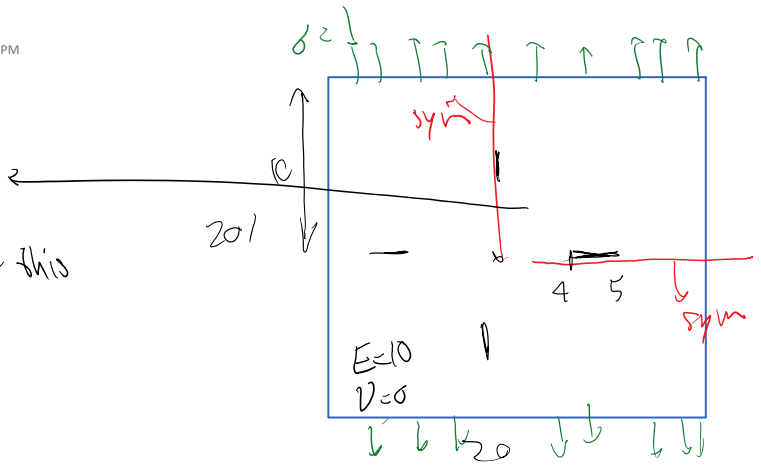
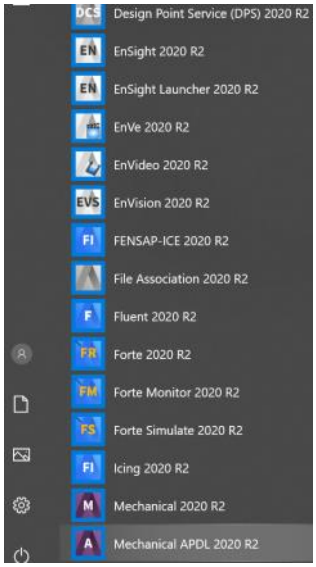
2020/10/21Ansys

Wednesday, October 21, 2020 3:15 PM

use ansys to solve this



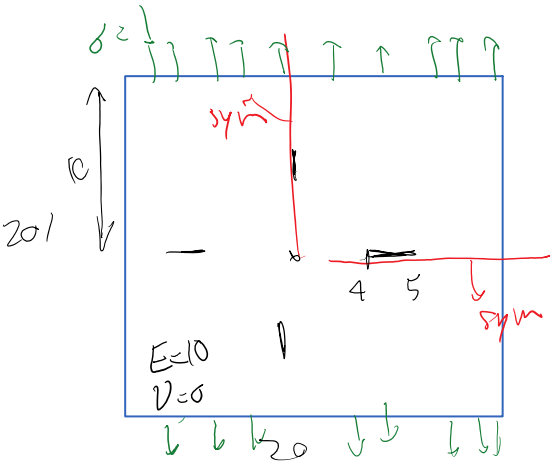


Compute K with two methods:

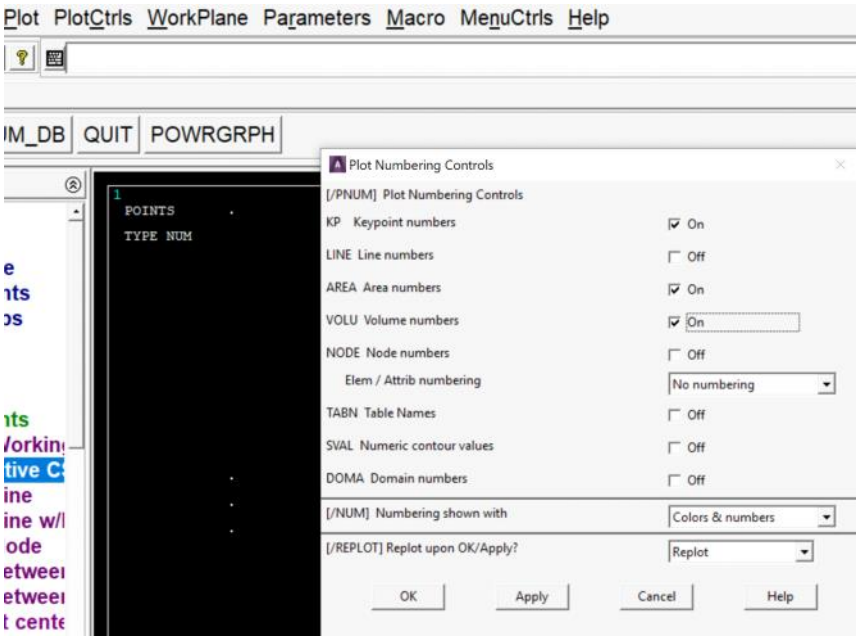
1. From displacement field
2. J integral

$$J = G = \frac{K^2}{E'} \rightarrow$$

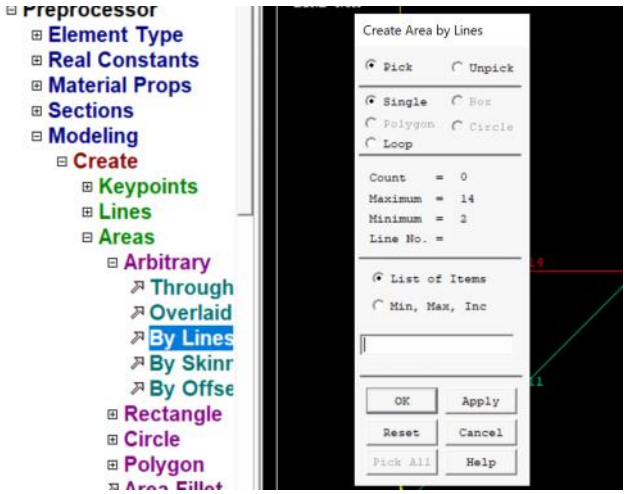
$$K = \sqrt{GE'}$$

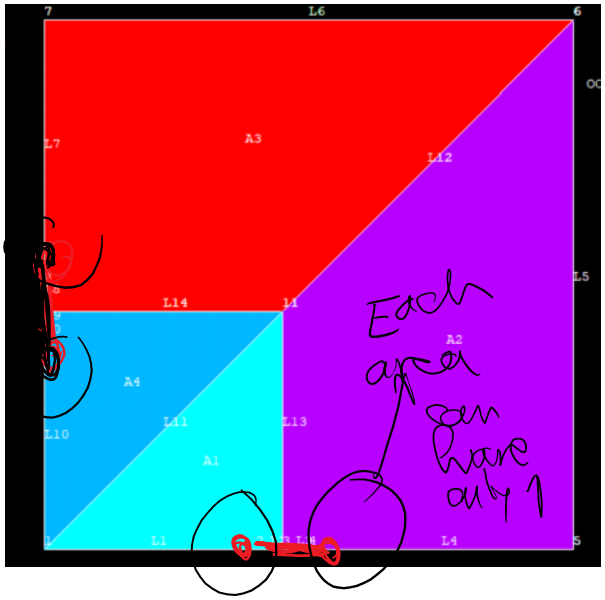



Real Constants
Material Props
Sections
Modeling
Create
Keypoints
On Workin
In Active C
On Line
Create Keypoints in Active Coordinate System
[K] Create Keypoints in Active Coordinate System
NPT Keypoint number
X,Y,Z Location in active CS
OK
Apply
Cancel
Help


Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help
IM_DB QUIT POWRGRPH
1
POINTS
TYPE NUM
Plot Numbering Controls
[/PNUM] Plot Numbering Controls
KP Keypoint numbers
On
LINE Line numbers
Off
AREA Area numbers
On
VOLU Volume numbers
On
NODE Node numbers
Off
Elem / Attrib numbering
No numbering
TABN Table Names
Off
SVAL Numeric contour values
Off
DOMA Domain numbers
Off
[/NUM] Numbering shown with
Colors & numbers
[/REPLOT] Replot upon OK/Apply?
Replot
OK
Apply
Cancel
Help


Modeling
Create
Keypoints
Lines
Lines
Straight Line


Preprocessor
Element Type
Real Constants
Material Props
Sections
Modeling
Create
Keypoints
Lines
Areas
Arbitrary
Through
Overlaid
By Lines
By Skinr
By Offse
Rectangle
Circle
Polygon
Create Area by Lines
Pick
Unpick
Single
Box
Polygon
Circle
Loop
Count = 0
Maximum = 14
Minimum = 2
Line No. =
List of Items
Min, Max, Inc
OK
Apply
Reset
Cancel
Pick All
Help



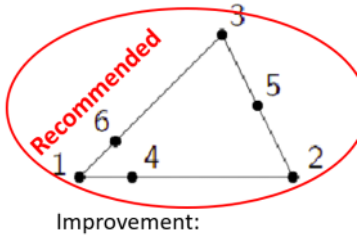
Defining element type

b) Quadratic

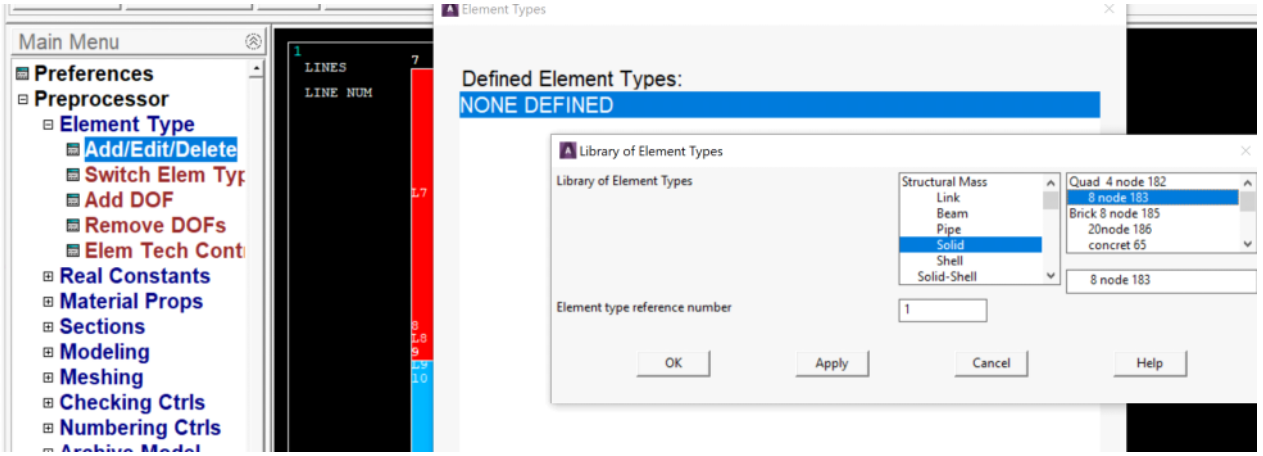
Physical space

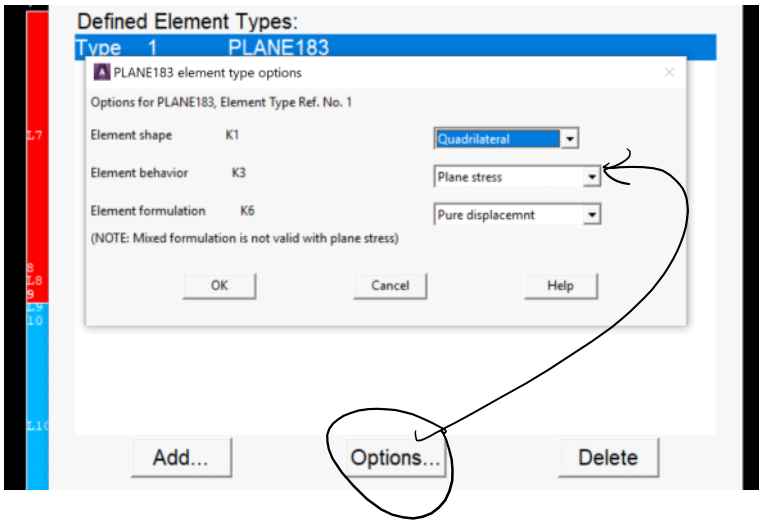
And this one around the crack tip

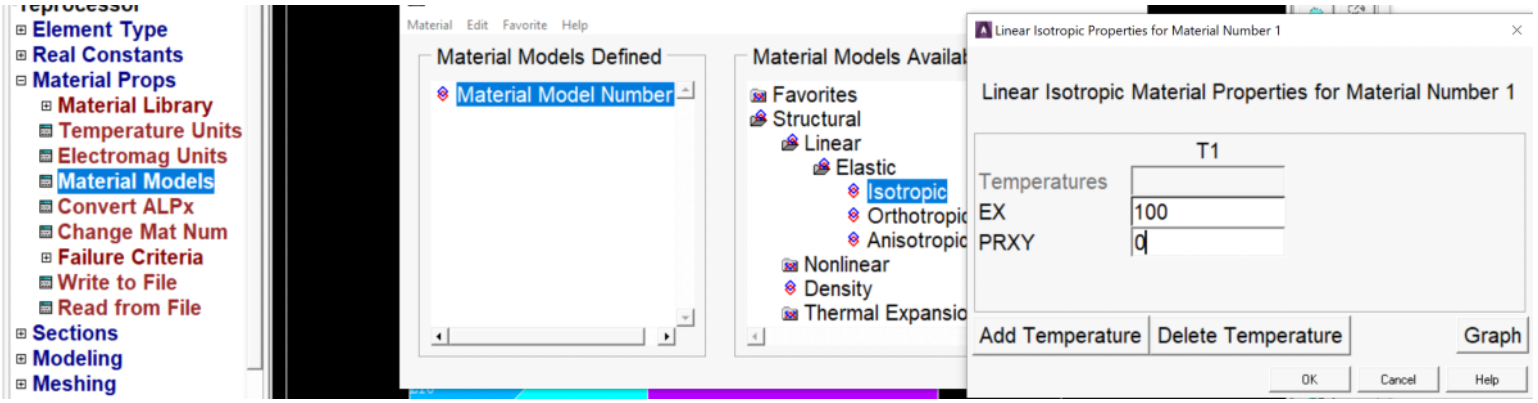
Quarter point
Tri element



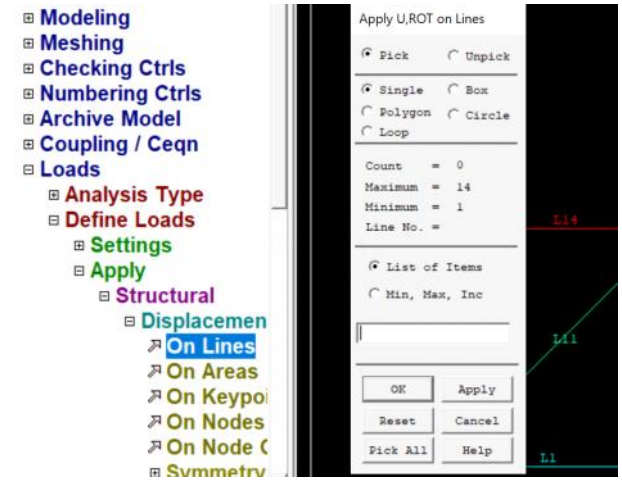
Improvement:
- Better accuracy and less mesh sensitivity

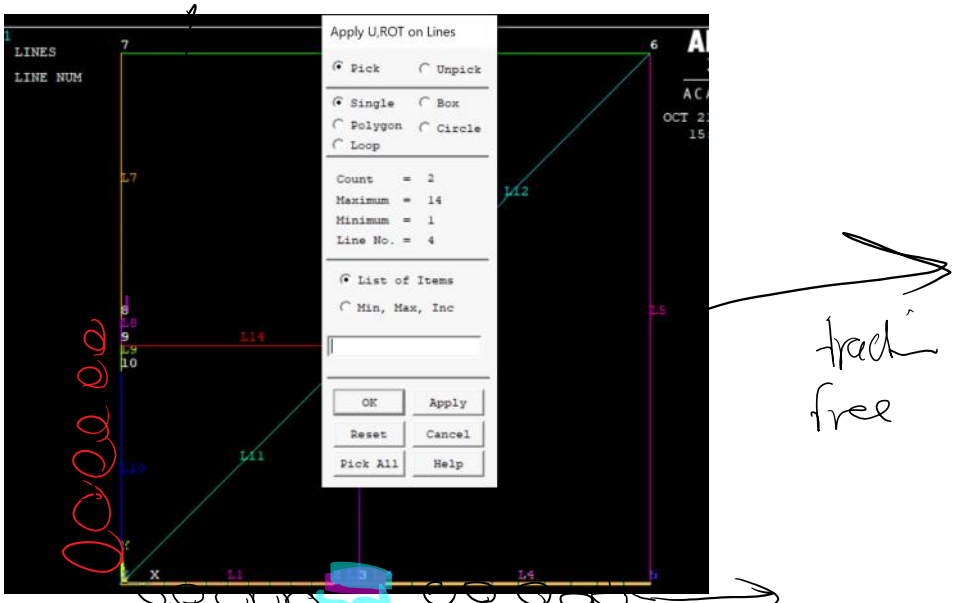


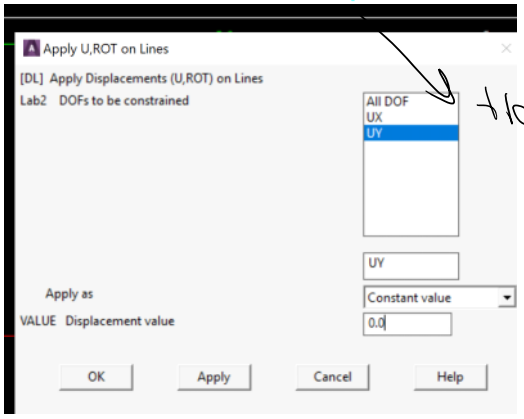




Apply the loads:

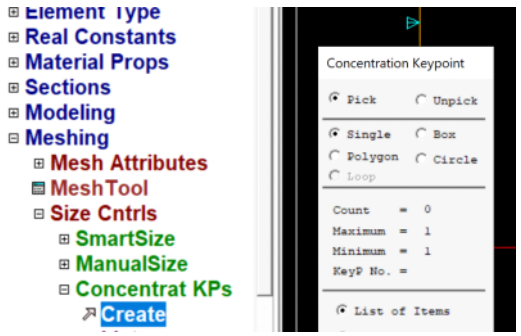


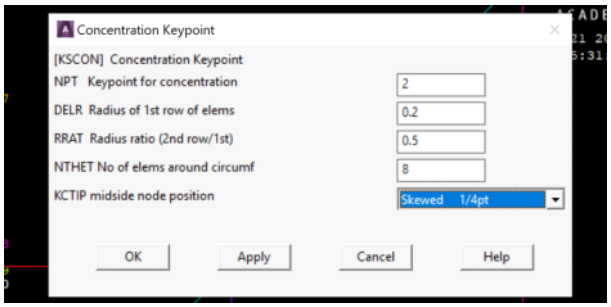




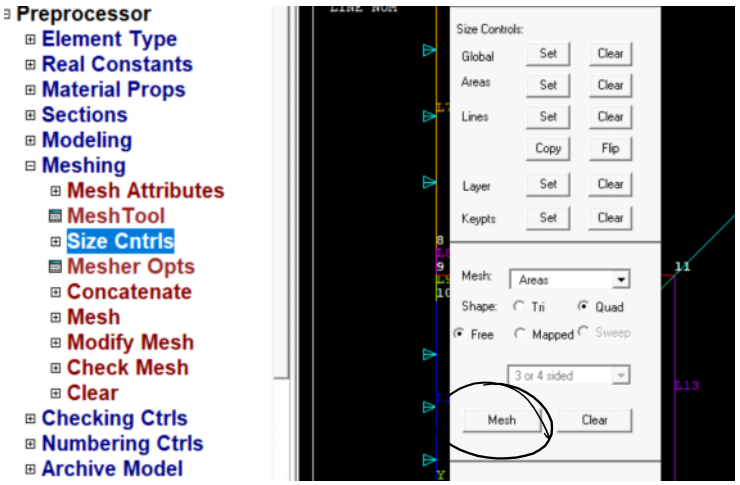
Top tensile load

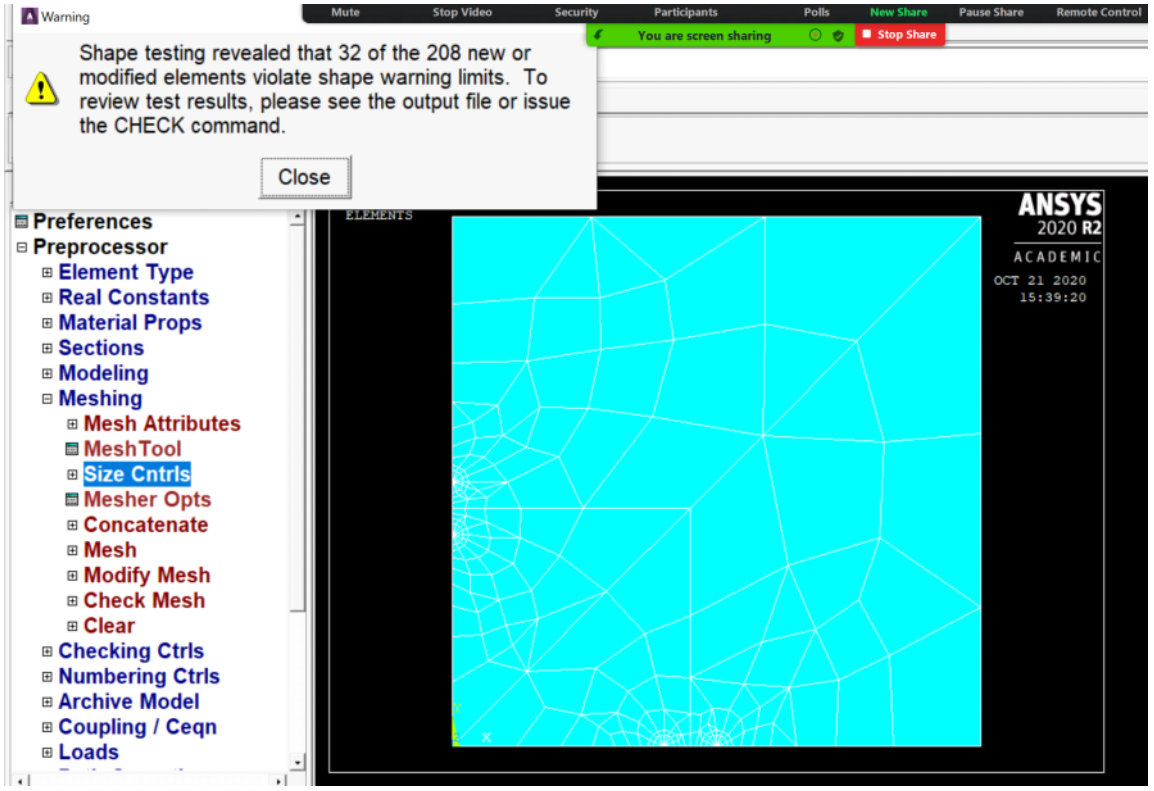
Create crack tip mesh concentration points:

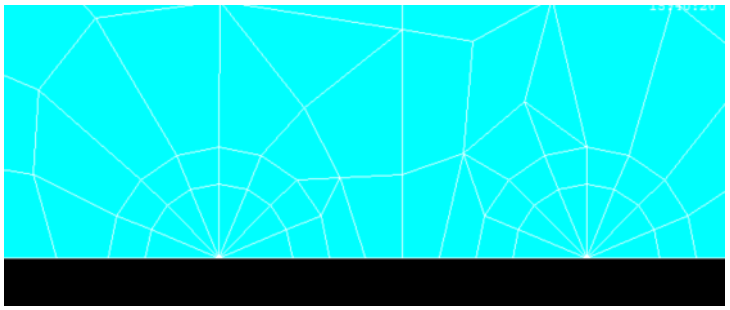




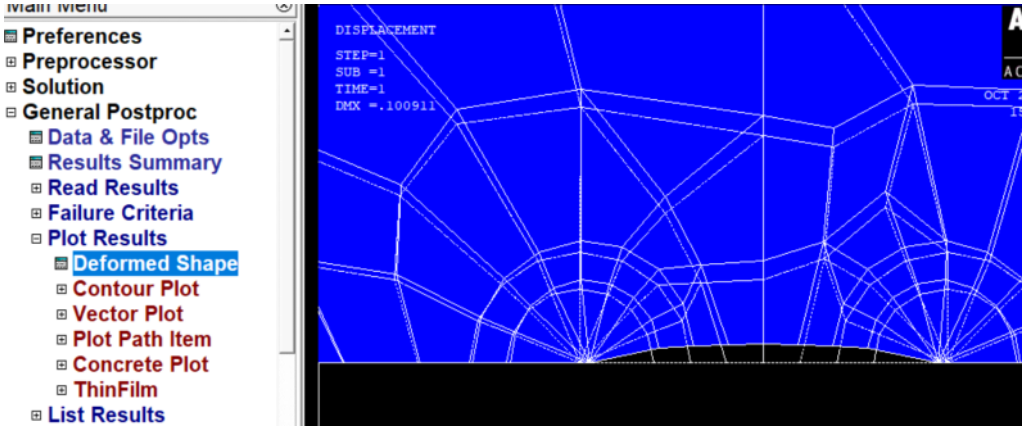
Next: We mesh this:

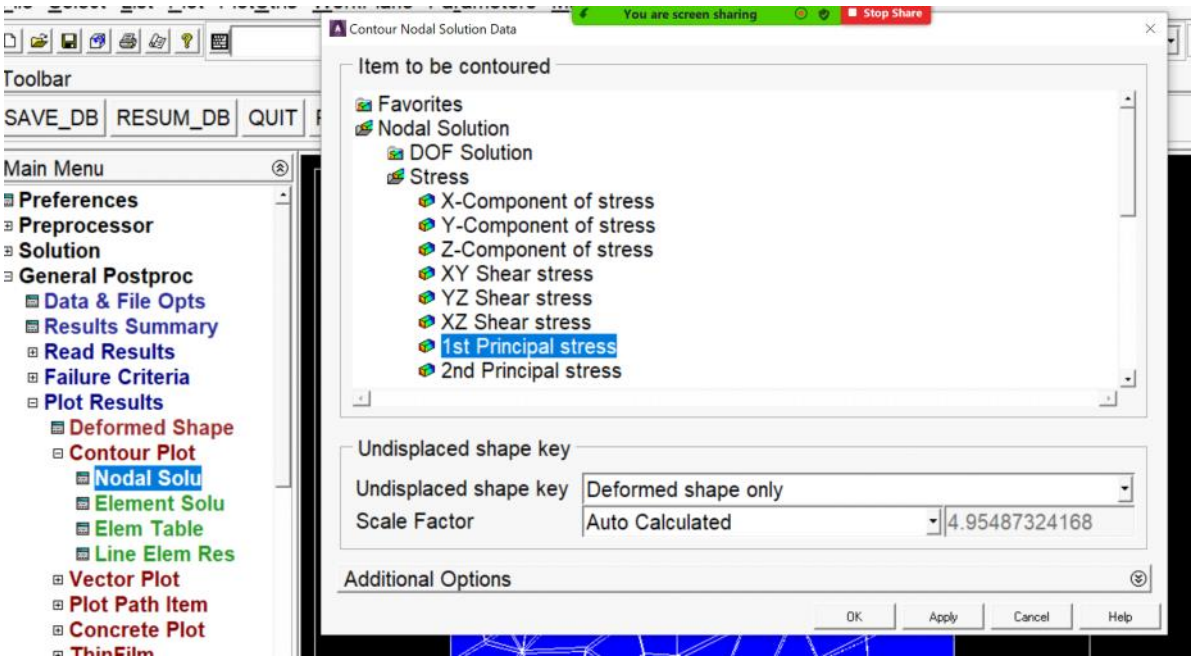


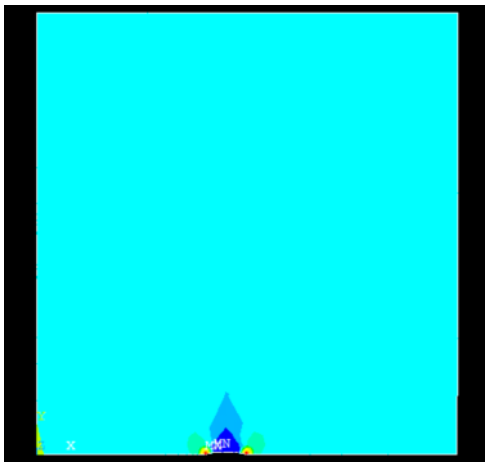


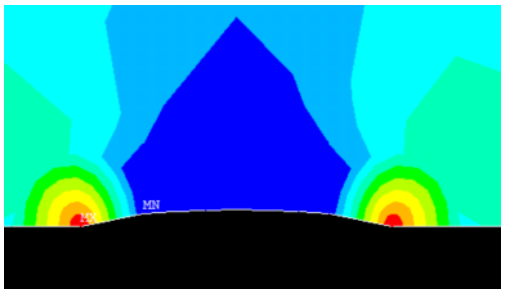


Now we are ready to solve this.









Calculating stress intensity factor

Method 1: Using displacement field around the crack tip

$$u = + \frac{K_{II}}{2G} \sqrt{\frac{r}{2\pi}} (1 + \kappa)$$

$$v = + \frac{K_{I}}{2G} \sqrt{\frac{r}{2\pi}} (1 + \kappa)$$

$$w = + \frac{2\ K_{III}}{G} \sqrt{\frac{r}{2\pi}}$$

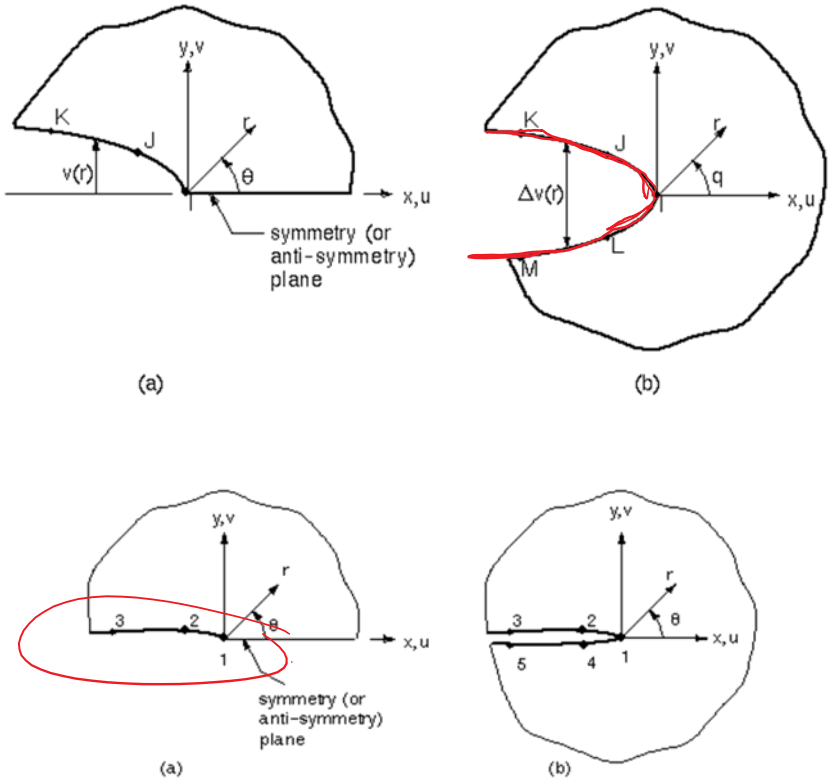
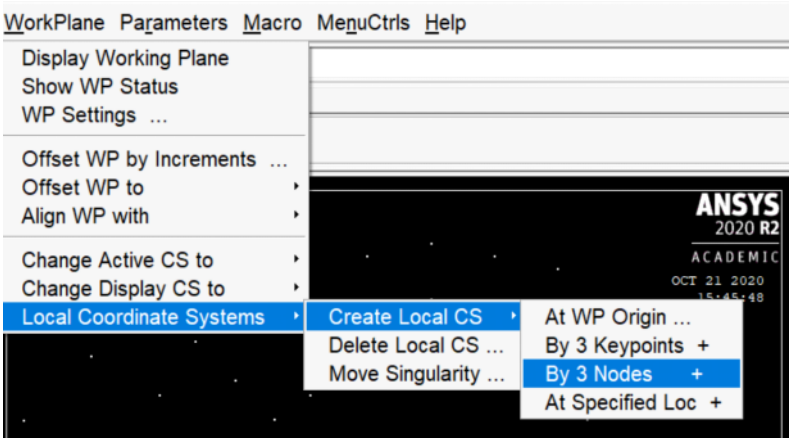
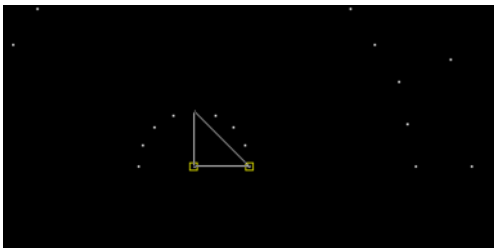


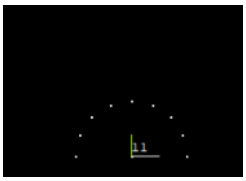
Figure 3) The paths defined for (a) a half-crack model and (b) a full-crack model

We need to

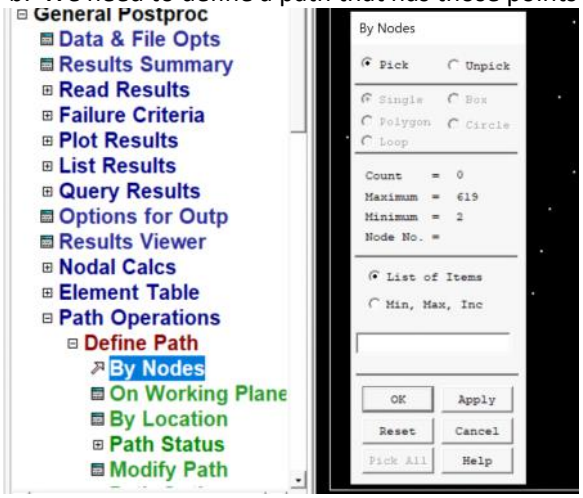
a. Define a local coordinate system
b. Define points 1 to 3 (or 1 to 5 for full crack)

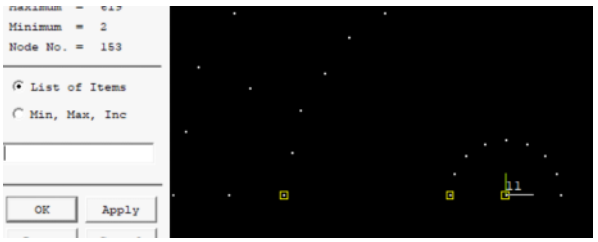




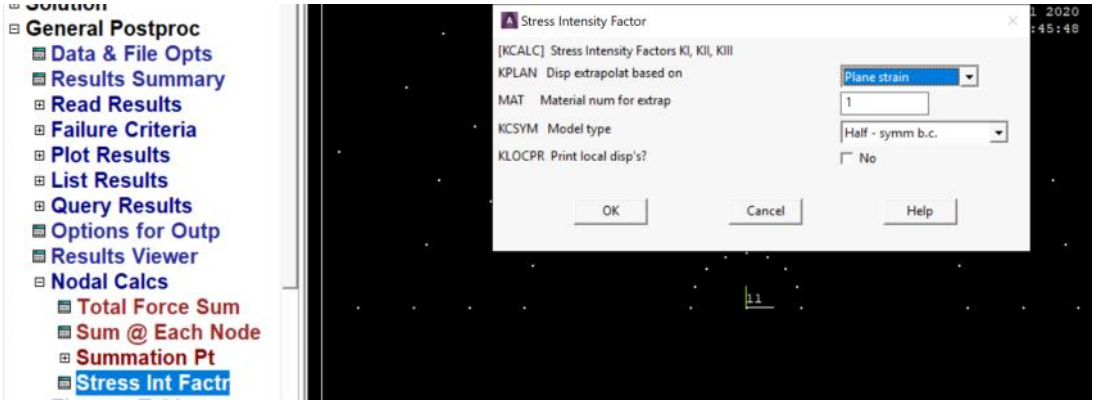


b. We need to define a path that has those points





```
 **********  PATH STATUS  **********
Path      nPts       nSets      nDiv
BR_CTP       3          30        20
```



KCALC Command

File

```
****  CALCULATE MIXED-MODE STRESS INTENSITY  FACTORS  ****
  ASSUME PLANE STRAIN CONDITIONS
  ASSUME A HALF-CRACK MODEL WITH SYMMETRY BOUNDARY CONDITIONS (USE 3 NODES)
  EXTRAPOLATION PATH IS DEFINED BY NODES:      147     154     153
   WITH NODE     147 AS THE CRACK-TIP NODE
  USE MATERIAL PROPERTIES FOR MATERIAL NUMBER     1
    EX =      100.00       NUXY =      0.0000      AT TEMP =  0.0000
****  KI =    1.2627      ,   KII =   0.0000    ,   KIII =   0.0000      ****
```

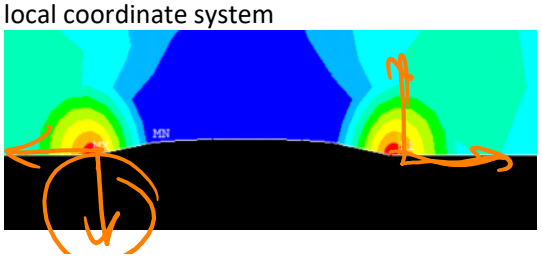
approximate infinite domain $K_I = \sigma\sqrt{\pi a}\,f$ (f → 1)

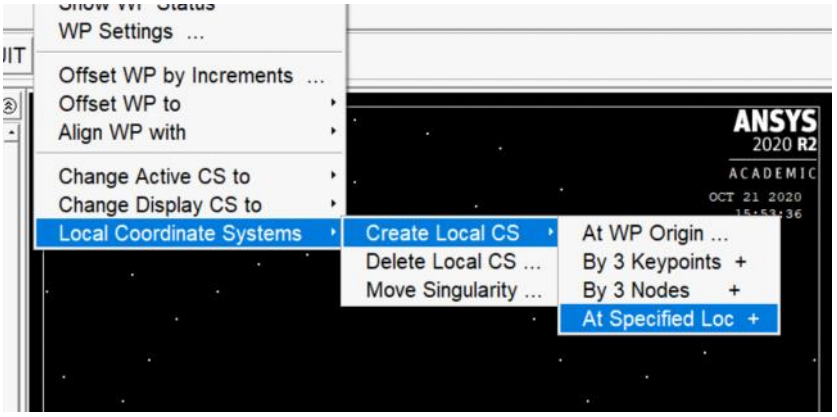
5 to 4 → 2a = 1

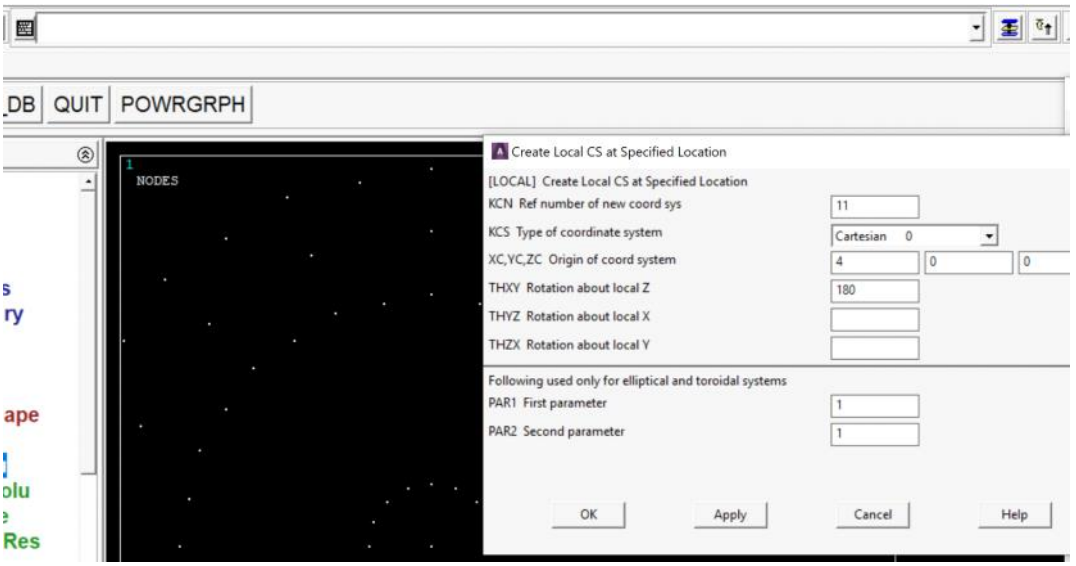
a = .5

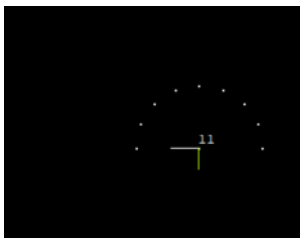
```
>> sqrt(pi * 0.5) * 1

ans =

    1.2533
```

Sometimes the third node does not exist to calculate a local coordinate system









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

ASSUME PLANE STRAIN CONDITIONS

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

EXTRAPOLATION PATH IS DEFINED BY NODES: 2 21 22
WITH NODE 2 AS THE CRACK-TIP NODE

USE MATERIAL PROPERTIES FOR MATERIAL NUMBER 1
EX = 100.00 NUXY = 0.0000 AT TEMP = 0.0000

**** KI = 1.2634 , KII = 0.0000 , KIII = 0.0000 ****

Method 2:
Calculating J integral and from that if needed, calculating K

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.

Must go to preprocessor and solve the problem again

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:

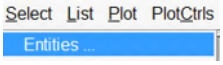
CINT,NEW,*id*

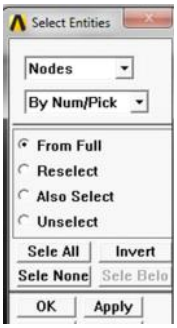
CINT,NEW,1

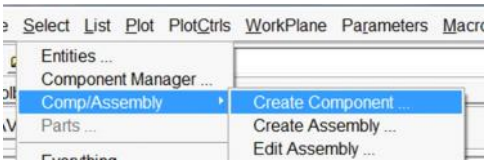
1. Specify the type of contour integral to be done (here it is the J-integral) with the command:

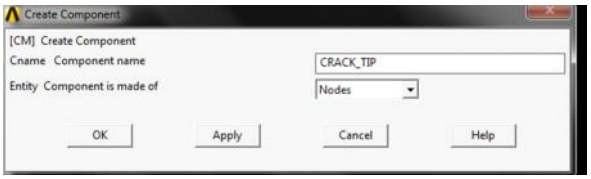
CINT,TYPE,JINT

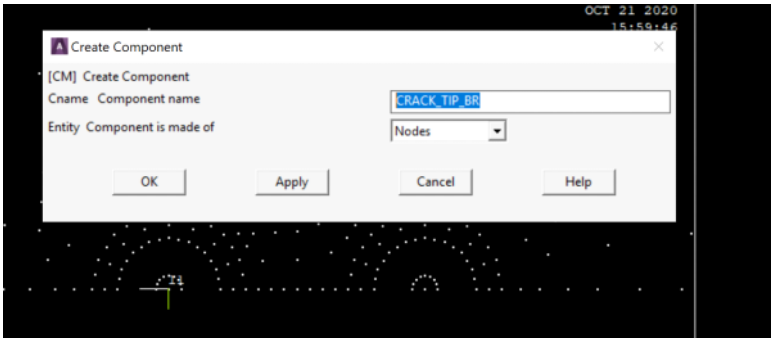
1. Define the node at the crack tip as a node component with the command:
   Note: to create component (corresponding to the crack tip), use select from menu and select the node, and create a component from that:







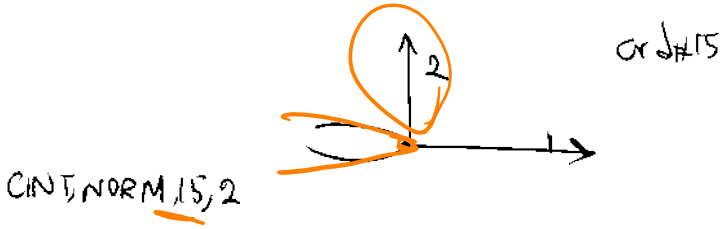




CINT,CTNC,CRACK_TIP_BR

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

   CINT,NORM,*par1*,*par2*

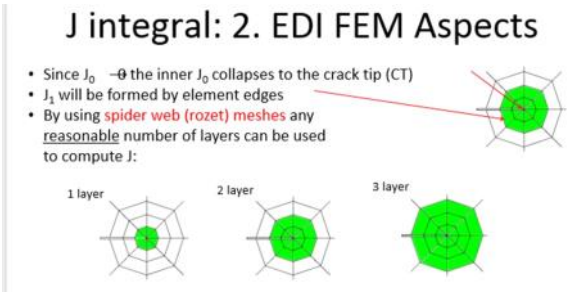


CINT,NORM,15,2

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

   CINT,NCON,*n*

   For example
   CINT,NCON,5



CINT,NCON,2

ONLY for symmetric cracks (not inside the domain)

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

   CINT,SYMM,ON

1. 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*

PRCINT,*1*