ENGN 2340

Computational Methods in Structural Mechanics

Generalized framework for solving 2D FE problems

Karthik Thalappully

Overview

The goal of the project was development of a FEA framework in C++ to solve a 2D contact problem. The project consists of the following two parts.

a) Implementation of a simple 2D FEA solver in C++ with the following capabilities.
   - Read an Abaqus style input file for model and history data.
   - Solve the system of equations.
   - Write an Abaqus ODB for visualizing results.

b) Expand the capabilities of the above framework to solve the following 2D contact problems.
   - Contact of one body with a rigid obstacle.
   - Contact of two deformable bodies.

Only part –a was implemented as part of this project.

Governing Equations [1]

The following governing equations will be solved for the contact problem to calculate the displacement field.

1. Linear momentum balance

$$\sigma_{kj,j}^{(i)} + f_k = \rho\, \ddot{u}^{(i)}{}_k$$

   Where the subscript (*I*) denotes body index = 1,2.

2. Initial and boundary conditions

$$\sigma_{kj}^{(i)} n_j^{(i)} = t_k^{(i)} \text{in } S_t$$

$$u_k^{(i)} = \bar{u}^{(i)}{}_k \ at\ S_d$$

   Where S_t and S_d represents the surfaces with prescribed traction and displacement boundary conditions respectively.

3. Strain displacement relations

$$\epsilon_{kj}^{(i)} = 0.5 * (u_{k,j}^{(i)} + u_{j,k}^{(i)})$$

4. Linear Elasticity (Constitutive equation)

$$\sigma_{mn}^{(i)} = C_{mnkl}^{(i)}\, \epsilon_{kl}^{(i)}$$

5. Contact conditions.

$$t_N \geq 0$$
$$g \leq 0$$
$$t_N * g = 0$$

Where $t_N(\omega) = -n(\omega).\sigma(\omega)n(\omega)$

$$g(\omega) = g_0 + u(\omega)n(\omega)$$

Here t_N represents the tractions generated in the body at the contact surface $\omega \in S_{contact}$ and g represents the gap (minimum distance) between a point on body 1 and the nearest point on body2.

Assumptions

a) The major assumption here for the contact problem is that the contact point on the impacting surface and the surface normal is not a function of the displacement.
b) The normals at the contacting point of the impacting bodies are assumed to be the same.
c) No friction in contact.

Implementation

The FEA frame work was developed in C++ with the use of the GNU Scientific library. The Abaqus python ODB API was used to generate the odb for visualizing the final results.

Supported Abaqus Keywords

- Model Data: - The following model data keywords are supported.
  *NODE (Array of node Id and the spatial coordinates.)
  *ELEMENT (Array of Element Id and the nodal connectivity)
  *NODE SET (Any set of nodes which can be used in other features)
  *ELEMENT SET (Any set of element which can be used to define other features)
  *AMPLITUDE (time dependent scaling to be applied to loads and boundary conditions)

- History Data: - History data describes the analysis to be performed and the loads and boundary condition on the model. The following keywords are supported by the framework.

  *STEP (defines the type of analysis and time parameters – Only *static works currently).

*CLoad (Concentrated force on the nodes/node sets).
*Boundary (displacement condition the nodes/node sets).

- Solver: -

  The default sparse solver from the GNU Scientific Library was used to solve the system of equations. The solver uses an iterative scheme using generalized minimal residual method (GMRES). No preconditioners have been implemented for the solver.

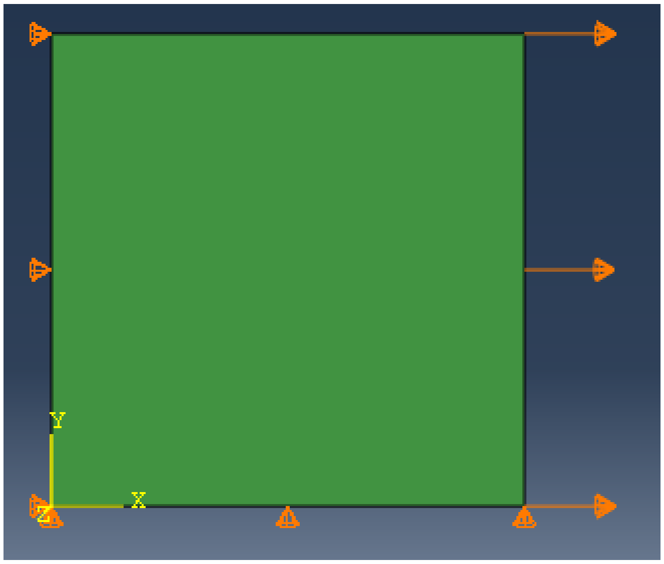
- Results: -
  The results were generated in the Abaqus ODB format using the Python Abaqus Scripting Interface. Only displacements and Stress outputs are currently supported.

Test Cases

Currently only linear elastic FEA is supported in the framework. All test cases were performed against solutions from ABAQUS FEA. Both displacements and the stresses were compared.

1. Single Element Test – With Prescribed displacements.

   Scenario – A fixed displacement was prescribed on the nodes on the right hand side as shown below and the nodes on the bottom and the left hand sides were constrained on along x and y respectively.



   Results :- The following results were obtained. The left hand image shows the results calculated using the framework and the right hand image shows the results from Abaqus FEA.

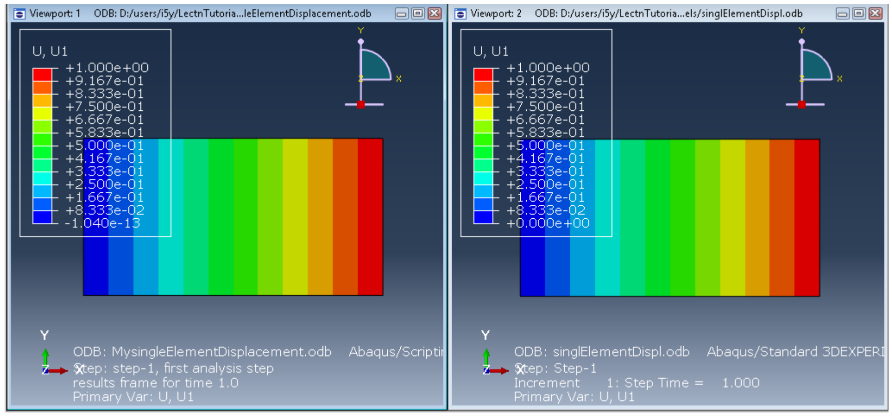


Figure 1 Comparison of U for a single 8 noded plane strain element.

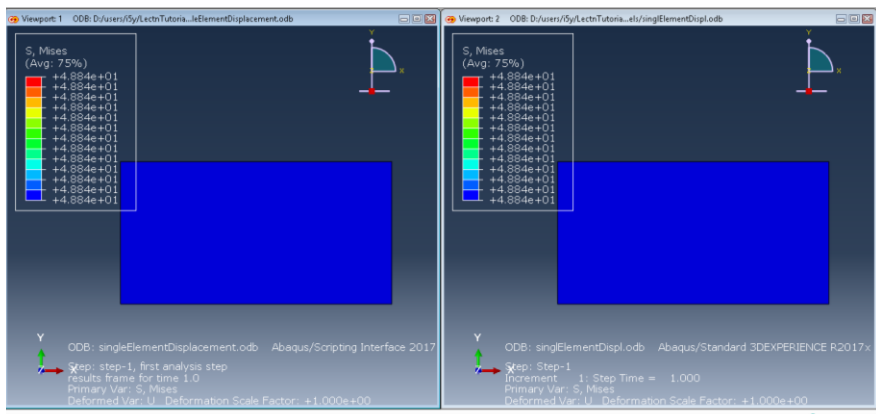
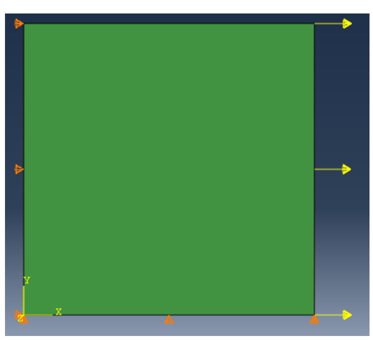


Figure 2 Comparison of Stress for a single element 8 noded plane strain element

2. Single Element Test – Prescribed Force.

Scenario:- A force was prescribed on the nodes on the right as shown below.



Results:- The following results were obtained. The left hand image shows the results calculated using the framework and the right hand image shows the results from Abaqus FEA.

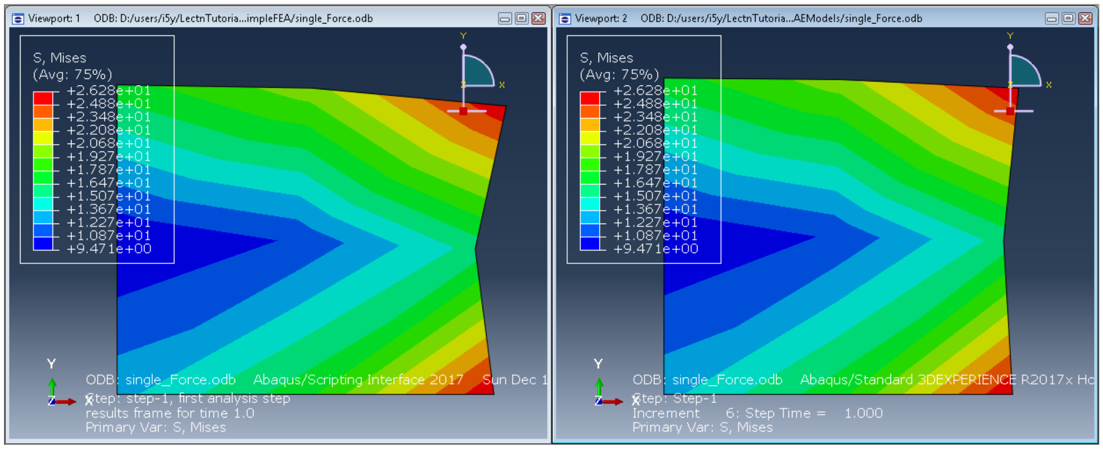


Figure 4 Comparison of Stress for a single element 8 noded plane strain element

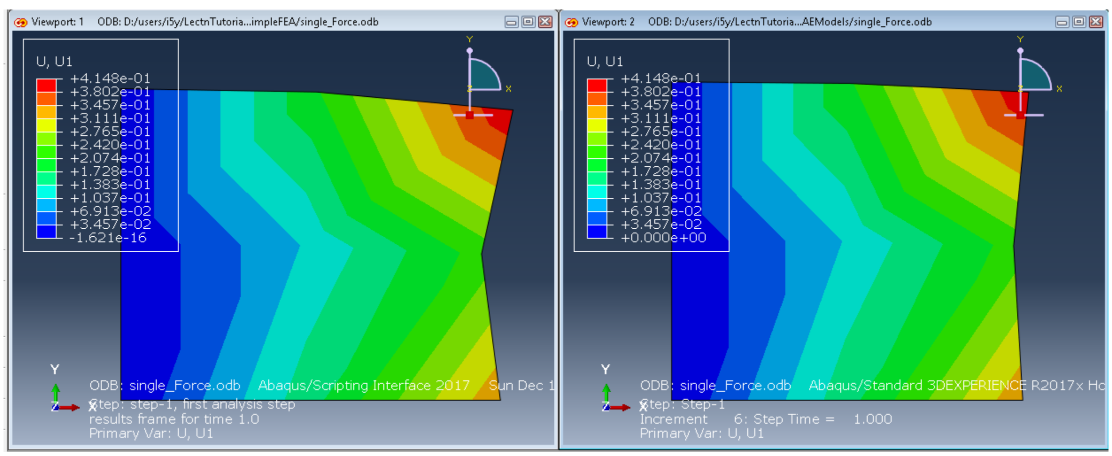
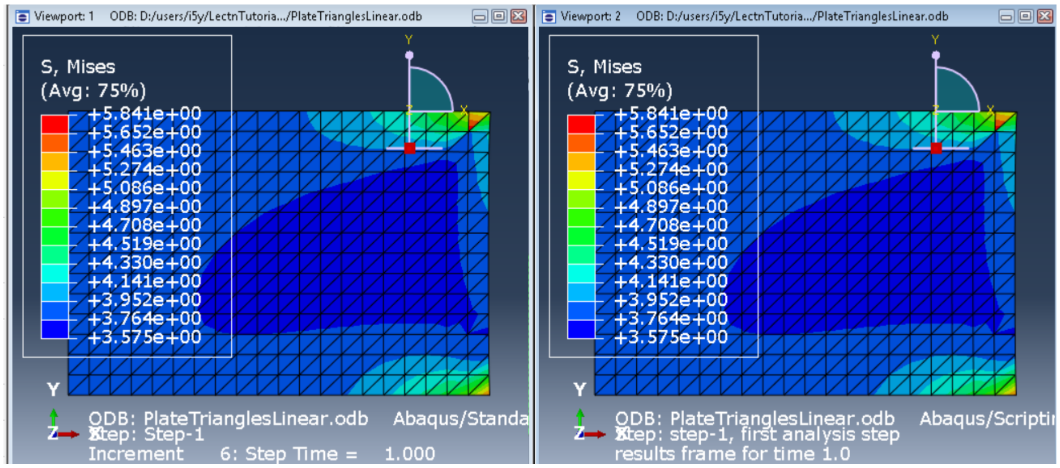


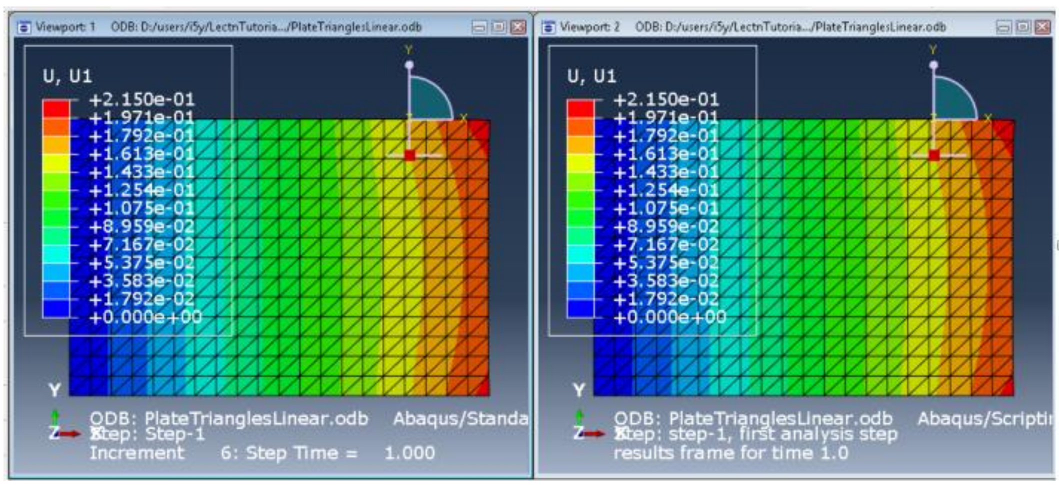
Figure 3 Comparison of displacements for a 8 noded plane strain element

3. Tests multi-element model with linear triangles.

   Scenario – A force load was applied to the right hand side of a multi-element model (560 elements).

   Results :- The following results were obtained for stress and displacements. The left hand image shows the results from Abaqus and the right hand image shows the results from the framework. Both the results were found to be in good agreement.





Conclusion

Currently the framework can only do linear FEA problems. For the various test cases performed using test cases the results were found to be matching exactly with Abaqus results.

Bibliography

[1] T.A.Laursen, Computational Contact and Impact Mechanics, 2002.