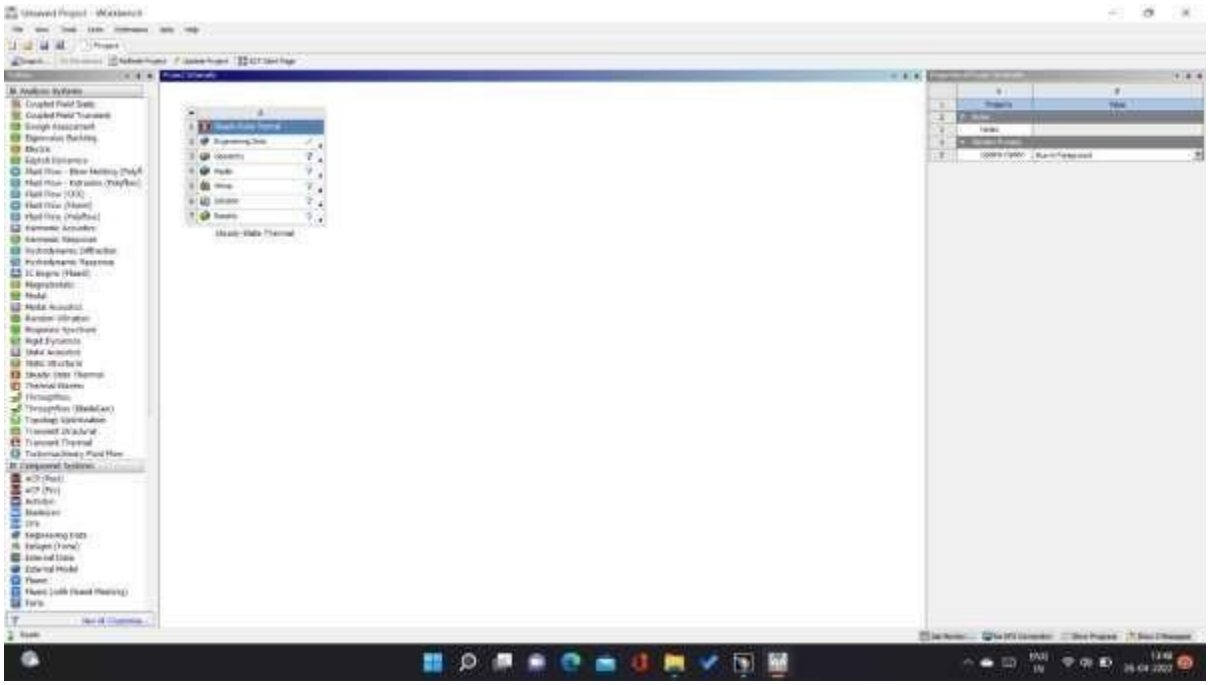
Name of Student:- Aniket Patil	**Class:TE Mech 2**
Semester/Year:- 6th SEM/3th YEAR	**Roll No:- 29**
Date of Performance:-	**Date of Submission:-**
Examined By:- Prof. B.R Pujari	**Experinment No:- 5**

Title: Thermal Analysis – Static/Transient Analysis.

FortherectangularcrossͲsessionoftheelementB =1mandH=0.5mandLength ofElement is 5m.Assign thermal Properties, Thermal Conductivity = 60.5 W/m∘C; Apply a constant temperature of 100∘C to the left side of the block (face 1) and a constant temperature of 300∘C to the right side of the block (face 3). All other faces are insulated by default. Find temperature distribution in the element.

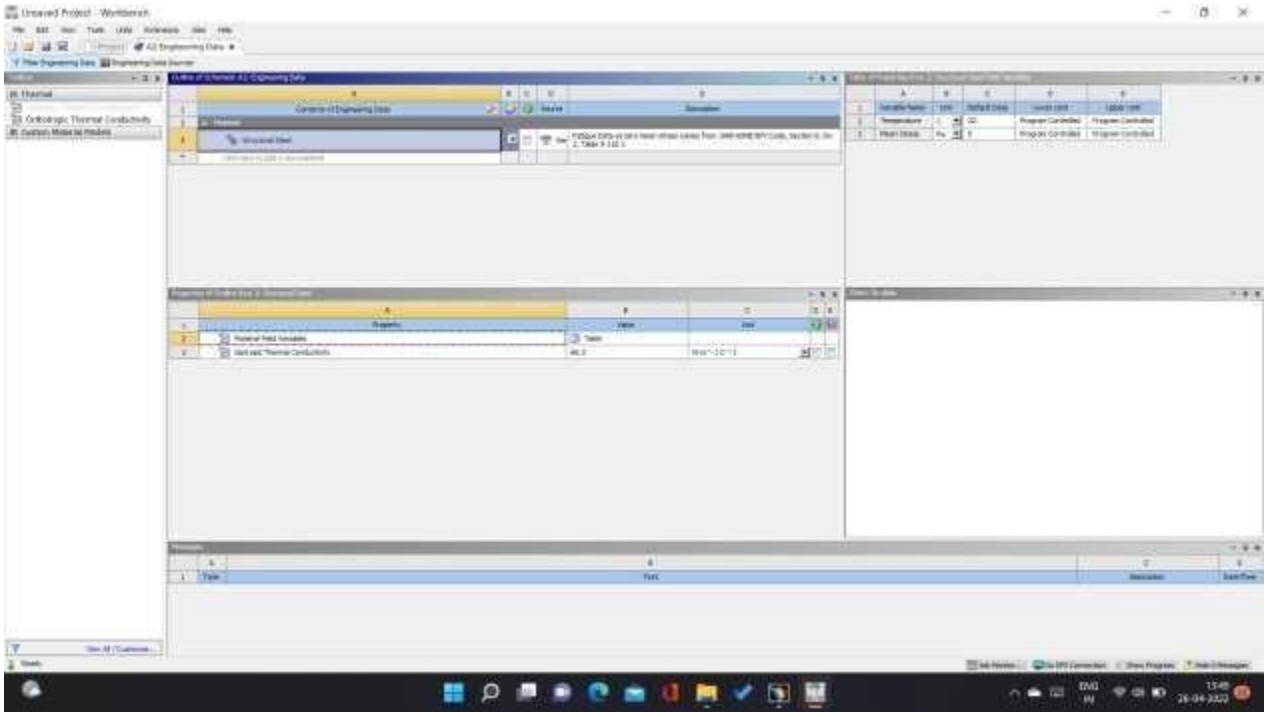
STEP 1:- Click on Start – ANSYS Workbench.

On left side double click on Steady state thermal.



STEP 2:-

Engineering Data: Double click on engineering data to select suitable material from library or add material and its material properties. Default material selected is structural steel. For thermal analysis, check or add isotropic thermal conductivity.

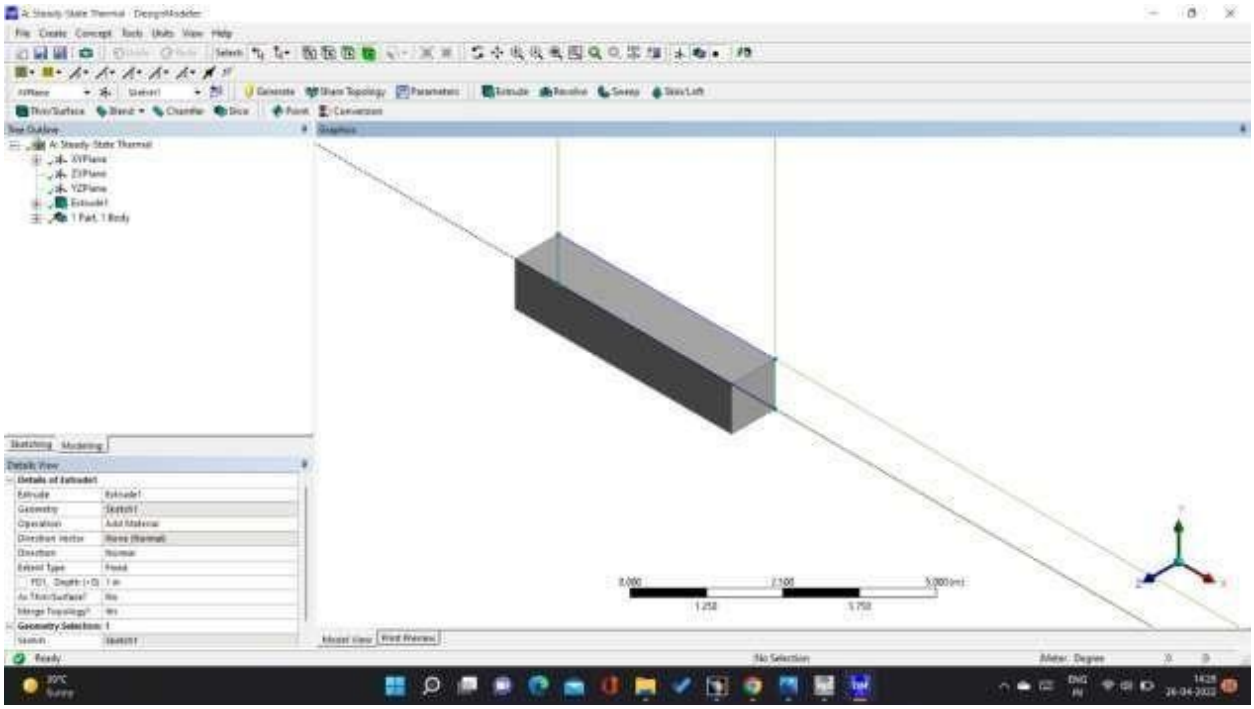


Click on project on upper left corner to go back.

STEP 3:-

Geometry: Doble click on geometry to prepare model as per given data

1. Select XY Plane – right click – select look at
2. Click on sketching on left middle

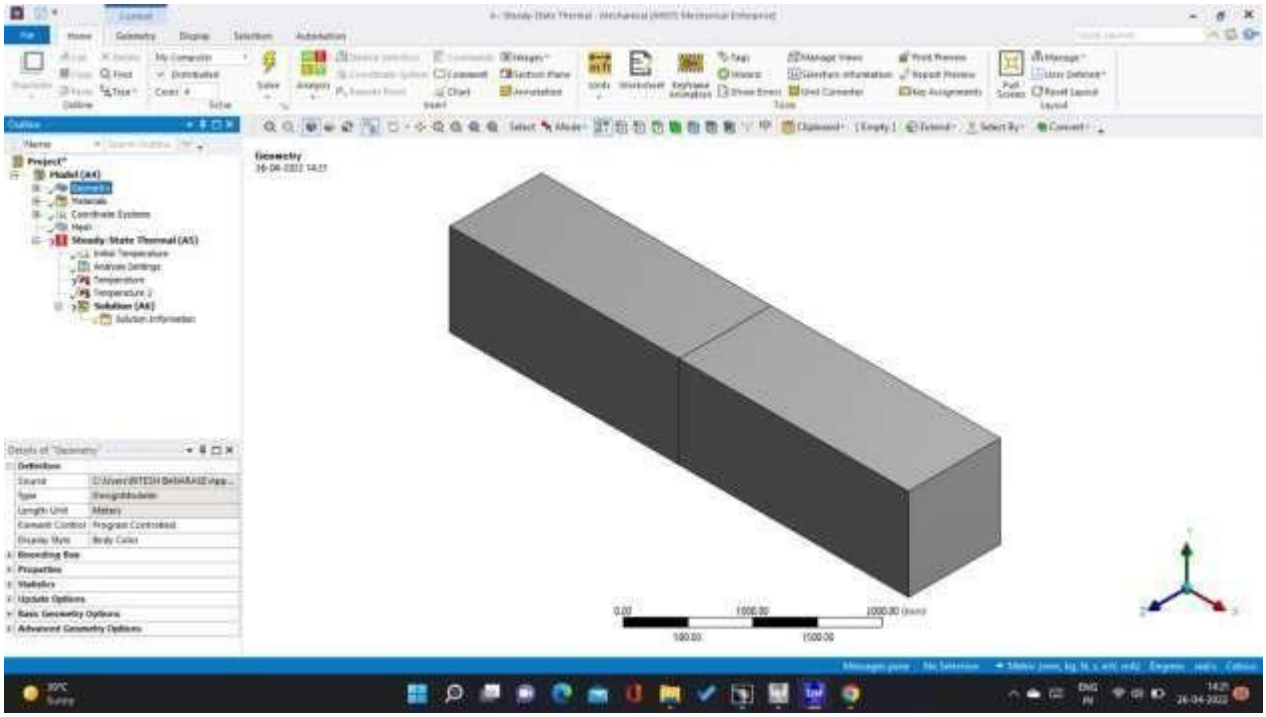


3. Click on Draw – line – draw in workspace of ansys
4. Click on dimensions – general – click on sketched line – give dimension 5 m
5. Click on concept on menu bar – lines from sketch – select sketch – apply – generate
6. Click on concept – cross-section – select rectangular corss-section – give dimensions
7. Click on part – line body – select cross section Rect1 – generate
8. Click on view in menubar – cross-section solids
9. Close design modeler environment

STEP 4:-

Model (Meshing): Double click on model to open analysis environment.

1. Right click on mesh – Generate mesh

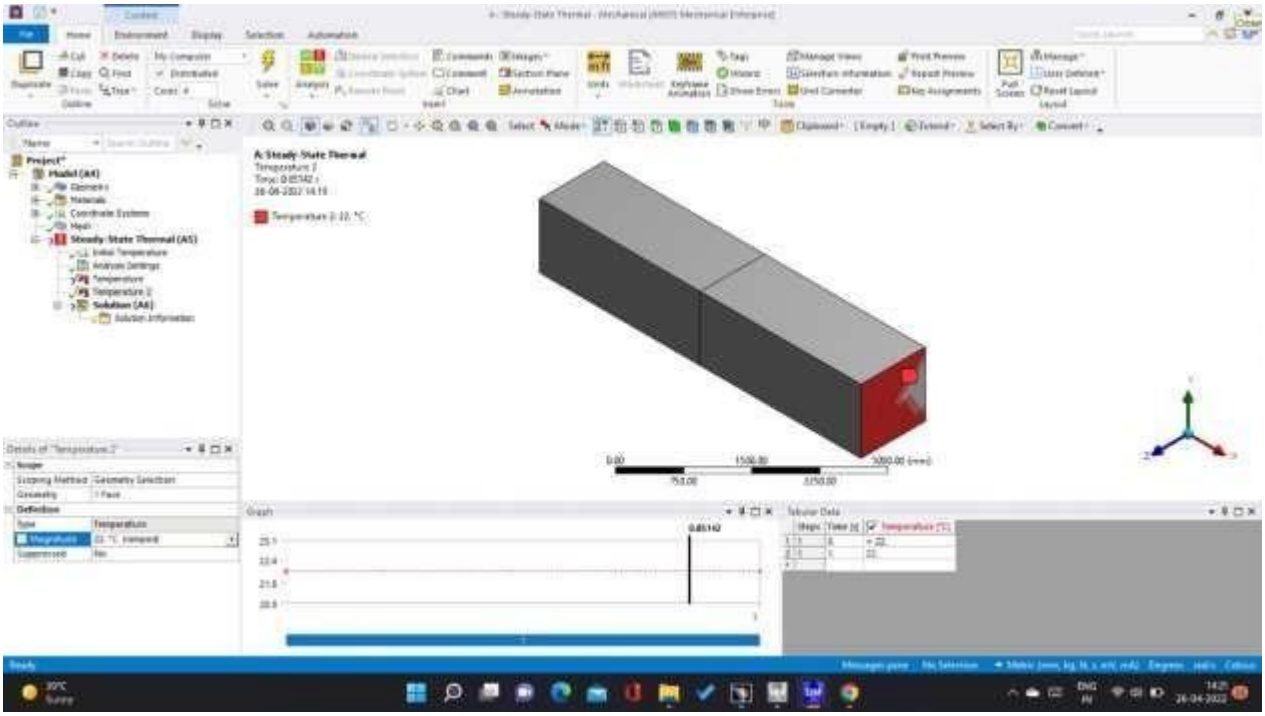


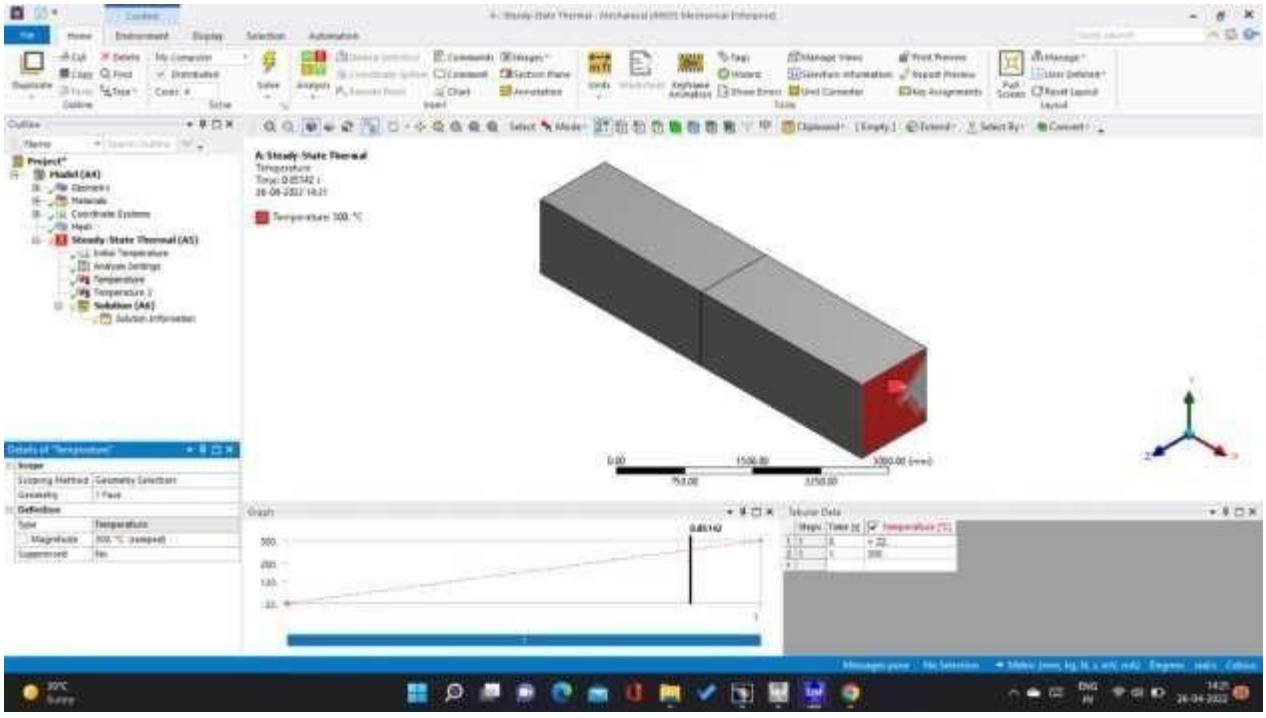
2. Check number of nodes and element in left bottom corner. This is 1d bar element with rectangular cross section have 1 element only.

STEP 4:-

Steady state thermal: To apply boundary condition.

1. Select edge in upper toolbox
2. Click on steady state thermal – select left edge – click on temperature –300^{0}C.
3. Click on steady state thermal – select left edge – click on temperature –100^{0}C.
4. Boundary condition complete





STEP 5:-

Solution:

1. Click on solution

2. On upper side select – Temperature
3. Thermal analysis completed