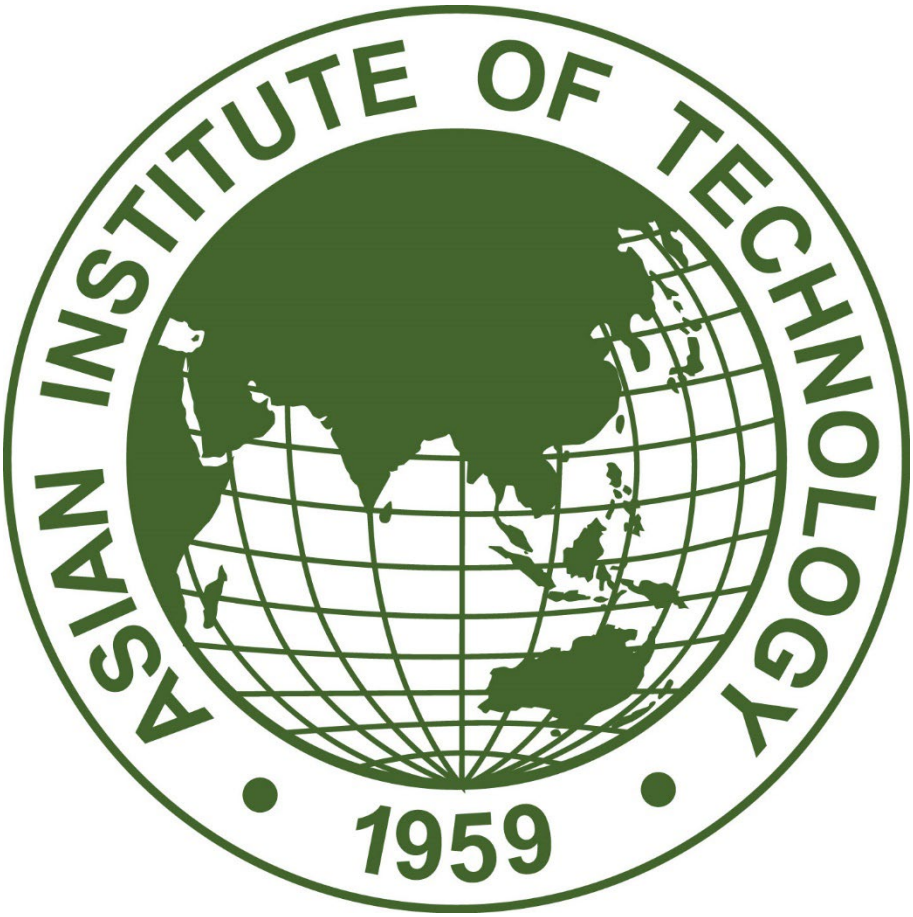


Finite Element Methods Group project

Application of cyclic loading on a cantilever column

Submitted by:

Group 02

PoPo Han Myint	st124271
Jutamas Taweesint	st124010
Peerawut Watsaratiyanont	st124069
Chhayleang Ly	st124112
Saurav Kumar Shrestha	st124344

Introduction

Our project is about applying a cyclic load at the top of a 3D cantilever column. The displacements required were in the multiple of 0.12m, both in the positive and the negative direction, which had to be applied in a number of load-steps. The challenge was that we had to do the unloading as well which was extremely tricky to figure out. Luckily, the stress-strain relationship for out project was linear. Therefore, we decided to just simply create load displacement data in a separate file and then call it to do the loading as well as the unloading part.

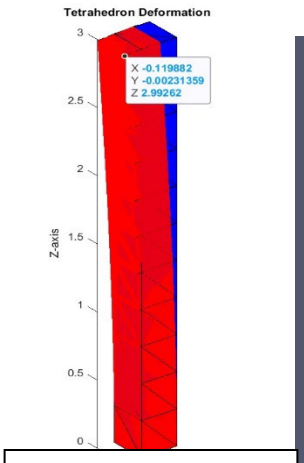
Objective

- To study the behavior of a cantilever beam exposed to cyclic load.
- To learn about linear tetrahedral elements and 3D FEM
- To learn about nature of cyclic load (how the loading and unloading part works)

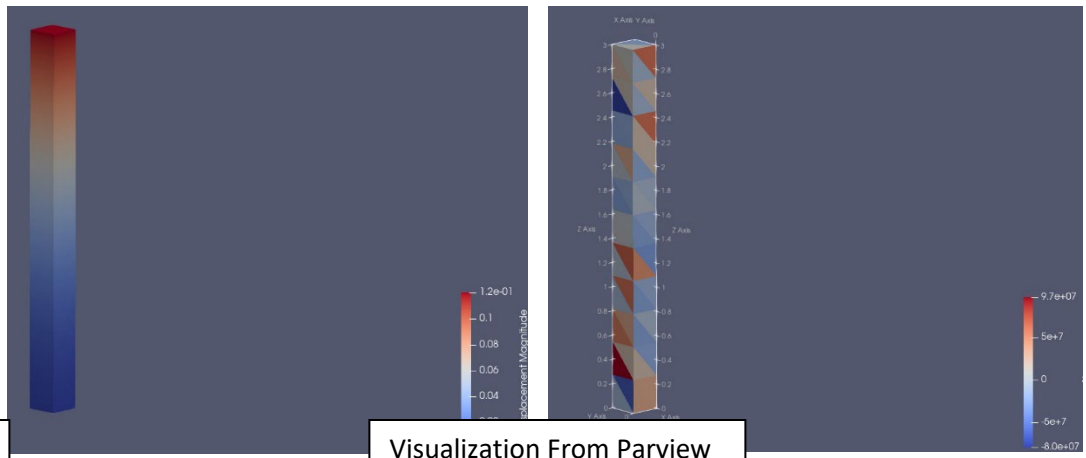
Methodology

- We started with hand-meshing to understand how a structure is discretized into smaller tetrahedral elements
- We moved on to GMSH to make a 3D plot of our column and discretize it to many elements. We limited our number of elements to a minimum to reduce the computational cost of our project.
- We then started the derivations of our shape functions, Jacobian, B-matrix, stiffness matrix and the force vector.
- We made a simple strategy and algorithm to break the problem down into smaller sections to make things easier for ourselves. (Preprocess, Solver and Post process)
- In the preprocess we defined our material properties, section size and element type along with the gmsh input file to make a 3D plot of our structure.
- In the solver part we solved the K*U=F equation to find the nodal displacement and force at each iteration or the step increment.
- In the post processor part we interpreted and visualized the result, we found the stress in addition to making a stress contour plot.

Results:



Deformation Result



Visualization From Parview