ARIZONA STATE UNIVERSITY

MAE 503

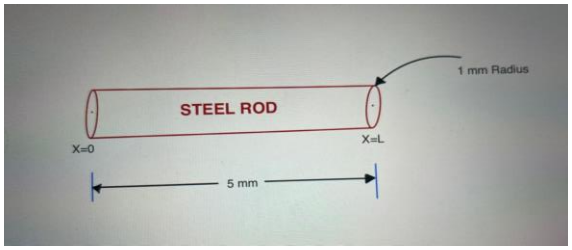
FINITE ELEMENTS IN ENGINEERING

Professor- Dr. Jay Oswald.

Omkar Rajendra Gaikwad | ogaikwa1 |1226843997
Harshal Rajesh Tingre | htingre | 1226039050

Task 1-

Restate the problem that you are solving in your own words. Fully write out the governing equation's strong form, including fully describing the boundary conditions. Write out the weak form of the governing equations and show the integral expressions for each of the finite element matrices you need to compute (i.e., conductivity matrices and element heat flux vectors). (*Lead: Omkar*)



Determine the steady-state temperature and heat flux fields of A circular steel rod with a 1 mm radius exposed to a volumetric heat load. The goal is to determine the steady-state temperature and heat flux fields in the rod using three methods:

1. Writing a finite element program in MATLAB.
2. Using ABAQUS or another finite element solver.
3. Solve the governing equations analytically with the help of a symbolic math program like the symbolic math toolbox in MATLAB.

Given Variables:

1. Length (L): 5 mm
2. Radius (r): 1 mm
3. Thermal Conductivity (k): 45 W/m^{0}C
4. Cross-sectional area $(A) = \pi r^2$
5. T_0= 10 ^{0}

Volumetric heat load defined by :

$$\boxed{s(x) = \sin(\omega x)}$$

The heat flux:

$q_{(x=0)} = h(T - T_0)$, Where ($h = 1\ W/mm^2$°C)

$q_{(x=L)} = 1\ W/mm^2$

*Assume there is no heat transfer to the surrounding air.

To find out:

1. Rod's steady-state temperature.
2. Heat flux fields.

The heat transfer in the steel rod is governed by the heat conduction equation:

$$\boxed{\rho * c_p * \frac{\partial T}{\partial t} = \nabla.(k * \nabla T) + q}$$

Where,

ρ = Density (kg/m^3)

c_p= Specific heat capacity $(J/kg \cdot K)$

T = Temperature ($°C$)

t = Time (s)

k = Thermal Conductivity ($W/m.K$)

q = Volumetric Heat Generation Rate (W/m^3)

In steady-state conditions,

$\nabla \cdot (k\ \nabla T) + q = 0$ *(time derivative = 0)*

Considering a one-dimensional heat transfer in the x-direction, the governing equation becomes:

$$\frac{d}{dx}(k\ \frac{dT}{dx}) + q = 0$$

The heat flux at x=0 depends on the temperature and is given by:

$$\boxed{q_{(x=0)} = h(T - T_0)}$$

Where,

$h =$ Heat transfer Co-efficient ($W/mm^2 \cdot k$)

T_0= Reference Temperature (°C)

The right end of the rod is actively cooled so the boundary condition is:

$q_{(x=L)} = -1$ *(Negative sign for a heat flux of 1 W/mm^2 leaving the bar)*

Where,

$q_{(x=L)}$is the heat flux at $(x = L)$.

At $(x = 0)$, we have a convective boundary condition:

$q_{(x=0)} = -h(T - T_0)$

Where,

$q_{(x=0)}$ is the heat flux at $(x = 0)$.

Assume, the rod is perfectly insulated at the lateral surface, so there is no heat transfer to the surrounding air.:

$\frac{dT}{dx}$= 0 *(lateral boundary conditions are adiabatic* $(x = \pm r)$

where,

$r = 1$ mm is the radius of the rod.

Energy Balance: Heat Flux

$$\boxed{S\left(x + \frac{\Delta x}{2}\right) * \Delta x + q(x) * A(x) - (x + \Delta x) * A\ (x + \Delta x) = 0}$$

Divide by Δx and after taking the limit $\rightarrow$ 0

$$S(x) - \frac{d}{dx}(qA) = 0$$

Where,

$$q = -k \frac{dT}{dx}$$

The strong form of the governing equation is:

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

Boundary Conditions:

At $(x = 0)$, $q = h(T - T_0)$

At $(x = L)$, $q = -1$

The weak form of the governing equation for steady-state heat conduction in the rod is obtained by multiplying the strong form by a test function, integrating over the domain, and applying the divergence theorem, so

Multiplying by a test function and Integrate over the domain we get,

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

After Integrating by parts, we get,

$$\boxed{-wAh\,\frac{dT}{dx} - \int_0^L \frac{dw}{dx}\left(Ak\frac{dT}{dx}\,dx\right) + \int_0^L wsdx = 0}$$

Rearranging the equation

$$\boxed{-\int_0^L \frac{dw}{dx}\left(Ak\frac{dT}{dx}\,dx\right) + \int_0^L ws\,dx - wAh(T - T_0) = 0}$$

Hence, the weak form of the governing equation

Find $\mathrm{T} \in C^o$ such that $(T = T_0)$

$$\boxed{\int_0^L \frac{dw}{dx}\left(Ak\frac{dT}{dx}\,dx\right) = \int_0^L ws\,dx - wAh(T - T_0)}$$

$\forall w \in C^o$ such that(w=0)

TASK 2-

Specify a consistent system of units. ***(Lead: Omkar)***

SI Units:

Mass	Length	Time	Temperature	Force	Energy	Density	Young's Modulus	Gravity	Stress
Kg	m	s	K	N	J	7.83e+03	2.07e+11	9.806	Pa

Task 3-

Clearly explain how you modelled the problem in ABAQUS: explain what type of elements you used, what assumptions you made, and how loads and boundary conditions were implemented. *(Lead: Omkar)*

To model the problem in ABAQUS, we first create a 3D axial-symmetric model of the steel rod with a circular cross-section, using axisymmetric elements.

Assumptions:

1. No heat transfer to the surrounding air.
2. Rod is homogenous and
3. Rod is isotropic, and that the
4. heat flux at x=0 is given by,
   $q_{(x=0)} = h(T - T_0)$

Step 1 - Create Part.

In module select Part →Create Part

Select Modeling Space → 3D Space

Type → Extrusion

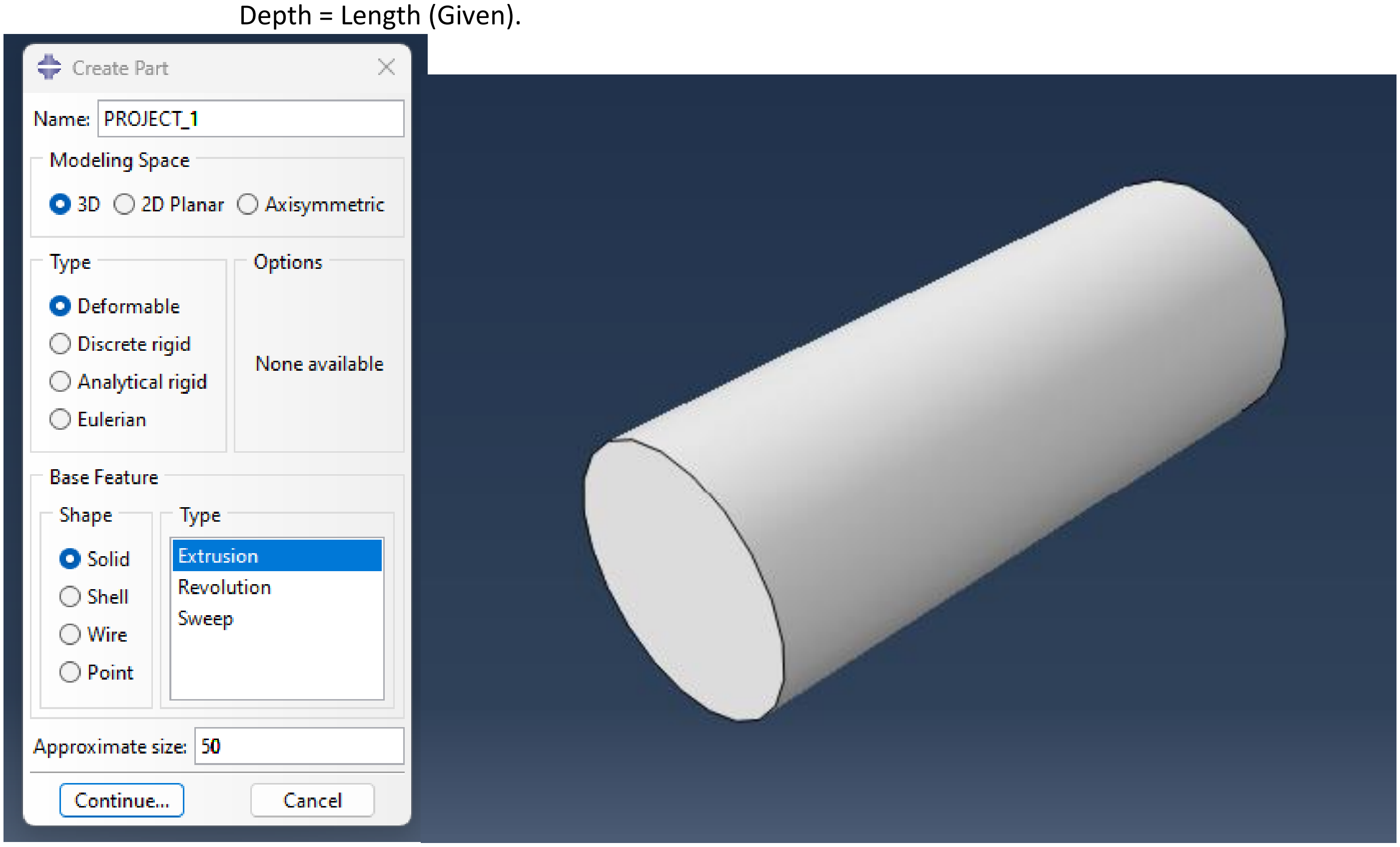
Shape → Solid

Now create the part.

Co-ordinates for center point = (0,0)

Create circle with given radius.

Depth = Length (Given).



Step 02 –

A. Create Material.

In module select Property → Create Material

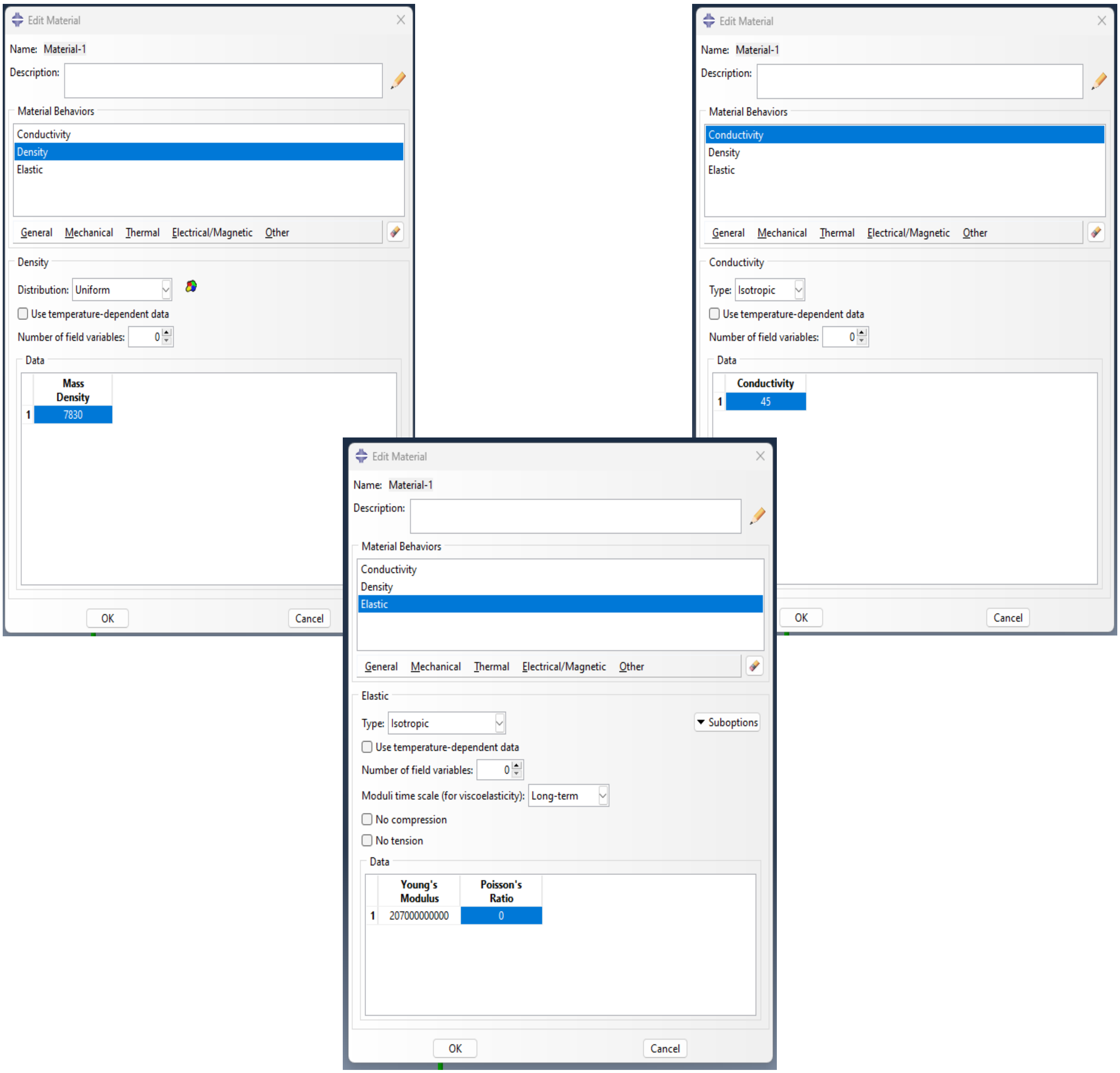
Enter the given properties of material (Steel).

Thermal → Conductivity → 45

General → Density →7830

Mechanical → Elasticity → Elastic → Young's Modulus → 207*10^9

Mechanical → Elasticity → Elastic → Poisson's Ratio → 0



B. Create Section.

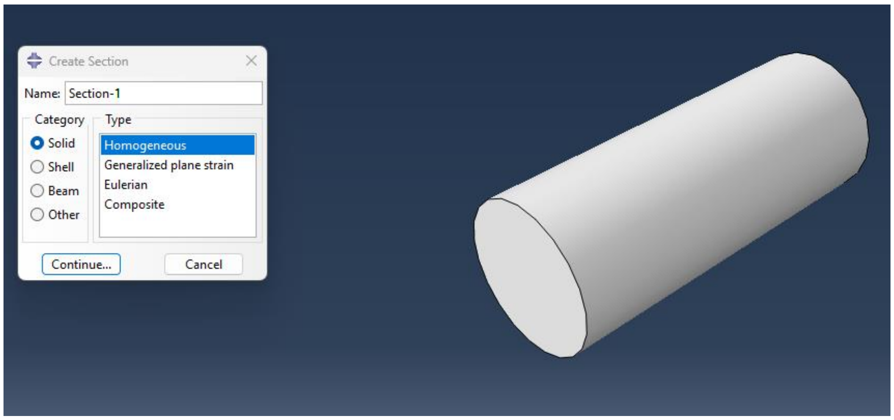
In module select Property → Create Section

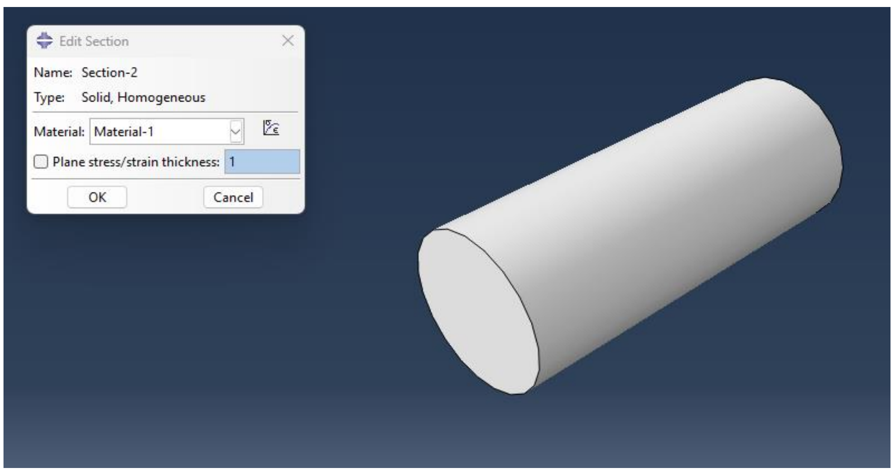
Category → Solid

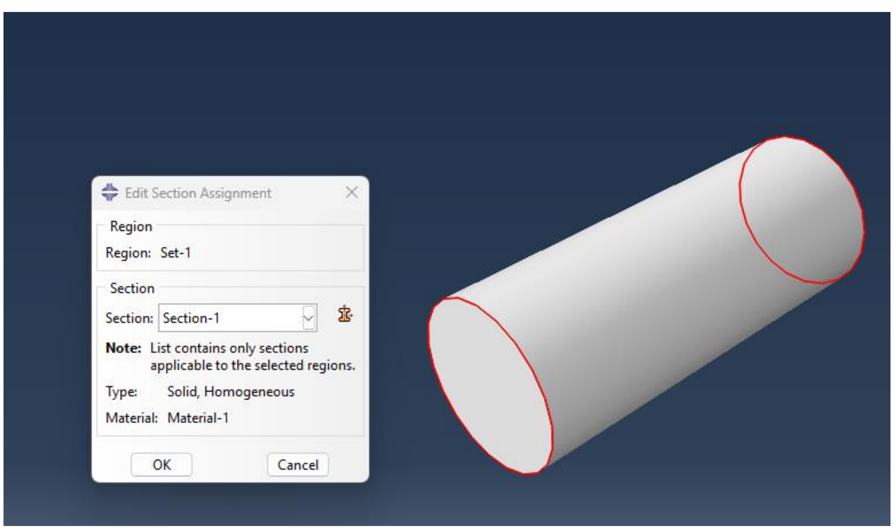
Type → Homogeneous

In module select Property → Assign Section

Select the part → Done.







Step 3 – Mesh.

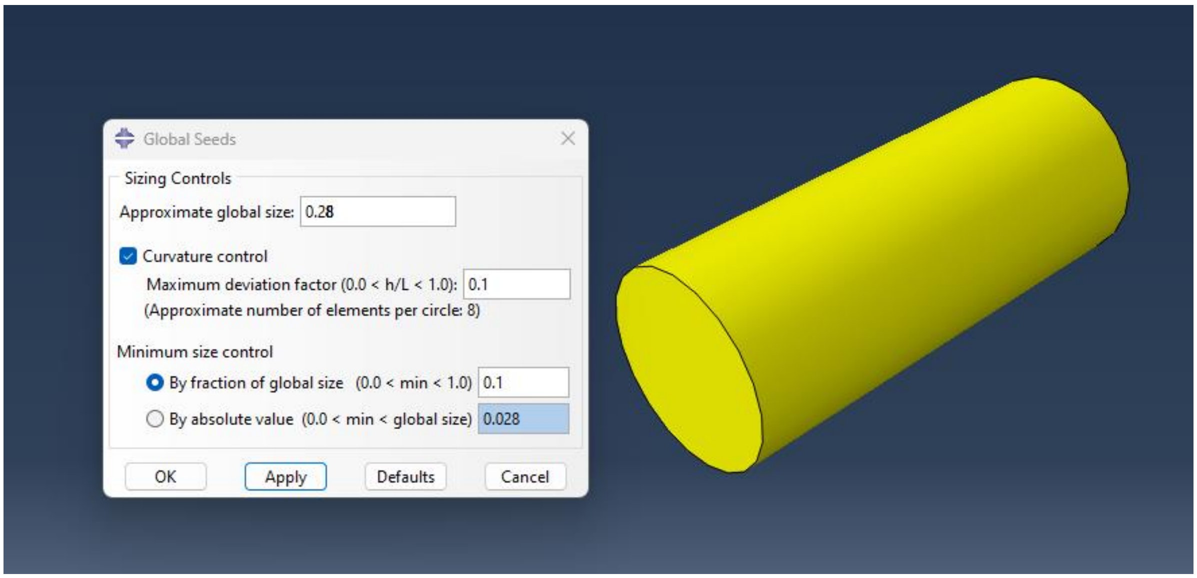
In module select Mesh ⟶ Seed Part Instance

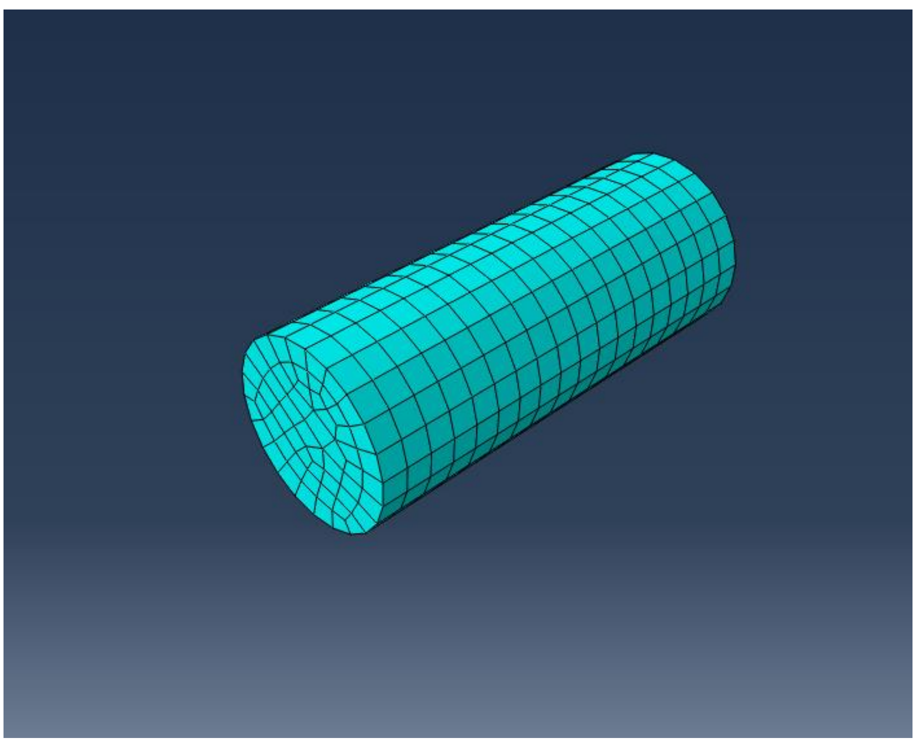
Approx. global size ⟶ 0.28 ⟶ Apply.

Mesh Part Instance ⟶ Yes

Assign element type ⟶ Select Part

253 elements have been generated on instance.





Step 4 – Assembly.

In module select Assembly → Create Instances

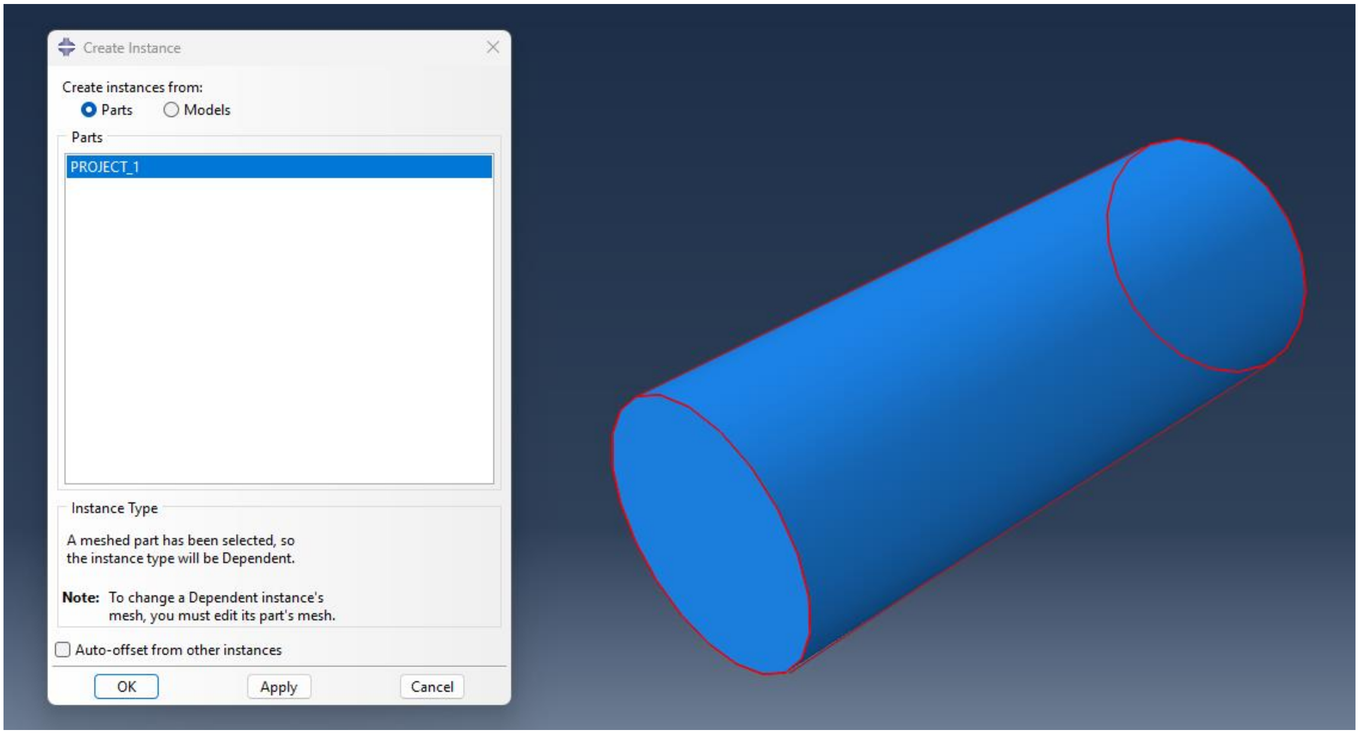
Select → Parts.

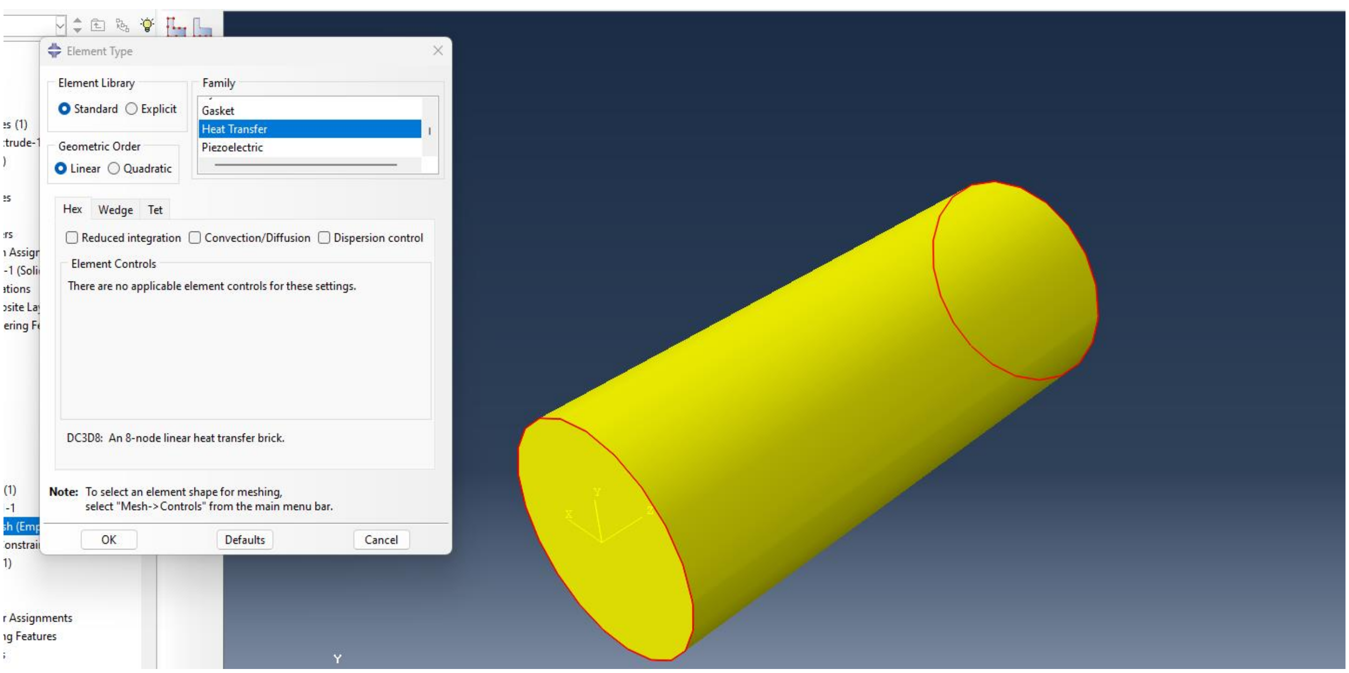
Instance Type → Independent.

Element Library → Standard.

Geometric Order → Linear.

Change Family → Heat Transfer.



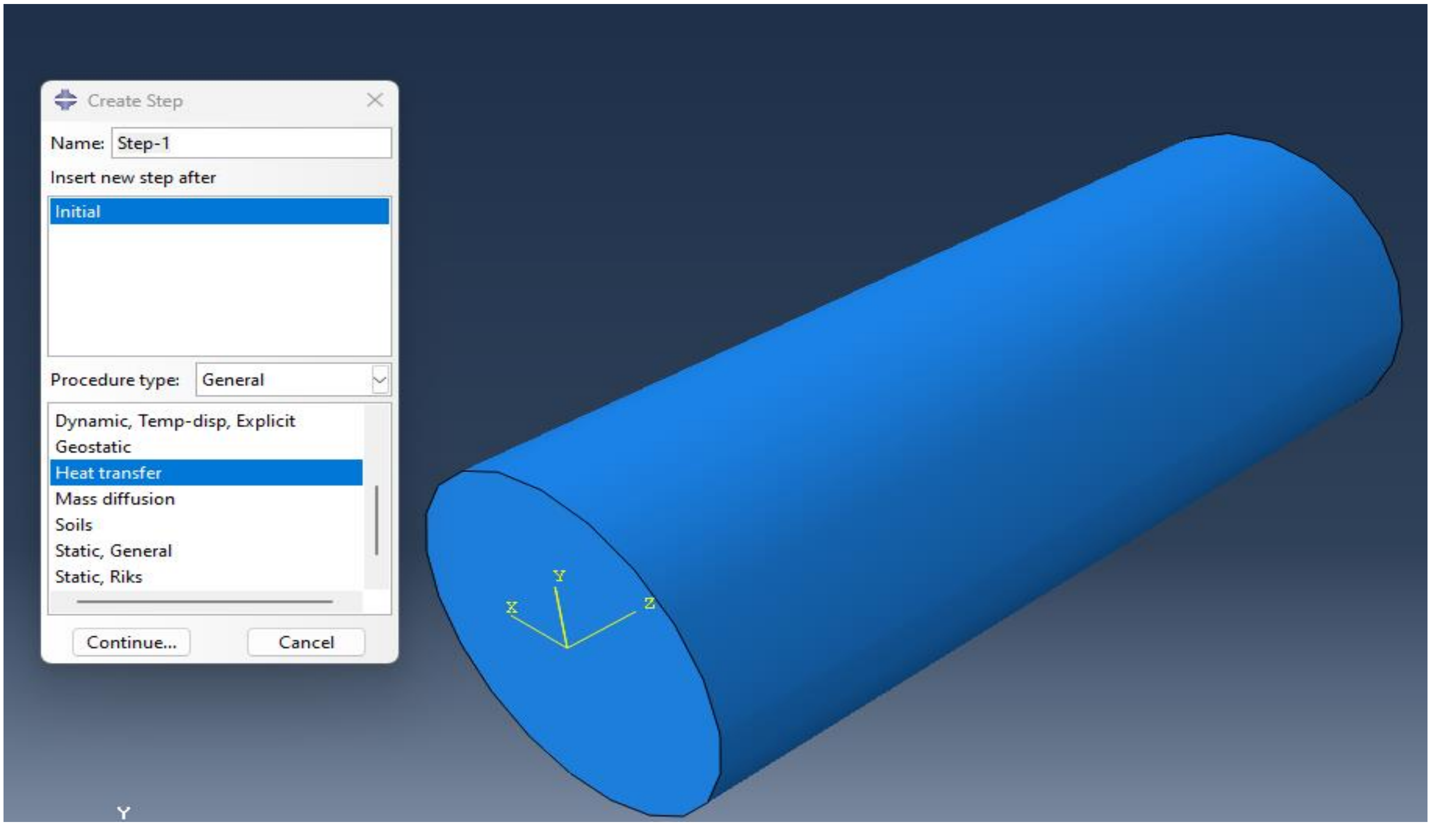


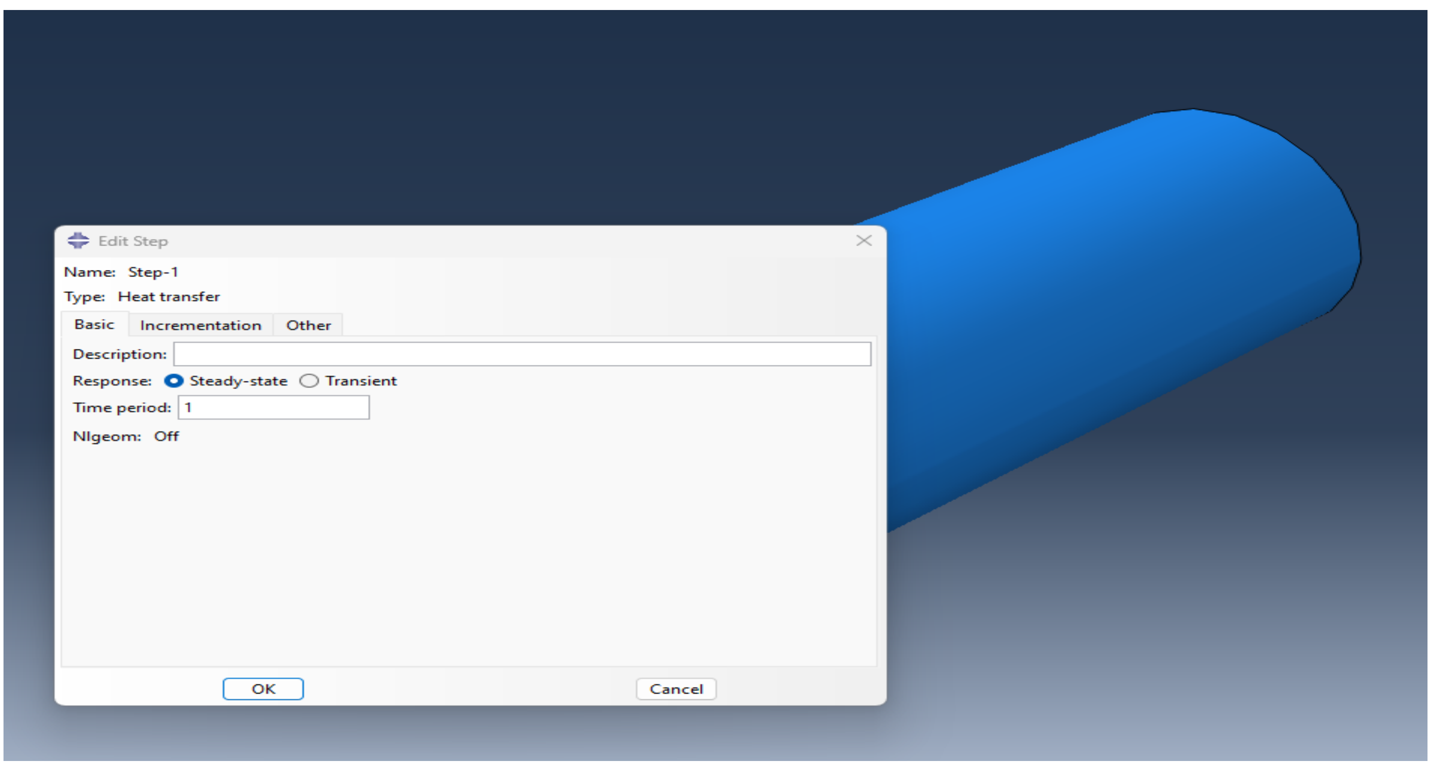
Step 5 – Step.

In module select Step → Create Step

Procedure Type →General →Heat Transfer.

Response → This is a Steady State Problem so select Steady State.





Step 6 – Boundary Conditions

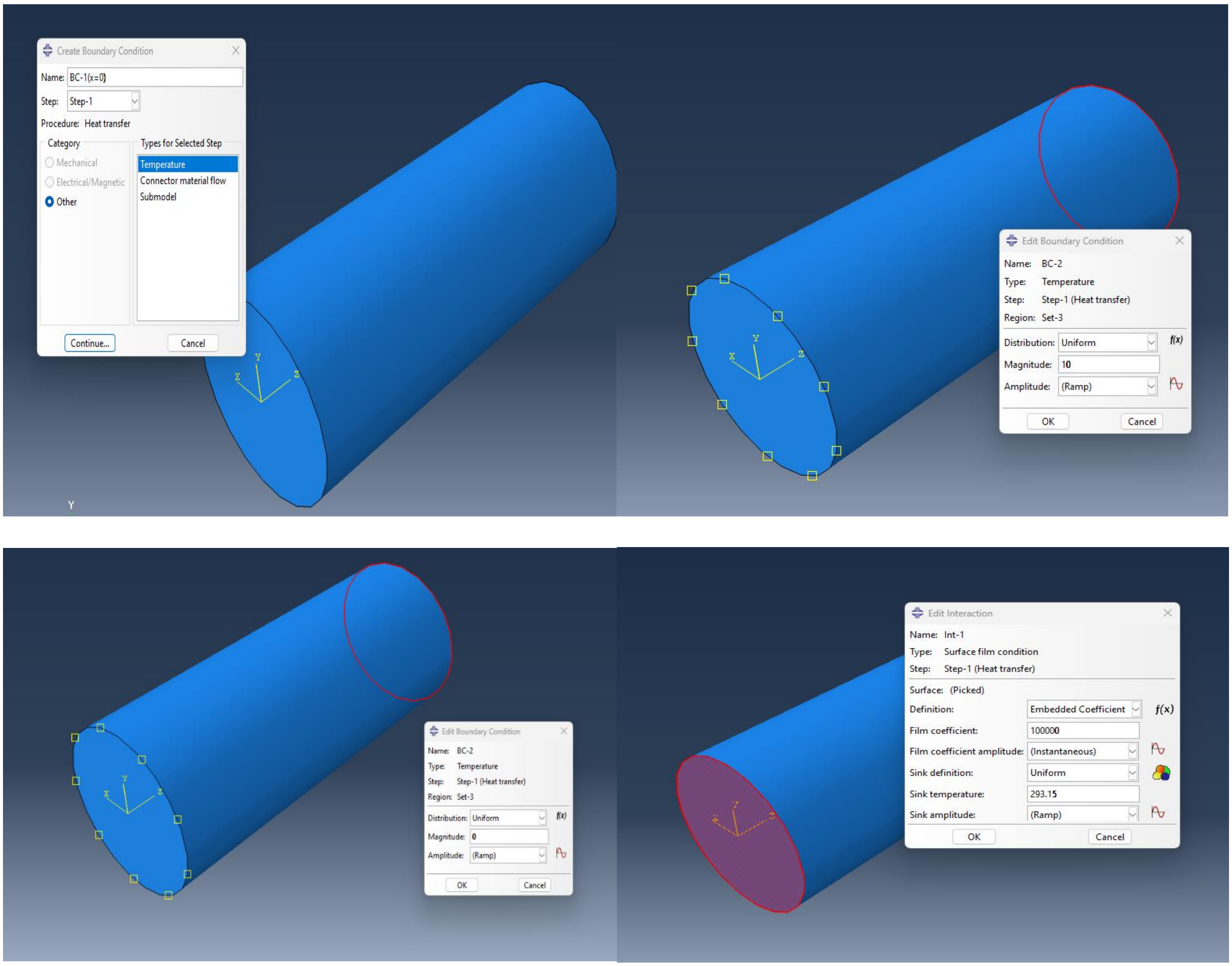
In module select Interaction → Create Interaction

Select → Surface Film Condition

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

Film Coefficient → 100000

Sink Temperature → 293.15.



Step 7- Load

In module select Load → Create Load

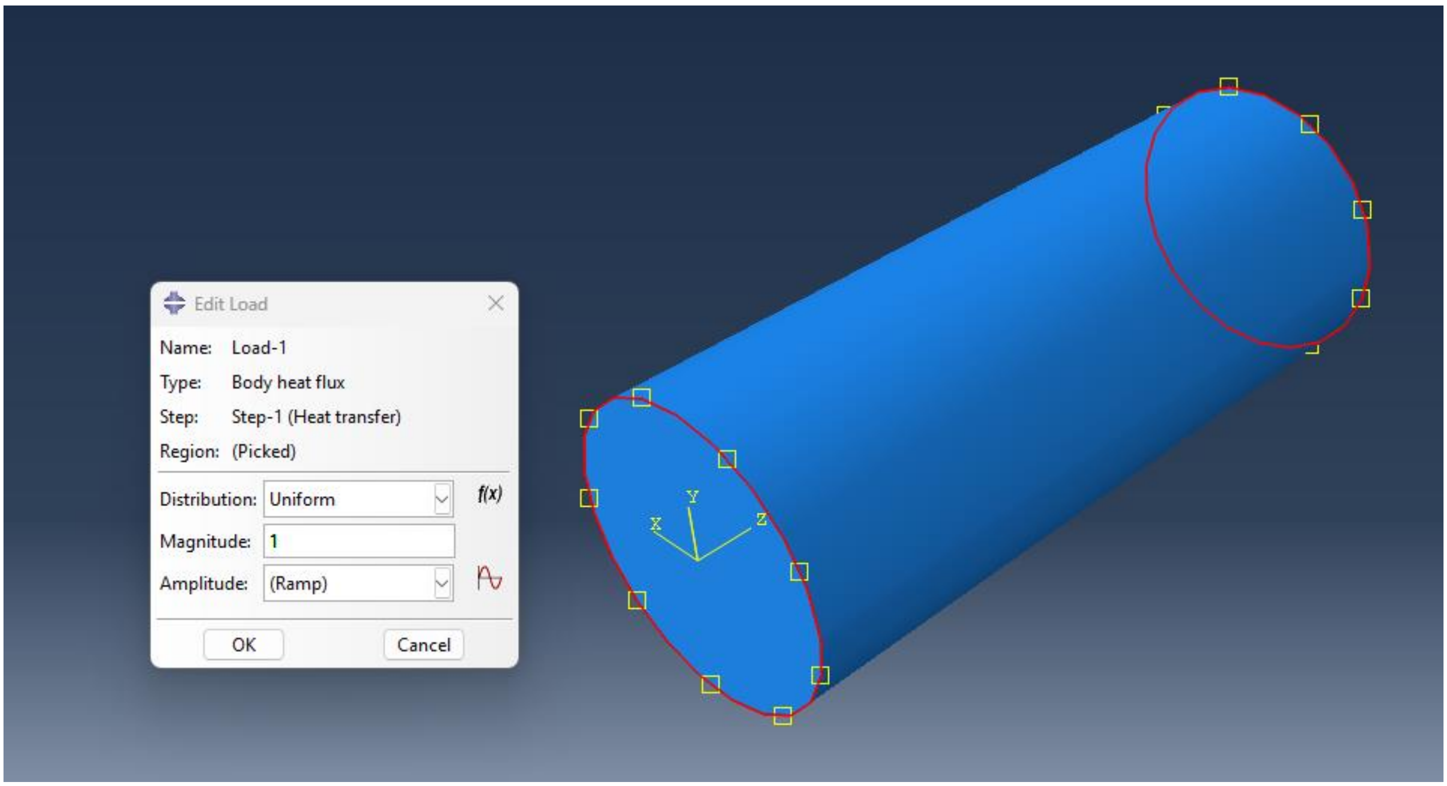
Category → Thermal

Type → Body Heat Flux

Continue → Select the Part

A uniform distribution.

Magnitude → 1



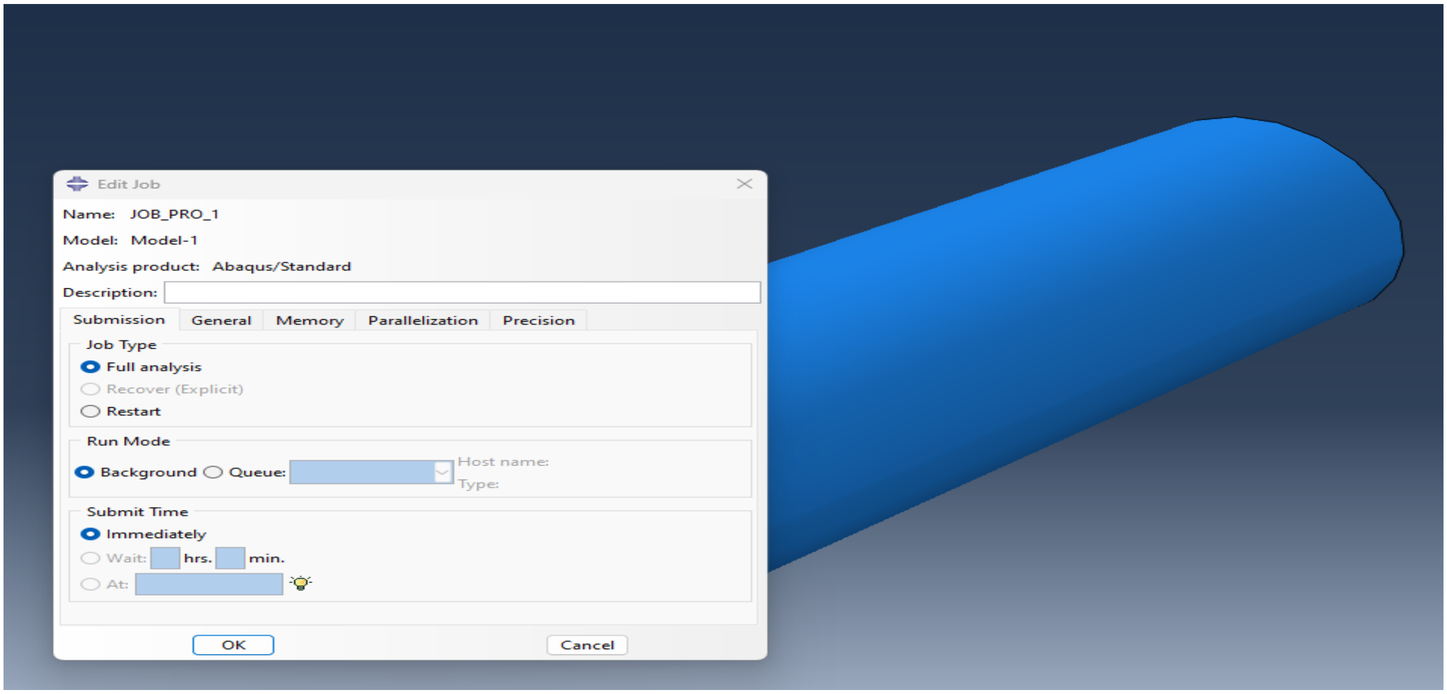
Step 8 – Job

In module select Job → Create Job

Create Job → Model-1 (JOB 1)

Job Manager → Submit (JOB 1)

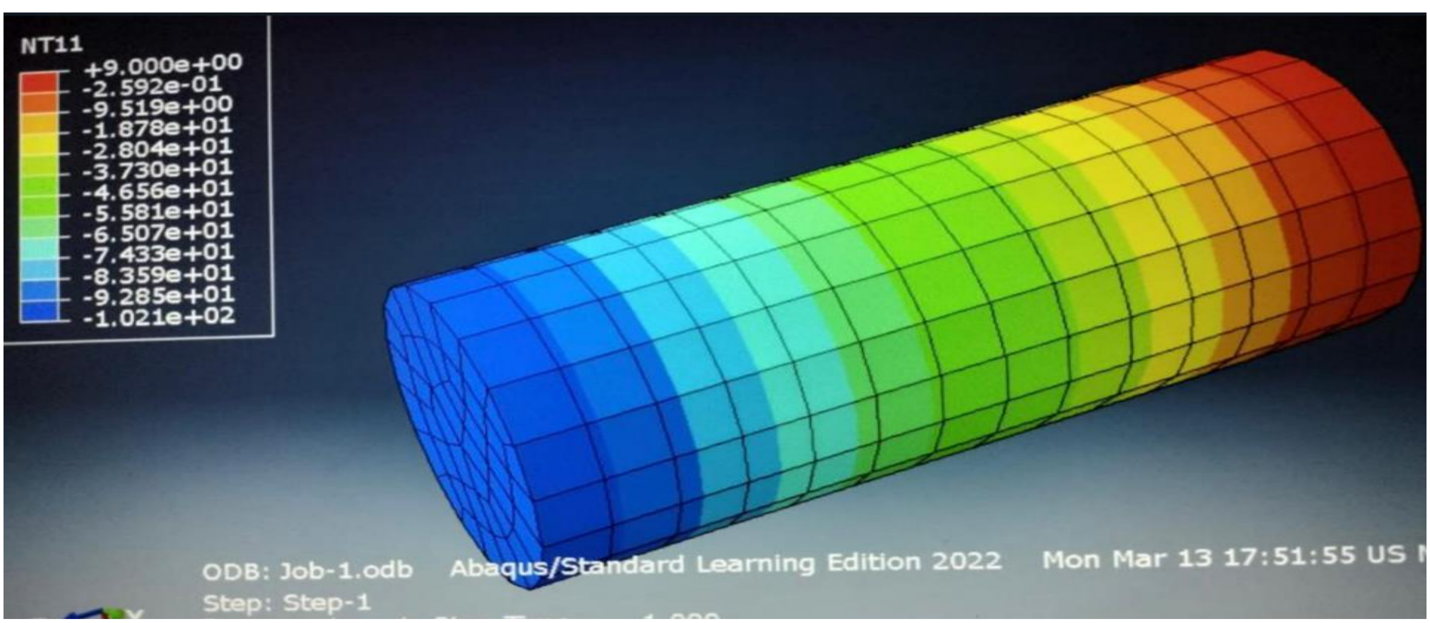
After Status is Completed check the result.



Step 9 – Visualizations

Check the Results and Plot the graphs.

Results



Step 10- Graphs

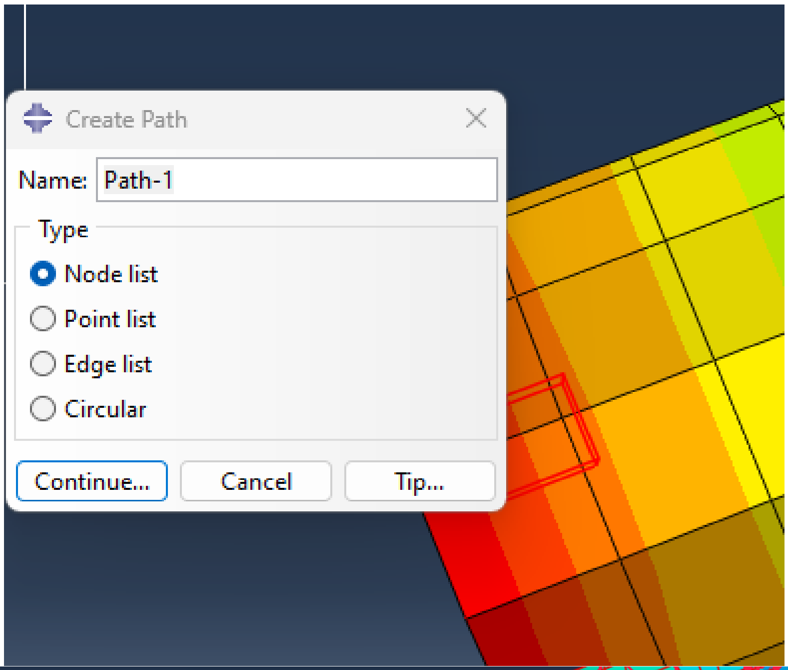
Select Path → Node List

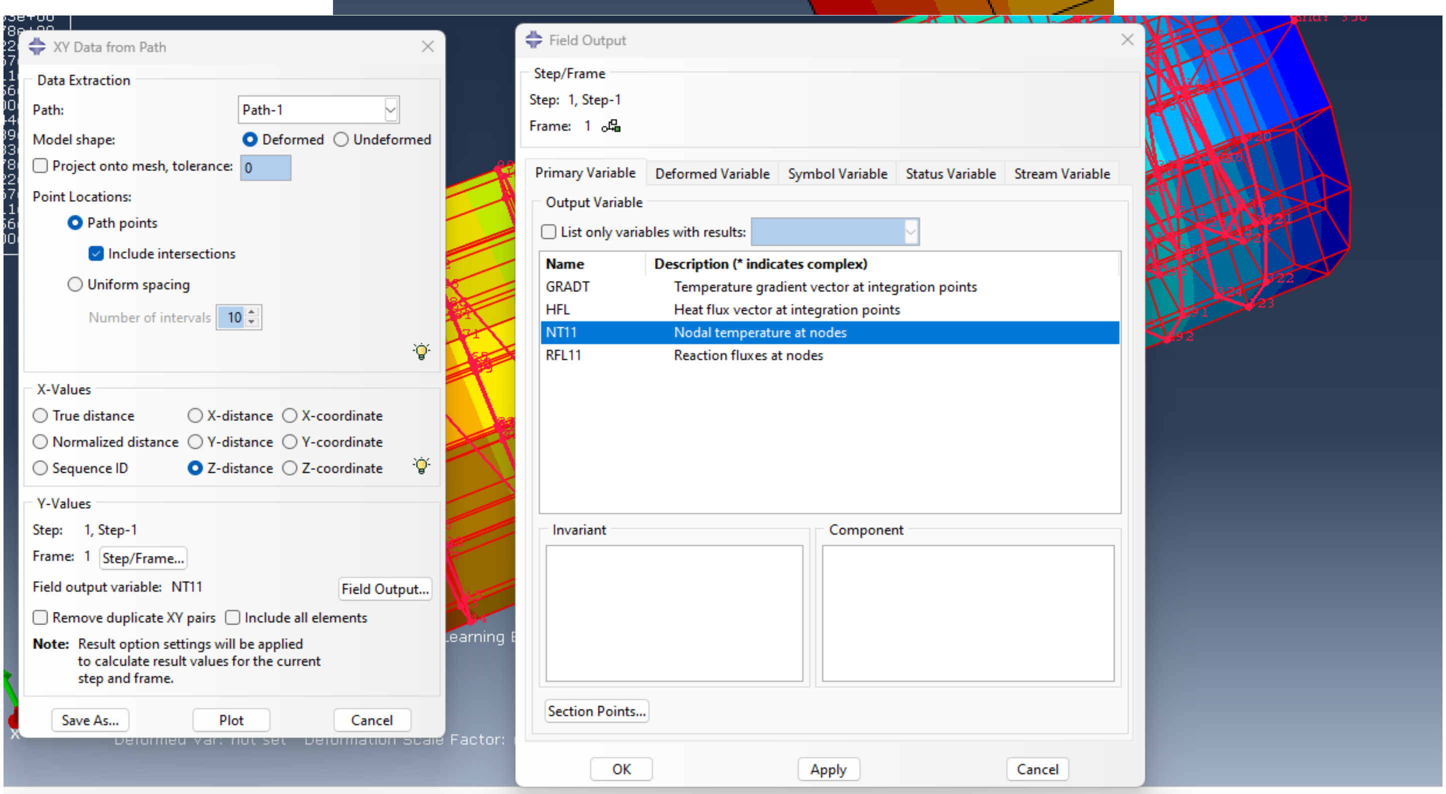
Select the Bottom node and Top node.

Select XY Data → Source → Path.

Point Location → Include Intersections

X-Values → Y-coordinate and then Plot.



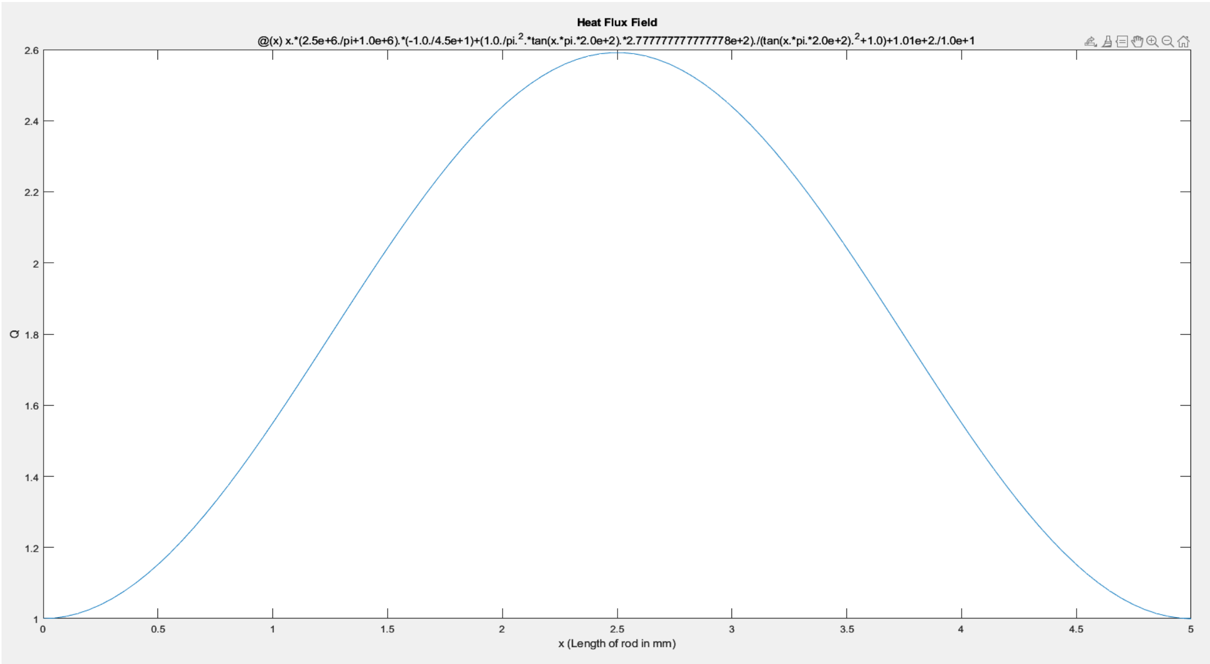


Task 4-

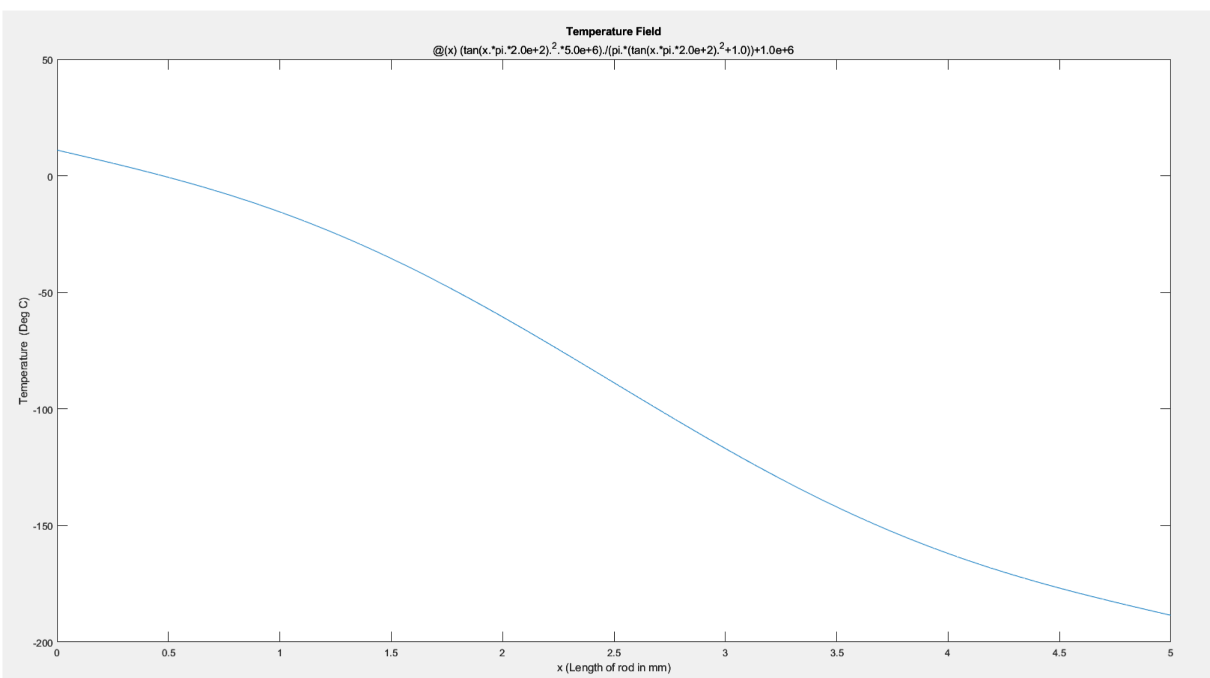
Analytical solution for the temperature and heat flux fields (show derivation). If using a symbolic math package include your code here and give the resulting solutions. *(Lead: Omkar & Harshal)*

```
%%%%%%%%%%%%%%%%%%%ANALYTICAL SOLUTION STARTS%%%%%%%%%%%%%%%%%%%%%%%%%%
%Created by Omkar Gaikwad ( ASU ID_1226843997)
%Collaboration with Harshal Tingre (ASU ID_1226039050)
%FEE_Project_1
%Date- March 12, 2023
syms x E1 E2
%Given Parameters
% Define all parameters (in consistent units)
% base units: mm
L = 5;              % Lenght of steel rod (mm)
R = 1;              % Radius of steel rod (mm)
A = (R^2)*pi;       % Cross sectional Area of Steel rod (mm^2)
k = 0.045;          % Thermal conductivity of steel (W/mm-deg C)
q = -1;             % Heat flux (W/mm^2) (Negative sign because flux is leaving the bar)
h = 1;              % Heat transfer coefficient (W/mm^2-k)
T0 = 10;            % Reference temperature at Left end of steel rod (deg C)
Q = q*A;            % (W)

% Steel rod is subjected to a volumetric heat load defined by
Omega = (0.4)*pi;    % (mm^-1)
S = A*sin(Omega*x);  % (W/mm^3)
% Integrating strong form (twice)
ALFA = int((int(-S/(A*k),x)+E1),x)+E2; %(T)
% Defining the boundary conditions.
BC_1 = subs(-A*k*diff(ALFA, x),x,L)==-Q;
BC_2 = subs(-k*diff(ALFA, x),x,0)==h*(subs(ALFA,x,0)-T0);
% Solving for constants of integration
solution = solve([BC_1,BC_2],[E1,E2]);
vpa(subs(subs(ALFA, solution),x,L));
NAN_xx = linspace(0,L,100);
ALFA_exact = matlabFunction(subs(ALFA,solution));
q_exact = matlabFunction(-k*subs(diff(ALFA,x),solution));

%Plotting graphs
%For Temperature Field
figure(1);
plot(NAN_xx, ALFA_exact(NAN_xx));
title('Temperature Field')
subtitle('@(x) (tan(x.*pi.*2.0e+2).^2.*5.0e+6)./(pi.*(tan(x.*pi.*2.0e+2).^2+1.0))+1.0e+6')
xlabel('x (Length of rod in mm)');
ylabel('Temperature  (Deg C)')

%For Heat Flux Field
figure(2);
plot(NAN_xx, q_exact(NAN_xx));
title('Heat Flux Field');
subtitle('@(x) x.*(2.5e+6./pi+1.0e+6).*(-1.0./4.5e+1)+(1.0./pi.^2.*tan(x.*pi.*2.0e+2).*2.777777777777778e+2)./(tan(x.*pi.*2.0e+2).^2+1.0)+1.01e+2./1.0e+1')
xlabel('x (Length of rod in mm)');
ylabel('Q');

%%%%%%%%%%%%%%%%%%%ANALYTICAL SOLUTION ENDS%%%%%%%%%%%%%%%%%%%%%%%%%%
```

$$q_{exact} = @(x)(pi.* 2.0 + 5.0)./(pi.* 2.0) - (5.0./2.0)./pi + (tan((x.* pi)./5.0).\hat{}2.* 5.0)./(pi.* (tan(* pi)./5.0).\hat{} + 1.0))$$



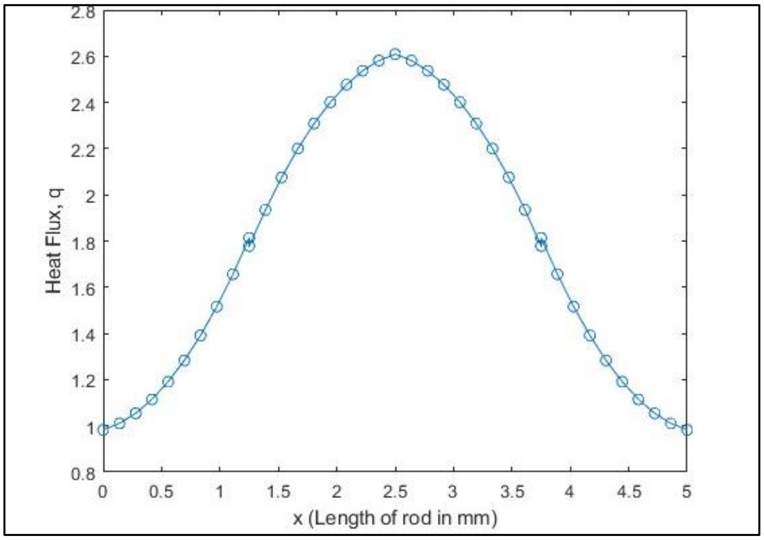
$$T_{exact} = @(x)(1.0./pi.\hat{}2.* tan((x.* pi)./5.0).* 2.777777777777778e + 2)./(tan((x.* pi)./5.0).\hat{}2 + 1.0) - (x.* (pi.* 2.0 + 5.0).* (1.0e + 2./9.0))./pi + 1.1e + 1$$



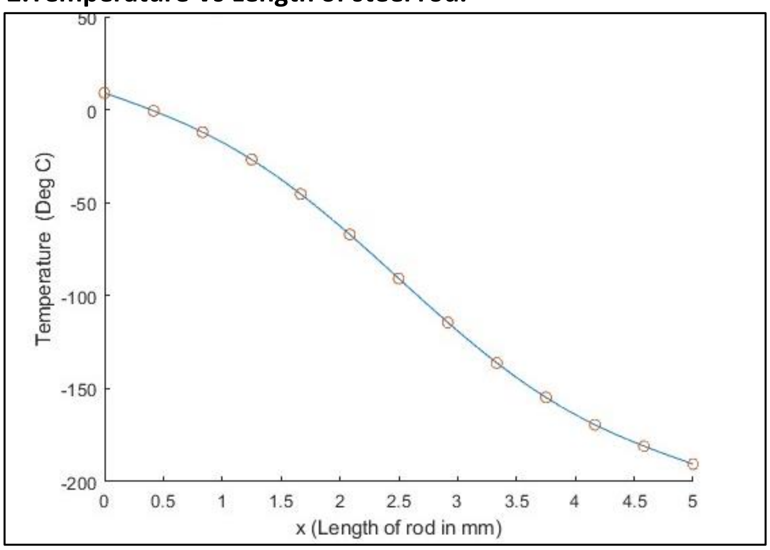
Task 5-

Show the temperature and heat flux fields comparing your analytical, ABAQUS and MATLAB solutions each on the same plot (using any order element). Use a fine enough mesh so that you can demonstrate whether all three approaches have the same solution. If one solution is different than another, explain why.

1.Heat Flux Vs Length of Steel rod.



2.Temperature Vs Length of steel rod.



Conclusion-

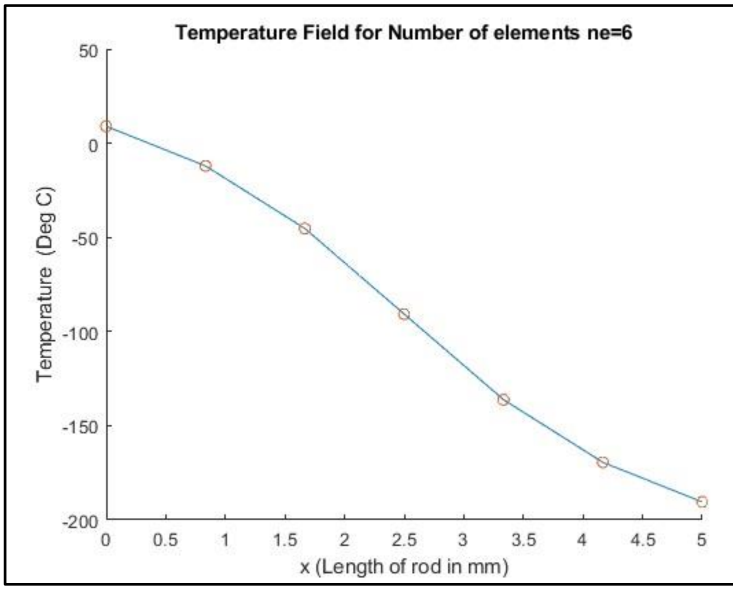
1.Above both Plots has same path and the same peak value for all 3 solutions your analytical, ABAQUS and MATLAB (Cubic solution with 1000 no of elements)

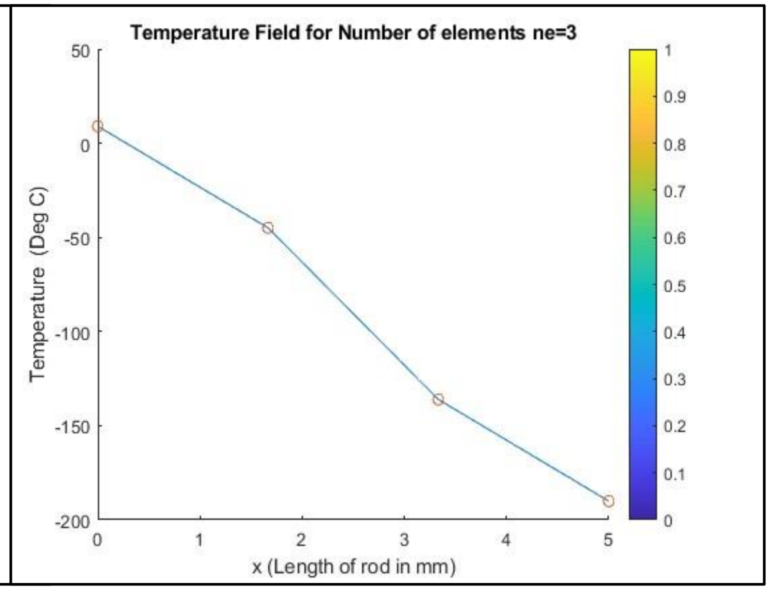
Task 6-

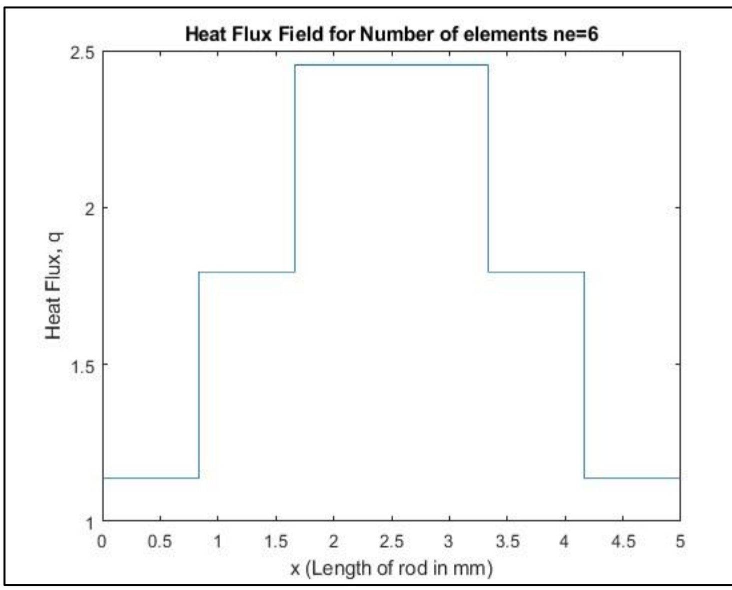
Plots of the temperature and heat flux fields solved using your MATLAB program for linear element solution with a mesh containing three and six elements. *(Lead - Harshal)*

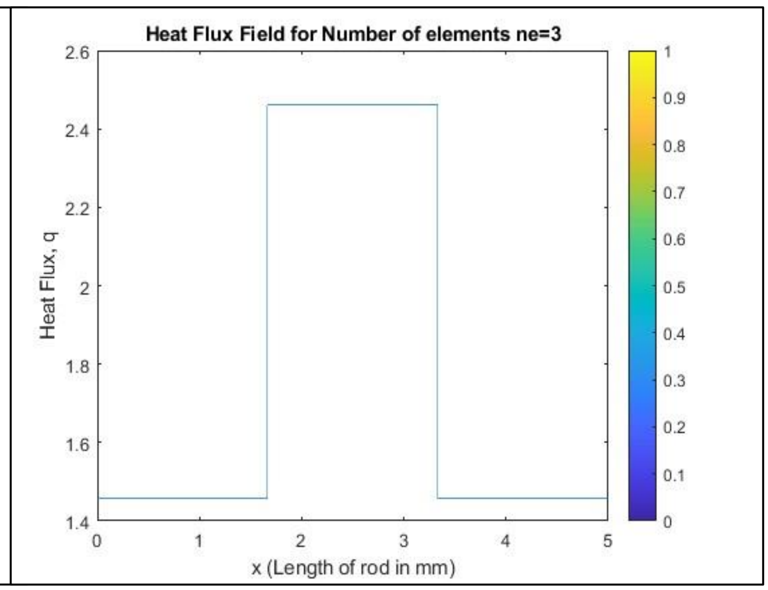
```
%%%%%%%%%%%%%%%%%%%%LINEAER ELEMENT SOLUTION STARTS%%%%%%%%%%%%%%%%%%%%%%%%%%
%Created by Omkar Gaikwad ( ASU ID_1226843997)
%Collaboration with Harshal Tingre (ASU ID_1226039050)
%FEE_Project_1
%Date- March 14, 2023
L = 5;                  % Lenght of steel rod (mm)
R = 1;                  % Radius of steel rod (mm)
A = (R^2)*pi;           % Cross sectional Area of Steel rod (mm^2)
k = 0.045;              % Thermal conductivity of steel (W/mm-deg C)
Omega = (0.4)*pi;       % Steel rod is subjected to a volumetric heat load defined by
h = 1;                  % Heat transfer coefficient (W/mm^2-k)
T0 = 10;                % Reference temperature at Left end of steel rod (deg C)
%We Can chnage ne = 3. For another result.
ne = 6;                    % number of elements.
nn = ne + 1;               % Number of nodes.
nne = 2;                   % Number of nodes per element.
X_Matrix = zeros(nn);
Y_Matrix = zeros(nn,1);
mesh.x = linspace(0, 5, nn);
mesh.conn = [1:nn-1; 2:nn];
Q1= -sqrt(1/3);
Q2= sqrt(1/3);
Quad_pts = [[Q1 Q2];[1 1]];
for c = mesh.conn
    xe = mesh.x(:,c);
    Ke = zeros(length(c));
    for q = Quad_pts
        [N, dNdp] = shape2(q(1));
        x = xe * N;
        j = xe * dNdp;
        s = dNdp / j;
        V=s*k*A*s'*j*q(2);
        Ke = Ke+V;
        Y_Matrix(c) = Y_Matrix(c) + N * A * j * sin(Omega*x) * q(2);
    end
    X_Matrix(c, c) = X_Matrix(c, c) + Ke;
end
Y_Matrix(1) = Y_Matrix(1) + h * A * T0;
Y_Matrix(nn) = Y_Matrix(nn) + A * -1;
X_Matrix(1,1) = X_Matrix(1,1) + A * h;
T = X_Matrix\Y_Matrix;
[xx, qq, TT] = findxq(mesh, k, T);
%Plotting Results for Number of elements ne=3
figure();
hold on
plot(xx, TT)
plot(mesh.x, T, 'o')
title('Temperature Field')
xlabel('x (Length of rod in mm)')
ylabel('Temperature  (Deg C)')
```

```
figure();
plot(xx, qq)
title('Heat Flux Field')
xlabel('x (Length of rod in mm)')
ylabel('Heat Flux, q')

format long

function [N, dNdp] = shape2(p)
         N1 = (1-p)/2;
         N2 = (1+p)/2;
         N = [N1; N2];
         dNdp = [-0.5; 0.5];
end

function[xx, qq, TT] = findxq(mesh, k, T)
         xx = [];
         TT = [];
         qq = [];
for c = mesh.conn
    xe = mesh.x(:,c);
    Te = T(c)';
    for p = linspace(-1,1,10)
        [N, dNdp]=shape2(p);
        Je = xe* dNdp;
        xx(end+1) = xe*N;
        TT(end+1) = Te*N;
        qq(end+1) = -k*Te*(dNdp/Je);
    end
end
end
%%%%%%%%%%%%%%%%%%%%%lINEAER ELEMENT SOLUTION ENDS%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
```









Conclusion-

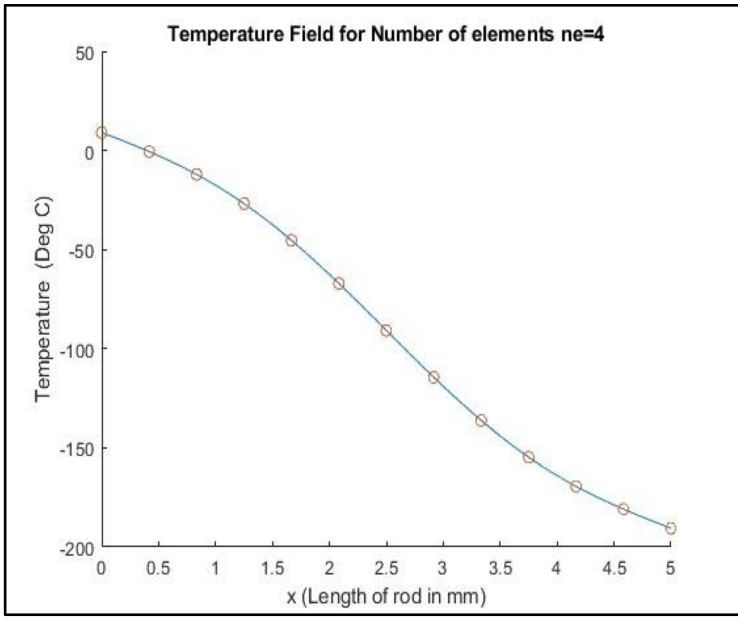
- As the number of elements increase, the temperature field becomes smoother and there is a slight decrease in temperature by a factor of 0.5.
- For the heat flux graph, the number of steps increases with an increase in the number of elements.
- At ne = 3, there is an increase in heat flux at x=5/3 and a decrease in heat flux to its original values at x=10/3.
- At ne = 6, there are increases in heat flux at x=5/6 and x=5/3, while decreases are noted at x=10/3 and x=25/6, going back to its original values at x=5 for both cases.
- There is a noticeable decrease in the heat flux at the beginning and the end of the steel rod from q=1.457 at ne = 3 to q=1.136 at ne = 6.

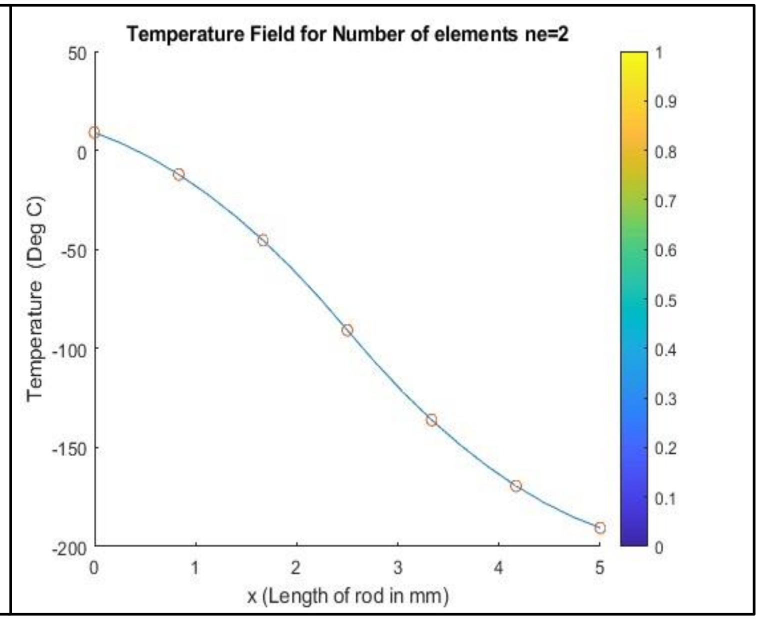
Task 3-

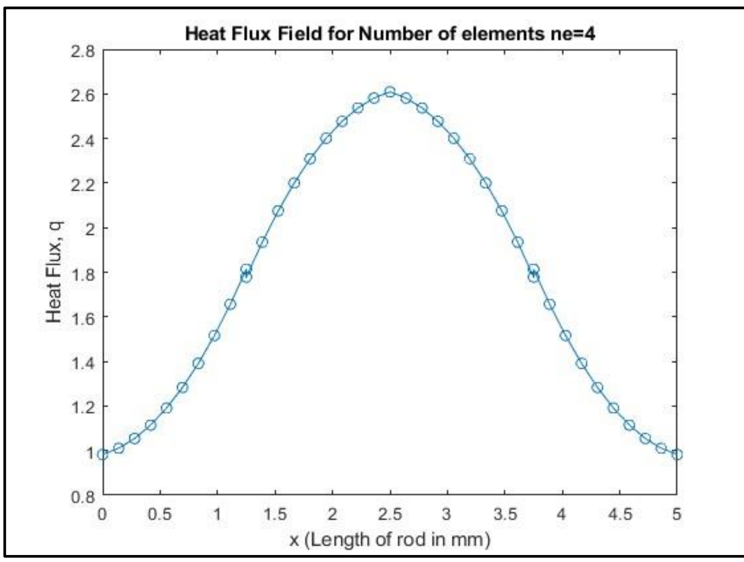
Plots of the temperature and heat flux fields solved using your MATLAB program for cubic element solution with a mesh containing two and four elements. *(Lead: Omkar & Harshal)*

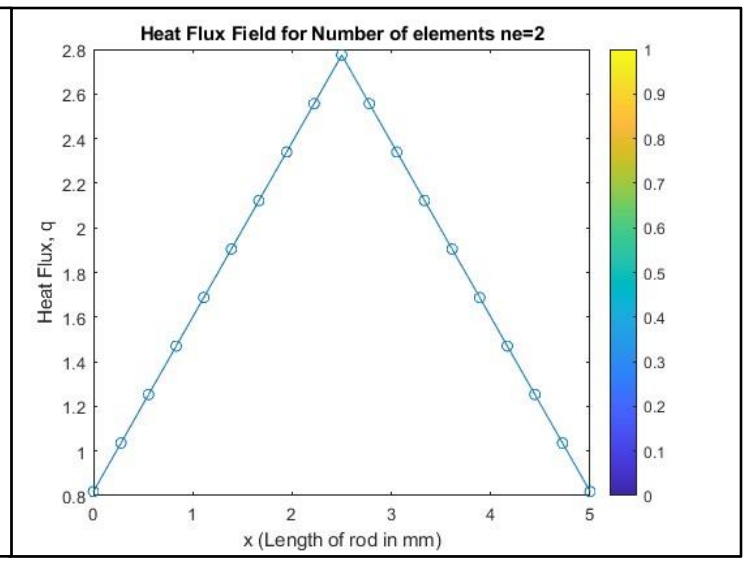
```
%%%%%%%%%%%%%%%%%%%CUBIC SOLUTION STARTS%%%%%%%%%%%%%%%%%%%%%%%%%%%
%Created by Omkar Gaikwad ( ASU ID_1226843997)
%Collaboration with Harshal Tingre (ASU ID_1226039050)
%FEE_Project_1
%Date- March 14, 2023
L = 5;               % Lenght of steel rod (mm)
R = 1;               % Radius of steel rod (mm)
A = (R^2)*pi;        % Cross sectional Area of Steel rod (mm^2)
k = 0.045;           % Thermal conductivity of steel (W/mm-deg C)
Omega = (0.4)*pi;    % Steel rod is subjected to a volumetric heat load defined by
h = 1;               % Heat transfer coefficient (W/mm^2-k)
T0 = 10;             % Reference temperature at Left end of steel rod (deg C)
%We Can chnage ne = 2. For another result.
ne = 2;             % number of elements.
nn = 3*ne + 1;      % Number of nodes.
nne = 4;            % Number of nodes per element.
X_Matrix = zeros(nn);
Y_Matrix = zeros(nn,1);
mesh.x = linspace(0, 5, nn);
M_1=(1:3:nn-3);
M_2=(2:3:nn-2);
M_3=(3:3:nn-1);
M_4=(4:3:nn);
mesh.conn = [M_1;M_2;M_3;M_4];
Q1= -sqrt(3/5);
Q2= sqrt(3/5);
W1=5/9;
W2=8/9;
Quad_pts = [[Q1 0 Q2]; [ W1 W2 W1]];
for c = mesh.conn
    xe = mesh.x(:,c);
    Ke = zeros(length(c));
    for q = Quad_pts
            [N, dNdp] = shape4(q(1));
            x = xe*N;
            j = xe*dNdp;
            s = dNdp/j;
            V =s*k*A*s'*j*q(2);
            Ke = Ke+V;
            DOD=N*A*j*sin(Omega*x)*q(2);
            F(c) = F(c)+DOD;
    end
    K(c,c) = K(c,c)+Ke;
end
F(1) = F(1) + h * A * T0;
F(nn) = F(nn) + A * -1;
K(1,1) = K(1,1) + A * h;
T = K\F;
[xx, qq, TT] = plotxq(mesh, k, T);
```

```
%Plotting Results
%Heat Flux Field
figure();
plot(xx, qq, '-o')
title('Heat Flux Field for Number of elements ne=2')
xlabel('x (Length of rod in mm)')
ylabel('Heat Flux, q')
%Temperature Field
figure();
hold on
plot(xx, TT)
plot(mesh.x, T, 'o')
title('Temperature Field for Number of elements ne=2')
xlabel('x (Length of rod in mm)')
ylabel('Temperature  (Deg C)')
format long
function [N, dNdp] = shape4(p)
                N1 = (-9/16)*(p + 1/3)*(p - 1/3)*(p - 1);
                N2 = (9/16)*(p + 1/3)*(p - 1/3)*(p + 1);
                N  = [N1;(27/16)*(p + 1)*(p - 1/3)*(p - 1); (-27/16)*(p + 1)*(p + 1/3)*(p - 1);N2];
               dNdp = [-((9*p)/16+3/16)*(p-1)-((9*p)/16+3/16)*(p-1/3)-(9*(p-1)*(p-1/3))/16;
                      ((27*p)/16+27/16)*(p-1)+((27*p)/16+27/16)*(p-1/3)+(27*(p-1)*(p-1/3))/16;
                      -((27*p)/16+27/16)*(p-1)-((27*p)/16+27/16)*(p+1/3)-(27*(p-1)*(p+1/3))/16;
                      ((9*p)/16+3/16)*(p+1)+((9*p)/16+3/16)*(p-1/3)+(9*(p+1)*(p-1/3))/16;];
end
function[xx, qq, TT] = plotxq(mesh, k, T)
xx = [];
TT = [];
qq = [];
    for c = mesh.conn
    xe = mesh.x(:,c);
    Te = T(c)';
        for p = linspace(-1, 1, 10)
        [N, dNdp] = shape4(p);
        ee = xe * dNdp;
        xx(end+1) = xe*N;
        TT(end+1) = Te*N;
        qq(end+1) = -k*Te*(dNdp/ee);
        end
    end
end
%%%%%%%%%%%%%%%%%%%CUBIC SOLUTION ENDS%%%%%%%%%%%%%%%%%%%%%%%%%%%
```


Temperature Field for Number of elements ne=4
Temperature (Deg C)
x (Length of rod in mm)


Temperature Field for Number of elements ne=2
Temperature (Deg C)
x (Length of rod in mm)

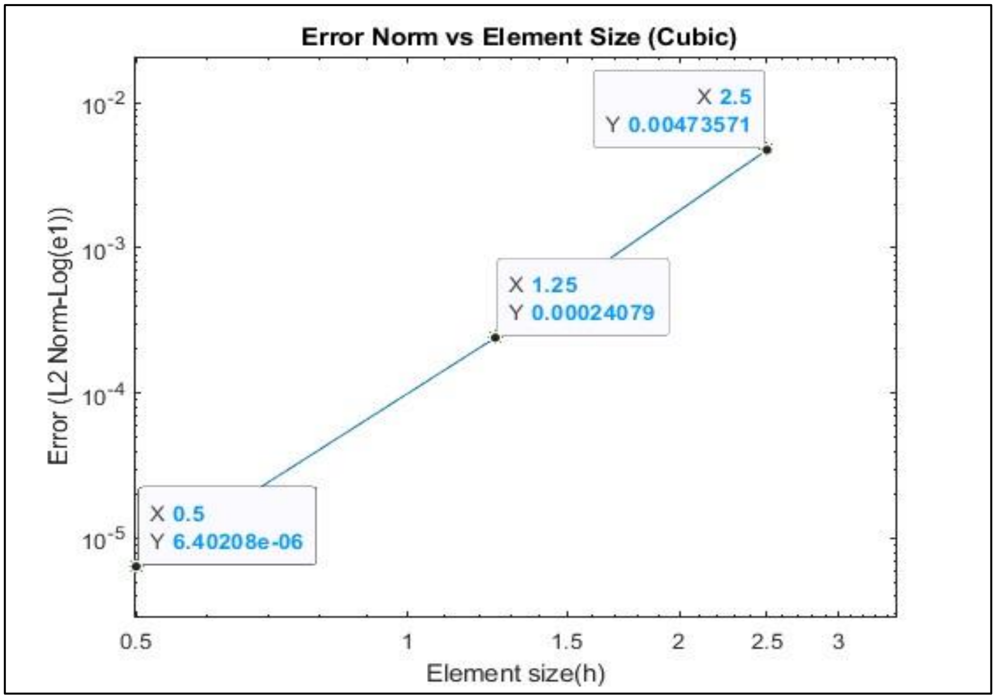

Heat Flux Field for Number of elements ne=4
Heat Flux, q
x (Length of rod in mm)

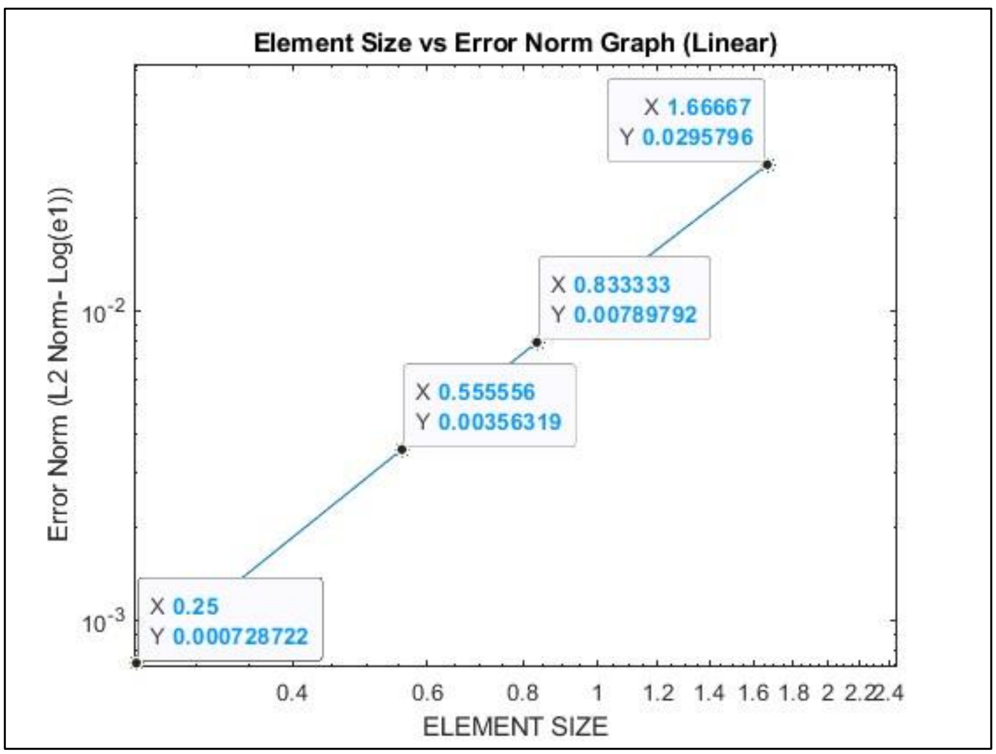

Heat Flux Field for Number of elements ne=2
Heat Flux, q
x (Length of rod in mm)

Task 8-

Plot of L2 error norm vs element size on a log-log plot and computed rate of convergence while using linear and cubic order elements. Use markers rather than lines in your plot so it is clear how many solutions you are using when computing the convergence rate. Explain whether the optimal convergence rates are achieved.

***(Lead: Omkar & Harshal*)**





The cubic function converges almost immediately while the linear function is much slower to converge on the actual solution.