NOVEMBER 14, 2021

ME4291 ASSIGNMENT 2

T3 VS T6 FEM CODE

GROUP 13

Goh Kheng Xi, Jevan	A0199806L
Lock Mei Lin	A0204751M
Tay Wei Le	A0124570M
Thin Rupar Win	A0213976W

1. Objective statement

Conduct a static FEM analysis for the displacement field of a 2D bar under plane stress using 3 and 6 node triangle elements.

2. Introduction

In this report, we attempt to solve a FE problem of a side loaded aluminum rod that is fixed to the wall on one end and loaded on the other. The rod is of length 0.3m and a square cross section of 0.1m x 0.1m. The rod is loaded at the top edge on the opposite end with a downward force of 5×10^8N. The unrealistically high load is meant to make the displacement more obvious, and would most likely not operate in the elastic regime or fracture in real life.

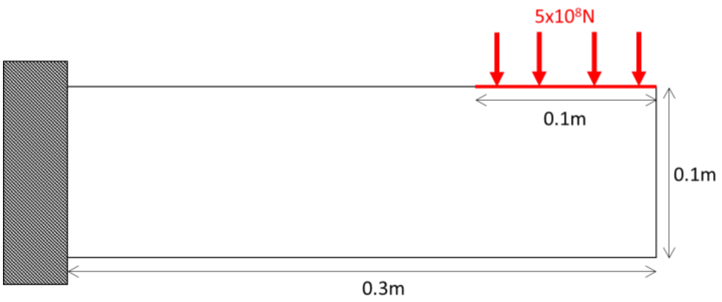


Figure 1. Diagram of FE problem.

To solve this FE problem, we manually meshed the rod using 3-node triangle element with 6-node triangle element and keeping the number of nodes in both cases the same. The material properties were obtained from solidworks using 1060 aluminum alloy which has a Poisson's ratio =0.33 and young's modulus = 6.9×10^{10} Pa. The fixed end of the rod was modelled using the Dirichlet boundary condition whereby the x and y displacement at the nodal points are set to 0. 4-point Gauss Quadrature for triangles is used to solve the numerical integration in the stiffness matrix.

3. 3-node Triangle Element (T3)

The 3-noded triangular element is a 2d linear shape, with 6 displacement degrees of freedom (DOFs) per element, as each node can move in the x and y directions. The three nodes are at the vertices of the triangle, which are numbered in an anti-clockwise direction. Figure 2 shows the mesh of the FE model comprising of 24 elements. The load is applied across nodes 4, 5, 6.

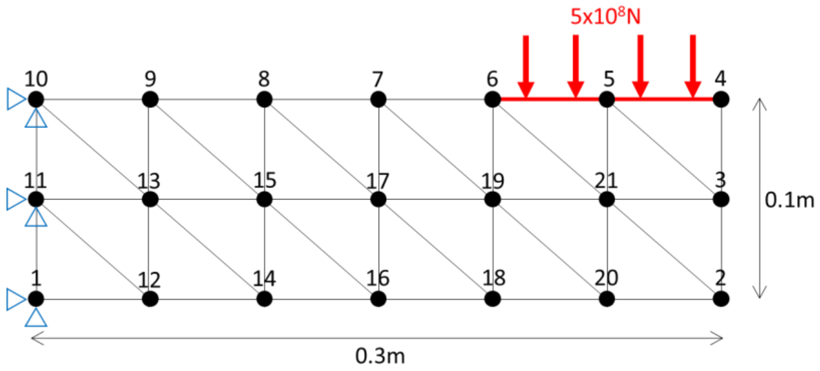


Figure 2. Schematic of T3 FE Model.

The shape functions for the 3-noded triangle element are given as:

$$N_1 = \frac{1}{2A}\{x_2y_3 - x_3y_2) + (y_2 - y_3)x + (x_3 - x_2)y\}$$

$$N_2 = \frac{1}{2A}\{x_3y_1 - x_1y_3) + (y_3 - y_1)x + (x_1 - x_3)y\}$$

$$N_3 = \frac{1}{2A}\{x_1y_2 - x_2y_1) + (y_1 - y_2)x + (x_3 - x_1)y\}\text{a}$$

and

$$A = \frac{1}{2}det\begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix}$$ is the area of the triangle element

The FEM model was ran on the base code in the file Triangle_Elem_Complete that was provided.

3.1. Results

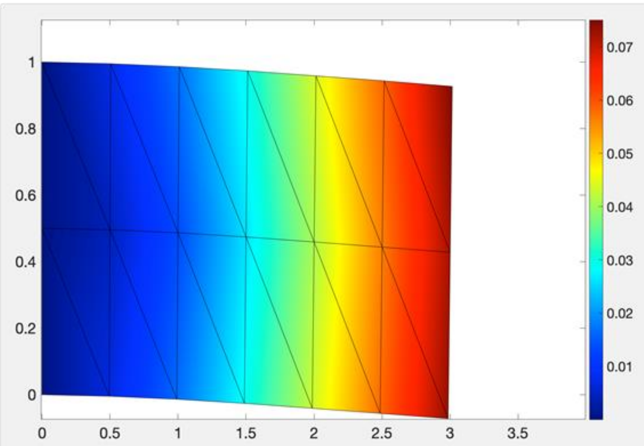


Figure 3. Displacement Results for T3 (Left). Stress Results for T3 (Right).

Figure 3 shows the euclidean displacement plotted as a colourmap. The max Euclidean displacement is 0.375m at node 4 where the x-displacement is 0.0796 and y-displacement is -0.367m.

4. 6-node Triangle Element (T6)

The model was meshed using a T6 element using a total of 6 elements as shown in Figure 4. Nodes 1, 2 and 3 were fixed with their displacement in the x and y direction set to zero. The load was applied across nodes 13,16 and 19.

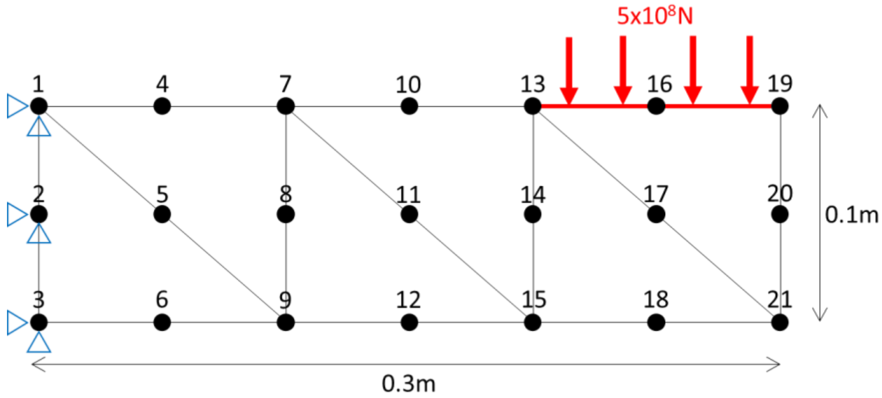
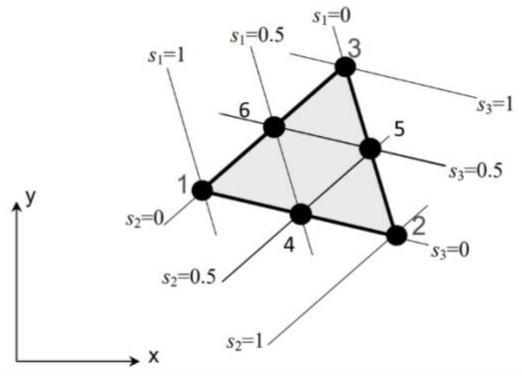


Figure 4. Schematic of T6 FE Model.

The base code in the file Triangle_Elem_Complete was used as a starting point. The main implementation of the T6 element is in the elemental stiffness matrix generation function (renamed to T.m) and Main_triangle.m was only amended slightly to be able to call the new function. Similar to the T3 element, the elemental stiffness matrix is given by K_e and the strategy to evaluate it will be to convert the integrand to intrinsic coordinates before solving it numerically via numerical integration using the Gauss formula. Thus, the goal is to obtain matrix C, B and |J| (determinant of Jacobian matrix). The next part will cover the formulation for matrix B and J. The formulation for the C matrix is identical to the original T3 code.

$$K_e = \iint_{x\text{-}y} B^T C h B \, dA_{x\text{-}y} \xrightarrow[\text{intrinsic coordinates}]{\text{Transformation to}} K_e = \iint_{s_1\text{-}s_2} B^T C h B |J| \, dA_{s_i} \xrightarrow[\text{(Gauss Formula)}]{\text{Numerical Integration}} K_e = 0.5h \sum_{i=1}^{n} W_i \, (B^T C B)_i \, |J|_i$$

To start off, we defined N, the shape functions of the T6 element, in intrinsic coordinates of s1, s2, s3. In the elemental stiffness matrix, the nodes within the element are ordered in an anticlockwise direction with the apex of the triangles ordered first followed by the side as per Figure 5.



Shape Function

$N_1 = s_1(2s_1\text{-}1)$
$N_2 = s_2(2s_2\text{-}1)$
$N_3 = s_3(2s_3\text{-}1)$
$N_4 = 4s_1s_2$
$N_5 = 4s_2s_3$
$N_6 = 4s_1s_3$
Where $s_3 = 1\text{-}s_1\text{-}s_2$

Figure 5. Naming convention and shape functions of T6 element.

The matrix B is derived from the stress-strain and strain-displacement relations in 2D.

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{Bmatrix} u_x \\ u_y \end{Bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{bmatrix} N_i & 0 & N_j & 0 & N_m & 0 \\ 0 & N_i & 0 & N_j & 0 & N_m \end{bmatrix} \begin{Bmatrix} u_{xi} \\ u_{yi} \\ u_{xj} \\ u_{yj} \\ u_{xm} \\ u_{ym} \end{Bmatrix}$$

$$\underbrace{\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix}}_{\boldsymbol{\varepsilon}} = \underbrace{\begin{bmatrix} \frac{\partial N_i}{\partial x} & 0 & \frac{\partial N_j}{\partial x} & 0 & \frac{\partial N_m}{\partial x} & 0 \\ 0 & \frac{\partial N_i}{\partial y} & 0 & \frac{\partial N_j}{\partial y} & 0 & \frac{\partial N_m}{\partial y} \\ \frac{\partial N_i}{\partial y} & \frac{\partial N_i}{\partial x} & \frac{\partial N_j}{\partial y} & \frac{\partial N_j}{\partial x} & \frac{\partial N_m}{\partial y} & \frac{\partial N_m}{\partial x} \end{bmatrix}}_{\mathbf{B}} \underbrace{\begin{Bmatrix} u_{xi} \\ u_{yi} \\ u_{xj} \\ u_{yj} \\ u_{xm} \\ u_{ym} \end{Bmatrix}}_{\mathbf{u}}$$

To obtain B and J, we differentiate N with respect to s_1 and s_2 and assemble the results in a matrix B_0.

$$\mathbf{B}_o = \begin{bmatrix} \frac{\partial N_1}{\partial s_1} & \frac{\partial N_2}{\partial s_1} & \frac{\partial N_3}{\partial s_1} & \frac{\partial N_4}{\partial s_1} & \frac{\partial N_5}{\partial s_1} & \frac{\partial N_6}{\partial s_1} \\ \frac{\partial N_1}{\partial s_2} & \frac{\partial N_2}{\partial s_2} & \frac{\partial N_3}{\partial s_2} & \frac{\partial N_4}{\partial s_2} & \frac{\partial N_5}{\partial s_2} & \frac{\partial N_6}{\partial s_2} \end{bmatrix}$$

The Jacobian is obtained by multiplying B_0 with the x-y nodal coordinates.

$$\mathbf{J} = \begin{bmatrix} \frac{\partial N_1}{\partial s_1} & \frac{\partial N_2}{\partial s_1} & \frac{\partial N_3}{\partial s_1} & \frac{\partial N_4}{\partial s_1} & \frac{\partial N_5}{\partial s_1} & \frac{\partial N_6}{\partial s_1} \\ \frac{\partial N_1}{\partial s_2} & \frac{\partial N_2}{\partial s_2} & \frac{\partial N_3}{\partial s_2} & \frac{\partial N_4}{\partial s_2} & \frac{\partial N_5}{\partial s_2} & \frac{\partial N_6}{\partial s_2} \end{bmatrix} \begin{bmatrix} x_1 & y_1 \\ x_2 & y_2 \\ x_3 & y_3 \\ x_4 & y_4 \\ x_5 & y_5 \\ x_6 & y_6 \end{bmatrix}$$

$|J^{-1}|B_0$ gives the values of elements in the B matrix that can be rearranged into the correct order. After which, the integrand is evaluated as per the original code using Gaussian Quadrature.

$$\mathbf{J}^{-1}\mathbf{B}_o = \begin{bmatrix} \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial x} & \frac{\partial N_3}{\partial x} & \frac{\partial N_4}{\partial x} & \frac{\partial N_5}{\partial x} & \frac{\partial N_6}{\partial x} \\ \frac{\partial N_1}{\partial y} & \frac{\partial N_2}{\partial y} & \frac{\partial N_3}{\partial y} & \frac{\partial N_4}{\partial y} & \frac{\partial N_5}{\partial y} & \frac{\partial N_6}{\partial y} \end{bmatrix}$$

4.1. Results

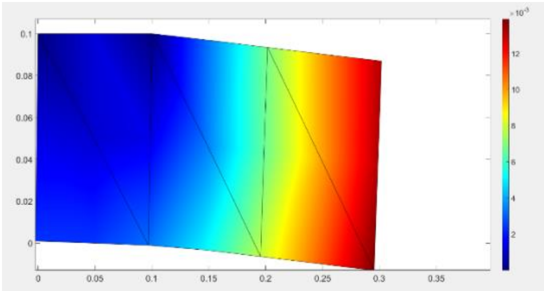


Figure 6. Displacement Results for T6 (Left).

From Figure 6, it can be observed that the maximum displacement occurs at the bottom right of the beam indicated by the dark red region which corresponds to a displacement of 0.00467m in the negative x direction and 0.0131m in the negative y direction. With reference to table, this maximum displacement point is represented by node 21.

5. Discussion

In this experiment we kept the DOF of the FE models the same. As a convergence study was not done, we are unable to ascertain experimentally whether the T3 or T6 element provides more accurate displacement values. Based on literature values, T6 elements can converge faster than T3 elements at the same DOF. However, it is observed that due to the increased number of elements in the T3 mesh, it can better replicate deformation as compared to the T6.

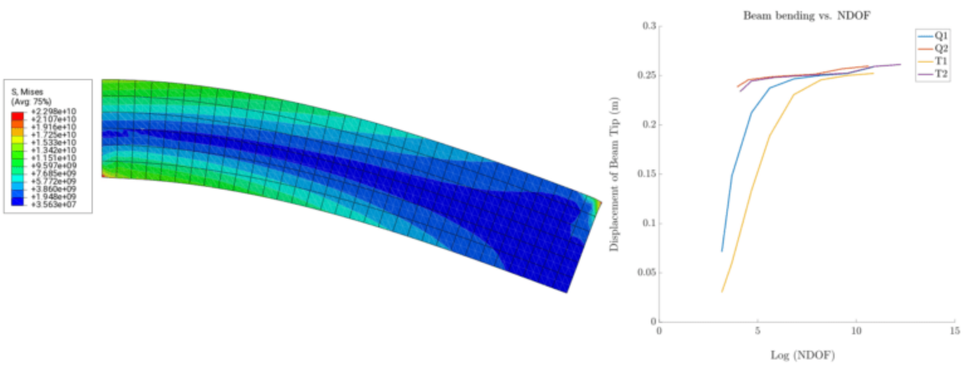


Figure 7. Literature comparison between T1 and T2 mesh (T3 and T6 in this report) [1].

Since the displacement solution is solved with a linear plane of deformation, the strain and stress over the whole element does not change. However, the stress will change along the 2D bar structure. Hence, the 3-noded triangular elements have the tendency to overestimate the stiffness of the structure and thus, underestimate the stress.

The loading force was varied for both models to observe the change in the maximum displacement.

Loading/N	T3	T6
5e2	3.75e-7	1.39e-8
5e4	3.75e-5	1.39e-6
5e8	0.375	0.0139

6. Conclusion

There is a significant difference in using the 3-noded triangular element and the 6-noded triangular element as both provided different displacement solutions as evident in the different maximum displacement values. In addition, the location of the maximum displacement differs between the two study. However, as convergence study was not done, it is difficult to conclude which element produced the more accurate solution.

Statement of contribution:

Everyone contributed to the debugging and the formulation of the code and report, as well as the slides.

Annex A: Elemental Stiffness Matrix Code

```
function Ke = T(node_1_coord, node_2_coord,
node_3_coord,node_4_coord,node_5_coord,node_6_coord )
%input 6 coordinates from main_triangle.m in the form of (x,y)

%Combines all input coordinates into a matrix, hardcoded initialize E, nu, h values.
coord = [node_1_coord;
        node_2_coord;
        node_3_coord;
        node_4_coord;
        node_5_coord;
        node_6_coord];

E = 6.9e10;
nu =0.33;
h = 0.1;
```

```
%Form C*h matrix
Ch = [E*h/(1-nu^2),    nu*E*h/(1-nu^2), 0;
      nu*E*h/(1-nu^2), E*h/(1-nu^2),    0;
      0,               0,               E*h/(2*(1+nu))];

%Shape functions in intrinsic coordinates s1,s2,s3
syms s1 s2
s3 = 1-s1-s2;
N1 = s1*(2*s1-1);
N2 = s2*(2*s2-1);
N3 = s3*(2*s3-1);
N4 = 4*s1*s2;
N5 = 4*s2*s3;
N6 = 4*s1*s3;

%Differentiate shape functions w.r.t. intrinsic coordinates
dN1_s1 = diff(N1,s1); dN1_s2 = diff(N1,s2);
dN2_s1 = diff(N2,s1); dN2_s2 = diff(N2,s2);
dN3_s1 = diff(N3,s1); dN3_s2 = diff(N3,s2);
dN4_s1 = diff(N4,s1); dN4_s2 = diff(N4,s2);
dN5_s1 = diff(N5,s1); dN5_s2 = diff(N5,s2);
dN6_s1 = diff(N6,s1); dN6_s2 = diff(N6,s2);

%Form matrix B0 which contains all the differentiated shape functions
B0 = [dN1_s1 dN2_s1 dN3_s1 dN4_s1 dN5_s1 dN6_s1;
      dN1_s2 dN2_s2 dN3_s2 dN4_s2 dN5_s2 dN6_s2];

%B0*coord converts dN/ds1 into dN/dx which is the Jacobian matrix
J =  B0 * coord;

%Forming the B matrix
temp = det(J)\B0;

B=[temp(1,1),0,temp(1,2),0,temp(1,3),0,temp(1,4),0,temp(1,5),0,temp(1,6),0;
0,temp(2,1),0,temp(2,2),0,temp(2,3),0,temp(2,4),0,temp(2,5),0,temp(2,6);
temp(2,1),temp(1,1),temp(2,2),temp(1,2),temp(2,3),temp(1,3),temp(2,4)
temp(1,4),temp(2,5),temp(1,5),temp(2,6),temp(1,6)];

%Initialize Gauss point data set
Gauss_data = load('GaussTri_04.txt');
Gauss_point_1 = Gauss_data(:,1);
Gauss_point_2 = Gauss_data(:,2);
Gauss_weight = Gauss_data(:,3);

%Set space in elemental stiffness matrix Ke
Ke = zeros(12,12);

%Loops to multiply
for j=1:length(Gauss_weight)
   digits(5);
   sum = 0.5 * Gauss_weight(j) * subs (B' * Ch * B * det(J),[s1 s2], [Gauss_point_1(j)
Gauss_point_2(j)]);
   Ke = Ke+ vpa(sum);
end

%Truncate values to 5 digit length
vpa(Ke);
end
```

References

[1]"What is convergence in finite element analysis? Simscale." SimScale, 10-Mar-2020. [Online] Available: https://www.simscale.com/blog/2017/01convergence-finite-element-analysis/.[Accessed: 14-Nov-2021].