

Finite element analysis of plate-hole problem using FORTRAN90 code

Srihari Sundar¹

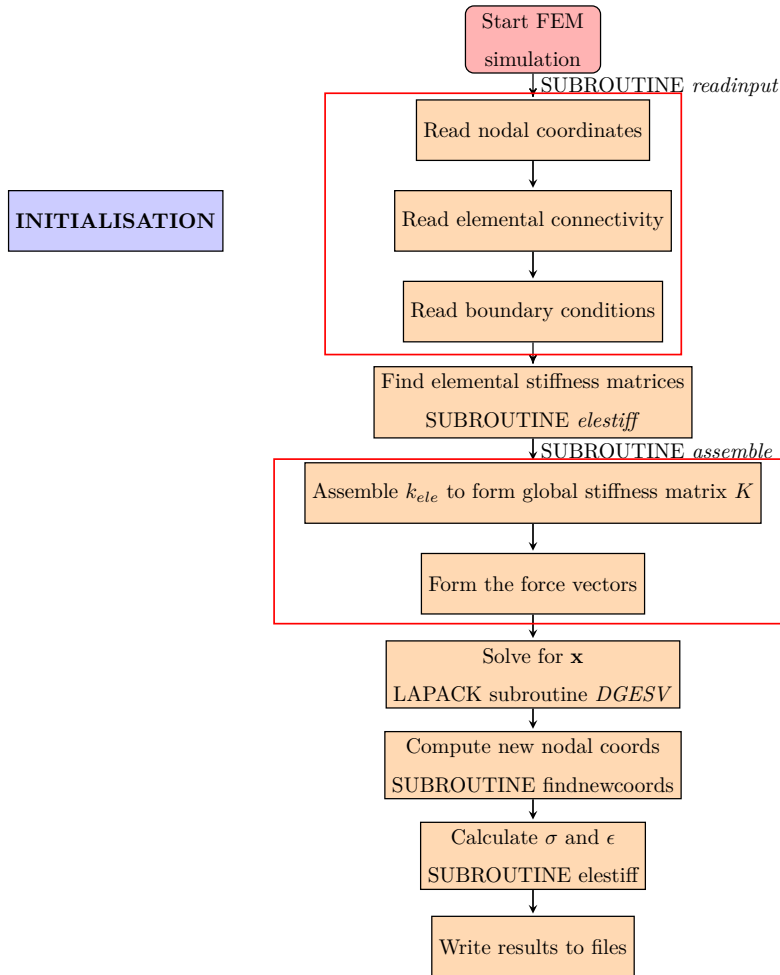
¹*Department of Metallurgical and Materials Engineering, Indian Institute of Technology Madras*

Abstract

A formula translator (FORTRAN) code is developed to perform finite element analysis (FEA) for the deformation of a 2D plate with a hole in the center. The results are compared with solution from ABAQUS.

Keywords: FEM,FORTRAN,plate-hole,ABAQUS

Flow of program



Structure of input file

1. nnodes,nelem
2. node numbers with nodal coordinates
 \vdots
3. elem numbers with nodes of element anti-clockwise direction starting from bottom left
 \vdots
4. number of fixed nodes
5. node number, a,b (a,b=0–constrained,1-unconstrained)
6. number of nodes with force BC
7. node number,forceX,forceY
8. number of nodes with displacement BC
9. node number, dispX, dispY

Calculation of elemental stiffness matrix

The element modeled here is a 4 noded element with 4 integration points [2].

The interpolation functions used are as follows:

$$N1 = \frac{1}{4}(1 - \xi)(1 - \eta)$$

$$N2 = \frac{1}{4}(1 + \xi)(1 - \eta)$$

$$N3 = \frac{1}{4}(1 + \xi)(1 + \eta)$$

$$N4 = \frac{1}{4}(1 - \xi)(1 + \eta)$$

where ξ and η are the x and y coordinates in elemental reference frame.

Over each integration point the following is carried out:

S and T are calculated which are ξ and η derivatives respectively. This is done in the subroutine

CALCULATE_ST

$$S_1 = -\frac{1}{4}(1 - \eta)$$

$$S_2 = \frac{1}{4}(1 - \eta)$$

$$S_3 = \frac{1}{4}(1 + \eta)$$

$$S_4 = -\frac{1}{4}(1 + \eta)$$

$$T_1 = -\frac{1}{4}(1 - \xi)$$

$$T_2 = -\frac{1}{4}(1 + \xi)$$

$$T_3 = \frac{1}{4}(1 + \xi)$$

$$T_4 = \frac{1}{4}(1 - \xi)$$

From these the ‘G’ matrix is assembled as seen in fig. 1, in the subroutine *CALCULATE_G*.

$$[G] = \frac{1}{4} \begin{bmatrix} -(1-\eta) & 0 & (1-\eta) & 0 & (1+\eta) & 0 & -(1+\eta) & 0 \\ -(1-\xi) & 0 & -(1+\xi) & 0 & (1+\xi) & 0 & (1-\xi) & 0 \\ 0 & -(1-\eta) & 0 & (1-\eta) & 0 & (1+\eta) & 0 & -(1+\eta) \\ 0 & -(1-\xi) & 0 & -(1+\xi) & 0 & (1+\xi) & 0 & (1-\xi) \end{bmatrix}$$

Figure 1: G matrix formulation

$$[A] = \frac{1}{\det J} \begin{bmatrix} J_{22} & -J_{12} & 0 & 0 \\ 0 & 0 & -J_{21} & J_{11} \\ -J_{21} & J_{11} & J_{22} & -J_{12} \end{bmatrix} \quad [D] = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{(1-\nu)}{2} \end{bmatrix}$$

Figure 2: 'A' matrix formulation

Figure 3: Plane stress elasticity matrix

The 'A' matrix is then calculated as seen in fig. 2, in the subroutine *CALCULATE.A*.

The Jacobians are calculated as:

$$J_{11} = \sum_{i=1}^8 s_i \times x_i \quad J_{12} = \sum_{i=1}^8 s_i \times y_i$$

$$J_{21} = \sum_{i=1}^8 t_i \times x_i \quad J_{22} = \sum_{i=1}^8 t_i \times y_i$$

where x'_i s and y'_i s are the coordinates of the nodes of the element in global frame.

The following operations are then carried out successively.

$$[B] = [A][G]$$

$$[C] = [D][B]$$

$$[KI] = [B^T][C]$$

Based on displacements at the nodes the stress and strain are calculated as shown below.

$$[\epsilon] = [B][du]$$

$$[\sigma] = [D][d\epsilon]$$

Finally, the elemental stiffness matrix is calculated by:

$$[K_{ele}] = [K_{ele}] + [KI] \times wt(ipt) \times \det J$$

The body forces are not presently considered.

The elemental stiffness matrices are assembled based on the nodal connectivity of each element and the boundary conditions to get the global elemental stiffness matrix. [1]

Simulation details

The code is checked with test cases for deformation of a plate descritized with 1 element, 2 elements and 4 elements, and displacement boundary conditions. The input files and results are in the respective input and output files.

Then, a plate hole geometry is generated in ABAQUS [3] and the node and element information are ported into the input file for the FEM solver.

Details:

- Number of nodes : 454
- Number of elements : 404
- Plate size : 12*12
- Bottom nodes are en-castrated
- Top nodes are displaced in the y direction by +3 units

Results

The x-displacement plots from ABAQUS 4 as well as the FORTRAN code 5 show near matching qualitatively. Quantitatively, there is a small error of 3%. The same cannot be said about the y-displacement plots 6 and 7. Though the region wise there is a match of displacements, the contours are not properly seen, to be able to make better assessments.

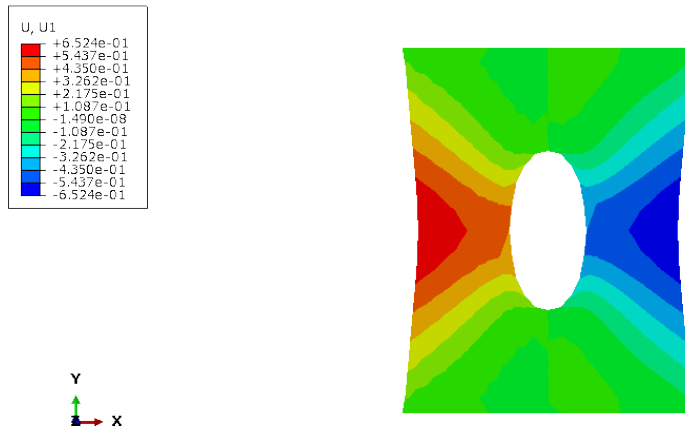


Figure 4: Displacement contour in x direction, ABAQUS

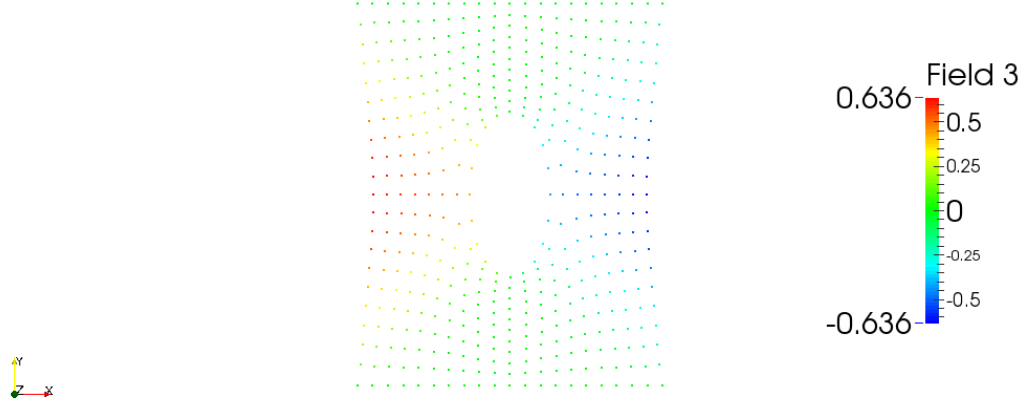


Figure 5: Displacement contour in x direction,FORTRAN code

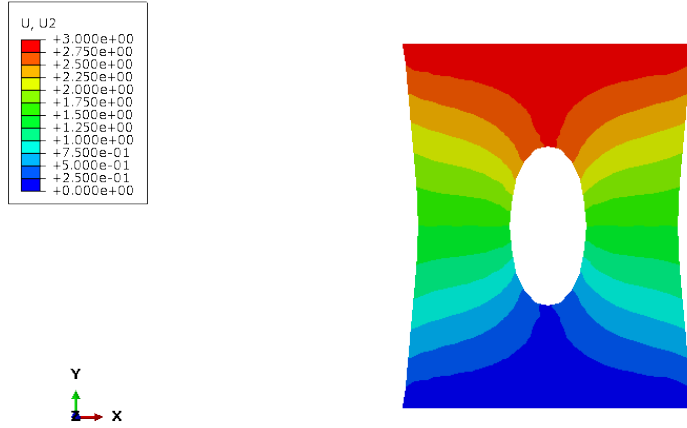


Figure 6: Displacement contour in y direction,ABAQUS

The stresses and strains at the integration points are also obtained, but these do not show sufficient agreement with the ABAQUS results.

References

- [1] I. M. Smith, D. V. Griffiths, Programming the finite element method, John Wiley & Sons, 2005.
- [2] K. Entwistle, Basic principles of the finite element method, IOM Communications, 1999.
- [3] Hibbit, Karlsson, Sorensen, ABAQUS/Standard Analysis User's Manual, Hibbit, Karlsson, Sorensen Inc., USA, 2007.

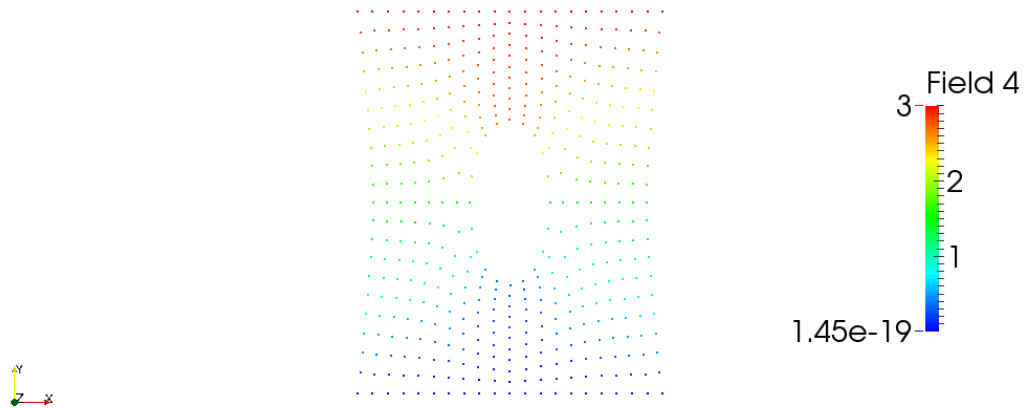


Figure 7: Displacement contour in y direction,FORTRAN code

Details of code

Main file - 79 lines

Number of subroutines - 15

Functions file (Containing subroutines) - approximately 350 lines