

University of Tennessee Space Institute

Department of Mechanical, Aerospace & Biomedical Engineering

Fracture Mechanics Course (ME 524)

**An Explanation on Computation of Fracture
Mechanics Parameters in ANSYS**

By Omid Omid

Teaching Assistant of Fracture Mechanics Course

Fall 2014

1. Introduction

ANSYS is capable of computing the stress intensity factors, the J-integral, energy release rate (G) and T-Stress. For more information, refer to Section “11.3. Numerical Evaluation of Fracture Mechanics Parameters” in ANSYS Help viewer.

In the following sections, the procedures on how to compute the Stress Intensity Factors and the J-integral are shortly presented.

2. Stress Intensity Factors (K)

There are two approaches in ANSYS to evaluate Stress Intensity Factors.

- Interaction integral method which is conceptually similar to J-integral
- Displacement extrapolation method

The method utilized in this report is the displacement extrapolation.

Step 1) Create mesh with singular elements around the crack-tips

To obtain a mesh singularity, second-order elements must be used and the elements must be collapsed as follows:

- ✓ Collapse one side of an 8-node isoparametric element (PLANE183, for example) so that all three nodes of one side have the same geometric location (on the crack tip).
- ✓ Move the midside nodes on the sides connected to the crack tip to the quarter point nearest the crack tip.

The size of the crack-tip elements influences the accuracy of the solutions. The smaller of the radial dimension is recommended. Reasonable results are obtained if typical elements around the crack tip subtend angles in the range of 10° (accurate) to 22.5° (moderately accurate). Since the crack tip causes a stress concentration, the stress and strain gradients are large as the crack tip is approached. The finite element mesh must be refined in the vicinity of the crack to get accurate stresses and strains. The singular elements are illustrated in Figure 1 for a 2-D plane element.

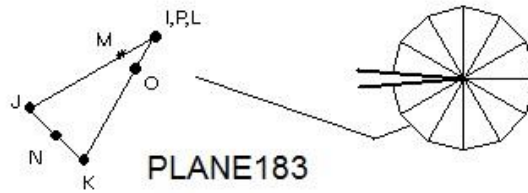


Figure 1) Singular elements for 2-D models using the element of PLANE183

In ANSYS, the command of KSCON assigns element division size around a keypoint and automatically generates singular elements around the specified keypoint. Other modeling guidelines for 2-D models are as follows:

- Take advantage of symmetry where possible.
- For reasonable results, the first row of elements around the crack tip should have a radius of approximately $L/8$ or smaller, where L is the crack length. In the circumferential direction, roughly one element every 30 or 40 degrees is recommended.
- The crack tip elements should not be distorted, and should take the shape of isosceles triangles.

Note: Mid-domain cracks:

For the cracks that are in the middle of domain the two end points are the same; however, in the middle of the cracks there should be duplicate nodes on the two faces of the crack. For example, middle of the crack surface is added for both faces the two mid-nodes have the same coordinate but different ids. Note that the mid-crack nodes do not even need to coincide.

How to differentiate mid-crack face nodes when they coincide:

In many instances, e.g. figure 3 for step 3, nodes from opposite side of crack surface are needed to be chosen. If they coincide, it will be difficult to choose the node on the right side (see figure 3). In this case, it is best to display deformed shape of the solution so these identical nodes can be separated due to their distinct displacements.

Step 2) Apply the boundary conditions and loadings

It is worth noting that the easiest way here is to apply boundary conditions and loadings to lines for 2-D and areas in 3-D. In such a condition, if one change the mesh it does not need to apply the boundary conditions and loadings again.

Step 3) Solve the problem as a static analysis

Step 4) Compute Stress Intensity Factors in general post processor

Once the static analysis is completed, in Post1 as the general postprocessor, the command of KCAL calculates the mixed-mode stress intensity factors KI, KII and KIII. This command is limited to linear elastic problems with a homogeneous, isotropic material near the crack region. It should be noted that to use the command properly, the following two steps must be considered:

- ✓ Define a local crack-tip or crack-front coordinate system, with X parallel to the crack face and Y perpendicular to the crack face, as illustrated in the figure below:

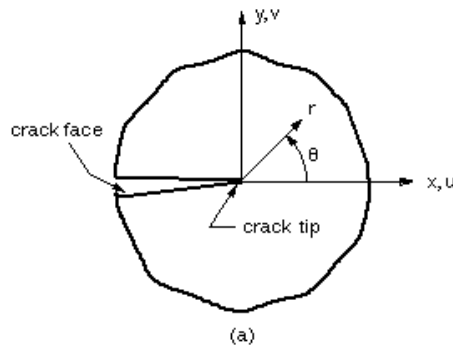


Figure 2) Crack coordinate systems for 2-D model

- ✓ Define a path along the crack face. The first node on the path should be the crack-tip node. For a half-crack model, two additional nodes are required, both along the crack face. For a full-crack model, where both crack faces are included, four additional nodes are required: two along one crack face and two along the other. Figure 3 shows the two cases for a 2-D model.

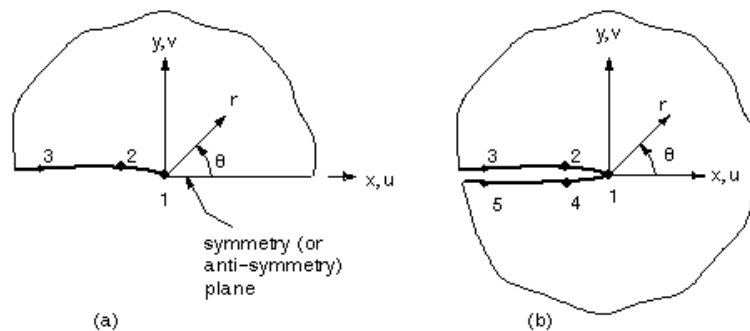


Figure 3) The paths defined for (a) a half-crack model and (b) a full-crack model

The command of KCAL calculates the stress intensity factors at a crack for linear elastic fracture mechanics analysis. The analysis uses a fit of the nodal displacements in the vicinity of the crack. The actual displacements at near a crack for linear elastic materials are as:

$$u = \frac{K_I}{4G} \sqrt{\frac{r}{2\pi}} \left((2\kappa - 1) \cos \frac{\theta}{2} - \cos \frac{3\theta}{2} \right) - \frac{K_{II}}{4G} \sqrt{\frac{r}{2\pi}} \left((2\kappa + 3) \sin \frac{\theta}{2} + \sin \frac{3\theta}{2} \right) + O(r)$$

$$v = \frac{K_I}{4G} \sqrt{\frac{r}{2\pi}} \left((2\kappa - 1) \sin \frac{\theta}{2} - \sin \frac{3\theta}{2} \right) - \frac{K_{II}}{4G} \sqrt{\frac{r}{2\pi}} \left((2\kappa + 3) \cos \frac{\theta}{2} + \cos \frac{3\theta}{2} \right) + O(r)$$

$$w = \frac{2K_{III}}{G} \sqrt{\frac{r}{2\pi}} \sin \frac{\theta}{2} + O(r)$$

where: u, v, w = displacements in a local Cartesian coordinate system
 r, θ = coordinates in a local cylindrical coordinate system
 G = shear modulus
 K_I, K_{II}, K_{III} = stress intensity factors relating to deformation shapes
 $\kappa = \begin{cases} 3 - 4\nu & \text{if plane strain or axisymmetric} \\ \frac{3-\nu}{1+\nu} & \text{if plane stress} \end{cases}$
 ν = Poisson's ratio
 $O(r)$ = terms of order r or higher

Evaluating the equations above at $\theta = +180^\circ$ & -180° and dropping the higher order terms yields:

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

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

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

For half-crack models the equations above can be reorganized to give:

$$K_I = \sqrt{2\pi} \frac{2G}{1+\kappa} \frac{|v|}{\sqrt{r}}$$

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

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

and for the case of full-crack models:

$$K_I = \sqrt{2\pi} \frac{G}{1+\kappa} \frac{|\Delta v|}{\sqrt{r}}$$

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

$$K_{III} = \sqrt{2\pi} \frac{G}{4} \frac{|\Delta w|}{\sqrt{r}}$$

where Δv , Δu and Δw are the motions of one crack face with respect to the other.

The nodes and their order for definitions used for the crack tip displacements needed for the equations above are illustrated in Figure 4 for both symmetric and full crack modeling.

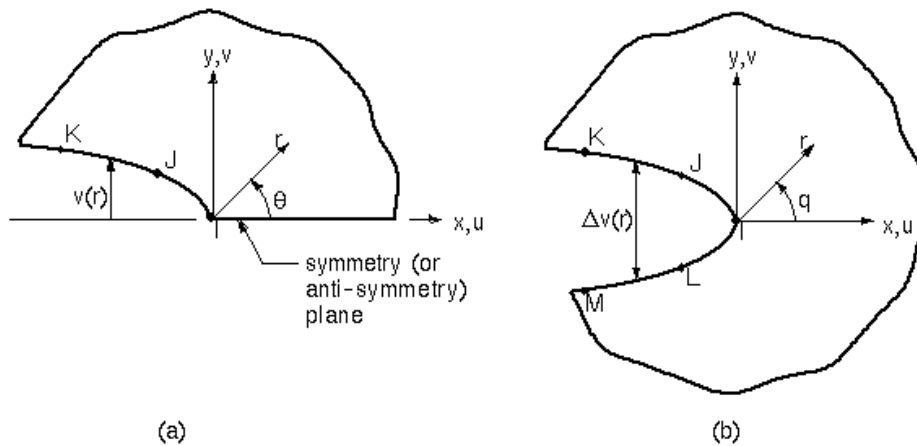


Figure 4) Nodes used for the approximate crack-tip displacements in (a) Half model and (b) Full model

In the first equation above, the final factor is $\frac{|\Delta v|}{\sqrt{r}}$, which needs to be evaluated based on the nodal displacements and locations. v is normalized so that v at node I is zero. Then A and B are determined so that the following equation is satisfied at points J and K.

$$\frac{|v|}{\sqrt{r}} = A + B r$$

Next, let r approach 0.0 that it becomes as:

$$\lim_{r \rightarrow 0} \frac{|v|}{\sqrt{r}} = A$$

Therefore, the first-mode stress intensity factor is computed as below:

$$K_I = \sqrt{2\pi} \frac{2 G A}{1 + \kappa}$$

The stress intensity factor of the other modes can also be calculated in the same manner.

Finally, the procedure which is utilized to determine the stress intensity factors based on displacement extrapolation approach using ANSYS can be summarized as the following:

- Create the geometry of the model by keypoints, line and area.
- Define the crack-tip which is to be meshed by singular elements. The post1 KSCON command specifies a keypoint about which an area mesh will be skewed. During meshing, elements are initially generated circumferentially about and radially away from the keypoint. It should be noted that only one concentration keypoint per unmeshed area is allowed. The parameters of the command are described as the following:

KSCON,NPT,DELR,KCTIP,NTHET,RRAT

where the parameters in this command are as:

- *NPT*, keypoint number at concentration.
 - *DELR*, radius of first row of elements about keypoint.
 - *KCTIP*, crack tip singularity key =1
 - *NTHET*, number of elements in circumferential direction.
 - *RRAT*, ratio of 2nd row element size to *DELR*
- Before meshing the area, the lines connected to the area that is to be meshed should be divided on the basis of element size or number of

divisions on the lines. The size of the elements influences the accuracy of the solutions.

- Select a plane element suitable for fracture analysis such as PLANE183.
- Mesh the area and refine it if it is possible.
- Apply boundary conditions as displacement and pressure on the lines in 2-D and areas in 3-D.
- Solve the problem as a static analysis.
- Define a local crack-tip coordinate system as discussed before. This coordinate system must be the active model coordinate system and results coordinate system when the KCAL command is issued.
- Define a path along the crack face as mentioned before. Its command is PATH.
- Calculate the stress intensity factors by the KCAL command.

3. J-integral

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

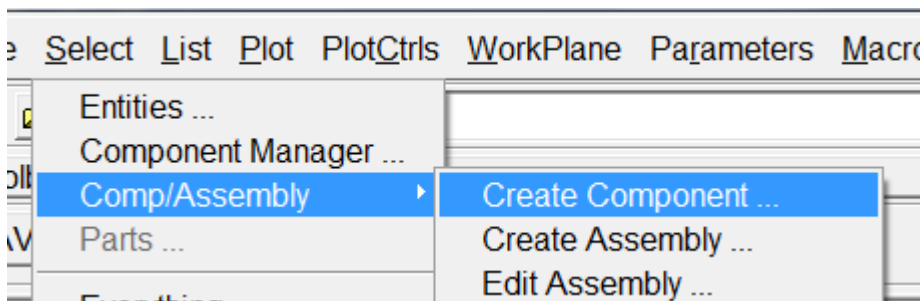
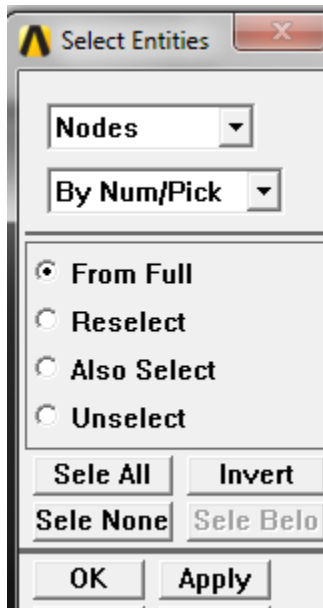
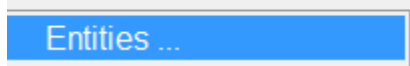
3. Specify the type of contour integral to be done (here it is the J-integral) with the command:

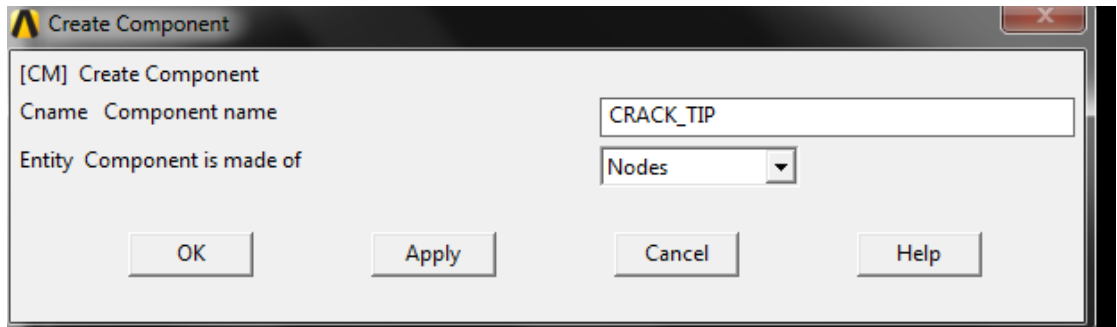
CINT,TYPE,JINT

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





CINT,CTNC,*name*

where "*name*" is the name of the crack tip node component name
so this becomes

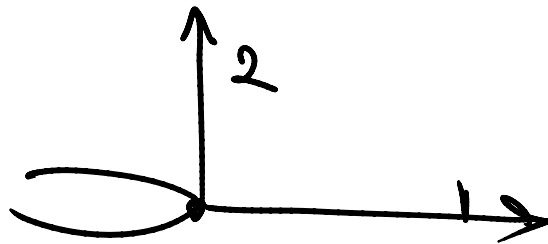
CINT,CTNC,CRACK_TIP

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*

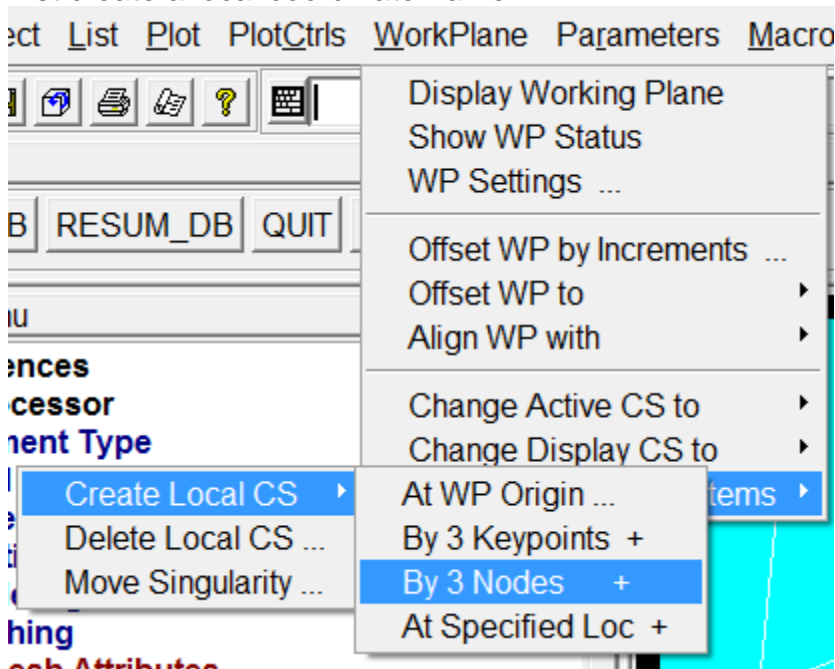
CINT,NORM,15,2



or J#15

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:



For example is local coordinate system is 11 and normal to crack direction is along its 2 direction this becomes:
CINT,NORM,11,2

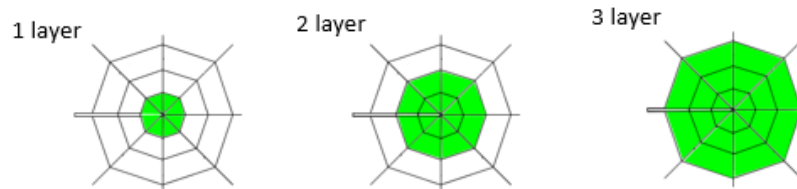
- Specify the number of contours “ n ” to compute with the command:

CINT,NCON, n

For example
CINT,NCON,5

J integral: 2. EDI FEM Aspects

- Since $J_0 \rightarrow \theta$ 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:



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

CINT,SYMM,ON

- Specify the output controls with the command:

OUTRES,CINT

- Solve the problem.

- 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*

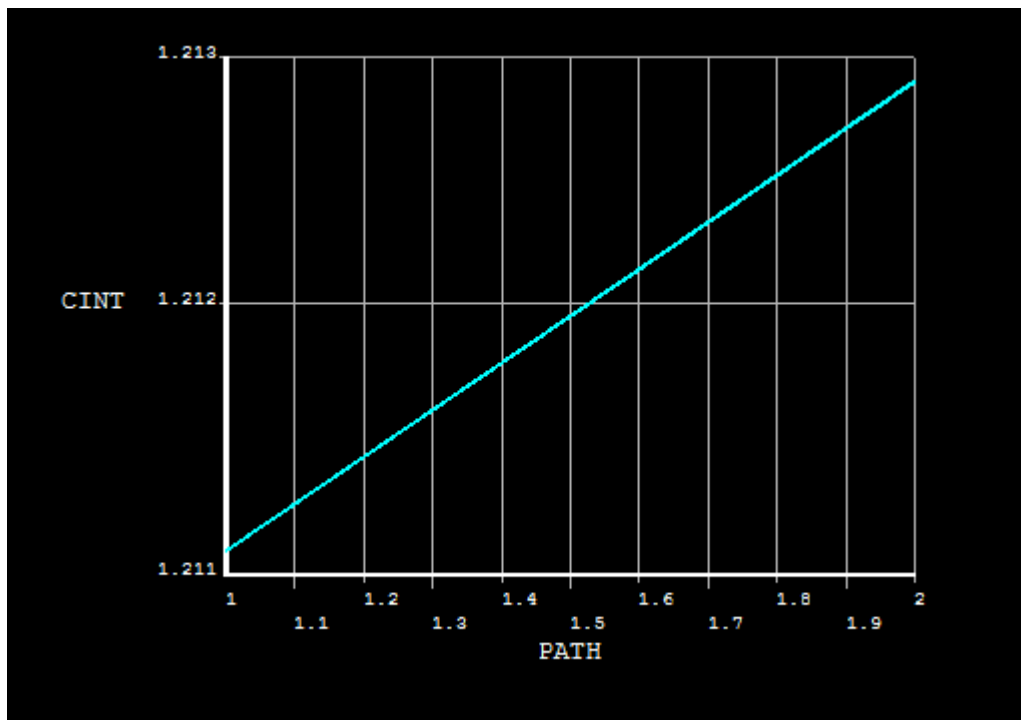
where “*id*” is the crack identifier.

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:

```
***** POST1 J-INTEGRAL   RESULT LISTING *****  
CrackID = 1  
Crack Front Node = 444  
Contour Values = 1.2113      1.2131
```

and



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.