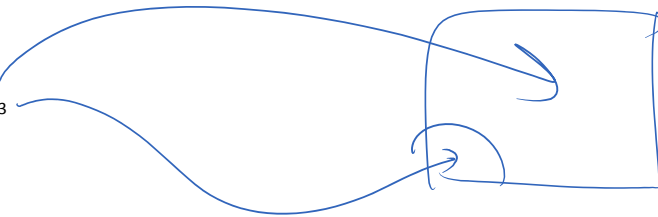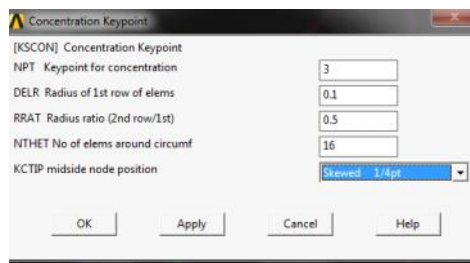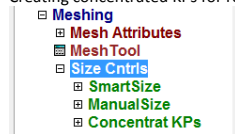Creating the areas (that will be meshed by FEs later)

- ⊟ **Modeling**
  - ⊟ **Create**
    - ⊞ **Keypoints**
    - ⊞ **Lines**
    - ⊟ **Areas**
      - ⊟ **Arbitrary**
        - ⬀ **Through KPs**
        - ⬀ **Overlaid on Area**
        - ⬀ **By Lines**
        - ⬀ **By Skipping**

Adding element type

Element Types

Defined Element Types:
NONE DEFINED

Add...   Options...   Close   Help

Library of Element Types

Library of Element Types

Structural Mass	Quad 4 node 182
Link	8 node 183
Beam	Brick 8 node 185
Pipe	20node 186
Solid	concret 65
Shell	
Solid-Shell	8 node 183

Element type reference number   1

OK   Apply   Cancel   Help

ANSYS
R18.2
Academic

Define materials

1. E = 100, nu = 0.3
2. E = 10000, nu = 0.3

Creating concentrated KPs for rozet meshes around CTs

- ⊟ **Meshing**
  - ⊞ **Mesh Attributes**
  - ▦ **Mesh Tool**
  - ⊟ **Size Cntrls**
    - ⊞ **SmartSize**
    - ⊞ **ManualSize**
    - ⊞ **Concentrat KPs**

Concentration Keypoint

[KSCON] Concentration Keypoint

NPT Keypoint for concentration	3
DELR Radius of 1st row of elems	0.1
RRAT Radius ratio (2nd row/1st)	0.5
NTHET No of elems around circumf	16
KCTIP midside node position	Skewed 1/4pt

OK   Apply   Cancel   Help

Assigning materials to different regions

Define materials

1. E = 100, nu = 0.3
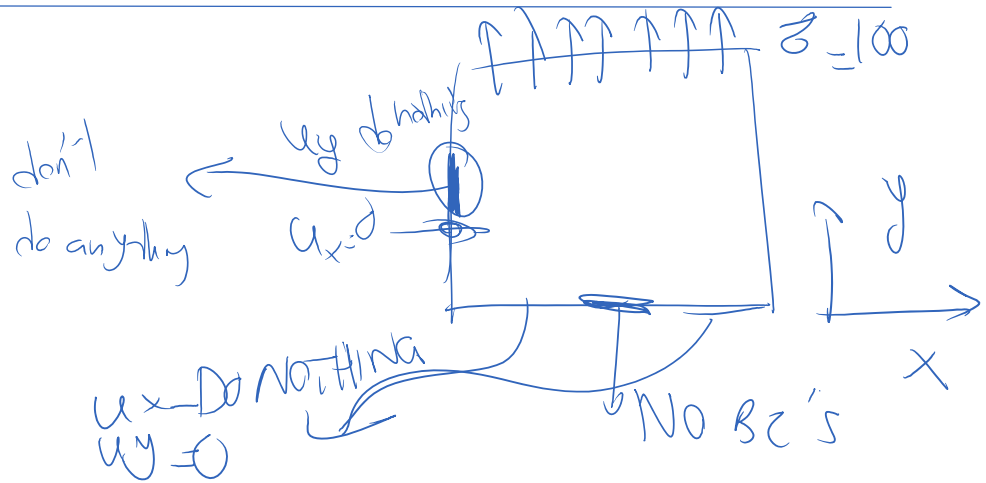2. E = 10000, nu = 0.3
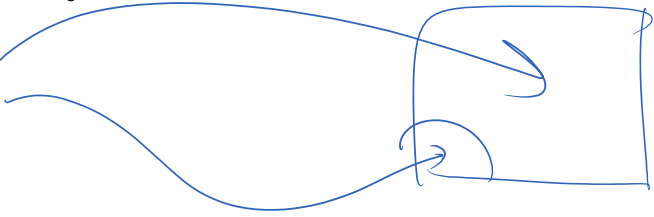
Assign materials by:

□ Meshing
  □ Mesh Attributes
    ▦ Default Attribs
    ▦ All Keypoints
    ↗ Picked KPs
    ▦ All Lines
    ↗ Picked Lines
    ▦ All Areas
   ↗ Picked Areas

---

Better to do everything before the actual meshing

□ Preprocessor
  ⊞ Element Type
  ⊞ Real Constants
  ⊞ Material Props
  ⊞ Sections
  ⊞ Modeling
  ⊞ Meshing
  ⊞ Checking Ctrls
  ⊞ Numbering Ctrls
  ⊞ Archive Model
  ⊞ Coupling / Ceqn
  ⊞ Multi-field Set Up
  □ Loads
    ⊞ Analysis Type
    □ Define Loads
      ⊞ Settings
      ⊞ Apply
      ⊞ Delete
      ⊞ Operate
      ▦ Domain Decomp

don't do anything

uy dof nothing
ux = 0

$\mathbb{Z} = -100$

ux → DO NOTHING
uy = 0

NO BC's

Displacement BCs needed for bottom and left lines:
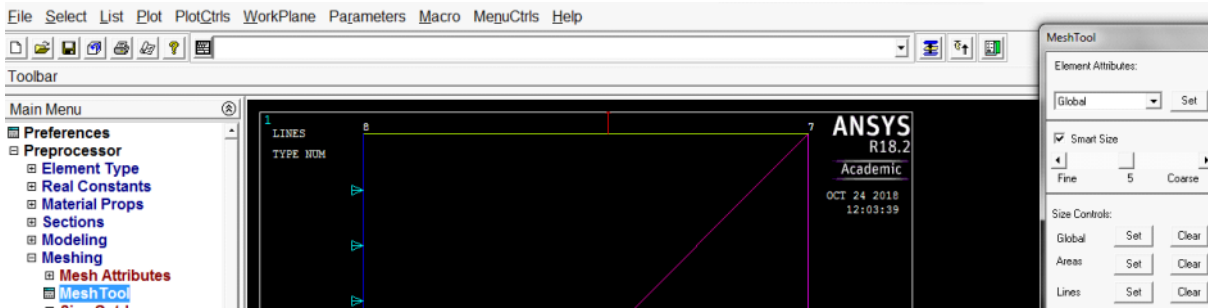
□ Loads
  ⊞ Analysis Type
  □ Define Loads
    ⊞ Settings
    □ Apply
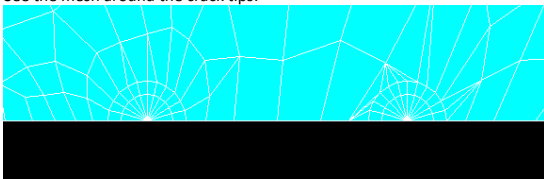   □ Structural
        □ Displacement
          ↗ On Lines
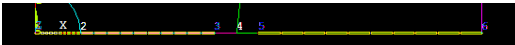
Load on the top surface

□ Multi-field Set Up
□ Loads
  ⊞ Analysis Type
  □ Define Loads
    ⊞ Settings
    □ Apply
      □ Structural
        ⊞ Displacement
        ⊞ Force/Moment
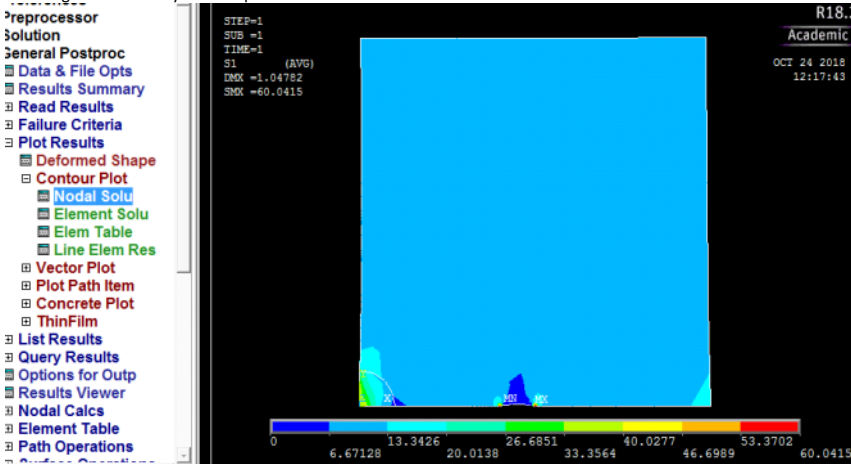   □ Pressure
          ↗ On Lines

Meshing the domain:

See the mesh around the crack tips:

Solution of the problem:

Checking the solution by contour plots

Now that we have the solution, we want to calculate K from displacements behind the crack tip:

crack face

r

θ
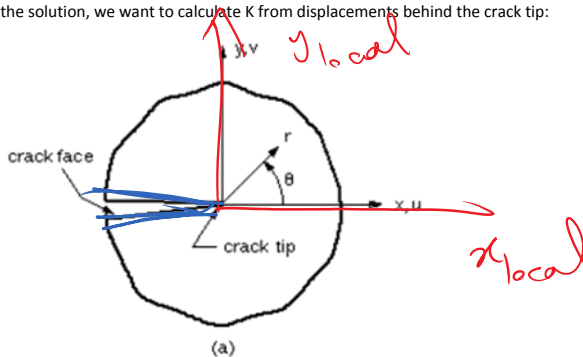
x, u

crack tip

y_{local}

x_{local}

(a)

Figure 2) Crack coordinate systems for 2-D model

a. Defining a local coordinate system

Defining the coordinate system by nodes:
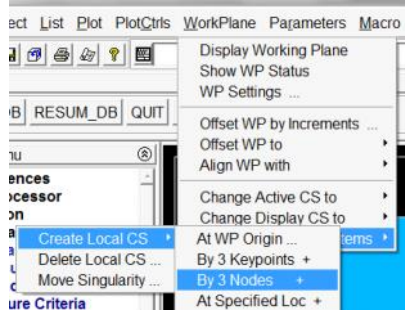
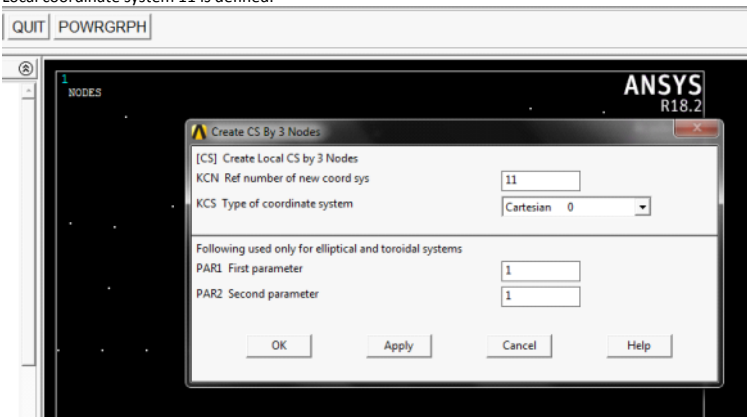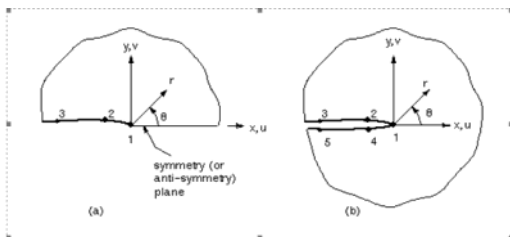Local coordinate system is defined as:
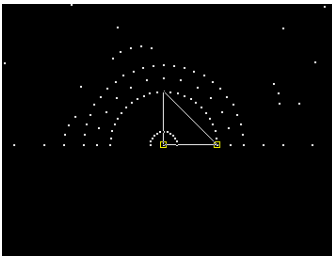
Local coordinate system 11 is defined:

Change the coordinate system to local coordinate system we defined:
csys,11

Changing the coordinate system where the results are provided at:
rsys,11

b. We deine a path on which the calculation of SIF is done

We do this process by defining a path:

Defining a path by nodes:
- **General Postproc**
  - Data & File Opts
  - Results Summary
  - Read Results
  - Failure Criteria
  - Plot Results
  - List Results
  - Query Results
  - Options for Outp
  - Results Viewer
  - Nodal Calcs
  - Element Table
  - Path Operations
    - Define Path
      - By Nodes

Choose 3 or 5 points needed, name it something you'll remember
- **General Postproc**
  - Data & File Opts
  - Results Summary
  - Read Results
  - Failure Criteria
  - Plot Results
  - List Results
  - Query Results
  - Options for Outp
  - Results Viewer
  - Nodal Calcs
  - Element Table
  - Path Operations
    - Define Path
      - By Nodes

Check if the path is there:

To calculate SIF:

The logic behind calculation of SIF:

1. K from local fields

1. Displacement

$$u_y(r, \theta = \pi) = \frac{4K_I\sqrt{r}}{\sqrt{2\pi}E'} \Rightarrow K_I = \lim_{r \to 0}\left[\frac{E'\,u_y}{4}\sqrt{\frac{2\pi}{r}}\right] \quad (\theta = \pi) \qquad E' = \begin{cases} E & \text{plane stress} \\ \frac{E}{1-\nu^2} & \text{plane strain} \end{cases}$$

or alternatively from the first quarter point element:

$$K_I = \frac{2G}{\kappa+1}\sqrt{\frac{2\pi}{L}}\left(-3\bar{v}'_A + 4\bar{v}'_B - \bar{v}'_C\right)$$

Mixed mode generalization:

$$\begin{Bmatrix} K_I \\ K_{II} \end{Bmatrix} = \frac{1}{2}\frac{2G}{\kappa+1}\sqrt{\frac{2\pi}{L}}\begin{bmatrix} 0 & 1 \\ 1 & 0 \end{bmatrix}\begin{bmatrix} -3\bar{v}'_A + 4\left(\bar{u}'_B - \bar{u}'_D\right) - \left(\bar{u}'_C - \bar{u}'_E\right) \\ -3\bar{v}'_A + 4\left(\bar{v}'_B - \bar{v}'_D\right) - \left(\bar{v}'_C - \bar{v}'_E\right) \end{bmatrix}$$

275

⊞ Preprocessor
⊞ Solution
⊟ General Postproc
   ▤ Data & File Opts
   ▤ Results Summary
   ⊞ Read Results
   ⊞ Failure Criteria
   ⊞ Plot Results
   ⊞ List Results
   ⊞ Query Results
   ▤ Options for Outp
   ▤ Results Viewer
   ⊟ Nodal Calcs
      ▤ Total Force Sum
      ▤ Sum @ Each Node
      ⊞ Summation Pt
      ▤ Stress Int Factr

Stress Intensity Factor

[KCALC] Stress Intensity Factors KI, KII, KIII

KPLAN   Disp extrapolat based on   Plane strain ▼

MAT   Material num for extrap   1

KCSYM   Model type   Half - symm b.c. ▼

KLOCPR Print local disp's?   ☐ No

OK   Cancel   Help

Stress Intensity Factor

[KCALC] Stress Intensity Factors KI, KII, KIII

KPLAN   Disp extrapolat based on   Plane stress ▼

MAT   Material num for extrap   1

KCSYM   Model type   Half - symm b.c. ▼

                                        Half - symm b.c.
                                        Half - asym b.c.
KLOCPR Print local disp's?   Full-crack model

OK   Cancel   Help

```
****  CALCULATE MIXED-MODE STRESS INTENSITY  FACTORS  ****

ASSUME PLANE STRESS CONDITIONS

ASSUME A HALF-CRACK MODEL WITH SYMMETRY BOUNDARY CONDITIONS (USE 3 NODES)

EXTRAPOLATION PATH IS DEFINED BY NODES:     444     453     452
WITH NODE     444 AS THE CRACK-TIP NODE

USE MATERIAL PROPERTIES FOR MATERIAL NUMBER     1
  EX  =    100.00      NUXY =    0.30000      AT TEMP =   0.0000
****  KI =   9.3391   ,   KII =   0.0000   ,   KIII =   0.0000     ****
```


$$K = \delta \sqrt{\pi a}$$

2nd method of calculating SIF from J

Select List Plot PlotCtrls WorkPlane Parameters Macro M
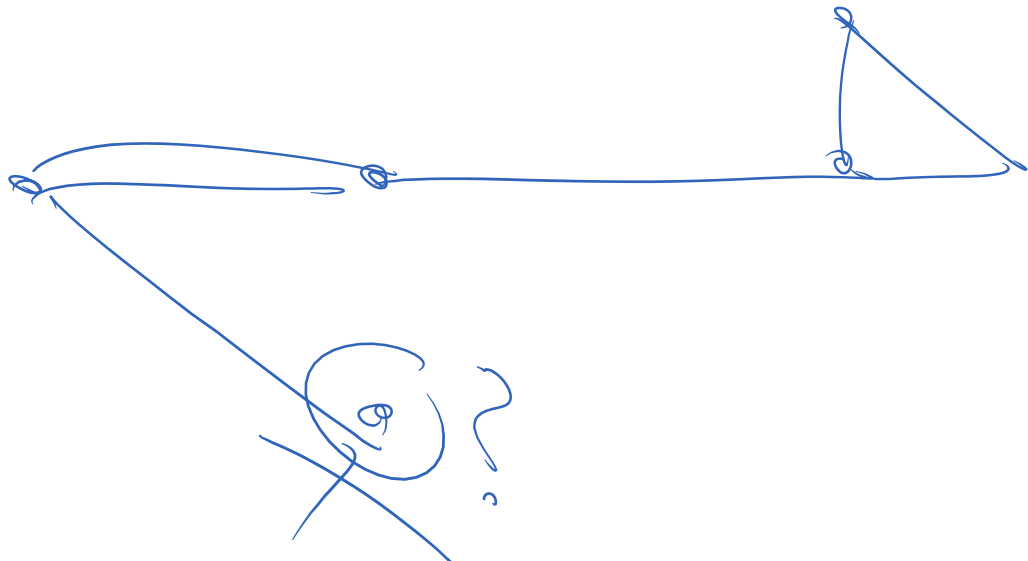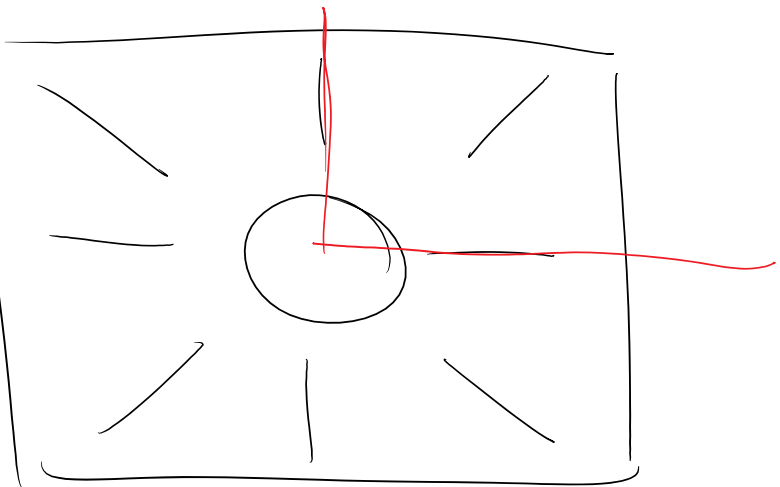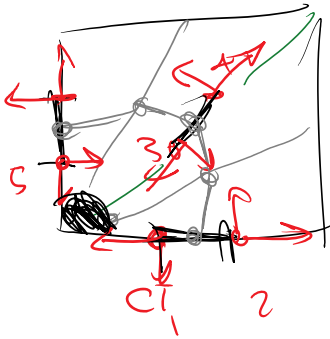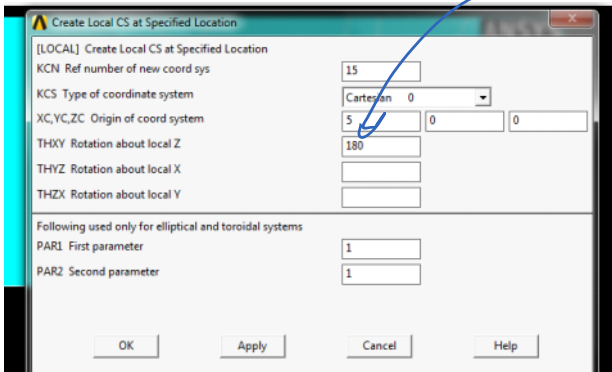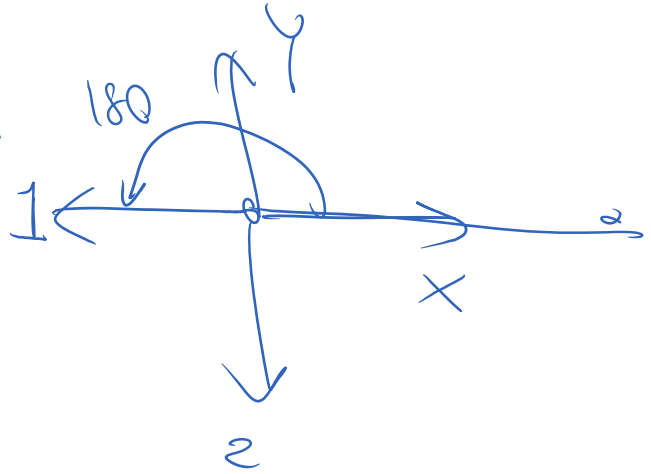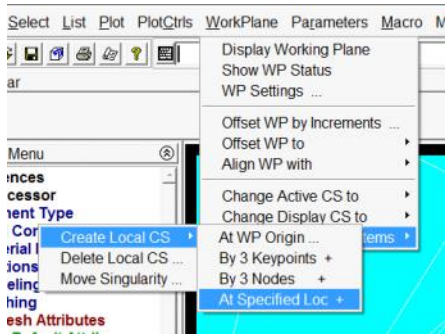
Computation of J-integral with comments and images included for some steps:
The J-Integral evaluation in ANSYS is based on the domain integral method. The domain integration formulation applies area integration for 2-D problems and volume integration for 3-D problems. In the following, the procedure to compute the J-integral is summarized.

It should be noted that the command syntax is all in UPPER CASE letters and the arguments which are entered by the user are in *lower case and italic*.

After creating the model (using keypoints, lines and areas), specifying the concentration keypoints to generate singular elements, defining the local coordinate for each crack tip and generating the mesh including singular elements around the crack tips, the following commands except the last one need to be issued in the command prompt in the utility menu at the preprocessor stage of the simulation. The last command (i.e., Step 8) is issued after solution and in the postprocessor stage. Unfortunately there is no way to apply these steps in GUI. It means these commands cannot be accessed from a menu.

1. Start the process by clicking on preprocessor option on the left side menu.
2. Start a new computation with the contour-based integral approach with the command:

    CINT,NEW,*id*

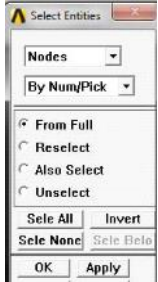   where "*id*" is an integer being specified to a crack tip

3. Specify the type of contour integral to be done (here it is the J-integral) with the command:
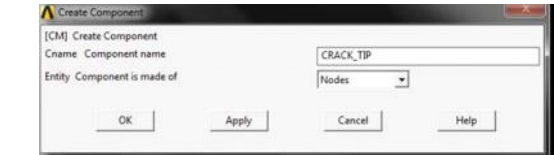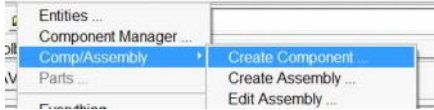
    CINT,TYPE,JINT

4. Define the node at the crack tip as a node component with the command:
   Note: to create component (corresponding to the crack tip), use select from menu and select the node, and create a component from that:

   Select   List   Plot   PlotCtrls
   Entities ...

   Select Entities
   Nodes ▾
   By Num/Pick ▾
   ⊙ From Full
   ○ Reselect
   ○ Also Select
   ○ Unselect
   Sele All   Invert
   Sele None   Sele Belo
   OK   Apply

   Select   List   Plot   PlotCtrls   WorkPlane   Parameters   Macro
   Entities ...
   Component Manager ...
   Comp/Assembly   ▸   Create Component ...
   Parts ...   Create Assembly ...
                            Edit Assembly ...

   Create Component
   [CM] Create Component
   Cname Component name   CRACK_TIP
   Entity Component is made of   Nodes ▾
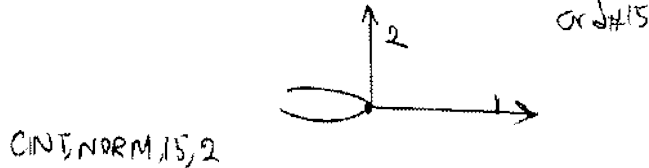   OK   Apply   Cancel   Help

   CINT,CTNC,*name*

   where "*name*" is the name of the crack tip node component name
   so this becomes
   CINT,CTNC,CRACK_TIP

   Make sure after this you select entities, nodes, all nodes.

5. Define the crack plane normal along with its local coordinate id with the command:

    CINT,NORM,*par1*,*par2*

   or ↓#15

   CINT,NORM,15,2

   where "*par1*" is a local coordinate system identifier and "*par2*" is the axis of the coordinate system normal to the crack face

   first create a local coordinate frame:

   ect   List   Plot   PlotCtrls   WorkPlane   Parameters   Macro
                                    Display Working Plane
                                    Show WP Status
                                    WP Settings ...
   B RESUM_DB QUIT
                                    Offset WP by Increments ...
                                    Offset WP to   ▸
   u   Align WP with   ▸
   ences
   cessor   Change Active CS to   ▸
   ent Type   Change Display CS to   ▸
          Create Local CS   ▸   At WP Origin ...   tems ▸
          Delete Local CS ...   By 3 Keypoints +
          Move Singularity ...   By 3 Nodes   +
   hing   At Specified Loc +
   ooh Attributoo
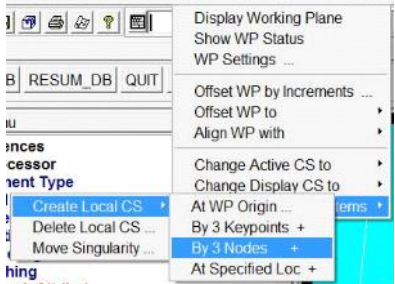
For example is local coordinate system is 11 and normal to crack direction is along its 2 direction this becomes:
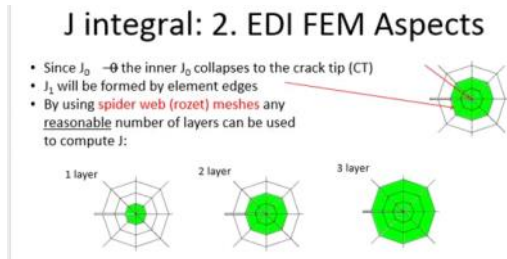CINT,NORM,*11,2*

6. Specify the number of contours "*n*" to compute with the command:

CINT,NCON,*n*

For example
CINT,NCON,5

J integral: 2. EDI FEM Aspects

- Since $J_0 \to \theta$ the inner J_0 collapses to the crack tip (CT)
- J_1 will be formed by element edges
- By using spider web (rozet) meshes any reasonable number of layers can be used to compute J:

1 layer   2 layer   3 layer

7. Activate the option for symmetry conditions, if present, with the command:

CINT,SYMM,ON

8. Specify the output controls with the command:

OUTRES,CINT

9. Solve the problem.
10. Finally, the results for the values of the J-integral may be listed or plotted in a graph for all considered contours around the crack tip with the command:
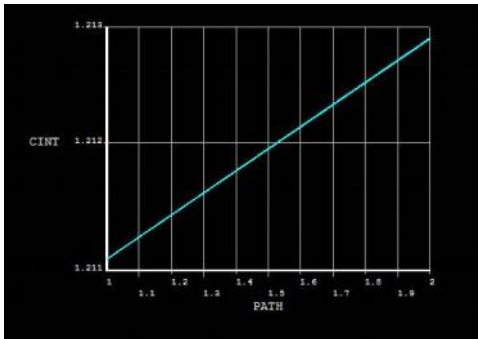
PRCINT,*id*
PLCINT,PATH,*id*

where "*id*" is the crack identifier.

For example 1 used above (id from step 2). Make sure before calling this command to go to postprocess node.

The result can be like:
and

***** POST1 J-INTEGRAL   RESULT LISTING *****

CrackID =   1
  Crack Front Node =   444
    Contour Values =   1.2113   1.2131

In order to delete a configured process for ANSYS to evaluate the J-integral or set up a new configuration for another crack tip, the following command needs to be used in the preprocessor stage.

CINT,DELE,*id*

where "*id*" is an integer being specified to a crack tip

Furthermore, one can define multiple crack tips with their own specifications (such as local coordinate, crack tip node component, number of contours and symmetry condition) in the preprocessor stage and then after the solution, the results for each crack tip "*id*" is available with the command "PRCINT,*id*". This way, only one simulation is needed to have the results for all crack tips.