ARIZONA STATE UNIVERSITY

MAE 503

FINITE ELEMENTS IN ENGINEERING

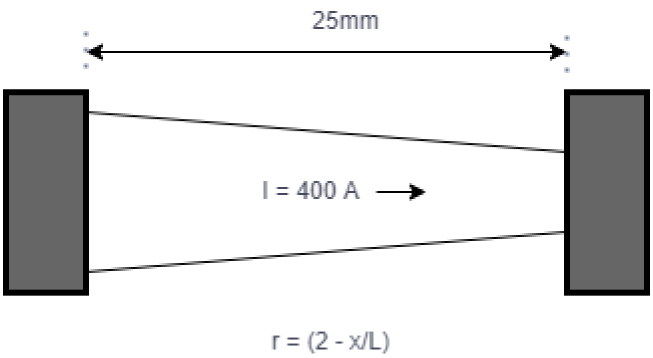
Professor- Dr. Jay Oswald

Onkar Chavan | ochavan2 | 1223314248

Mangirish Kulkarni | mkulka17 | 1223229852

Problem Statement-

Restate the problem that you are solving. Fully write out the strong form of governing equations-



We first determine the steady-state temperature field of an aluminum rod with a tapering cross-sectional area. The rod is induced to constant current which passes through it which results in internal heat generation. Rod is fixed between two thermal reservoirs. Next, we calculate the stress generated within the rod due to thermal expansion. Then we need to solve for both the steady-state temperature field and the stress in the rod using three different methods. First step by writing out Finite Element Program in MATLAB to approximate the solution, Next step to use Abaqus software to approximate solutions, and lastly solve the governing equations analytically using Mathematica.

Variable Notation and Definition-

- Length (L)- 0.025 m
- Current (I)- 400 A
- Temperature on the Boundary (T)- 293.15K
- Young's Modulus (E)-70*10^9
- Electrical Resistance (p^e)- 2.65*10^-08 Ωm
- Heat transfer Co-efficient (h)-500 kW/m2K
- Heat conductivity (k)- 205 W/mK
- Coefficient of Thermal Expansion (α)-21*10^-06 K-1
- Location Point on the Rod-x
- Radius (r)- (2-x/L)
- Cross Sectional area- $A=\pi r^2$
- Internal Heat Generation- $s=(I^2/A) * p^e$.

Strong Form and Resulting Governing Equations-

Energy Balance: Heat Flux

$$S(x + \frac{\Delta x}{2})\,\Delta x + q(x)A(x) - q(x + \Delta x)A(x + \Delta x) = 0$$

Divide by Δx and take the limit → 0

$$S(x) - \frac{d}{dx}(qA) = 0 \; where \; q = -k\,\frac{dT}{dx}$$

Strong Form and Governing Equations

$$\boldsymbol{\frac{d}{dx}\left(Ak\frac{dT}{dx}\right) + S}$$

Boundary Condition $q = -h\,(T - \bar{T})$ at x = 0, L

Units Specification-

Specify a consistent system of units-

We have used SI units for during our calculation and simulations. Following are the units we used-

- Mass -Kg
- Length- m
- Temperature- K
- Force- N
- Stress- Pa
- Energy- J
- Power- W
- Resistance- Ω
- Current- A

Weak Form and Integral Expressions-

Write out the weak form of governing equations and show the integral expression for each of the finite element matrices required to compute.

Strong Form and Governing Equations

$$\frac{d}{dx}\left(Ak\frac{dT}{dx}\right) + S$$

Boundary Condition $q = -h(T - \bar{T})$ at x = 0, L

Multiply by a test function and Integrate over the domain

$$\int_0^L w\left[\frac{d}{dx}\left(Ak\frac{dT}{dx}\right) + s\right]dx = 0$$

Integrate by Parts

$$-wAh\frac{dT}{dx}|_{\Gamma} - \int_0^L \frac{dw}{dx}Ak\frac{dT}{dx}dx + \int_0^L wsdx = 0$$

Rearrange the Equation

$$-\int_0^L \frac{dw}{dx} Ak \frac{dT}{dx} dx + \int_0^L wsdx - wAh((T - \bar{T})|_{\Gamma} = 0$$

Weak form of governing equations

Find $T \in C^0$ such that $T - \bar{T}$ on Γ and

$$\int_0^L \frac{\boldsymbol{dw}}{\boldsymbol{dx}} \boldsymbol{Ak} \frac{\boldsymbol{dT}}{\boldsymbol{dx}} \boldsymbol{dx} = \int_0^L \boldsymbol{wsdx} - \boldsymbol{wAh}((\boldsymbol{T} - \bar{\boldsymbol{T}})|_{\boldsymbol{\Gamma}}$$

$\forall\ w \in C^0$ such that w = 0 on Γ_q

Abaqus Report-

Below are the detailed steps followed for Abaqus software Simulation-

- **Part Creation**

In module select Part => Create Part

Here Select Modelling Space => Axisymmetric

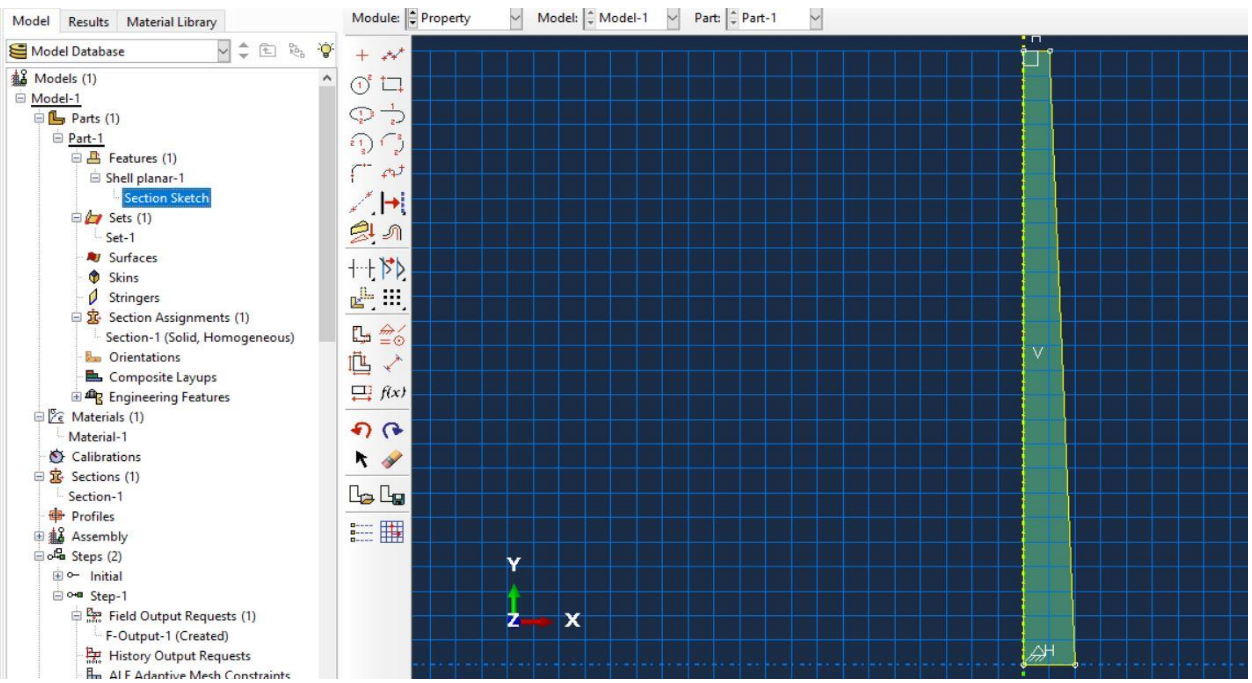
Type => Deformable

Shape => Shell

Now create the part using X-Y co-ordinates

Select Create Line => Enter co-ordinates to create the part.

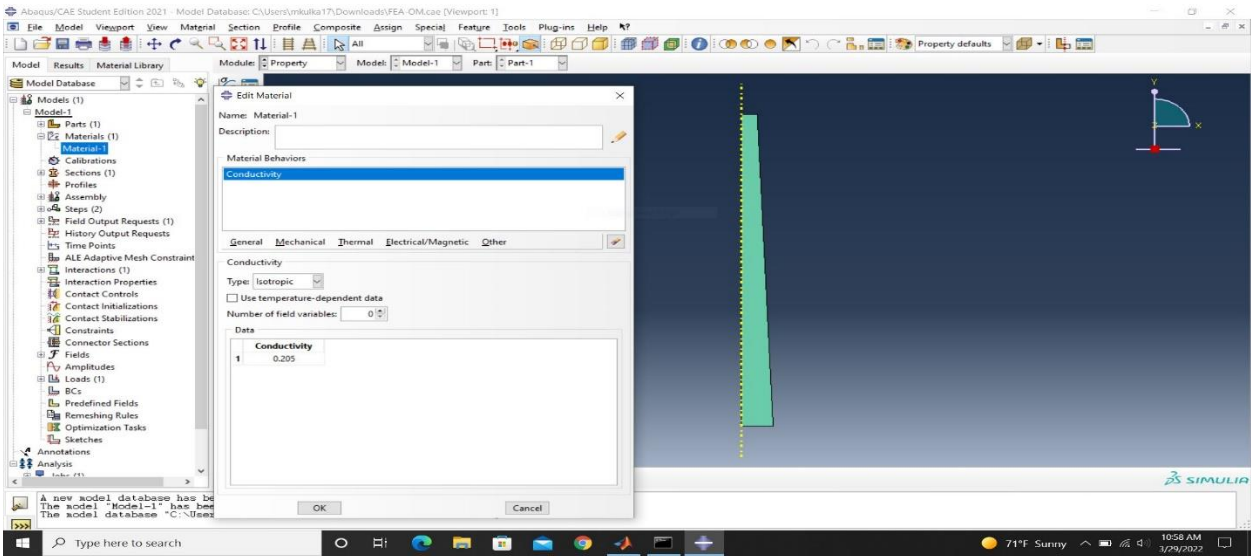
Co-ordinates = (0,0), (0.002,0), (0.001,0.025), (0,0.025), (0,0)



- **Material Creation**

1. Create Material

In module select Property => Select create Material



Here we entered the given properties of material as below-

Thermal => Conductivity => 205 Watts/(meter-Kelvin)

Mechanical => Elasticity => Elastic => Young's Modulus => 70*10^9 Pascal or Newton/m^2

Mechanical => Elasticity => Elastic => Poisson's Ratio -> 0

Mechanical => Expansion => Expansion Coeff => 2.1*10^-05 Kelvin^-1

2. Create Section

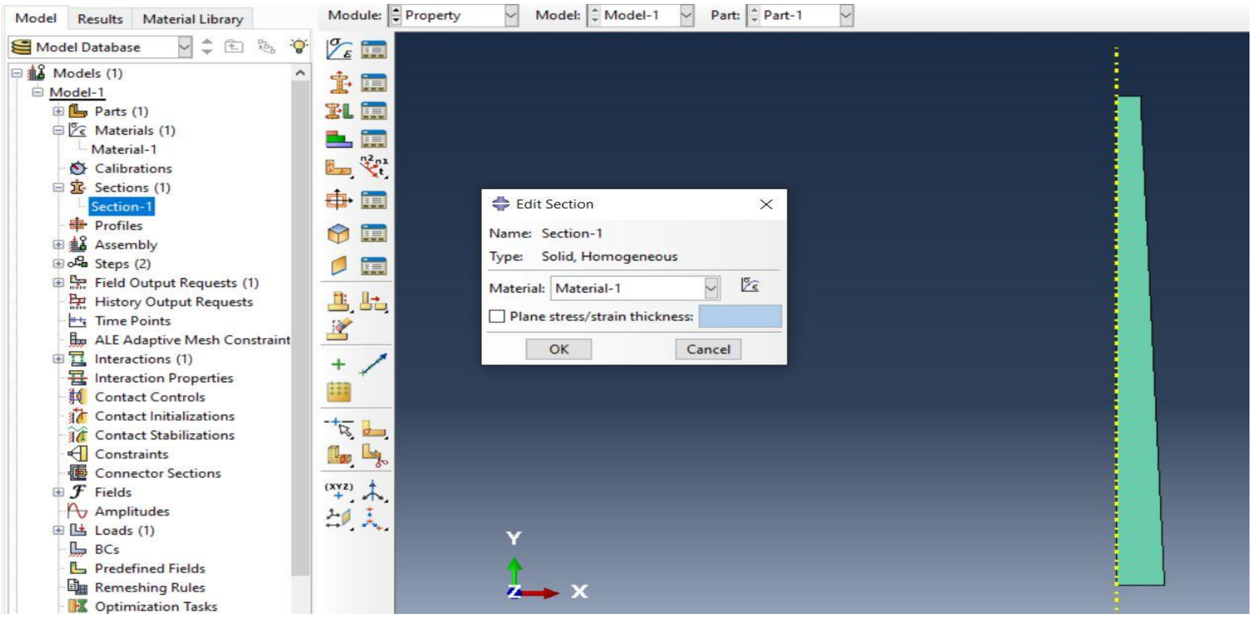
In module select Property => Create Section

Category => Solid

Type => Homogeneous

In module select Property => Assign Section

Select the part.

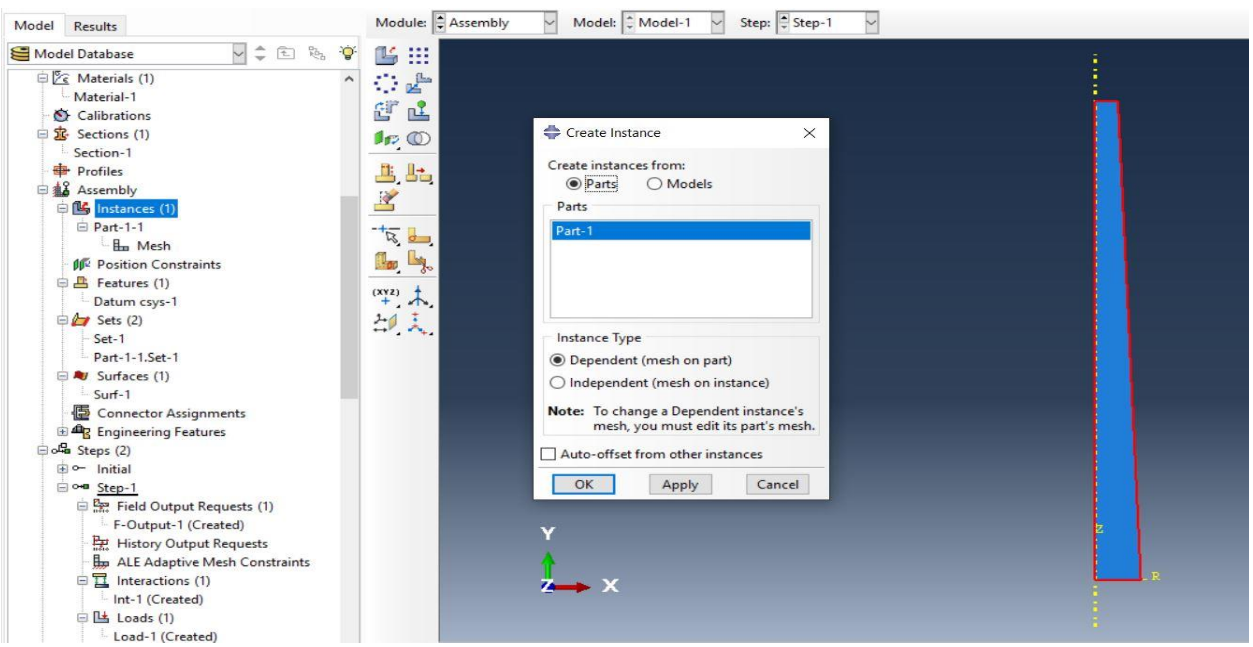


- **Assembly**

In module select Assembly => Create Instances

Select => Parts

Instance Type => Independent

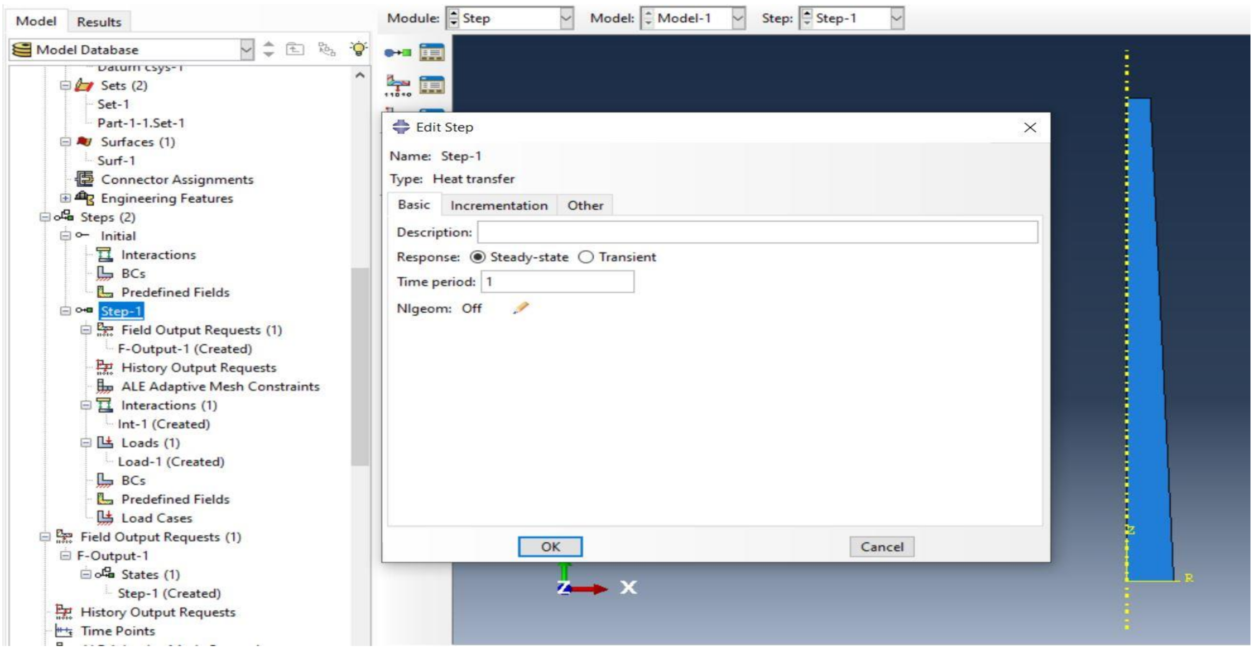


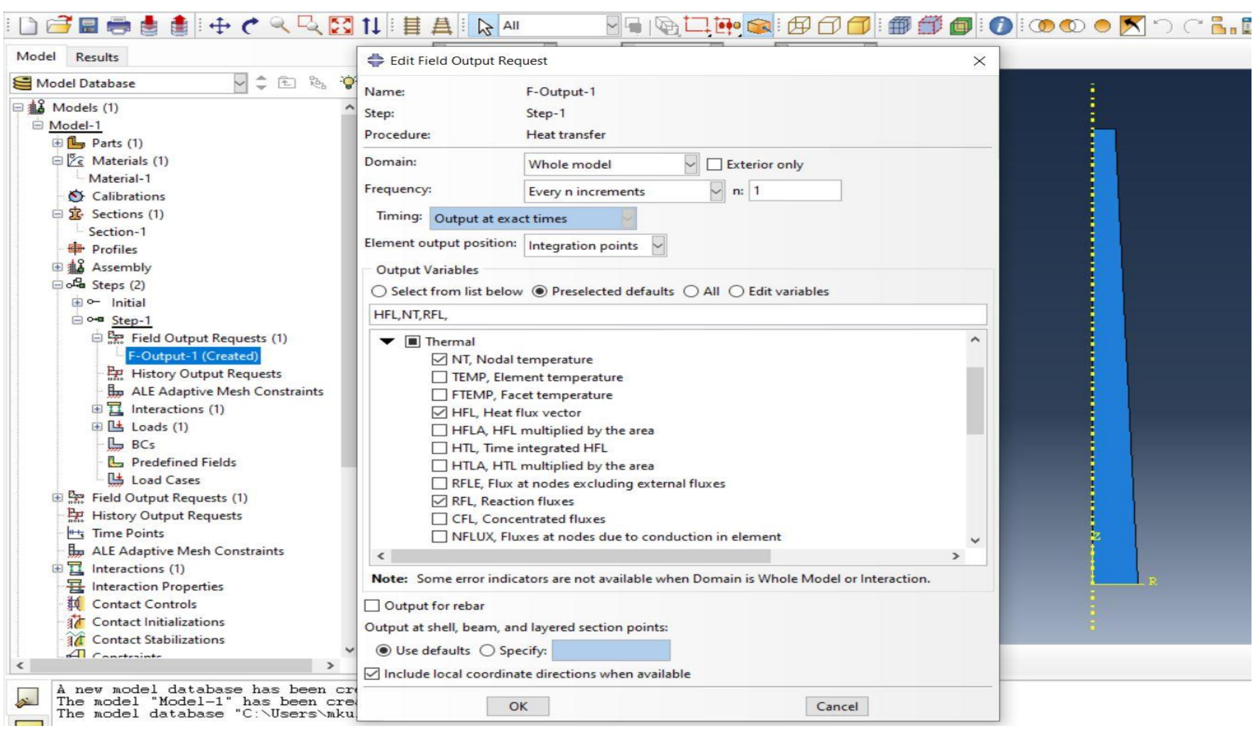
- **Step**

In module select Step => Create Step

Type => Coupled temperature displacement

This is Steady State Problem so select Steady State in Response.





- **Boundary Conditions**

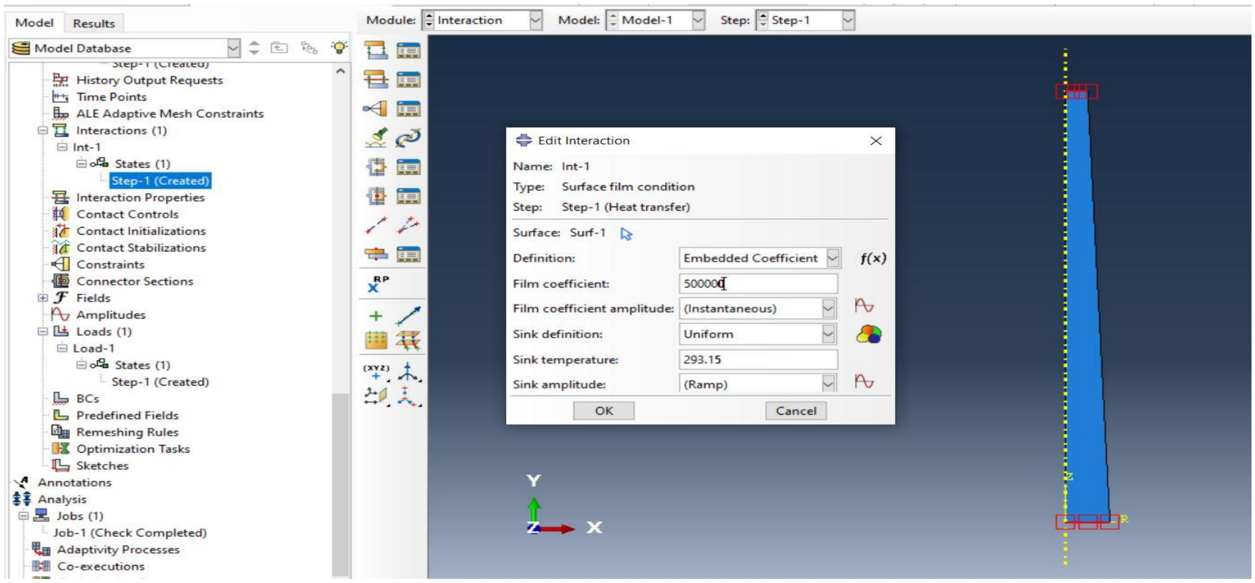
In module select Interaction => Create Interaction

Select => Surface Film Condition

And apply to the top and bottom surface and select Done.

Film Coefficient => 500000

Sink Temperature => 293.15



- **Loads**

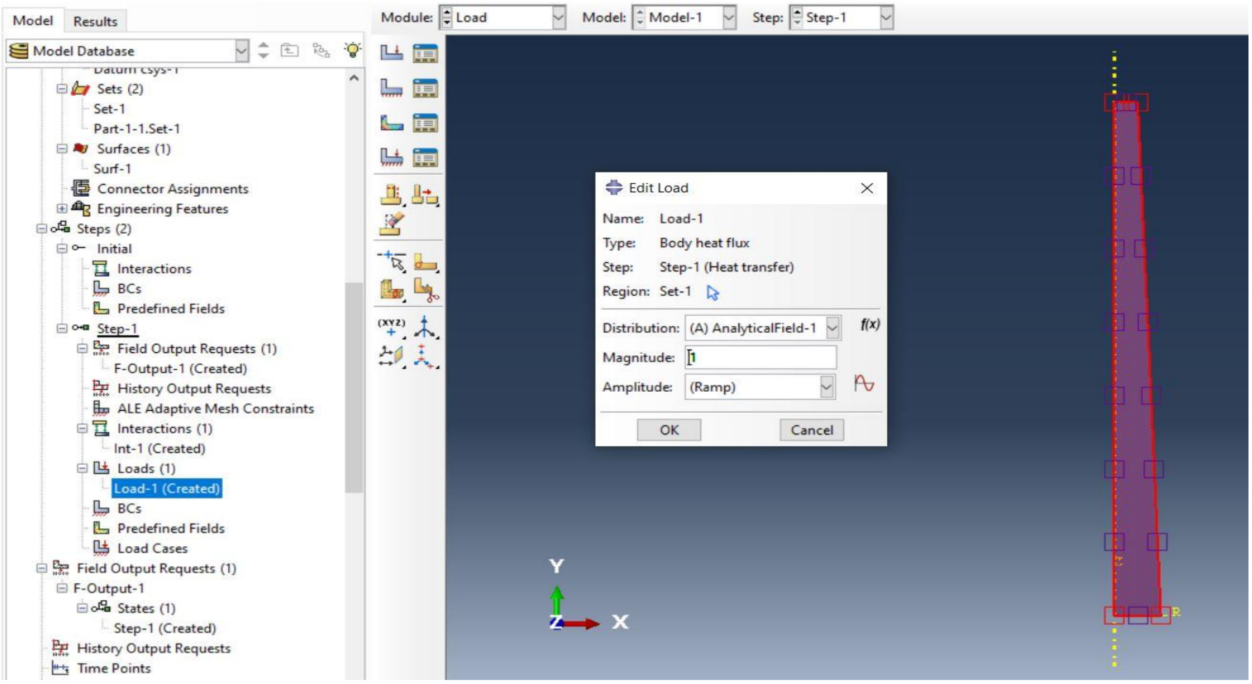
1) Create Load

In module select Load => Create Load

Category => Thermal

Type => Body Heat Flux

Continue => Select the Part

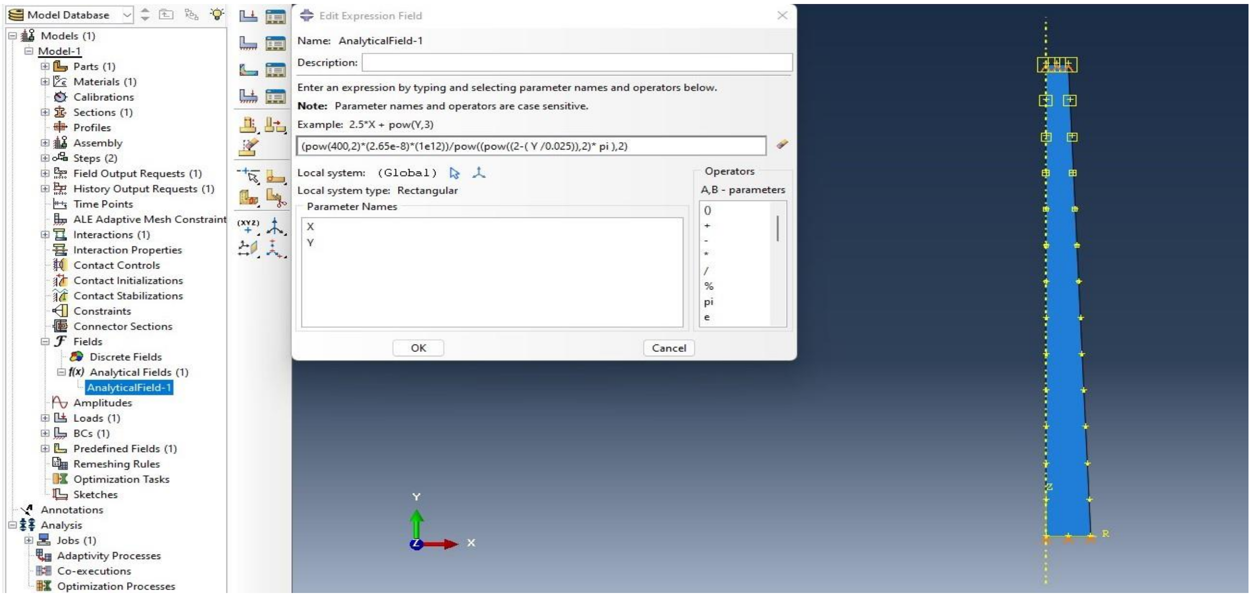


This problem does not have a uniform distribution.

Therefore, we create an **Analytical Field**

In Edit Load select Distribution => Analytical Field

Magnitude => 1

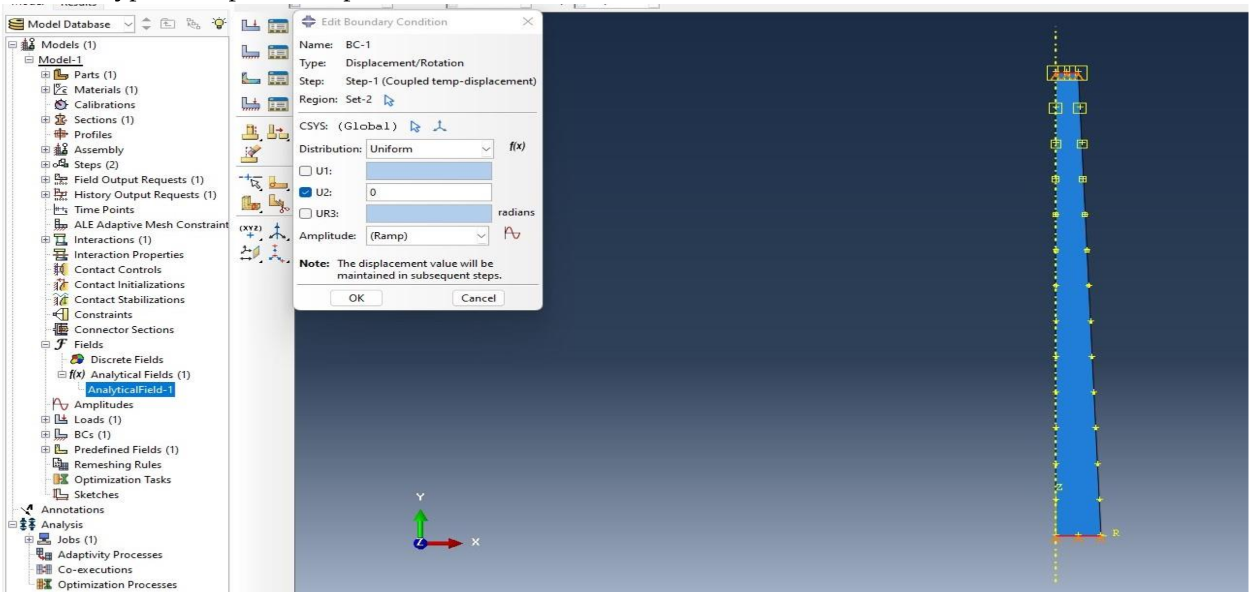


A. Create Boundary Condition

In Create Boundary Condition

Category => Mechanical

Type of steps => Displacement/Rotation



B. Create Predefined Field

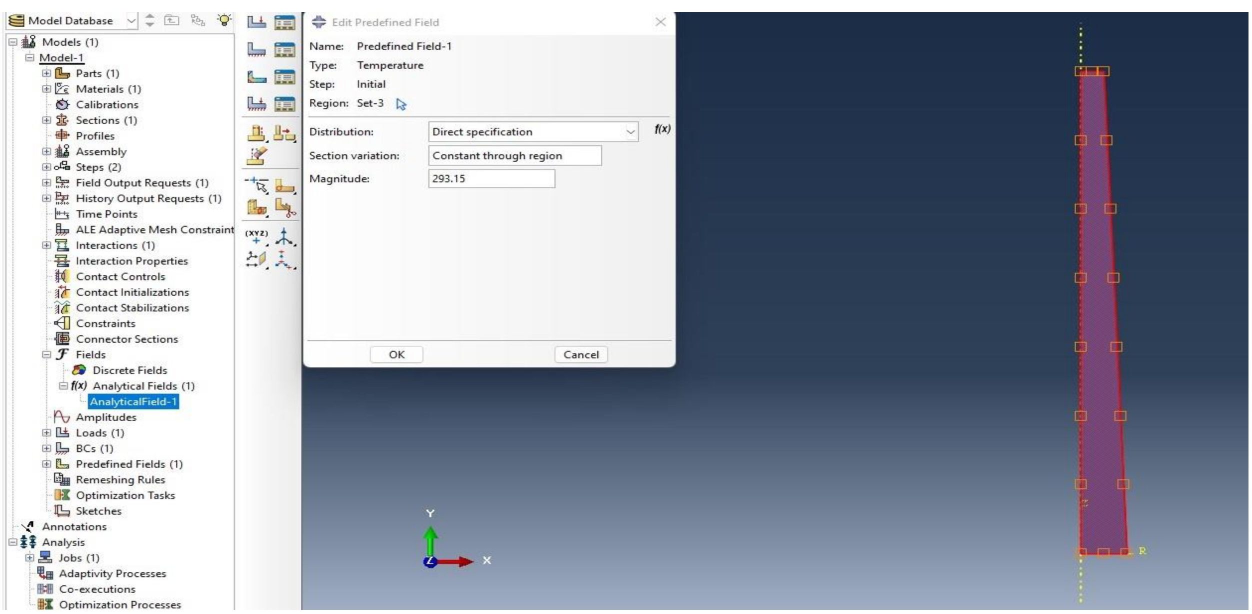
Select Predefined Field => Step => Initial

Category => Other

Types for selected steps => Temperature

Select Part

Magnitude => 293.15



- **Meshing**

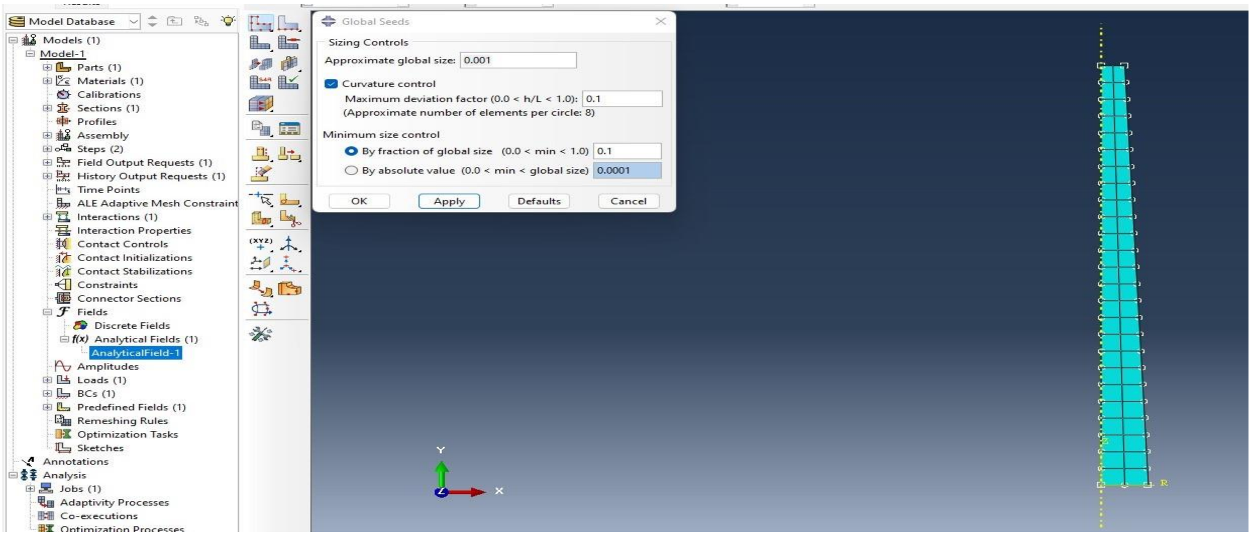
In module select Mesh => Seed Part Instance

Approx. global size => 0.001 => Apply

Mesh Part Instance => Yes

Assign element type => Select Part

In Element Type => Family => Coupled Temperature Displacement



- **Job**

In module select Job => Create Job

Create Job => Model-1 (Job 1)

Job Manager => Submit (Job 1)

- **Visualizations**

Verify the Results and Plot the graphs.

Select Path -> Node List

Select the Bottom node and Top node

Select XY Data -> Source -> Path

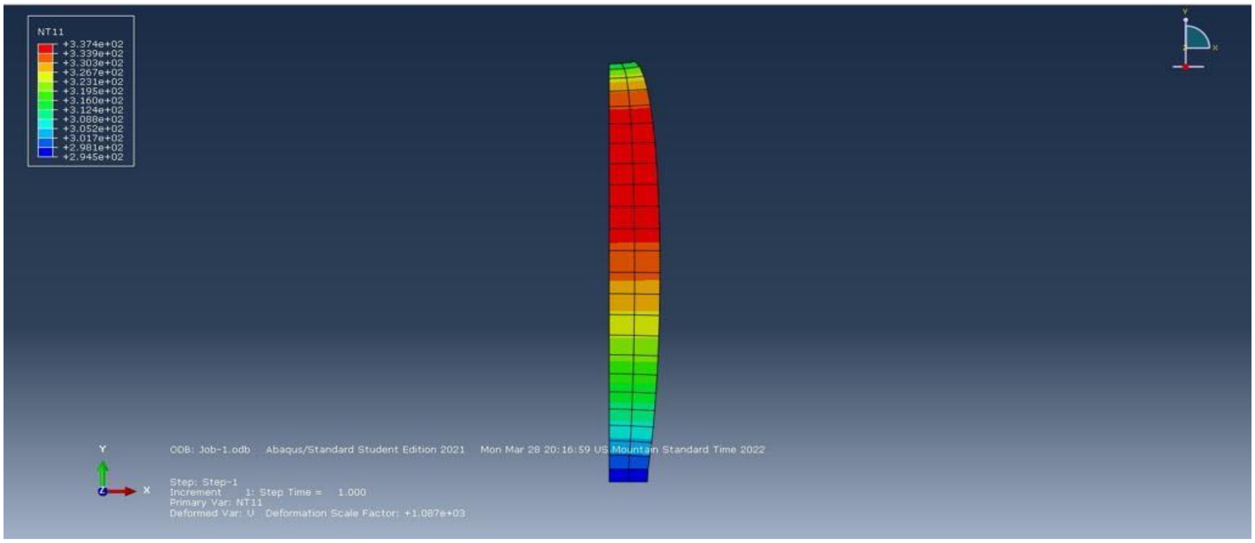
Point Location -> Include Intersections

X-Values -> Y-coordinate

And Plot

Results-

Temperature-



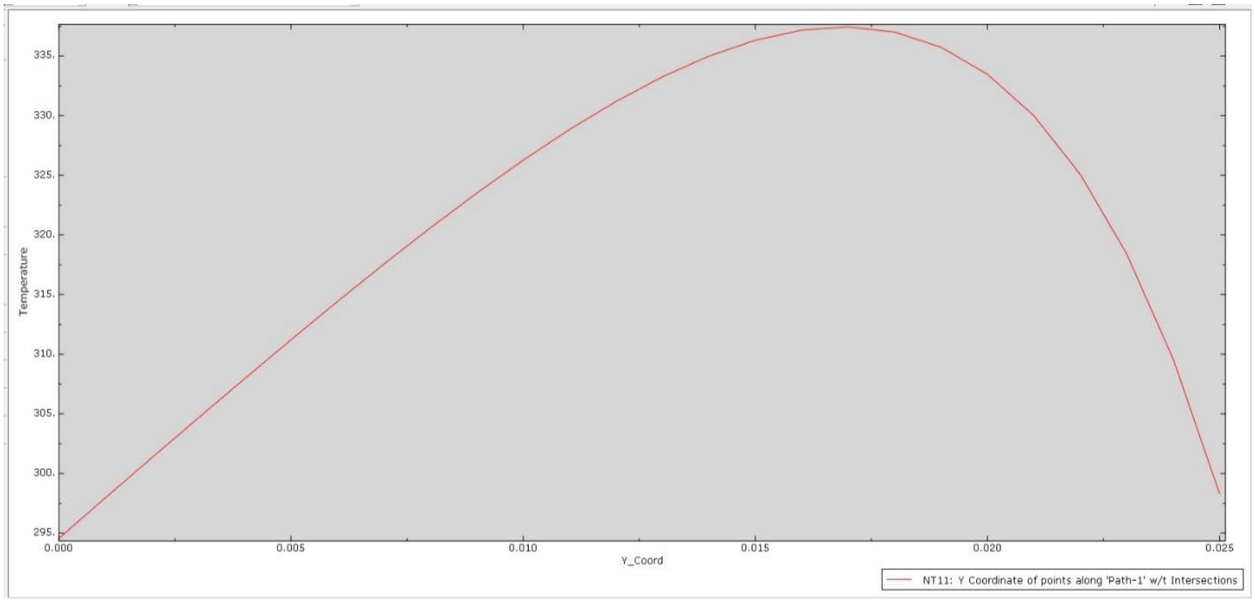
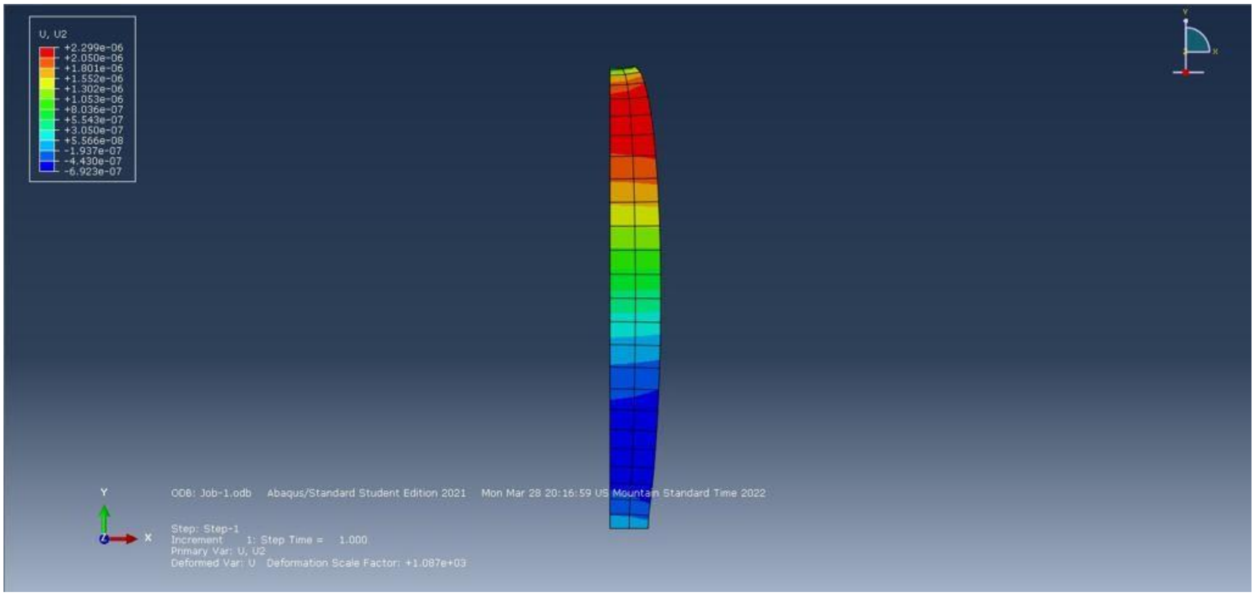


Figure 1 -Temperature Plot

Displacement-



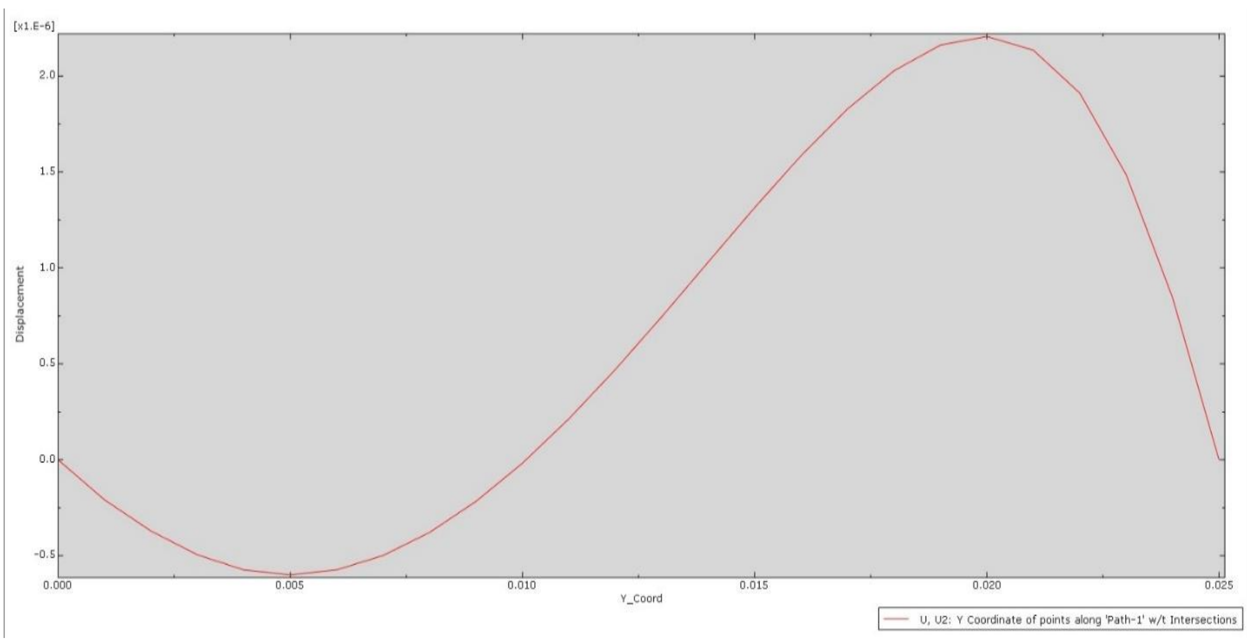
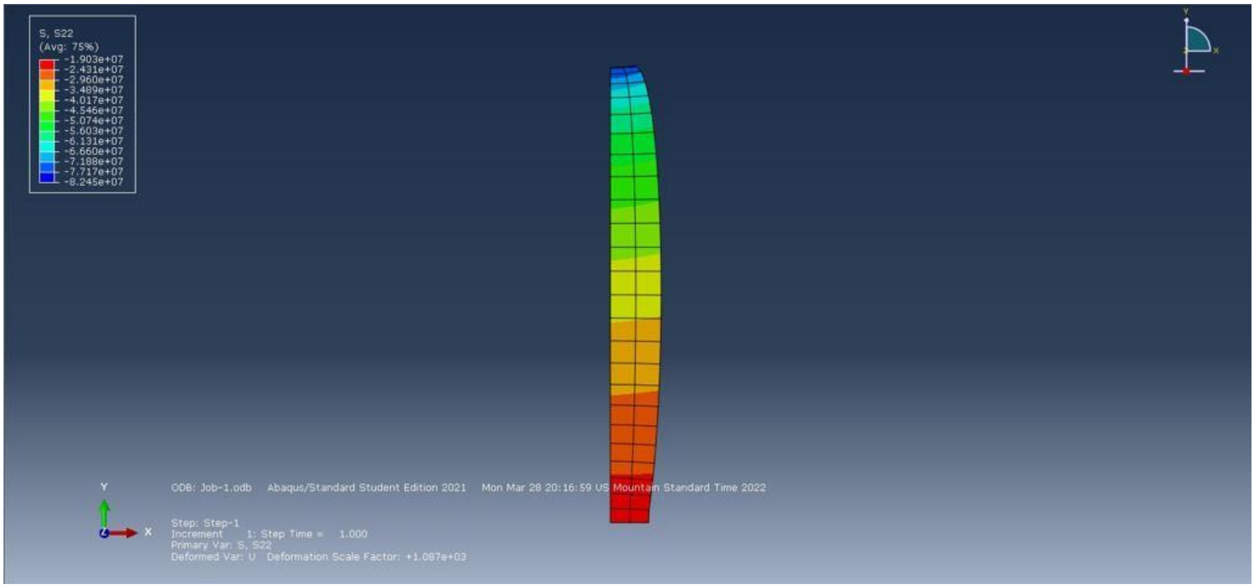


Figure 2- Displacement Plot

Stress Field-



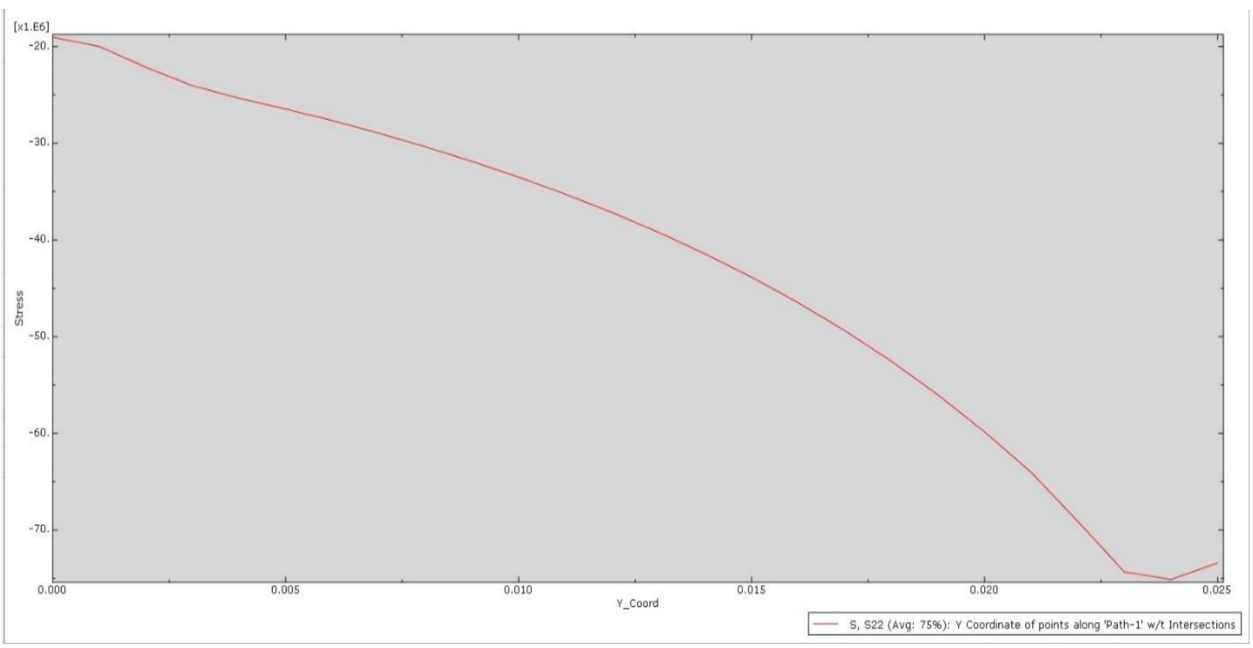


Figure 3- Stress Field Plot

Mangirish Kulkarni (ASU ID – 1 223 229 852)
Onkar Chavan (ASU ID – 1 223 314 248)

```
L = 0.025;
k = 205;
i = 400;
ro = 2.65 * 10^-8;
h = 500;
A = (Pi * (2 - x / L) ^2) * 10^-6;
S = ro * (i / A) ^2;
tsol = Integrate[((Integrate[((-S) * A), x] + c1) / (k * A)), x] + c2;
m = (-k) * D[tsol, x];
La = ((-h) * 10^3) * ((tsol /. x → 0) - 293.15);
Lb = (h * 10^3) * ((tsol /. x → 0.025) - 293.15);
Ra = m /. x → 0;
Rb = m /. x → 0.025;
TbcSolve = Solve[La - Ra == 0 && Lb - Rb == 0]
TemperatureField = tsol /. TbcSolve;
Tleft = tsol /. TbcSolve /. x → 0
TRight = tsol /. TbcSolve /. x → 0.025
Disp = Integrate[(n1 / (k * A) + (21 * 10^-6) * (TemperatureField - 293.15)), x] + n2
DbcSolve = Solve[(Disp /. x → 0) == 0 && (Disp /. x → 0.025) == 0]
DisplacementField = Disp /. DbcSolve
Strain = D[DisplacementField, x]
Stress = ((70 * 10^9) * (Strain - (21 * 10^-6) * (TemperatureField - 293.15)))

Plot[Stress, {x, 0, 0.025}, PlotTheme → "Detailed"]
Plot[TemperatureField, {x, 0, 0.025}, PlotTheme → "Detailed"]
Plot[DisplacementField, {x, 0, 0.025}, PlotTheme → "Detailed"]
```

Out[•]= $\{\{c1 \to 25.505,\ c2 \to -36.7851\}\}$

Out[•]= $\{294.524\}$

Out[•]= $\{298.393\}$

Out[•]= $\left\{n2 - 0.00692864\,x + \frac{1.\ \left(8.59531 \times 10^{-6} - 0.970457\,n1\right)}{-0.05 + 1.\,x} - 0.000519781\,\mathrm{Log}[0.05 - 1.\,x]\right\}$

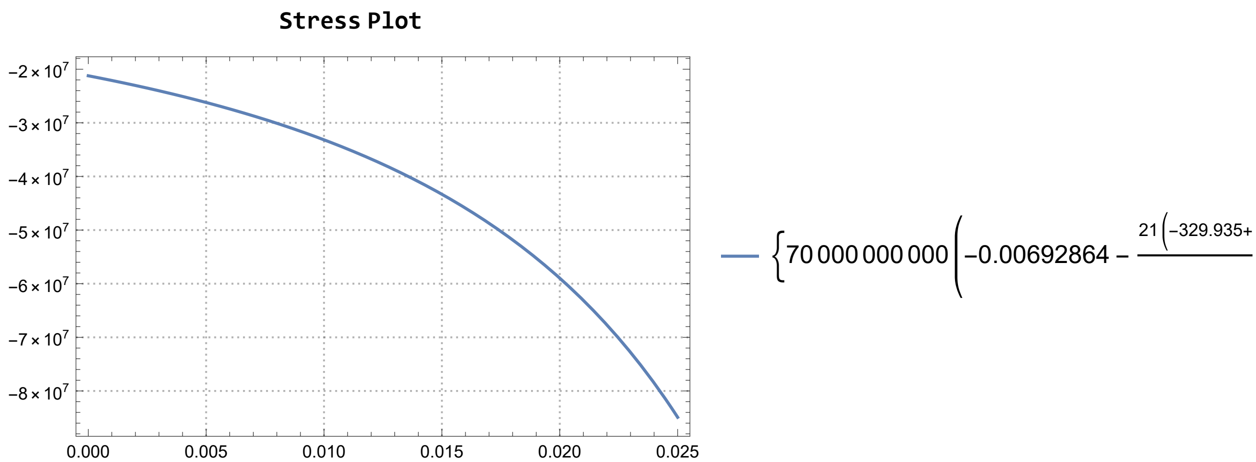
Out[•]= $\left\{\left\{n1 \to -7.81209 \times 10^{-7},\ n2 \to -0.00137006\right\}\right\}$

Out[•]= $\left\{\left\{-0.00137006 - 0.00692864\,x + \frac{9.35344 \times 10^{-6}}{-0.05 + 1.\,x} - 0.000519781\,\mathrm{Log}[0.05 - 1.\,x]\right\}\right\}$

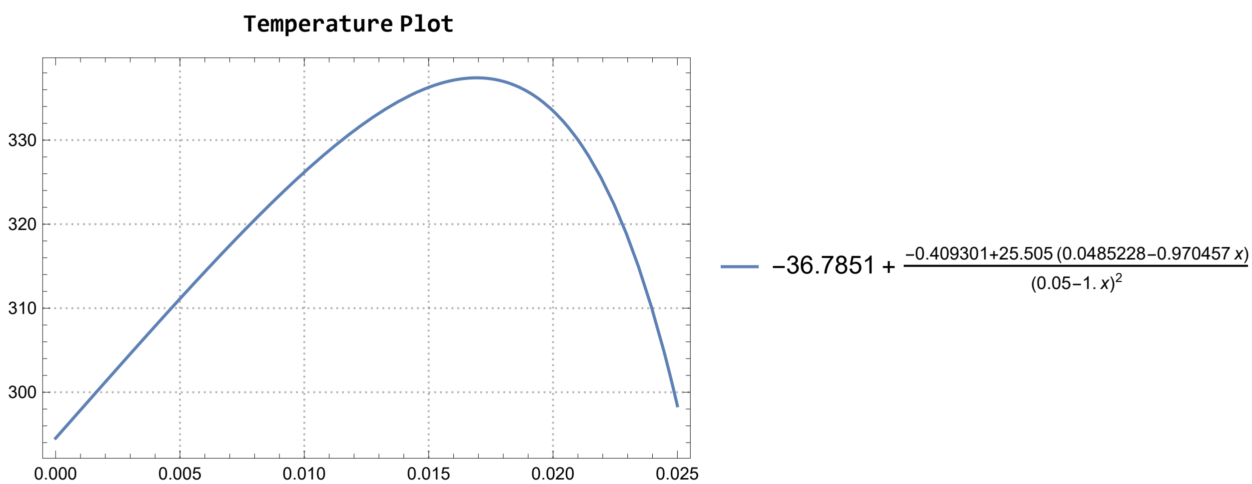
Out[•]= $\left\{\left\{-0.00692864 + \frac{0.000519781}{0.05 - 1.\,x} - \frac{9.35344 \times 10^{-6}}{(-0.05 + 1.\,x)^2}\right\}\right\}$

Out[ ]= $$\left\{\left\{70\,000\,000\,000\left(-0.00692864-\frac{21\left(-329.935+\frac{-0.409301+25.505\,(0.0485228-0.970457\,x)}{(0.05-1.\,x)^2}\right)}{1\,000\,000}+\frac{0.000519781}{0.05-1.\,x}-\frac{9.35344\times10^{-6}}{(-0.05+1.\,x)^2}\right)\right\}\right\}$$

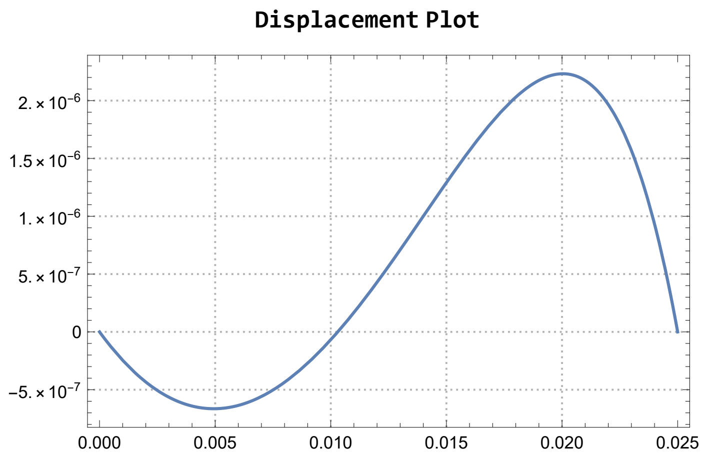
Out[ ]=



Out[ ]=

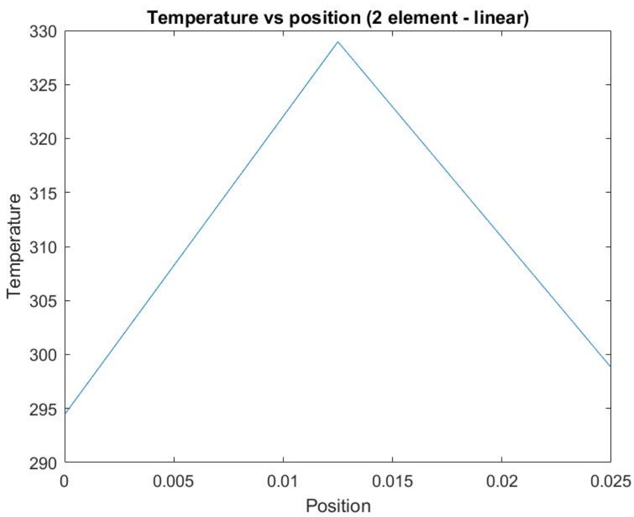


Out[ ]=

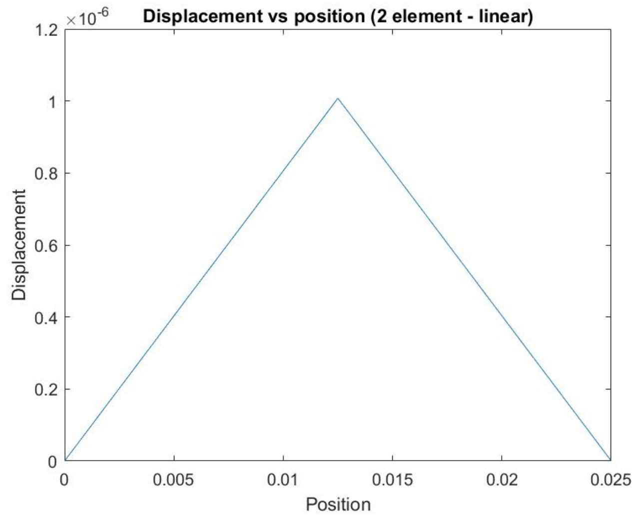


$$\left\{-0.00137006 - 0.00692864\,x + \frac{9.35344\times10^{-6}}{-0.05+1.\,x}\right.$$

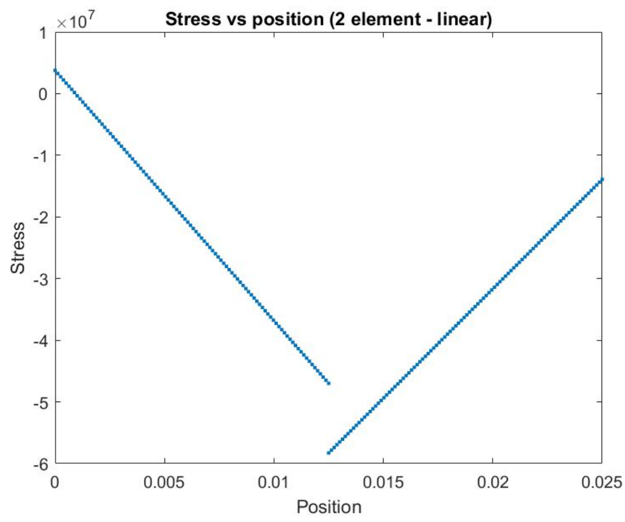
MatLab plot:



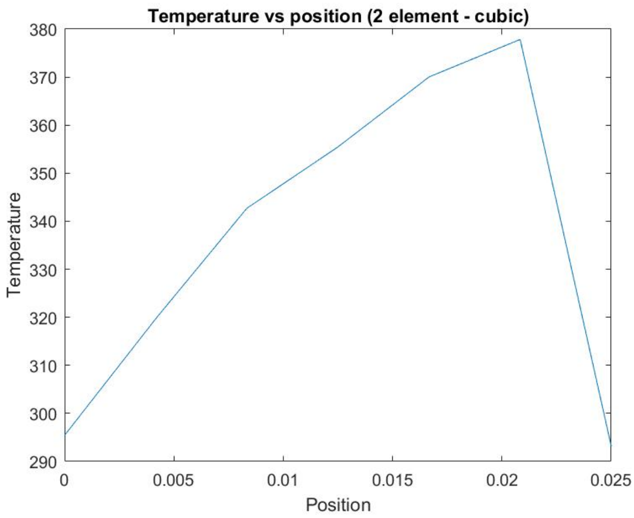
Temperature vs position (2 element - linear)
Temperature
Position
330
325
320
315
310
305
300
295
290
0
0.005
0.01
0.015
0.02
0.025



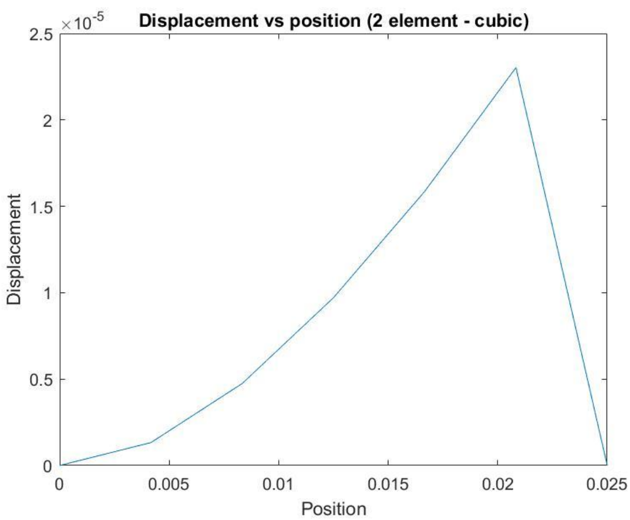
Displacement vs position (2 element - linear)
×10^-6
Displacement
Position
1.2
1
0.8
0.6
0.4
0.2
0
0
0.005
0.01
0.015
0.02
0.025



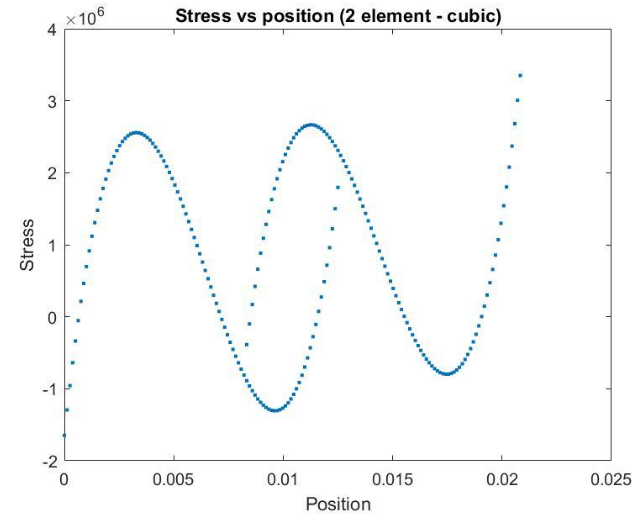
Stress vs position (2 element - linear)
×10^7
1
0
-1
-2
-3
-4
-5
-6
Stress
0
0.005
0.01
0.015
0.02
0.025
Position

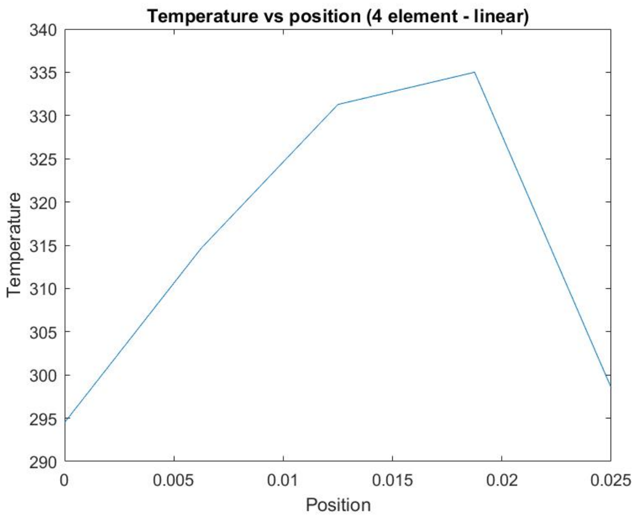


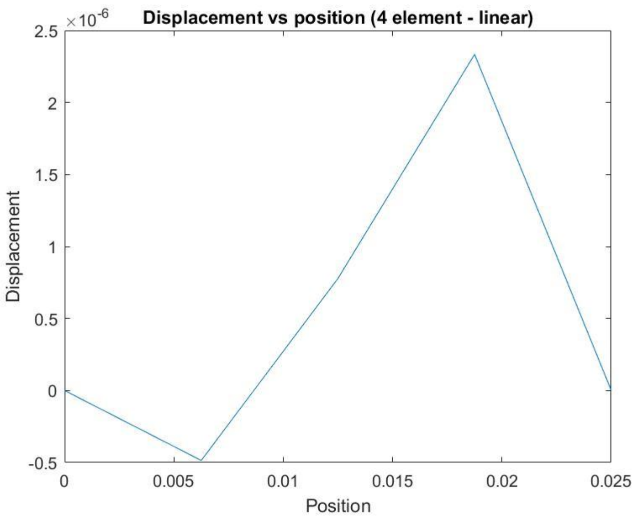
Temperature vs position (2 element - cubic)
380
370
360
350
340
330
320
310
300
290
Temperature
0
0.005
0.01
0.015
0.02
0.025
Position

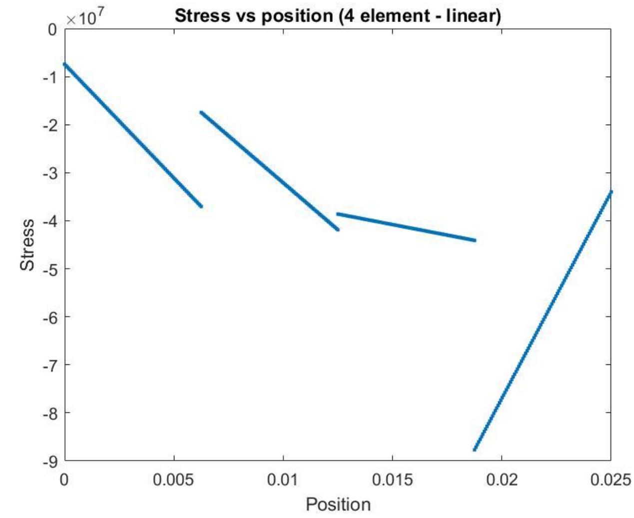


Displacement vs position (2 element - cubic)
×10^-5
2.5
2
1.5
1
0.5
0
Displacement
0
0.005
0.01
0.015
0.02
0.025
Position

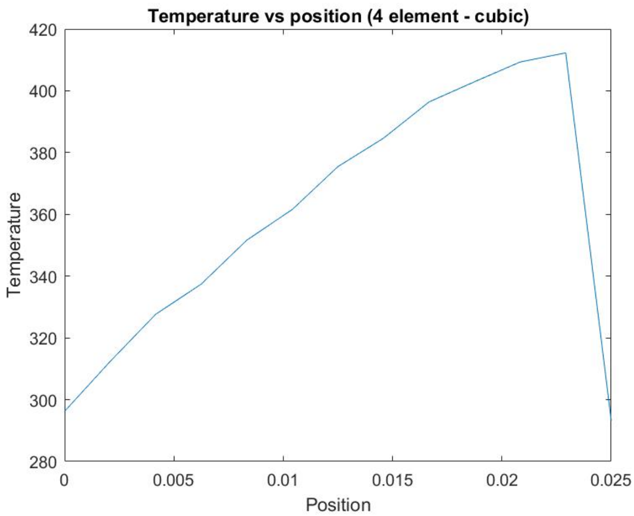

Stress vs position (2 element - cubic)
×10^6
Stress
Position


Temperature vs position (4 element - linear)
Temperature
Position

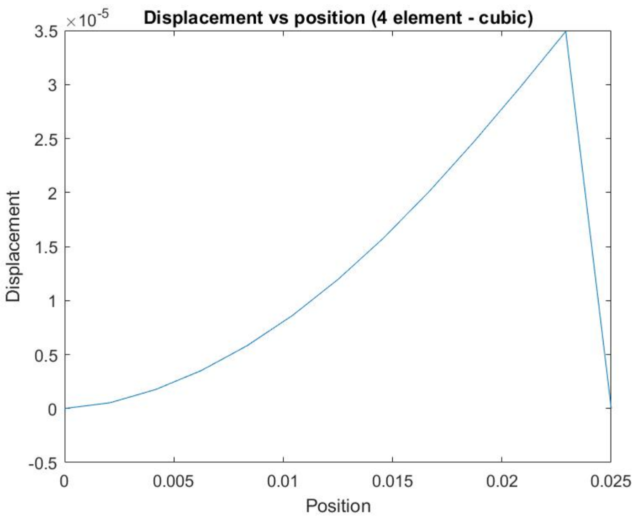

Displacement vs position (4 element - linear)
×10^-6
Displacement
Position



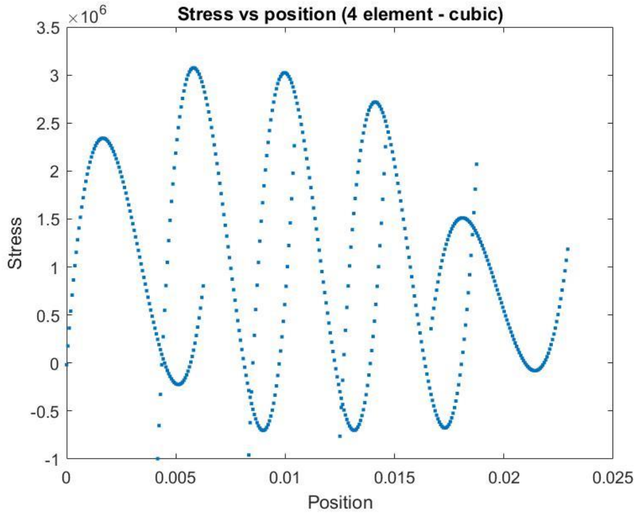
Stress vs position (4 element - linear)
×10^7
Stress
Position
0
-1
-2
-3
-4
-5
-6
-7
-8
-9
0
0.005
0.01
0.015
0.02
0.025



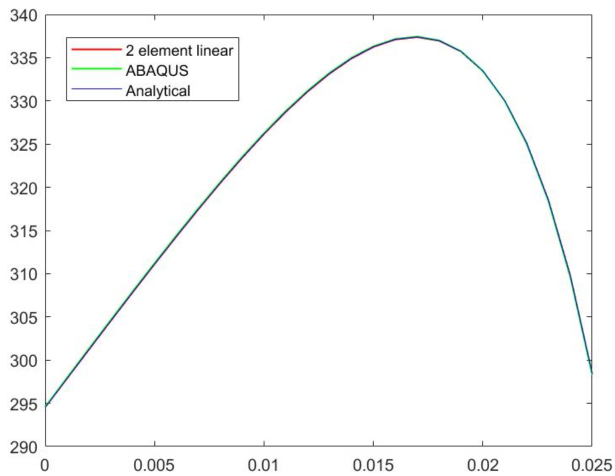
Temperature vs position (4 element - cubic)
Temperature
Position
420
400
380
360
340
320
300
280
0
0.005
0.01
0.015
0.02
0.025

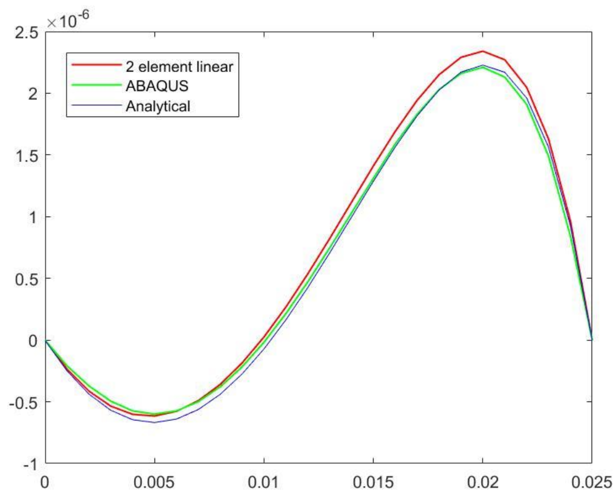


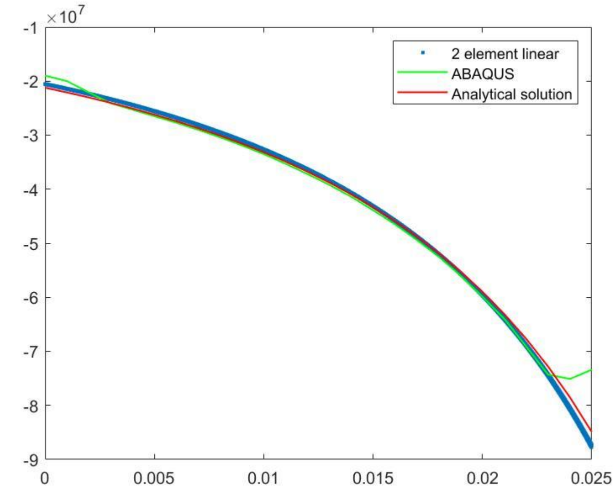
Displacement vs position (4 element - cubic)
×10^-5
Displacement
Position
3.5
3
2.5
2
1.5
1
0.5
0
-0.5
0
0.005
0.01
0.015
0.02
0.025



Comparison plots:

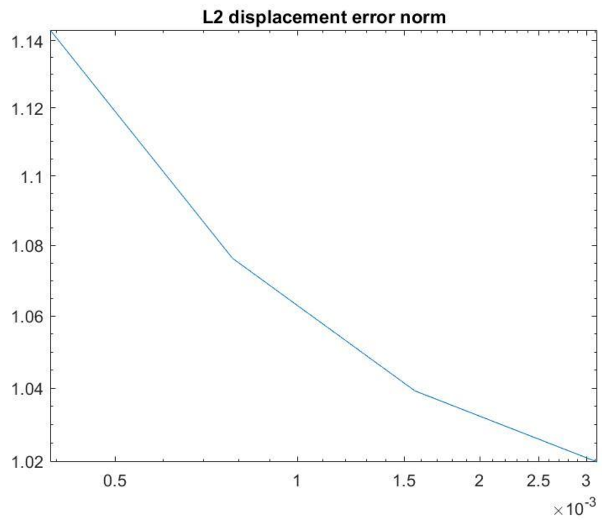






As can be seen above, all three approaches have the same solution for temperature and stress. The displacement plot diverges at peak points, which may occur due while rounding off in calculation.

L2 displacement error norm plot:



Above is the plot for displacement error norm for linear elements.