MAE 503

FINITE ELEMENTS IN ENGINEERING

SPRING 2022

PROJECT #1

28/03/2022

RESTATING THE PROBLEM:

Problem Statement:

We have an Aluminum rod that is fixed between two rigid plates which are at a fixed temperature of 293 K, and we have current passing through the rod. This can be considered a Heat Transfer problem in Finite Element. Given the property of the material, the aluminum rod will offer resistance to the current, which in result will acts as a source of internal heat generation within the rod. At the ends, we have heat flux which is a result of convection occurring on the end, at the left end it's from the fixed plates to the rod and at the other end it's from the rod to the fixed plate. So, we basically we have heat flux boundary conditions at the two ends of the rod and internal heat generation which will cause a temperature distribution over the length of the rod. At the same time, since the rod is being heated, there would stress induced in the rod reason being the rod would try to expand because of the change in temperature distribution, but at the same time it is being fixed, so the resulting strain would be the difference between the two. So, using the finite element method we will be solving for the temperature firstly. Then, using the temperature distribution we will compute stress for the aluminum rod.

Methodology:

1. We would first model our problem and then start with finding the temperature distribution over the length of the rod. Knowing the heat flux boundary conditions at the two end we can solve and find the temperature over the length of the rod.
2. Once we found out the temperature distribution, we can then use this to find the strain, we can solve for strain given the fixed boundary conditions at the end of the rod. Once we have computed the strain, we can then move on to compute the stress generated within the body.

Strong form of the governing equation:

$\frac{d}{dx}$ (A K $\frac{dt}{dx}$) + S = 0,

where.

A is the area.

S is the internal heat generation within the element.

K is the thermal conductivity.

This is the strong form for heat equation.

Now,

For the Elasticity strong form.

$$\frac{d}{dx}\left(EA\left(\frac{du}{dx} - \alpha\Delta T\right)\right) = 0,$$

where.

E is the young's modulus.

α is the thermal heat transfer coefficient.

ΔT is the temperature difference between the body and the plate temperature.

This is a no body force in the strong form as we assume it to be very less and then solve the following strong form.

SPECIFYING THE UNITS:

Units:

We have used the following consistent units for Mass, Length, time, force, stress, and energy while modeling the problem both in Abaqus and MATLAB.

MASS	LENGTH	TIME	FORCE	STRESS	ENERGY
kg	m	s	N	Pa	J

Weak form of the governing equation:

1. Firstly, we start with the strong form

$$\frac{d}{dx}\left(A\,K\,\frac{dt}{dx}\right) + S)\ dx = 0,$$

Now, we multiply strong form by a test function and integrate the following.

$$\int_0^L w\left(\frac{d}{dx}\left(A\,K\,\frac{dt}{dx}\right) + S\right)$$

where.

W is the test function.

2. Integrating by parts.

$$\int_0^L \frac{d}{dx}\left(w\,A\,K\,\frac{dt}{dx}\right) - \frac{dw}{dx}\,A\,K\,\frac{dt}{dx} + w\,S\ dx = 0$$

we get,

$\int_0^L \frac{dw}{dx}\, A\, K\, \frac{dt}{dx}$ dx = $\int_0^L w\, S$ dx + w A q/q (boundary condition on q)

where.

q is the convective heat flux we have at the boundary.

q = $\overline{-h}$ (T – T∞)

$\therefore \int_0^L \frac{dw}{dx}$ A K $\frac{dt}{dx}$ dx = $\int_0^L w\, S$ dx + w A $\overline{-h}$ (T – T∞)

However, the LHS of the equation can be written as;

$B^e = \frac{dNe}{dx} = \frac{dNe}{dP} / J$

$$w^T \sum_0^e (L^e) \int_{x1e}^{x2e} \underbrace{(B^e)^T\ A\ K\ B\ dx}_{K_e}.\ L^e\ .\ T$$

Now,

$\int_{x1e}^{x2e} (B^e)^T$ A K B^e dx = K^e (Element conductivity matrix),

This becomes our conductivity matrix for each element. For the whole body, we can loop over the connectivity matrix and find the conductivity of the whole body.

$\int w\, S\, dx$ can be written as;

$w^T \sum_0^e . \int_{x1e}^{x2e} (L^e)^T (N_e^T).$ S*A dx $\longrightarrow$ (we multiply by area to make the units compactible)

Where;

F_e is the term used for heat flux vector for each element.

If we loop over to the connectivity matrix, we can find it for the whole body.

Now, we have also have this term on the RHS:

-w A h (T-T∞)

This can be written as;

$w^T N^T$ A h N T/q (boundary condition at q) @x=0 &x=L

This can be considered as boundary condition which we apply on the stiffness matrix.

@x=0, $N^T N$ gives a global conductivity matrix with the first element of matrix one.

So, this can be written as;

K (1,1) = K (1,1) + A(h) $\longrightarrow$ at x=0

Similarly, at x = L;

$N^T N$ gives a global conductivity matrix with the last element of matrix one.

K (L, L) = K (L, L) + A(h)

(This index however corresponds to the last node number)

Similarly for the heat flux vector this can be written as;

f (1) = f (1) + A(h)*293 $\longrightarrow$ at x=0

f (L) = f(L) + A(h)*293 $\longrightarrow$ at x=L

Now, we have built both our element conductivity matrix and heat flux vector. So, we can solve for T as follows;

KT = f

∴ T = K\f

Here, we will get the temperature distribution over the body.

Now, we have the following as a strong form for Elasticity;

$$\frac{d}{dx}(E\,A\,(\frac{du}{dx} - \alpha\,\Delta T)) = 0$$

For the weak form, we multiply strong form by w and integrate.

$$\therefore \int_0^L w\,\frac{d}{dx} \Big|\; E\,A\,(\frac{du}{dx} - \alpha\,\Delta T) = 0$$

We integrate by parts and then get the final equation as.

$$\therefore \int_0^L \frac{dw}{dx}\,EA\,(\frac{du}{dx} - \alpha\Delta T)\,dx = 0$$

$$\therefore \int_0^L \frac{dw}{dx}\,EA\,\frac{du}{dx}\,dx = \int_0^L \frac{dw}{dx}\,E\,A\,\alpha\Delta T\,dx$$

$$W^T \sum_0^e L^{e\,T} \int_{x1e}^{x2e} (B^e)^T {}^* E\,A\,(B^e)dx\,(L^e)d = w^T \sum_0^e (L^e)^T \int_{x1e}^{x2e} (B^e)^T\,E\,A\,\alpha\,\Delta T$$

Now,

$$\int_{x1e}^{x2e} (B^e)^T\,E\,A\,(B^e)dx \longrightarrow K^e$$

This is our element stiffness matrix. Now, if we integrate over the connectivity matrix, we will get stiffness matrix for the whole body.

$\int_{x1e}^{x2e} (B^e)^T \, E \, A \, \alpha \, \Delta T \longrightarrow f^e$

This is our element force vector. Now, if we integrate over the connectivity matrix, we will get stiffness matrix for the whole body.

For boundary condition d is fixed on both the side.

∴ For the force vector

f (1) = 0

f(L) = 0

This corresponds to the last node number

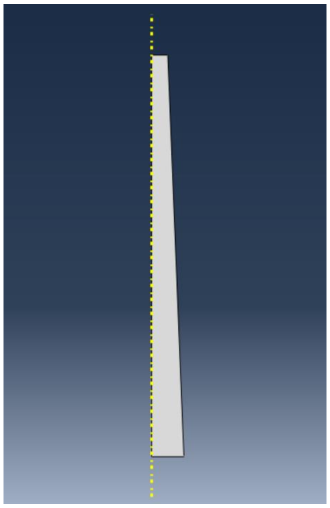
By using this formula, we can find our displacements.

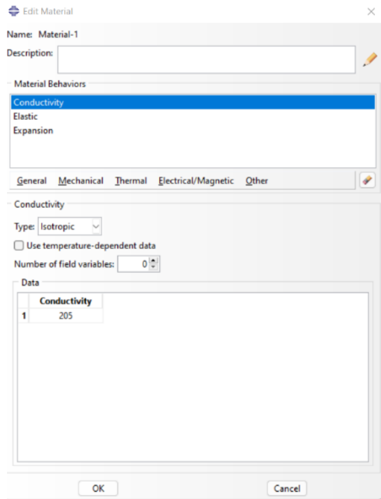
K d = f

d = k\f

MODELLING THE PROBLEM IN ABAQUS:

a) We have maintained the same consistent units as above for modelling the problem in Abaqus.
b) We firstly created the Axisymmetric part, according to the given dimensions.
c) Then we defined the material properties for the material, gave the young's modulus, Thermal conductivity and the thermal coefficient expansion as an input and created the section for the given material properties and applied it to the model.
d) After giving the material properties, we go to assembly and create instance for part 1 and select the instance type ("Independent" mesh on instance).
e) The next stage is to create a step, while creating a step we select coupled thermal displacement and then steady state to evaluate all the required fields.
f) After this, we go to interaction to apply the convective heat flux at each end of the rod, we select surface film condition and input the values for heat transfer coefficient.
g) Next, we move on to apply the boundary conditions at the initial predefined field and set the value for temperature over there, then we move on to apply the boundary conditions for displacement in the y direction (set U2=0) for the two fixed ends at the top and bottom (since in Abaqus it symmetric in Y).
h) Then we move on to apply the loads, for loads we will apply a body heat flux using the given analytical equation for S.
i) Next, we move on to apply mesh for the model. We have selected Linear Element type for the model and selected a family of coupled temperature displacement.
j) After this is done, we submit our job to the job manager and plot the results. For the results, we create a path from the bottom node to the top node and according plot for Temperature, Displacement and Stress.





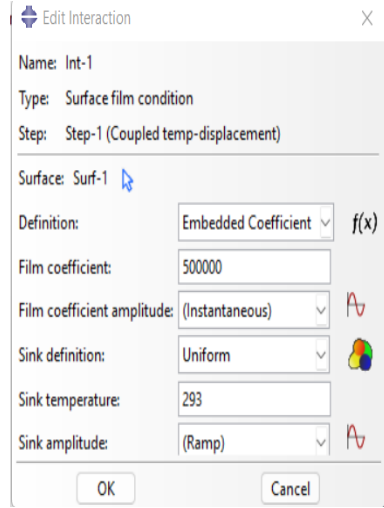
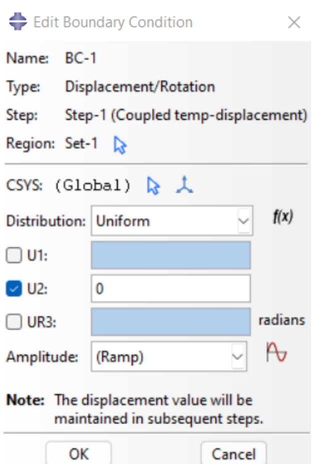
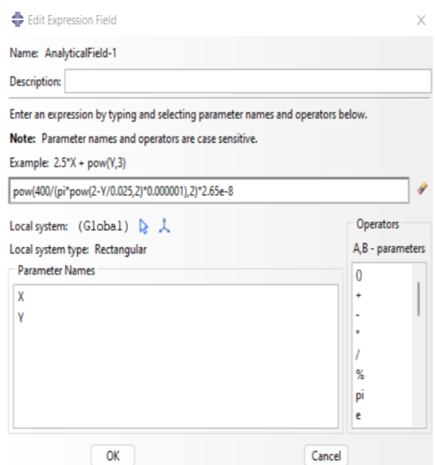


Fig. a) Part.

Fig. b) Material

Fig. c) Interaction(heat flux)





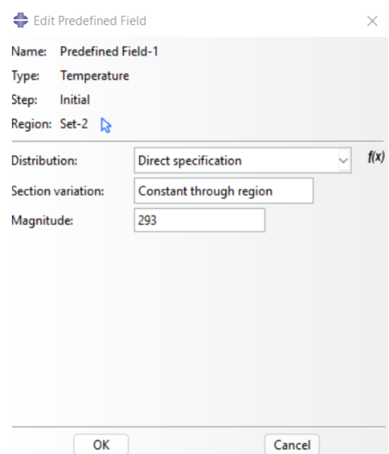
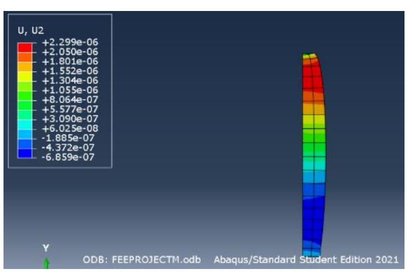


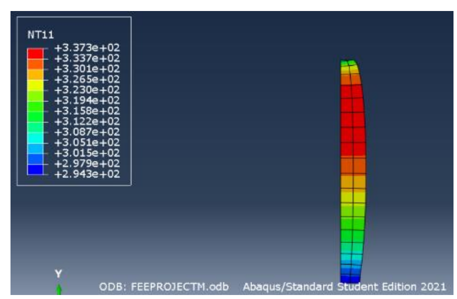
Fig. d) Boundary conditions

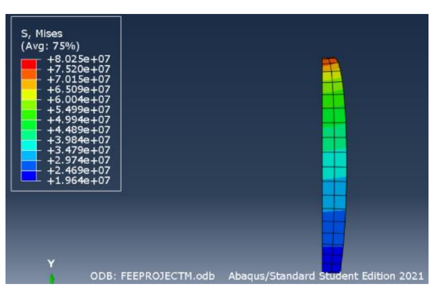
Fig. e) Internal heat generation

Fig. f) Predefined field



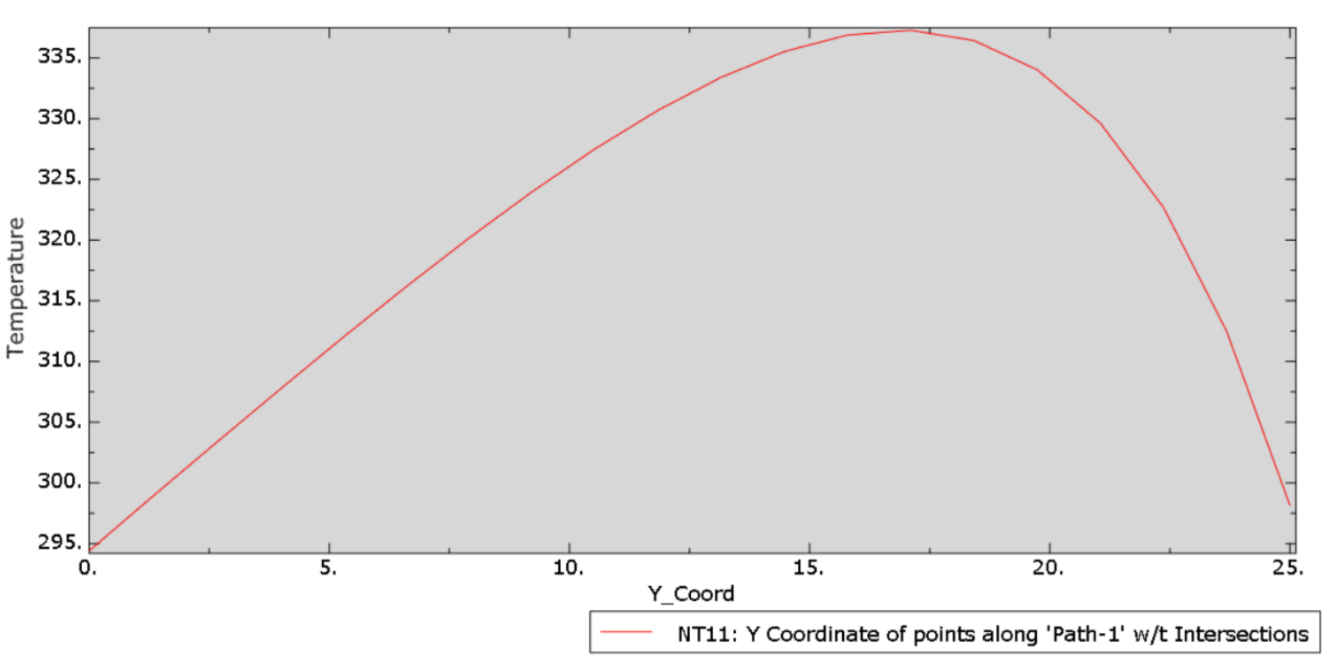
OUTPUTS



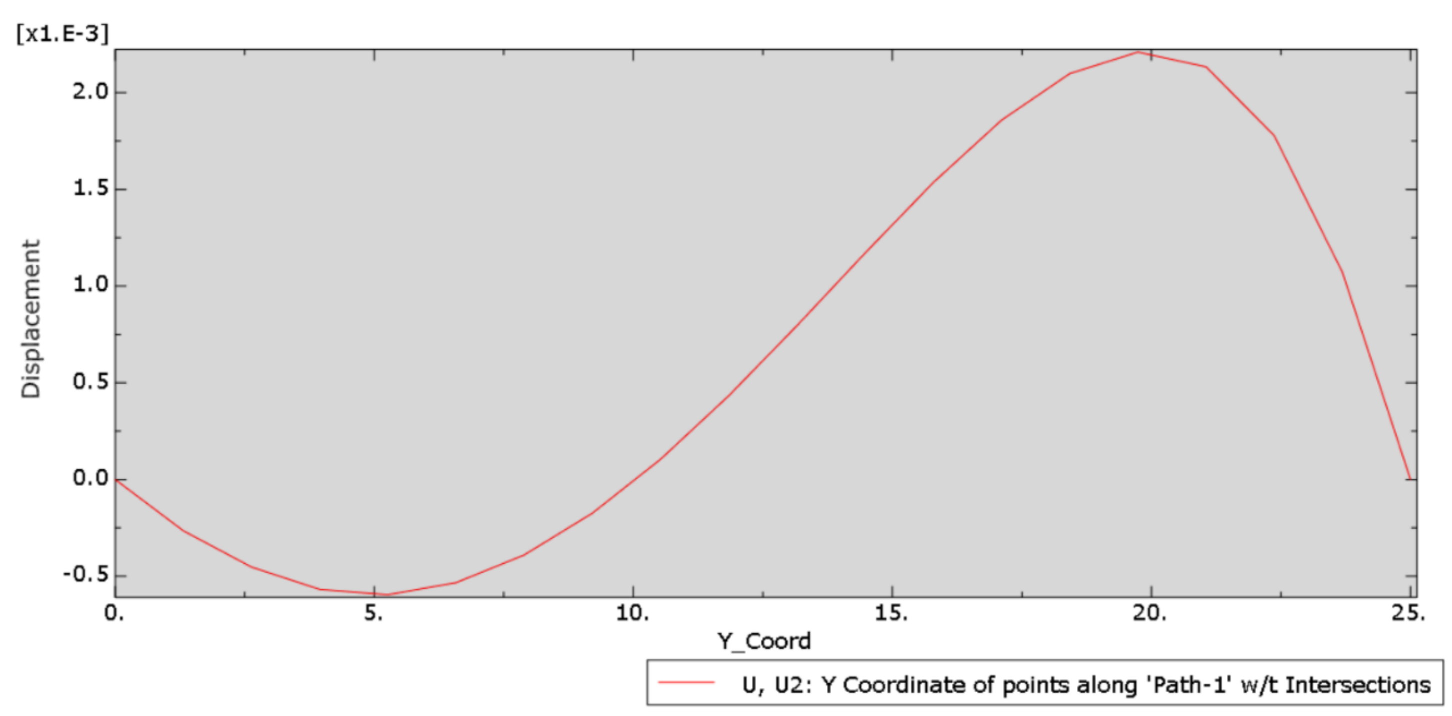


RESULTS FOR ABAQUS:

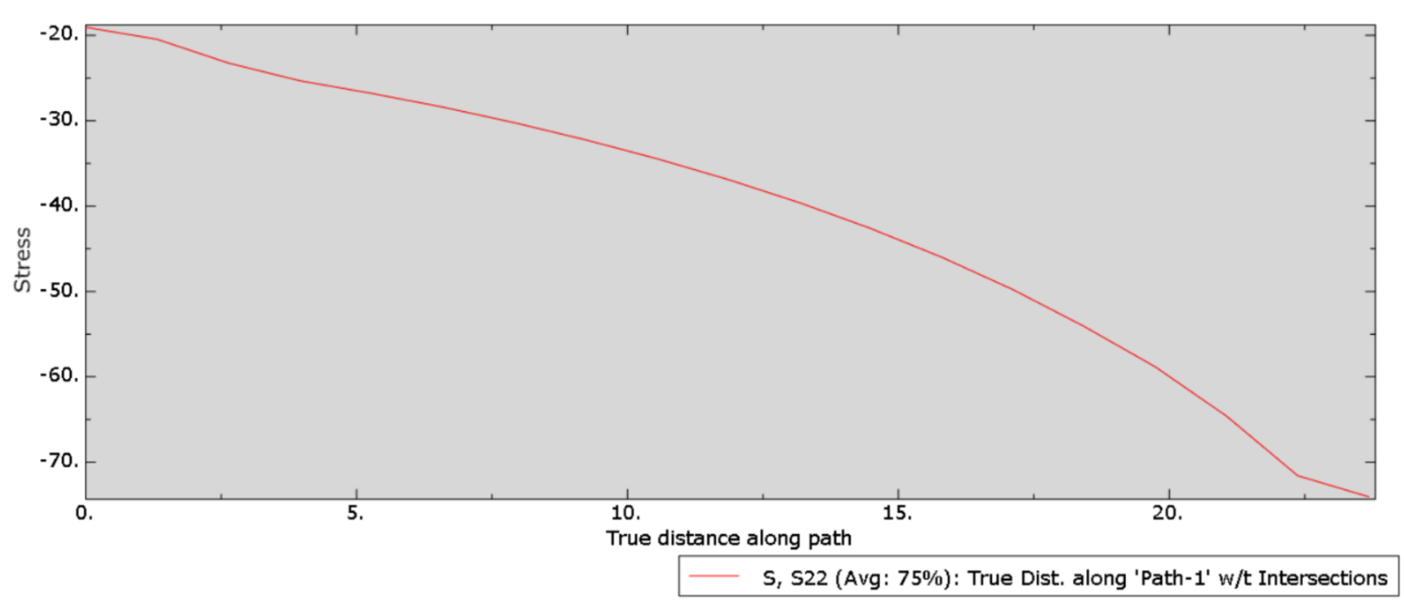
1. TEMPERATURE:



2. DISPLACEMENT:



3. STRESS:



-20.
-30.
-40.
-50.
-60.
-70.
Stress
0.
5.
10.
15.
20.
True distance along path
S, S22 (Avg: 75%): True Dist. along 'Path-1' w/t Intersections

ANALYTICAL SOLUTION:

$\frac{d}{dx}$ (A K$\frac{dt}{dx}$) + S = 0

$\frac{d}{dx}$ (A K $\frac{dT}{dx}$) = - S

$\int d$ (A K $\frac{dt}{dx}$) = $\int -S$ A dx $\longrightarrow$ We multiply by area to make the units compatible.

A K $\frac{dT}{dx}$ = -S(x) A + c_1

$\therefore \frac{dt}{dx} = \frac{-s(x)\ A + c1}{A\ K}$ dx

$T = \frac{-s(x)\ A + c1}{A\ K}$ +c_2

Now, this are just the steps we follow in Mathematica and evaluate the analytical solution. These are just the steps and not the exact integrals.

We have given the boundary condition to find c_1 and c_2.

@x = 0, the heat flux is given by;

-k $\frac{dt}{dx}$ = -h (T-293)

At x = L, the heat flux is given by;

-k $\frac{dt}{dx}$ = h (T-293)

For displacement.

$\frac{d}{dx}$ (E A ($\frac{du}{dx}$ - $\alpha\Delta$T)) = 0

Integrating's on both sides,

E A ($\frac{du}{dx}$ - $\alpha\Delta$T) = c_1

$\therefore \frac{du}{dx} = \frac{c1}{EA} + \alpha\Delta T$

$\int du = \int (\frac{c1}{EA} + \alpha\Delta T)$ dx

Integrating this we get.

U = ($\frac{c1}{EA}$ + $\alpha\Delta$T) + c_2

For c_1 and c_2, we apply boundary conditions,

At x = 0 u = 0 & x = L u = L

Now, this are just the steps we follow in Mathematica and evaluate the analytical solution. These are just the steps and not the exact integrals.

After computing U and T, stress can be computed as;

$$\sigma = E\left(\frac{du}{dx} - \alpha(T - 293)\right)$$

We have followed the same method of integration on wolfram and integrated.

WOLFRAM MATHEMATICA — Plan: Arizona State University | Documentation | Language Intro | Quick Links | Cloud Files

FEE PROJECT.nb — File Edit Format Insert Evaluation View Help Share Publish

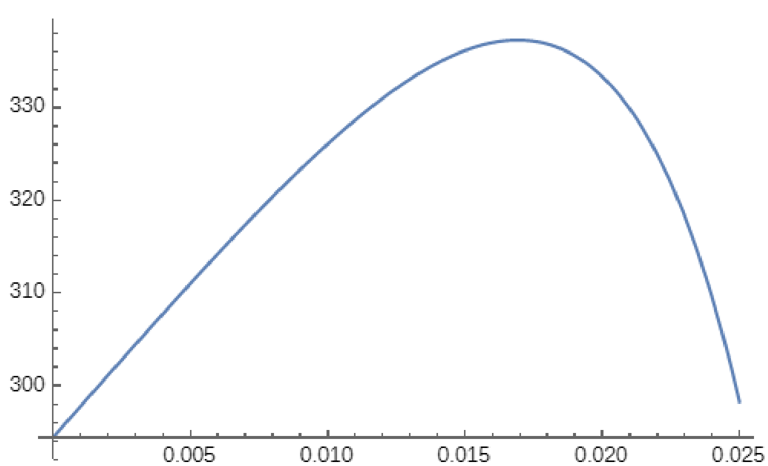
```
In[1]:= A = Pi * (2 - x / l) ^2 * 10^-6;

s = (current / A) ^2 * rho;

param = {l → 0.025, rho → 2.65 * 10^-8, current → 400, k → 205};
Tat = Integrate[(Integrate[-s * A, x] + c1) / (A * k), x] + c2 /. param;
q = -k * D[Tat, x] /. param;

bc = Solve[ (q /. x → 0) == (-500000 * ((Tat /. x → 0 ) - 293)) && (q /. x → 0.025) == (500000 * ((Tat /. x → 0.025) - 293))];
TAT = Tat /. bc;
CForm[TAT]
Plot[{TAT}, {x, 0, 0.025}]

U = Integrate[( c3 / (205 * A) + (21 * 10^-6) * (TAT - 293)), x] + c4 /. param;
bc2 = Solve[( U /. x → 0) == 0 && (U /. x → 0.025) == 0];
UAT = U /. bc2;
CForm[UAT]
Plot[ {UAT}, {x, 0, 0.025}]

Stress = 70 * 10^9 (D[UAT, x] - (21 * 10^-6) * (TAT - 293));
CForm[Stress]
Plot [Stress, {x, 0, 0.025}]
```

TEMPERATURE:

Out[25]//CForm=

```
List(-36.93507240899844 - (0.30890604769005425*
      (1.3250000000000004 + 80.12622478386169*(-0.05 + x)))/Power(-0.05 + x,2))
```

Out[26]=

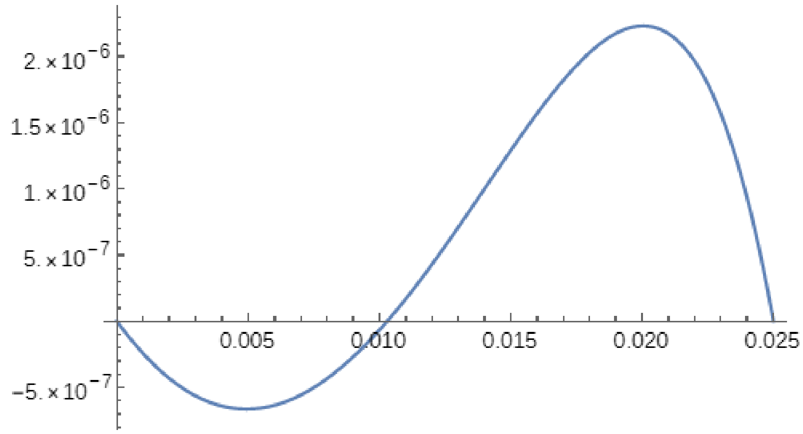


DISPLACEMENT:

Out[30]//CForm=

```
List(List(-0.0013700558577051827 - 0.000016635608667726764/(-0.05 + x) -
     0.006928636520588968*x + 0.000025989049185022965/(-0.05 + 1.*x) -
     0.0005197809837004593*Log(0.05 - 1.*x)))
```

Out[31]=

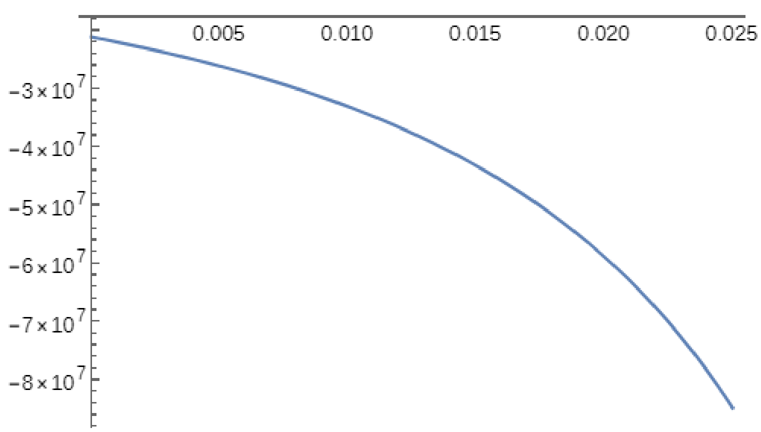


STRESS:

Out[33]//CForm=

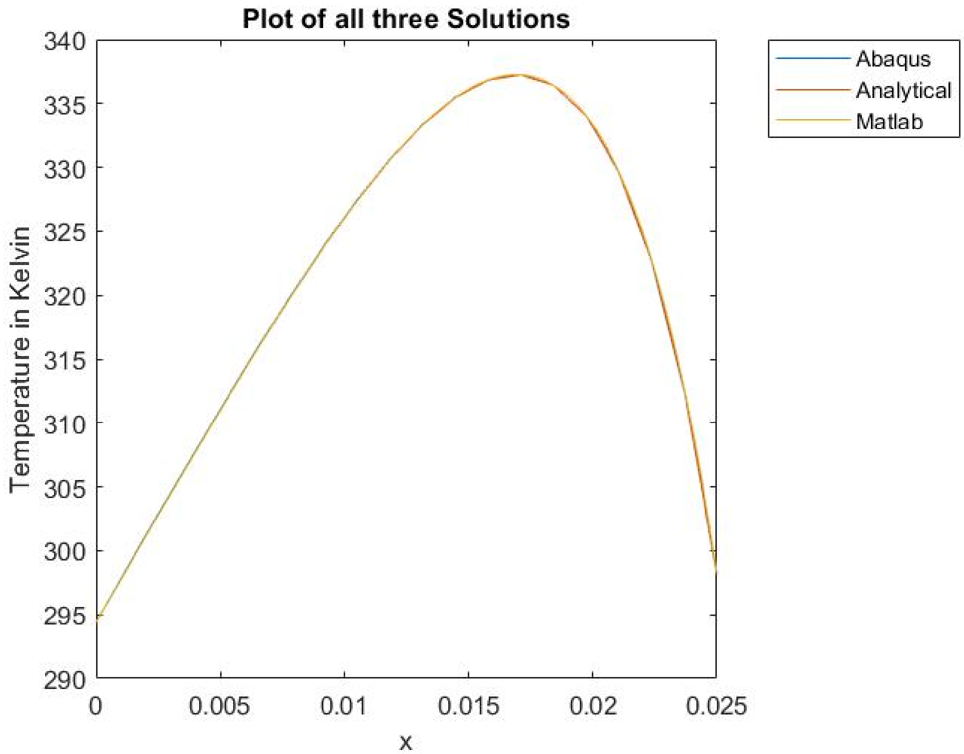
```
List(List(70000000000*(-0.006928636520588968 -
       (21*(-329.93507240899845 - (0.30890604769005425*
              (1.3250000000000004 + 80.12622478386169*(-0.05 + x)))/
            Power(-0.05 + x,2)))/1.e6 + 0.0005197809837004593/(0.05 - 1.*x) +
       0.000016635608667726764/Power(-0.05 + x,2) -
       0.000025989049185022965/Power(-0.05 + 1.*x,2))))
```

Out[34]=

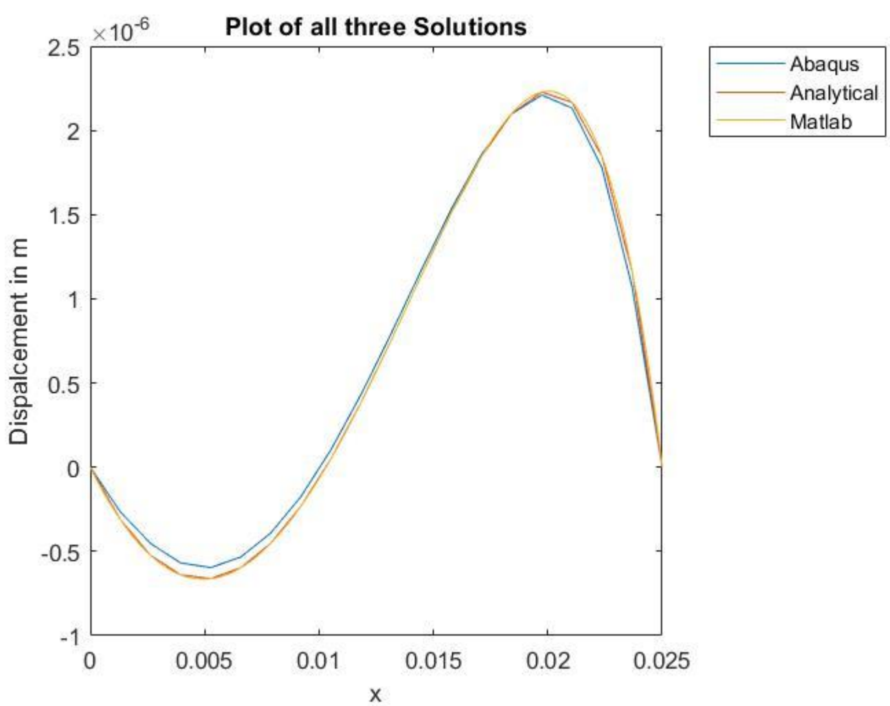


COMBINED PLOT FOR ALL THE THREE PARAMETERS USING MATLAB, ANALYTICAL AND ABAQUS:

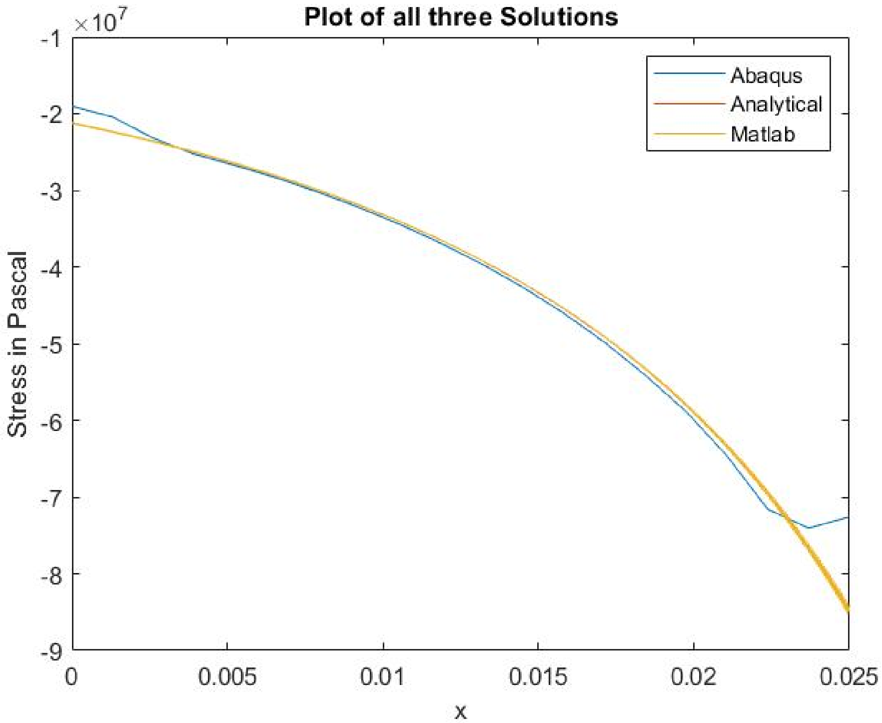
TEMPERATURE:



DISPLACEMENT:



STRESS:

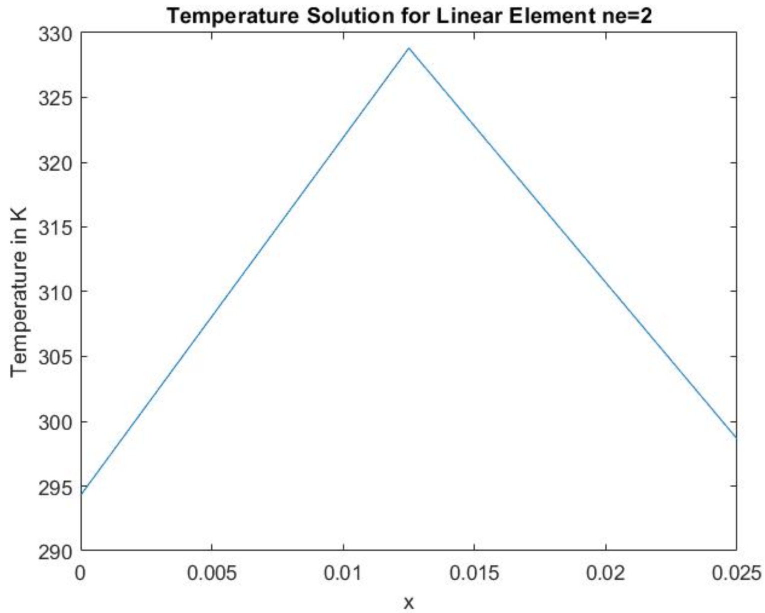


Explanation: We have used a fine enough mesh in our finite element code to make sure the solution converges to the same as Abaqus and Analytical. The graphs are almost overlapping on one another, however, at a micro level there is definitely a difference in graph. Reason being, we built our finite element code using the weak form so all the conditions are weakly satisfied. So, compared to the analytical solution which consist of the strong form, the boundary conditions being strongly satisfied causes a very minute difference in the solution which is almost negligible. Now, when it comes to comparing the solution with Abaqus, we have modelled our problem in MATLAB in 1D, whereas Abaqus models in 2D, which causes the difference and is very much evident in the stress plot.

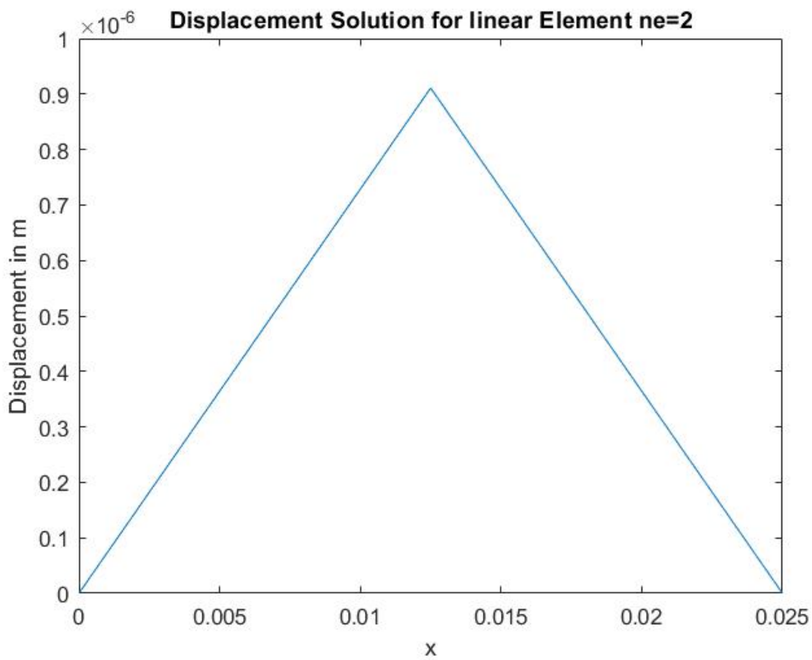
PLOT OF TEMPERATURE, DISPLACEMENT AND STRESS USING MATLAB FOR LINEAR ELEMENT SOLUTION:

- FOR N_e=2

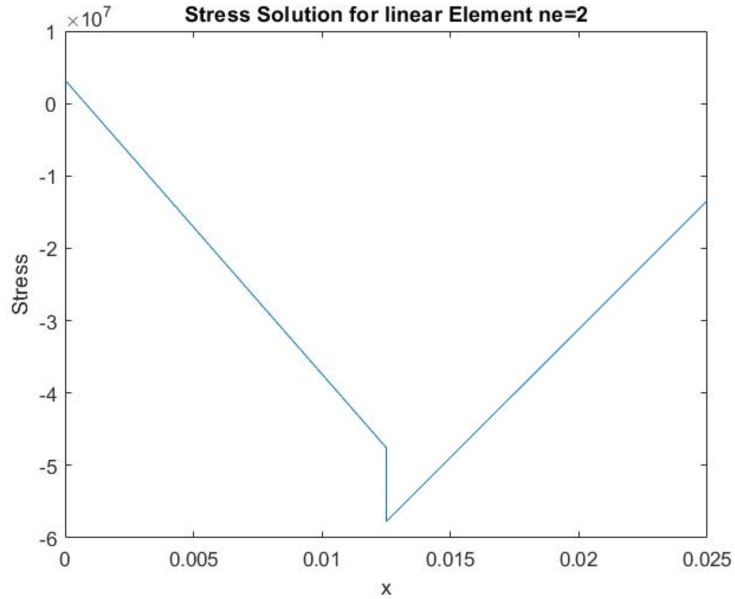
TEMPERATURE:



DISPLACEMENT:

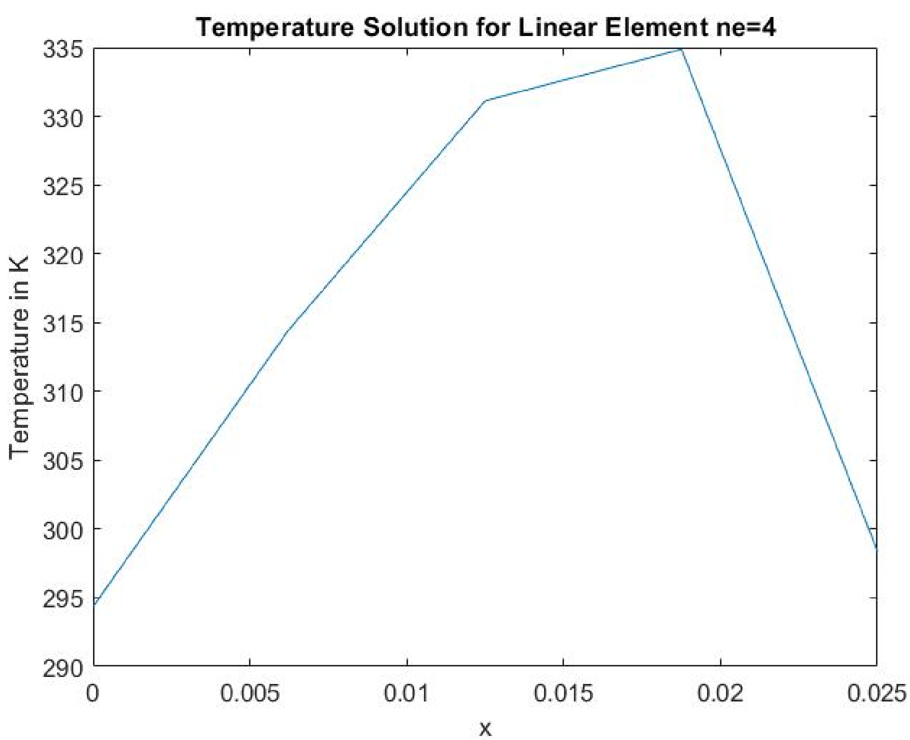


STRESS:

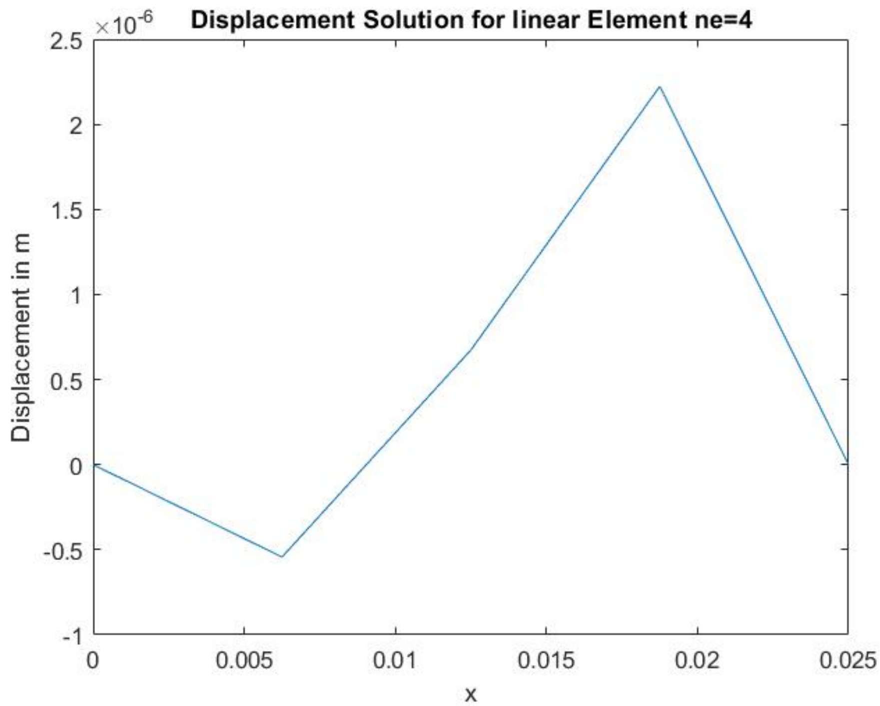


- FOR N_e=4

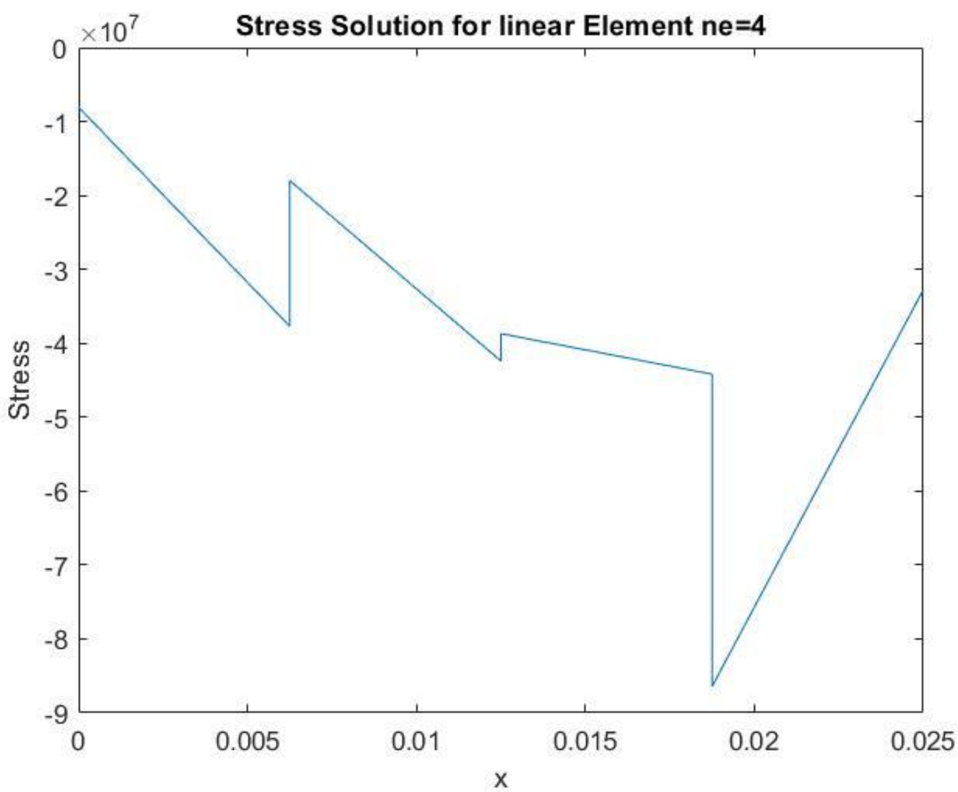
TEMPERATURE:



DISPLACEMENT:



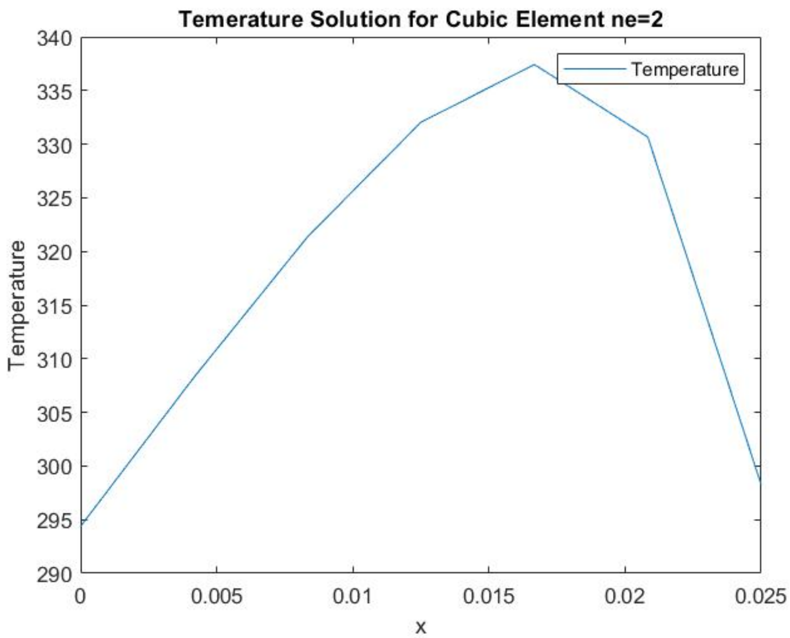
STRESS:



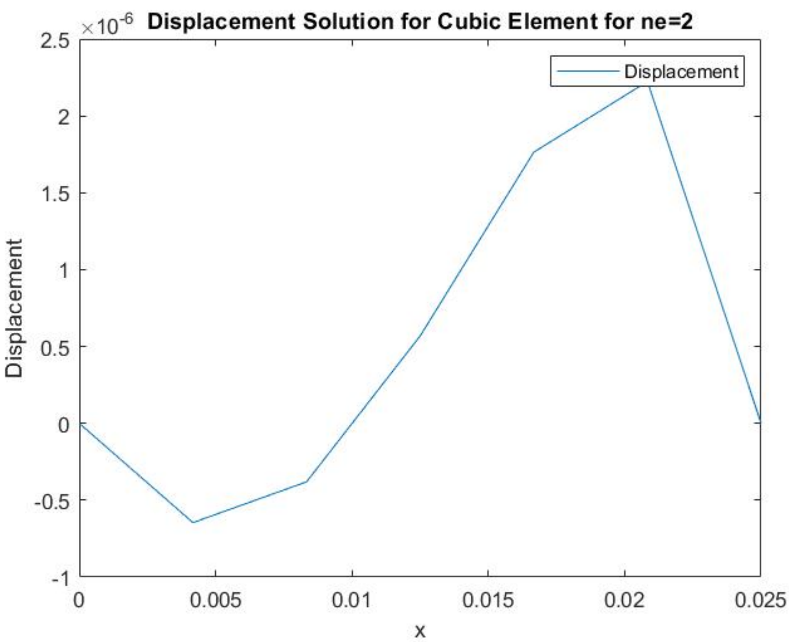
PLOT OF TEMPERATURE, DISPLACEMENT AND STRESS USING MATLAB FOR CUBIC ELEMENT SOLUTION:

- $N_e = 2$

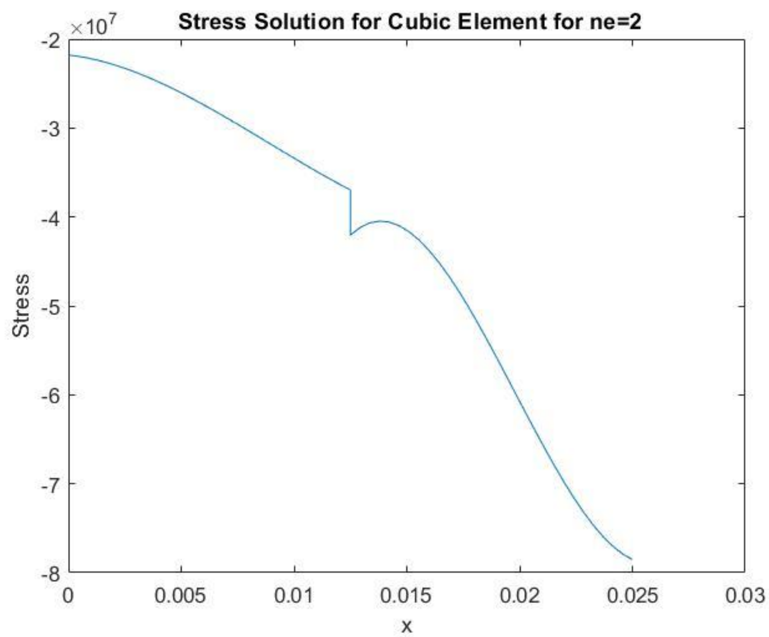
TEMPERATURE:



DISPLACEMENT:

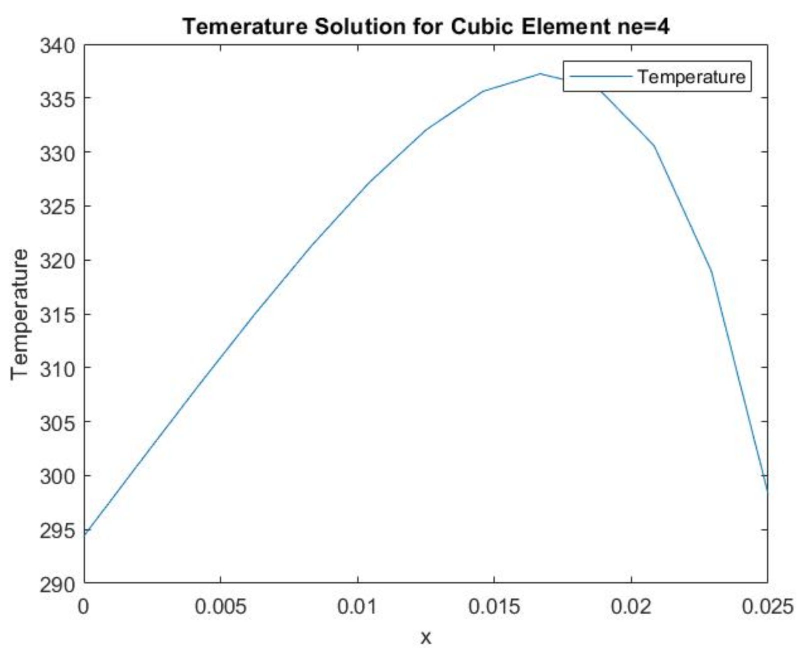


STRESS:

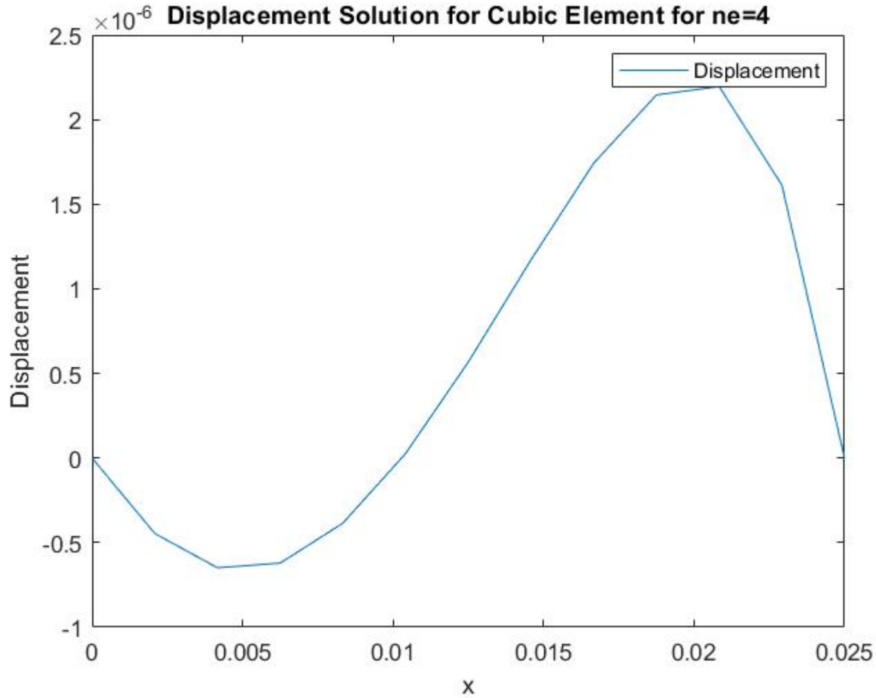


- FOR N_e=4

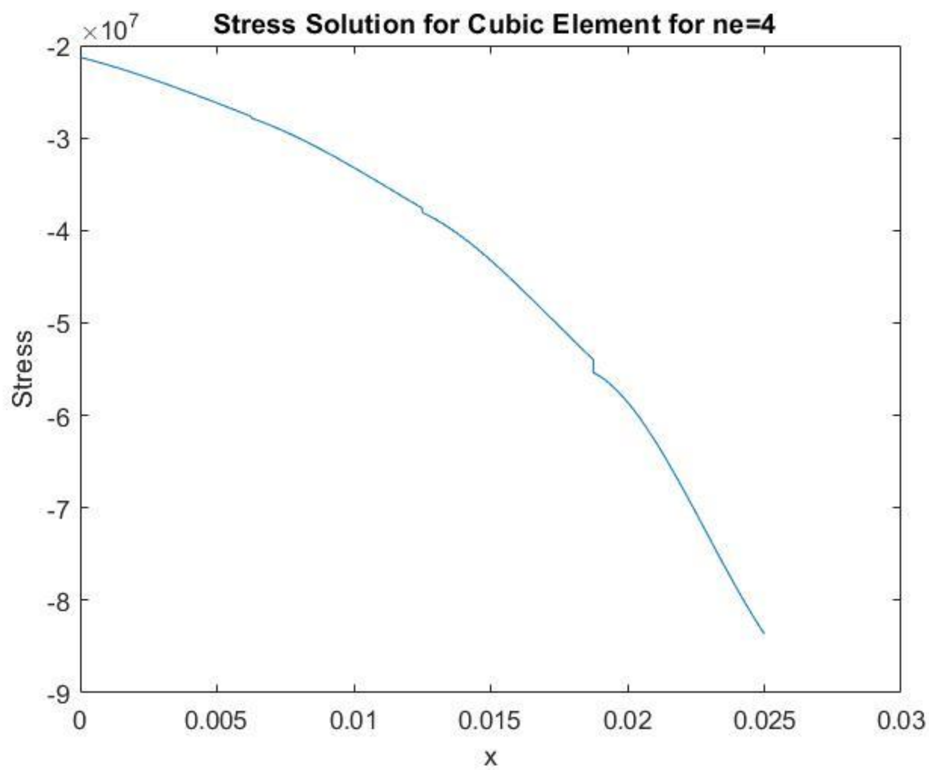
TEMPERATURE:



DISPLACEMENT:

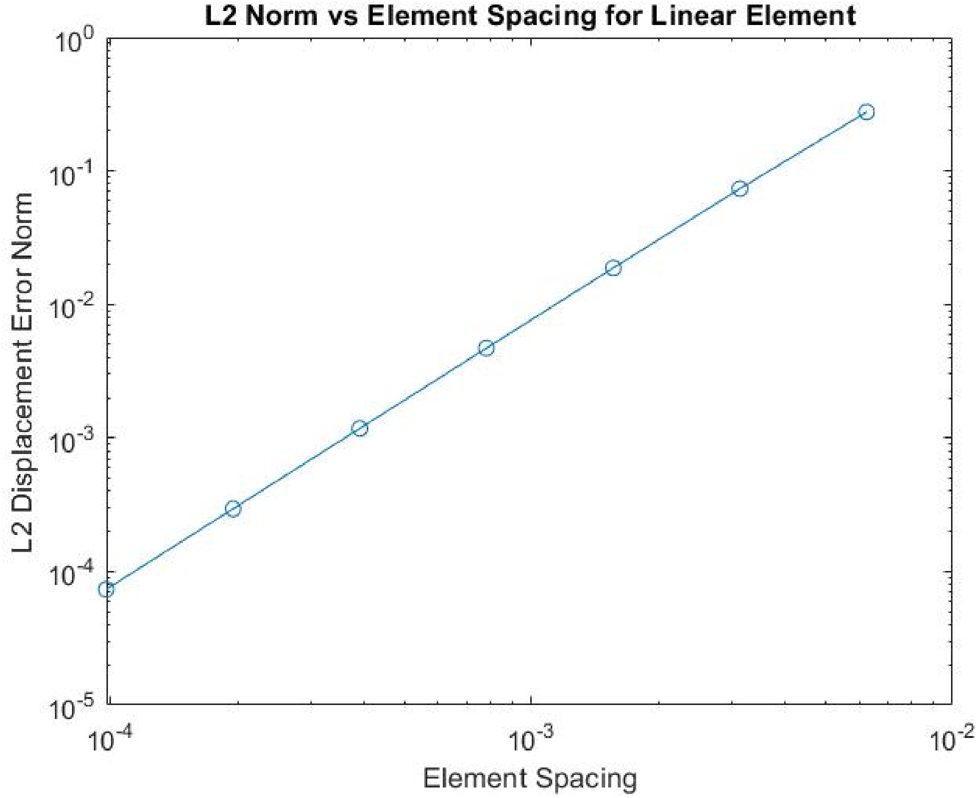


STRESS:



PLOT OF L2 DISPLACEMENT NORM

(LINEAR ELEMENT):



OUTPUT-TABLE:

```
The Slope of the line for L2 Norm of Linear Element is 1.983682
L2Norms =

  7×3 table
```

No_of_Elements	Element_Spacing	L2_Norm
4	0.00625	0.276298841138483
8	0.003125	0.0736102812396437
16	0.0015625	0.018734590947128
32	0.00078125	0.00470540560869933
64	0.000390625	0.00117772822164411
128	0.0001953125	0.000294518376973228
256	9.765625e-05	7.36349934443162e-05

PLOT OF L2 DISPLACEMENT NORM

(QUADRATIC ELEMENT):

PLOT:

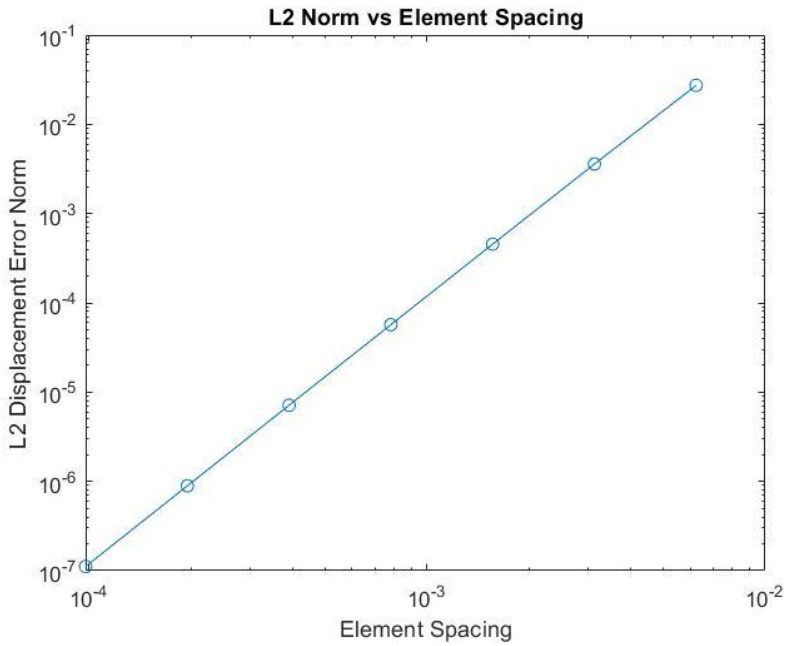


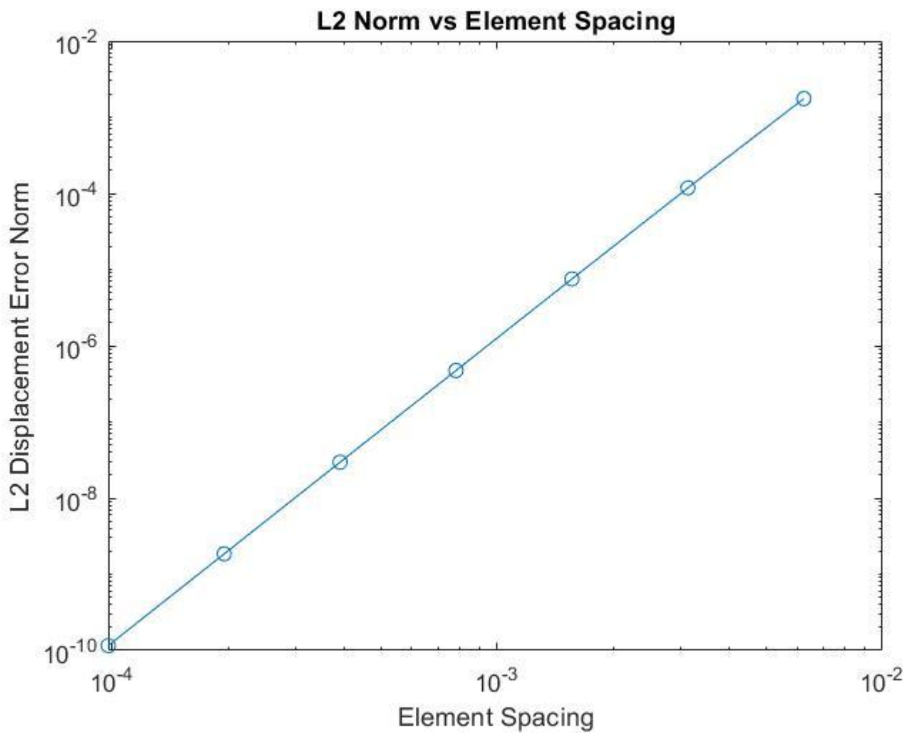
TABLE:

```
The Slope of the line for L2 Norm of Cubic Element is 2.987411
L2Norms =

  7×3 table
```

No_of_Elements	Element_Spacing	L2_Norm
4	0.00625	0.0272953233740021
8	0.003125	0.00358879168757701
16	0.0015625	0.000454585736385706
32	0.00078125	5.70144051072487e-05
64	0.000390625	7.13280855069027e-06
128	0.0001953125	8.91789048444755e-07
256	9.765625e-05	1.1147947917721e-07

PLOT OF L2 DISPLACEMENT NORM

(CUBIC ELEMENT):



OUTPUT-TABLE:

```
The Slope of the line for L2 Norm of Cubic Element is 3.980994
L2Norms =

  7×3 table
```

No_of_Elements	Element_Spacing	L2_Norm
4	0.00625	0.00175288865869849
8	0.003125	0.000118023532294715
16	0.0015625	7.53067074276297e-06
32	0.00078125	4.73184668491937e-07
64	0.000390625	2.96138280564262e-08
128	0.0001953125	1.85148590555298e-09
256	9.765625e-05	1.15730287752271e-10