

---

## Engineering Analysis using the Finite Element Method

University of California, Berkeley, Spring 2019

Instructor: Sanjay Govindjee

GSI: Linus Mettler

### Lab 5

---

This lab is concerned with the modeling of macroscopic cracks using finite elements. This is demonstrated on the example of a simple 2D plate with crack under plane stress. In the course of the lab you will

- generate a 2D geometry and FE model
- compute the stress intensity factor and J-integral in Ansys workbench using command snippets
- compare the FE estimate with a theoretical approximation for the given geometry
- examine the dependence of the stress intensity factor on the mesh size
- examine the dependence of the von Mises stress on the mesh size

No report is required to check off this lab.

## 1 Overview: linear elastic fracture mechanics

Macroscopic and microscopic cracks are found in virtually all man-made and natural structures, including for example bridges or airplanes. The field of fracture mechanics is concerned with the analytical description of the propagation of cracks. Classical elasticity theory predicts locally infinite stresses at sharp corners. And yet, most materials can withstand loads even when a crack is present. In ductile materials this is due to plastic deformation at the crack tip, which causes a redistribution of stress.

### 1.1 Stress intensity factor

Early research by A.A. Griffith and G.R. Irwin has shown that for purely elastic (i.e. brittle) materials, or when the plastic zone at the crack tip is small compared to the crack length, the stress field around the crack tip is well approximated by

$$\sigma_{ij} = \frac{K_I}{\sqrt{2\pi r}} f_{ij}(\theta) \quad (1)$$

where  $r$  and  $\theta$  are the directions of a cylindrical coordinate system centered at the crack tip,  $f_{ij}(\theta)$  is a dimensionless function describing the variation of stress in circumferential direction and  $K_I$  is the so-called *stress intensity factor*. The subscript  $I$  stands for “mode 1”, which describes a deformation that leads to crack opening, i.e. with tensile stress normal to the plane of the crack.

The approximation in equation [1] breaks down as  $r \rightarrow 0$  and the predicted stress goes to infinity. However, it was found that the stress intensity factor  $K_I$  for a given static load can be compared with a material property, called fracture toughness  $K_{Ic}$ , to predict whether a crack will or will not propagate, which is clearly a very pressing question in engineering design. Note that  $K_I$  depends on geometry *and* load, while  $K_{Ic}$  is a property of the material.

## 1.2 J-integral

Another common measure to predict the evolution of cracks is the so-called *J-integral*.  $J$  is a path integral around the crack tip that represents the net change of potential energy caused by growth of the crack. It can thus be compared with the energy required to create new crack surface for a given material, in order to predict whether or not the crack will grow. In the limit of linear elastic material behavior, the  $J$ -integral can be related to the stress intensity factor directly by

$$K_I = \sqrt{E J_I} \quad (\text{plane stress}) \quad \text{and} \quad K_I = \sqrt{\frac{E}{1-\nu^2} J_I} \quad (\text{plane strain}) \quad (2)$$

## 2 FE model

### 2.1 Geometry

We are modeling a 2D plane stress geometry<sup>1</sup> depicted in the following figure, where the dimensions are given (in millimeter) by  $h = 50$ ,  $b = d = 50$ ,  $2a = 20$ . In order to sketch the crack in DesignModeler, create a rhombus-shaped hole in the center of the plate with width (in millimeter)  $w = 2a = 20$  and height  $h = 0.1$ .

<sup>1</sup>As before, this calls for the *behavior* of the geometry to be set to 2D

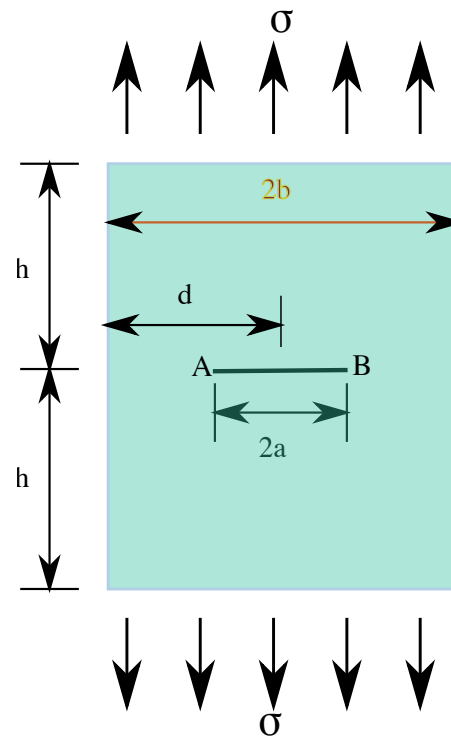


Figure 1: Plate in uniaxial stress with central mode I crack. Source: Wikipedia.

Assign a material (e.g. structural steel) and make sure the behavior is defined to be linear elastic.

## 2.2 “Exact” solution

Approximate solutions for  $K_I$  for simple geometries and loads can be found in any fracture mechanics textbook. In our case the stress concentration factor is approximately

$$K_I = \sigma \sqrt{\pi a} \frac{1 - \frac{a}{2b} + 0.326\left(\frac{a}{b}\right)^2}{\sqrt{1 - \frac{a}{b}}} \quad (3)$$

## 2.3 Mesh

Since we expect stress concentrations at the two tips of the centered crack, our mesh should be refined around those corners. Apply appropriate body and edge sizing, for example using 0.2mm mesh size along the crack and a bias factor of 100. Generate the mesh.

By zooming in on the crack tip, note that some of the elements erroneously “reach across the crack” and connect with nodes on the other side of the crack face. This is caused by automatic geometry defeaturing performed by the meshing algorithm. In order to fix the issue we need a “patch conforming” mesh (as opposed to patch independent). To do so, insert mesh control → method → multizone quad / tri method, and specify details → advanced → mesh based defeaturing → off. With these settings, all faces and boundaries of the geometry are respected by the mesh.

## 2.4 Loads

Apply a uniform tensile stress (of a suitable magnitude) to the top and bottom edges. Recall that the model is insufficiently supported, so apply appropriate supports (e.g. if you chose to apply weak springs you can click on analysis settings → solver controls → weak springs → on, and keep spring stiffness → program controlled).

## 2.5 Coordinate system

Create a new coordinate system centered at the crack tip on the right of our crack (by RMB in the tree on coordinate systems → insert → coordinate system → define by → geometry selection → pick the crack tip. Set the coordinate system ID to 12, if this is not already the case. This is needed for the later commands to be able to identify the relevant coordinate system.

## 2.6 Crack tip named selection

In order for our code to identify the node corresponding with the crack tip, we need to create a named selection. RMB in the tree on model → insert → named selection → pick the crack tip on the right side of the crack. Rename the named selection to CrackTip.

## 2.7 Command snippets

Insert the following commands by RMB in the tree on static structural → insert → commands:

```
CINT,NEW,1 ! define a new crack ID
CINT,TYPE,JINT ! J-integral calculation
CINT,CTNC,CrackTip,NODE ! define the crack tip using named selection
CINT,NCON,10 ! define number of contour lines for J-integral
CINT,NORMAL,12,2 ! define normal plane to crack
OUTRES,CINT,ALL ! output result at all time steps
```

```
CINT,NEW,2 ! define a new crack ID
CINT,TYPE,SIFS ! KI stress intensity for mode 1
CINT,CTNC,CrackTip,NODE ! define the crack tip using named selection
CINT,NCON,10 ! define number of path integrals
CINT,NORMAL,12,2 ! define normal plane to crack
OUTRES,CINT,ALL ! output result at all time steps
```

Insert the following commands under solution:

```
PRCINT,1,,JINT
PRCINT,2,,K1
```

Once your model is fully defined, click solve.

## 2.8 Post-processing

You can find the computed  $K_I$  and  $J_I$  values in the tree solution  $\rightarrow$  solution information. This is the solver output file into which we have printed the values using PRCINT. Perhaps it helps to search for POST1 or CrackID in order to find the data. There should be 10 values each, because we've set the number of contour integrals to perform around the crack tip to 10. Both  $K_I$  and  $J_I$  should converge to a constant value.

To get checked off, answer the following questions:

- (i) What are the values of  $K_I$  and  $J_I$  found by your FE model? Include the correct units.
- (ii) What is the relative error between the FE approximation and the exact solution [3] for both  $K_I$  and  $J_I$ ?
- (iii) Verify that relation [2] holds for the finite element result (this is expected because the material behavior is purely elastic).
- (iv) Plot the von Mises equivalent stress. Repeat the calculation with a finer mesh. How does the stress change? How do  $K_I$  and  $J_I$  change?