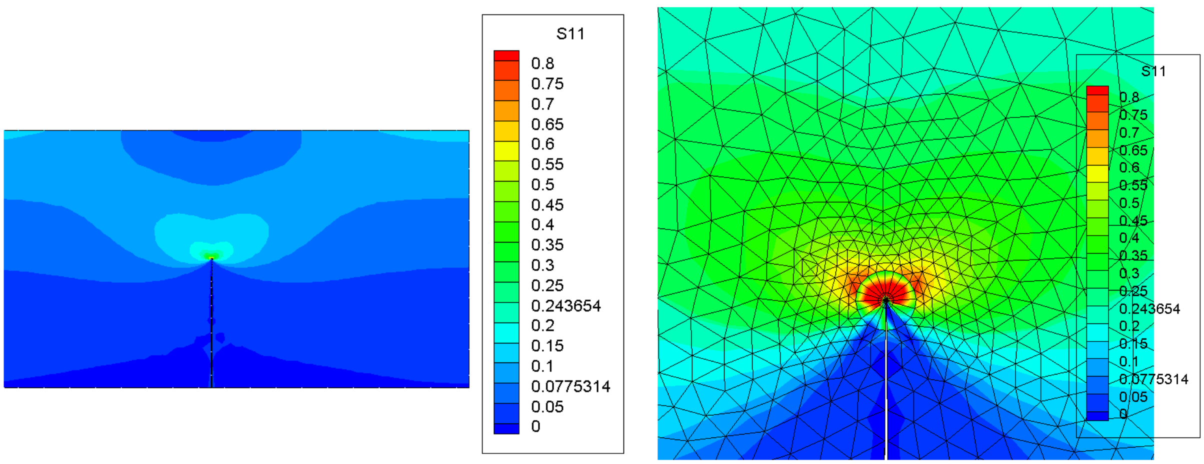
School of Engineering
Brown University

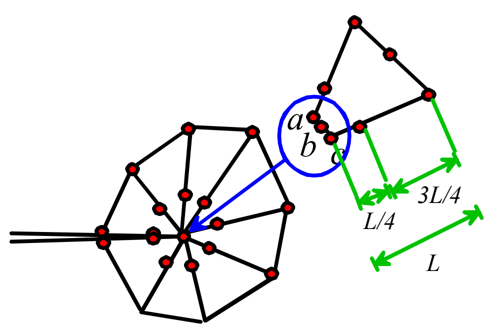
EN234: Computational methods in Structural and Solid Mechanics

Homework 3: Crack tip elements and the J-integral
Due Wed Oct 7, 2015



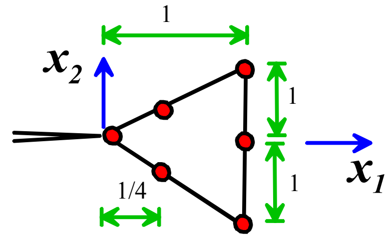
The purpose of this homework is to help understand how to handle element interpolation functions and integration schemes in more detail, as well as to explore some applications of FEA to fracture mechanics.

In this homework you will solve a simple linear elastic fracture mechanics problem. You might find it helpful to review some of the basic ideas and terminology associated with linear elastic fracture mechanics here (in particular, recall the definitions of stress intensity factor and the nature of crack-tip fields in elastic solids). Also check the relations between energy release rate and stress intensities, and the background on the J integral here.



1. One of the challenges in using finite elements to solve a problem with cracks is that the stress field at a crack tip is singular. Standard finite element interpolation functions are designed so that stresses remain finite everywhere in the element. Various types of special 'crack tip' elements have been designed that incorporate the singularity. One way to produce a singularity (the method used in ABAQUS) is to mesh the region just near the crack tip with 8 noded elements, with a special arrangement of nodal points: (i) Three of the nodes (nodes 1,4 and 8 in the figure) are connected together, and (ii) the mid-side nodes 2 and 7 are moved to the quarter-point location on the element side. This magically produces the necessary singularity in the strain.

In this problem you will demonstrate the singularity by plotting the strain distribution in a single element, shown in the figure. Write a MATLAB code that will plot the strain (you can plot stress as well if you like) as a function of distance *x* ahead of the crack tip:



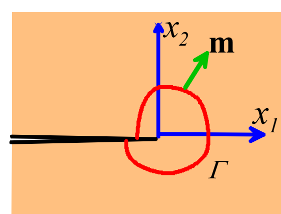
- Do a parametric plot as a function of distance along the line $\xi_2 (or\ \eta) = 1/2$ in normalized coordinates.
- Place nodes 1, 4, 8 at the origin, nodes 5, 7 at $(0.25, \mp 0.25)$, nodes 2,3 at $(1, \mp 1)$ and node 6 at (1,0)
- Assume that the nodes have displacements that are consistent with the crack tip fields, i.e.

$$u_1 = \frac{K_I}{\mu}\sqrt{\frac{r}{2\pi}}\left[1-2\nu+\sin^2\frac{\theta}{2}\right]\cos\frac{\theta}{2}$$

$$u_2 = \frac{K_I}{\mu}\sqrt{\frac{r}{2\pi}}\left[2-2\nu-\cos^2\frac{\theta}{2}\right]\sin\frac{\theta}{2}$$

  where (r,θ) are the polar coordinates of the node. You can assume $K_I / \mu = \sqrt{2\pi}$ and take $\nu = 0$ for simplicity.
- You already know how to calculate the strain in the element – in fact you have code that does this in your 2D elasticity element from HW3. You will have to make some small tweaks to the code to convert to MATLAB – for example MATLAB accesses every other element in a vector using $vector(1:2:15)$ while in Fortran the equivalent is $vector(1:15:2)$.
- MATLAB functions for the shape functions and their derivatives can be cut and pasted from the sample elastic MATLAB FEA code shown in class (posted here)
- Use a log-log scale for the plot so you can see the square root singularity.

2. Finite element simulations are often used to calculate stress intensity factors for cracks in elastic solids. This raises another question: what is the best way to determine the stress intensity factor from a finite element computation? The usual approach is to make use of the relationship between stress intensity and the energy release rate for the crack.



$$G = \frac{1-\nu^2}{E}K_I^2$$

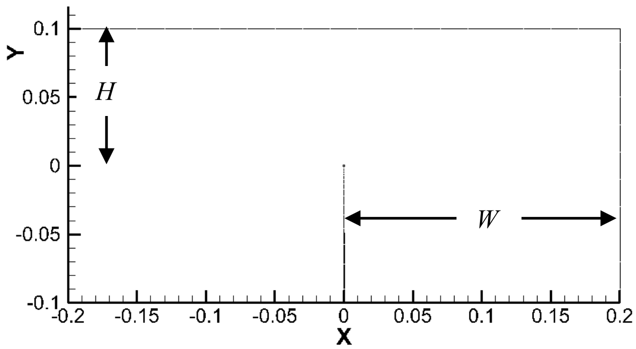
The energy release rate can be calculated using the famous 'J integral'

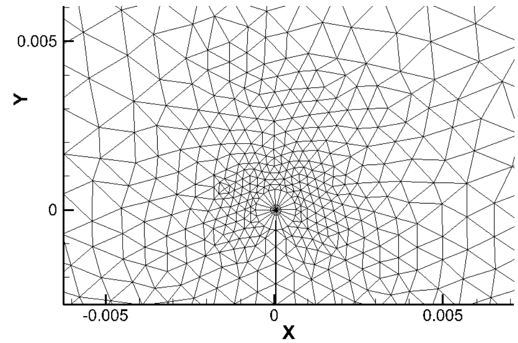
$$G = \int_{\Gamma} (W\delta_{j1} - \sigma_{ij}\frac{\partial u_i}{\partial x_1})m_j ds$$

where $W = \sigma_{ij}\varepsilon_{ij}/2$ is the strain energy density, σ_{ij} is the stress field, u_i is the displacement field, m_i is a unit vector normal to Γ, and the $\mathbf{e}_1$ basis vector is parallel to the direction of crack propagation. For finite element calculations, it is convenient to re-write the J integral as an equivalent *area* integral (which is much easier to calculate in an FE code). For this purpose we select a convenient annular region surrounding the crack tip, shown in the figure, and introduce an arbitrary smooth weighting function $q(\mathbf{x})$ defined on this domain. This function must be selected so that $q = 1$ on the inner contour, and $q=0$ on the outer contour. The inner contour can be shrunk all the way down to the crack tip, if desired. The energy release rate can then be calculated from

$$G = \int_{A} (\sigma_{ij}\frac{\partial u_i}{\partial x_1} - W\delta_{j1})\frac{\partial q}{\partial x_j} dA$$

Your goal in this problem is to use this procedure to calculate the crack tip stress intensity factor for a crack in a rectangular plate, shown in the figure below





You can find a finite element mesh for this problem in the file called crack_tri6.in. The mesh contains two regions: elements 1 to 1246 are regular 6 noded triangles, and mesh the region outside r=0.0006. Elements 1247-1256 are crack tip elements, which lie inside a circular domain with radius 0.0006 units. You can thus evaluate the domain J integral over the circular region of crack tip elements using

$$q = 1 - \frac{r}{r_0}$$

where $r_0 = 0.0006$.

You should implement the J integral calculation in the file called user_print.f90. A subroutine called compute_J_integral has been created for you for this purpose. You will need to uncomment the lines in crack_tri6.in that read

% USER PRINT FILES
% END USER PRINT FILES

and enter an appropriate output file name that will record the value of the J integral. The code will call the user print subroutine if it finds that you have requested user print files.

You will need to edit the code to compute the relevant area integral. A few tips on doing the coding:

- Note that since the crack propagates in the x_2 direction, you must compute

  $$G = \int_A \left(\sigma_{ij}\frac{\partial u_i}{\partial x_2} - W\delta_{j2}\right)\frac{\partial q}{\partial x_j}\, dA$$

  The derivatives $\partial q / \partial x_j$ can easily be computed analytically in the region $r < r_0$ ($\partial r / \partial x_i = x_i / r$).

  All integrals over crack tip elements (including those calculating the stiffness) should be done with 9 integration points, instead of the usual 4, to ensure that the integral is evaluated accurately. You will need to edit the element stiffness routine that you wrote last week to take care of this.
- Much of the code you need to do the integrals over each element can be cut-and pasted from the 2D linear elasticity subroutine – the procedures for calculating the stress and strain are identical; and the same integration scheme can be used.
- When you run your code, you will find that the stress contours in the crack tip elements look a little strange – this is because the procedure used to project stresses from integration points to nodes for printing does not work properly for crack tip elements (there is no way to get a stress for the crack tip node, for example).
- To check your answer, you can compare the value you compute with the solution obtained by ABAQUS. For a displacement of 0.003 (arbitrary units) applied to the rightmost boundary, ABAQUS gives a J integral value of 1.2×10^{-3} (arbitrary units) for the geometry and material parameters specified in the input file. If you are curious you can download the .cae file that sets up this problem from the homework page of the website. You can run the analysis – J integral data can be found in the History output – just select the J integral variable from the list.