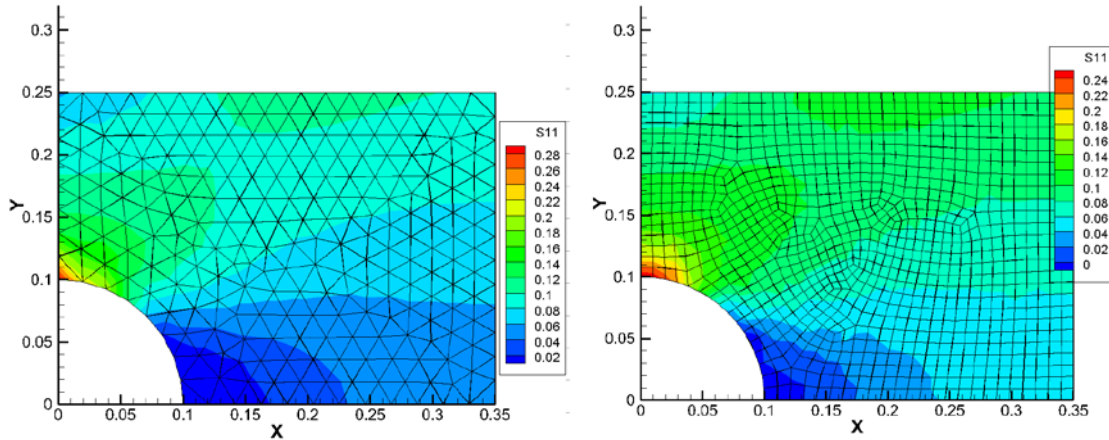




EN234: Computational methods in Structural and Solid Mechanics

Homework 3: Introduction to coding elements Due Wed Oct 7, 2015

School of Engineering
Brown University



The purpose of this homework is to get comfortable with basic FEA code development.

1. If you have not already done so, install a FORTRAN compiler, TECPLOT, Eclipse, create yourself a GitHub account, and download EN234_FEA from the course website (there are detailed instructions on the programming page of the website).
2. Work through the FEACHEAP tutorial (first link on the programming page) to make sure you can run EN234FEA successfully.
3. Then implement a general 2D fully integrated (i.e. use enough integration points to evaluate the stiffness exactly) static plane elasticity element in the code. The easiest way to do this is:
 - a. Copy the 3D linear elasticity element to a new file called `el_linelast_2Dbasic.f90`
 - b. Modify the 3D subroutines `el_linelast_3dbasic` and `fieldvars_linelast_3dbasic` (and re-name them appropriately) to create corresponding 2D versions. (you should also rename the subroutine called `el_linelast_3dbasic_dynamic`, but since you won't be doing a dynamic analysis you can leave the code unchanged). Note that the subroutines that initialize values for integration points and shape functions (see the `Element_Uutilities` module) check the dimensions of the arrays you are using for coordinates and shape functions, and will automatically return the 2D versions of the relevant variables for you. You will need to modify the definitions at the top of the file to use the 2D versions of shape functions and coordinates, and also change the B matrix and the D matrix. If you want to plot the out-of-plane stress or strain, or the Mises stress, you will have to do some simple post-processing in the `fieldvars_linelast_2dbasic` subroutine to calculate their values.
 - c. Add lines to `user_codes/src/user_element.f90` to call your new subroutines, as described in the EN234_FEA tutorial.
 - d. A set of input files have been provided to test your element (you can create your own if you prefer)
 - e. Modify `src/main.f90` to read your input files.

Please write code to solve at least 2D plane strain problems.

To test your code, you can:

- (1) Set up an input file solving uniaxial tensile straining of one or two elements, and compare the predicted displacements and stresses with the exact solution
- (2) If you would like to solve a more interesting problem, will find input files for EN234FEA that set up a simple boundary value problem (plate with a hole, as usual), with 3 and 6 noded triangular elements, and 4 and 8 noded rectangular elements, in the input_files/ directory of EN234FEA.
- (3) You can also generate your own meshes with ABAQUS/CAE and run problems of your own design if you wish.

You can edit the main.f90 code to cycle over as many input files as you like, as verification (you should edit the input files so they don't all save the output to contourplots.dat, of course)

As a solution to this homework, please hand in:

1. A one or two page description of the tests that verify your code (upload to canvas). Please include the link to your GitHub repository.
2. Push your code up to your GitHub fork of EN234FEA. I will pull your codes and run them. Please don't upload a code that takes six weeks to run!

Optional: - if this was too easy, you could code your element to solve plane strain, plane stress, and axisymmetric problems. To allow the user to control the choice through the input file, you can either add a variable to the list of element properties (this has to be a number, strings can't be used as properties); alternatively you can use the integer valued 'element identifier' to control the type of analysis.

Note that the procedure used by EN234FEA to project element state variables to nodes does not distinguish between axisymmetric and plane elements – they are all treated as plane elements. Consequently, when computing the quantity

$$f^a = \int_{A_{el}} f(x) N^a(x) dA$$

to project some variable $f(x)$, you do not need to include the factor $2\pi r$ that appears in the integrals for the stiffness matrices for axisymmetric elements.