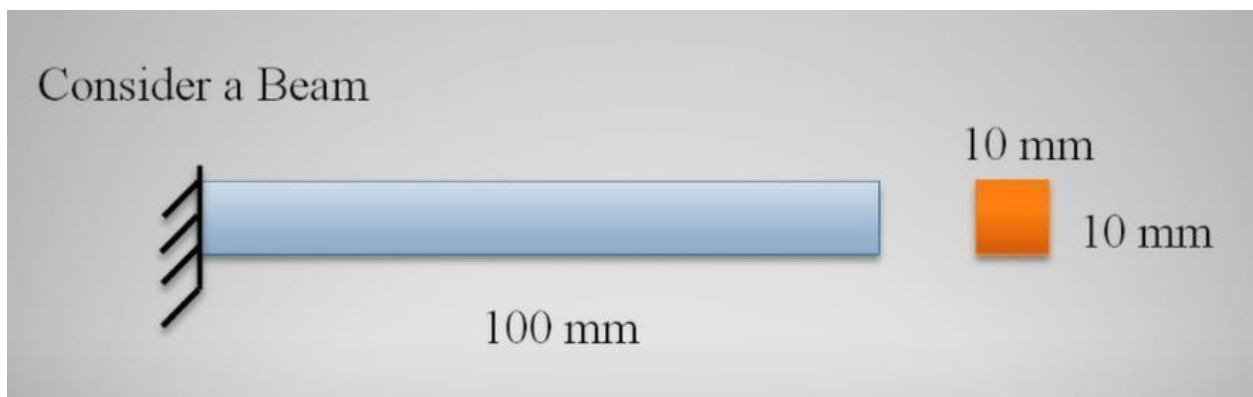


Name of Student: Aniket Patil	Class: TE MECH 2
Semester/ Year: 6 <sup>th</sup> / 3 <sup>rd</sup>	Roll No: 29
Date of performance:	Date of Submission
Examined by: Prof. B.R Pujari	Expt No: 7

**Title:** Modal analysis – Cantilever beam. Aim: To determine natural frequency of cantilever beam using FEA package.

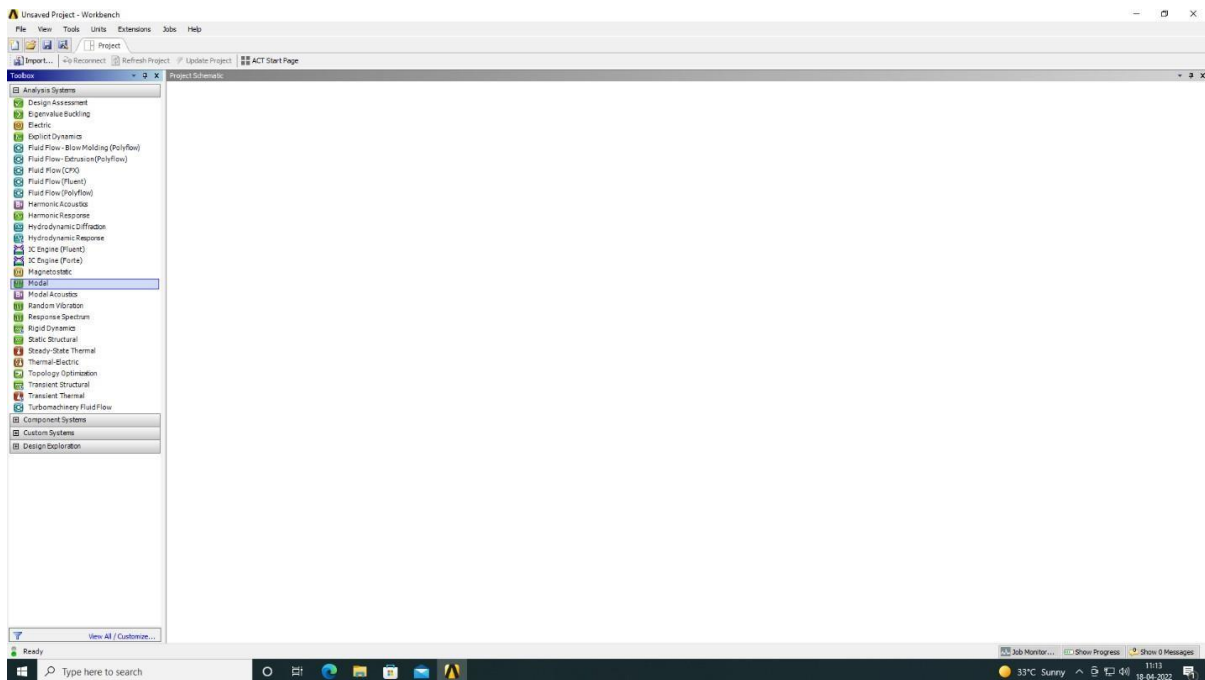
**FEA Package Used:** Ansys 2022 Workbench.

**Problem Statement:** A cantilever beam is shown in the figure. Length of the beam is 150 mm. Young's modulus is 200GPa. Poisson's ratio is 0.3. Find first five natural frequencies for beam. The cross-section area of the beam is rectangle of 10 × 10 mm.

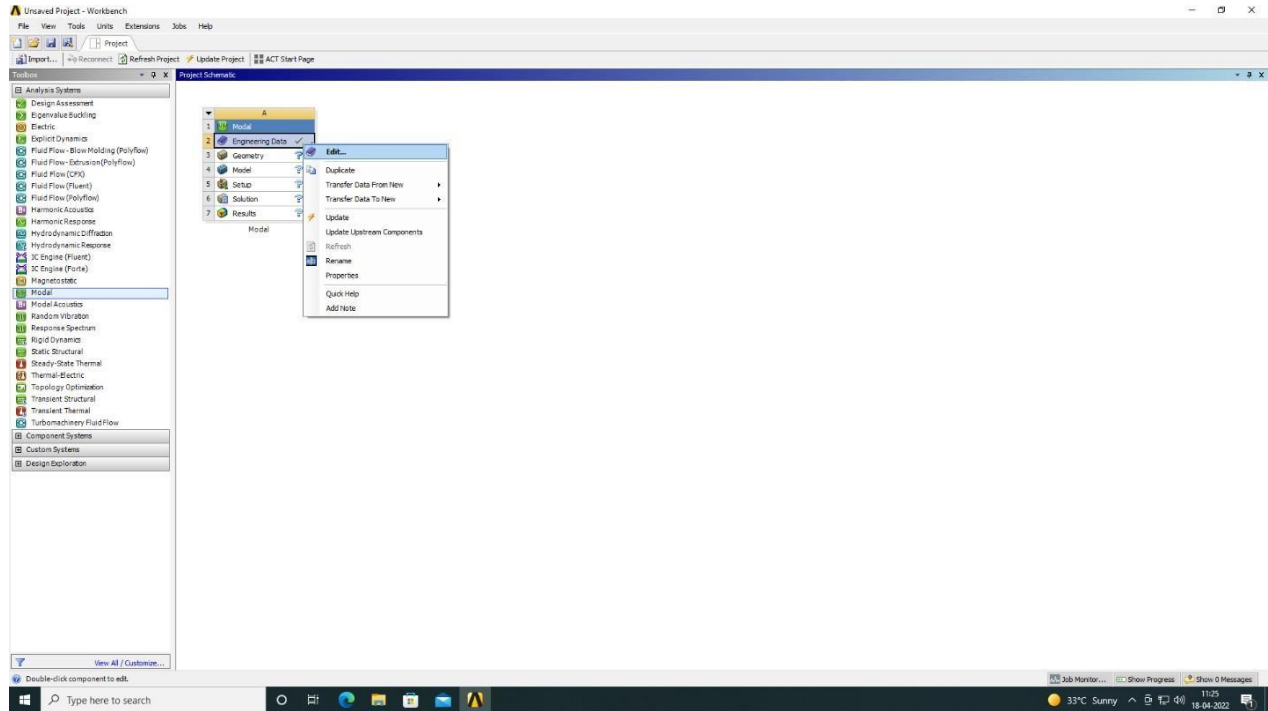


### Software Solution:

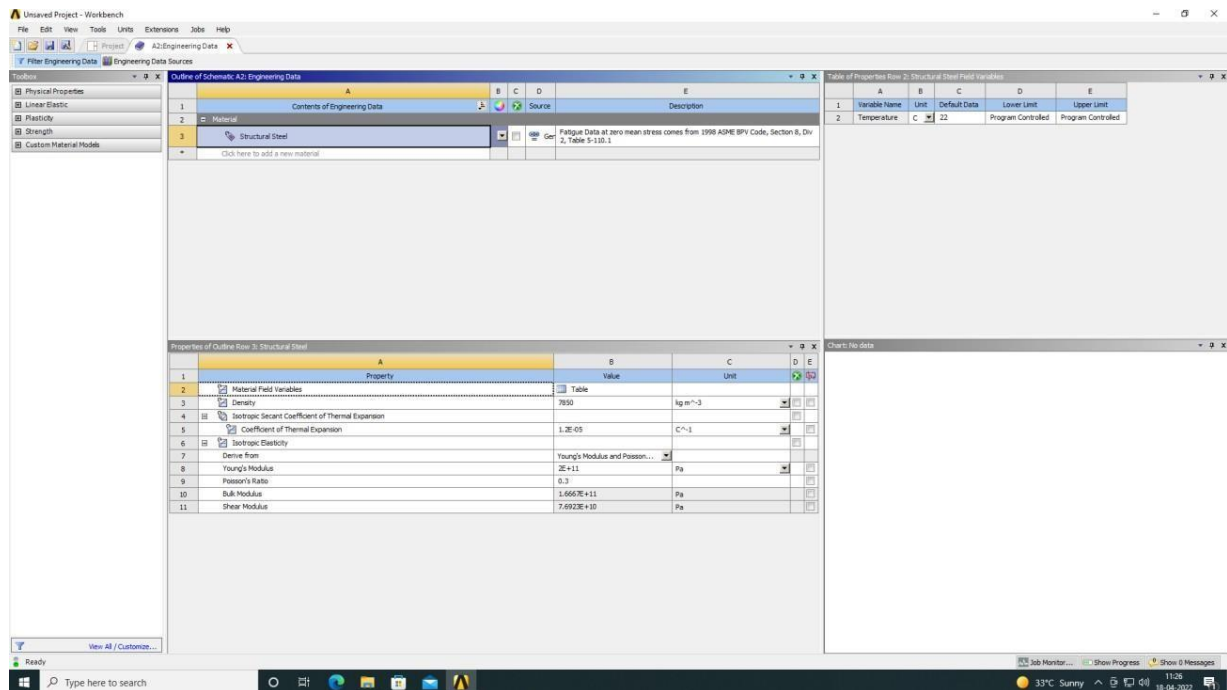
Step 1: Open Ansys 2022 Workbench > Analysis Systems > Modal > Ok.



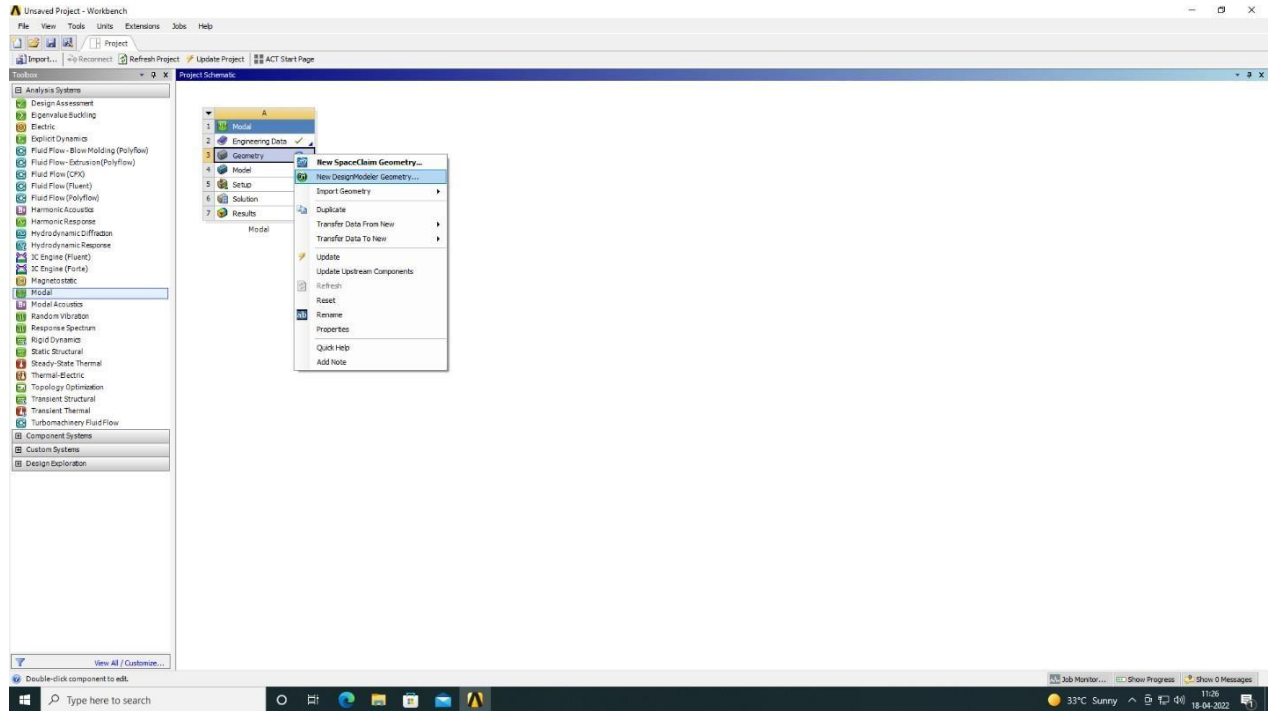
## Step 2: Engineering data > Edit > Ok



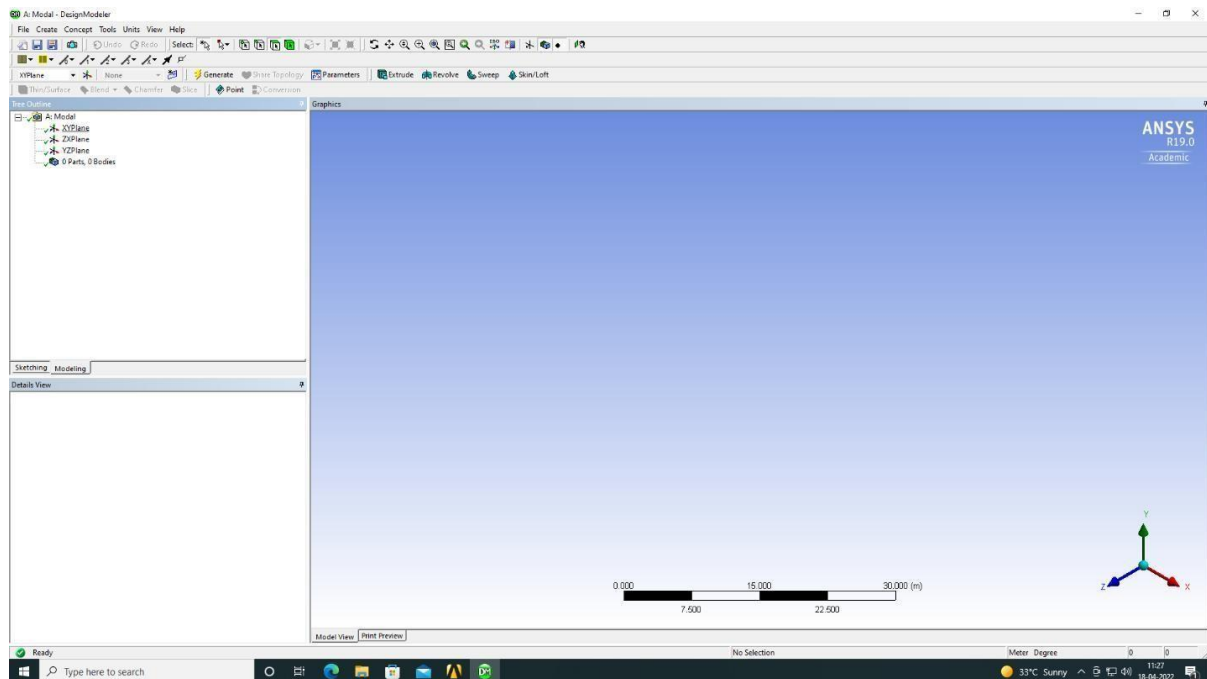
## Step 3: Check data > Close



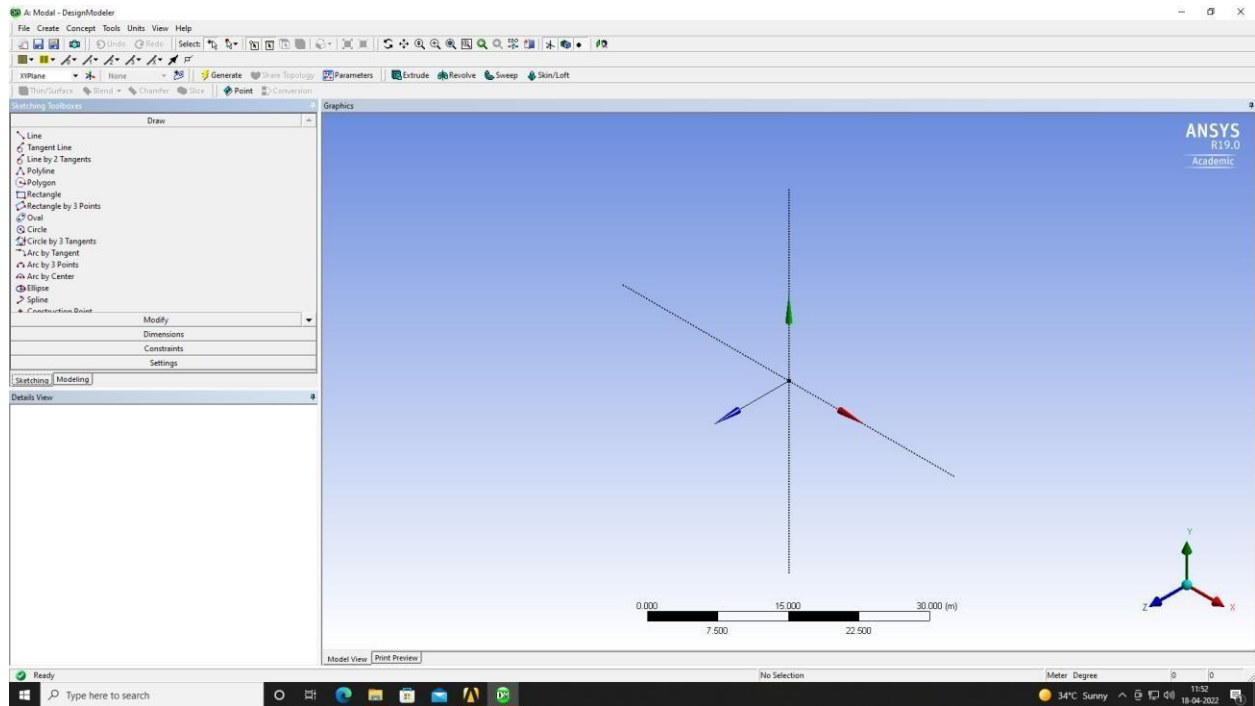
## Step 4: Geometry > New Designmodeler Geometry > Ok.



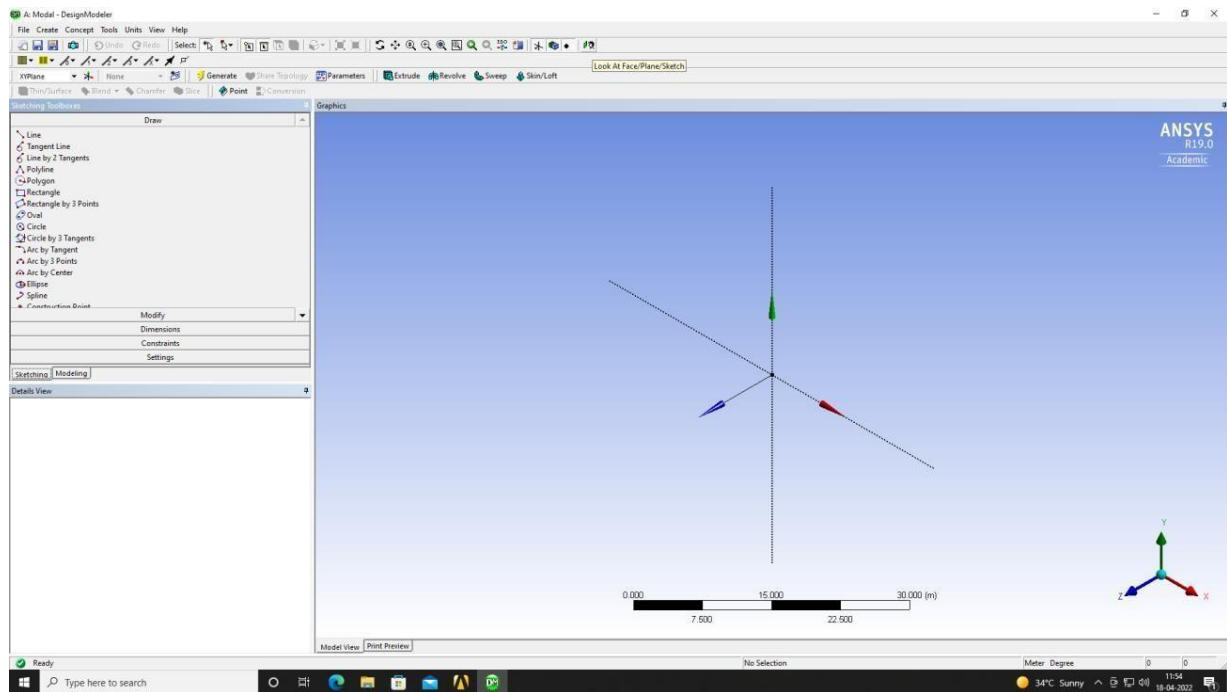
## Step 5: Select XY Plane.



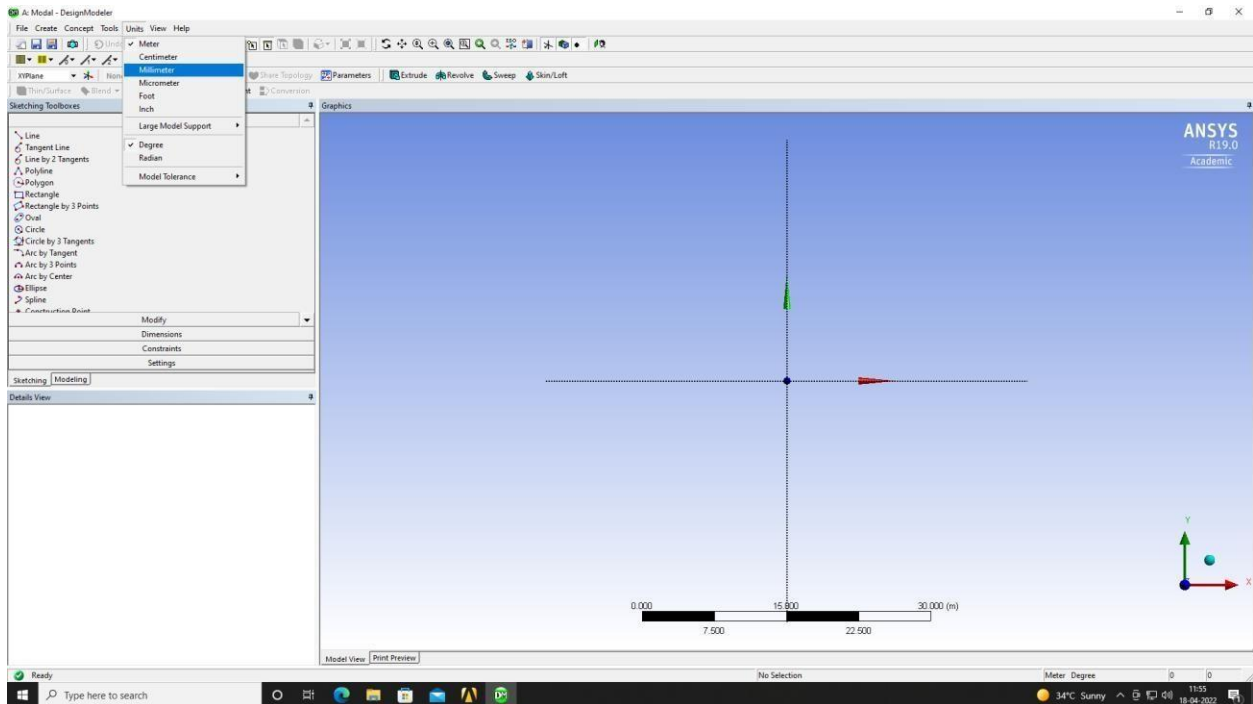
## Step 6: Select Sketching



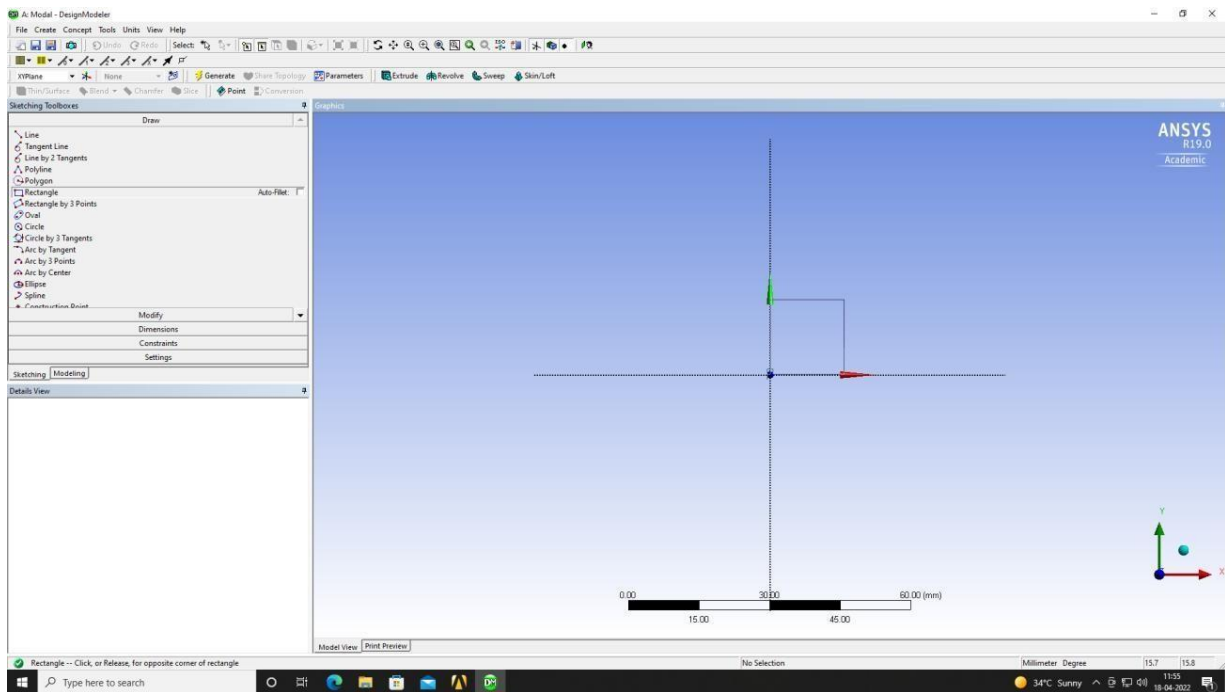
## Step 7: Select Look At Face.



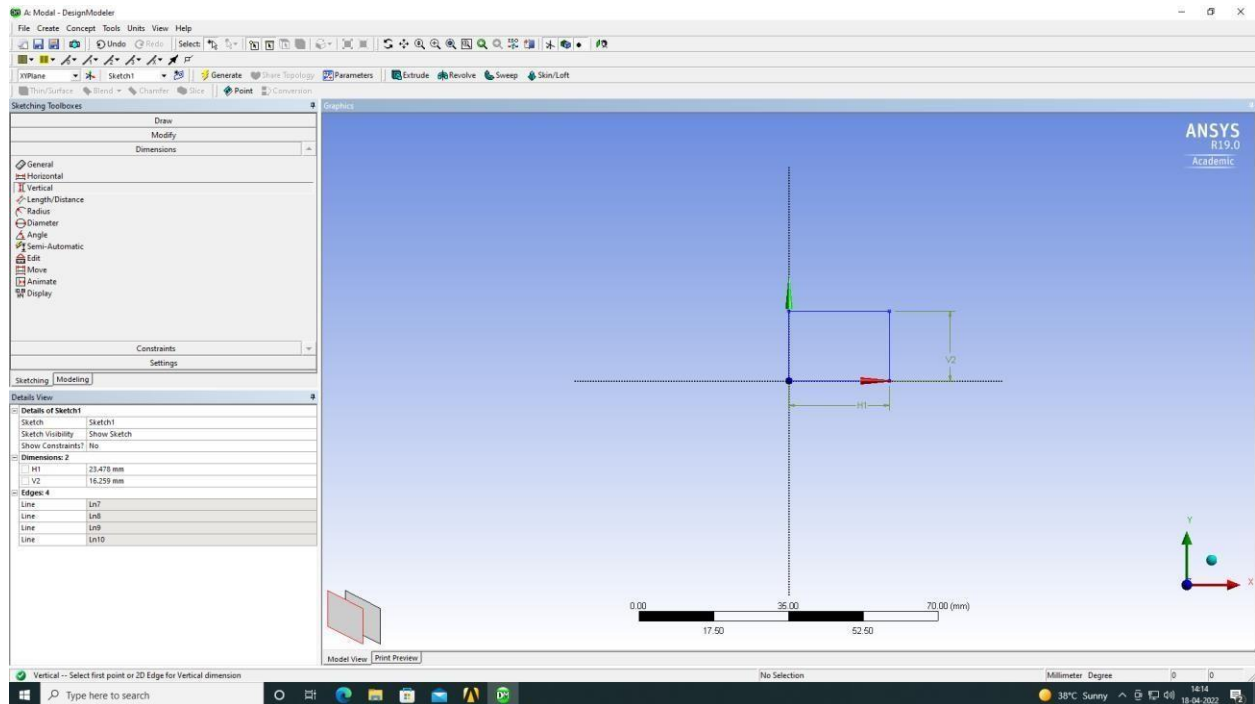
Step 8: Change unit to millimeter.



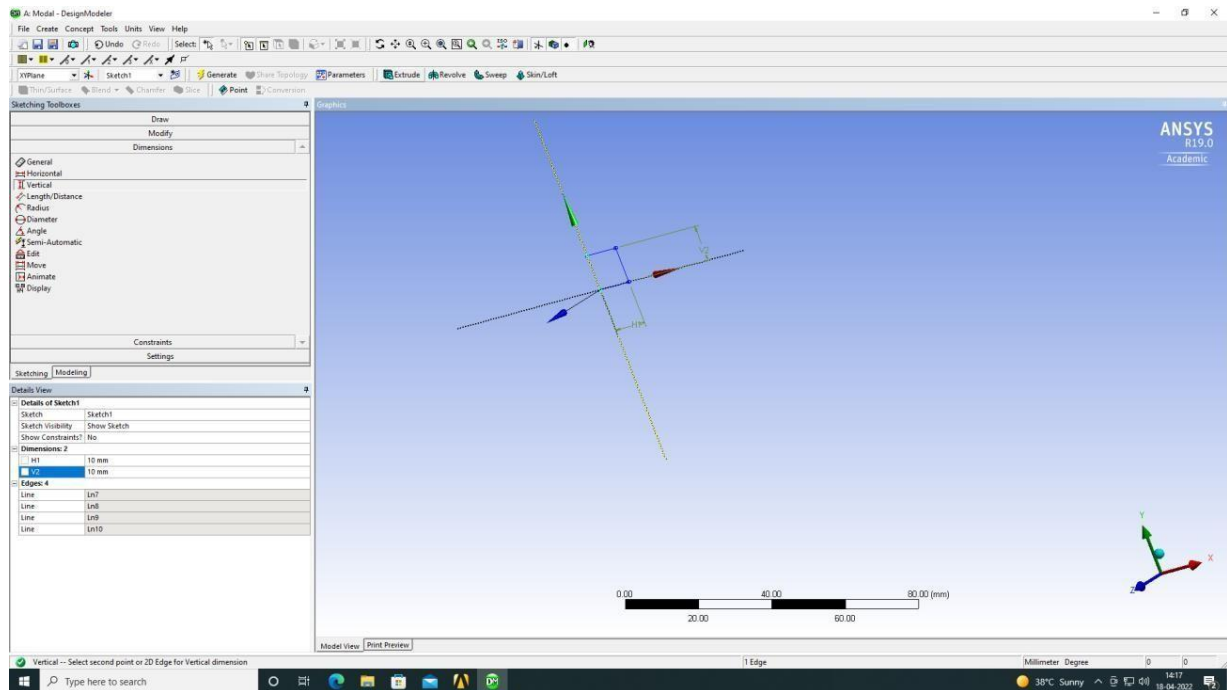
Step 9: Select and draw rectangle from origin.



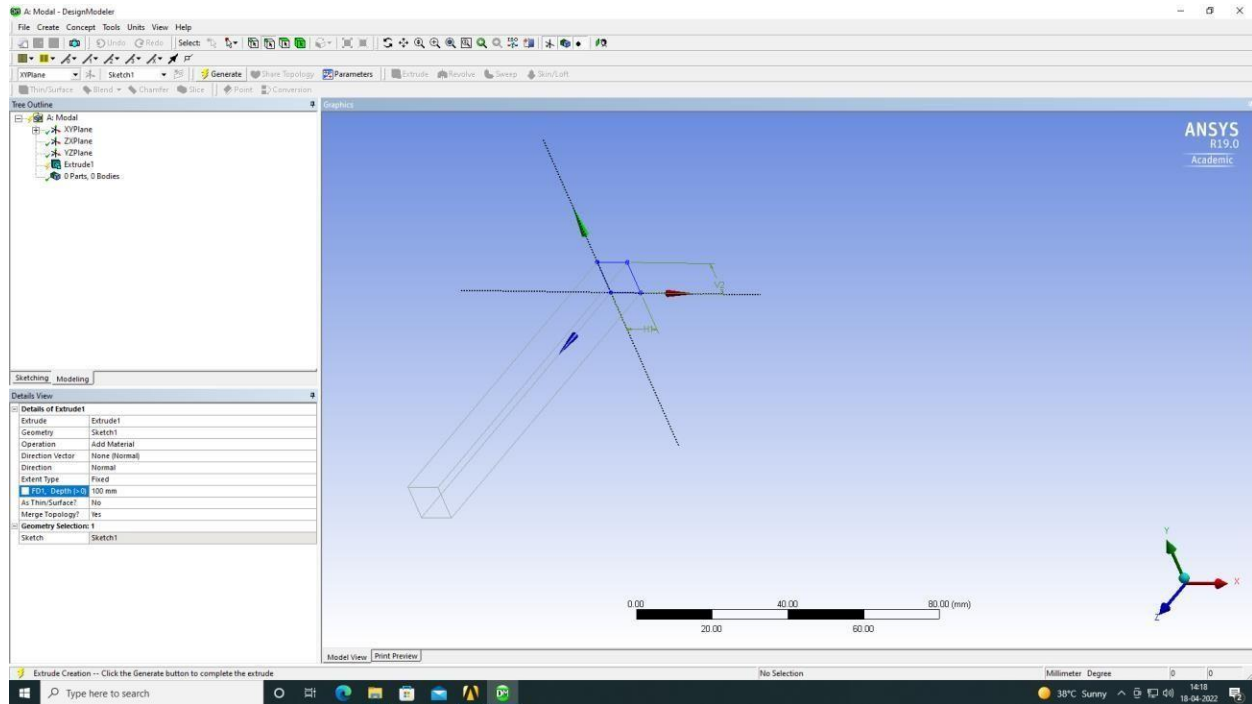
Step 10: Select dimension > Allot Vertical and Horizontal dimensions to rectangle.



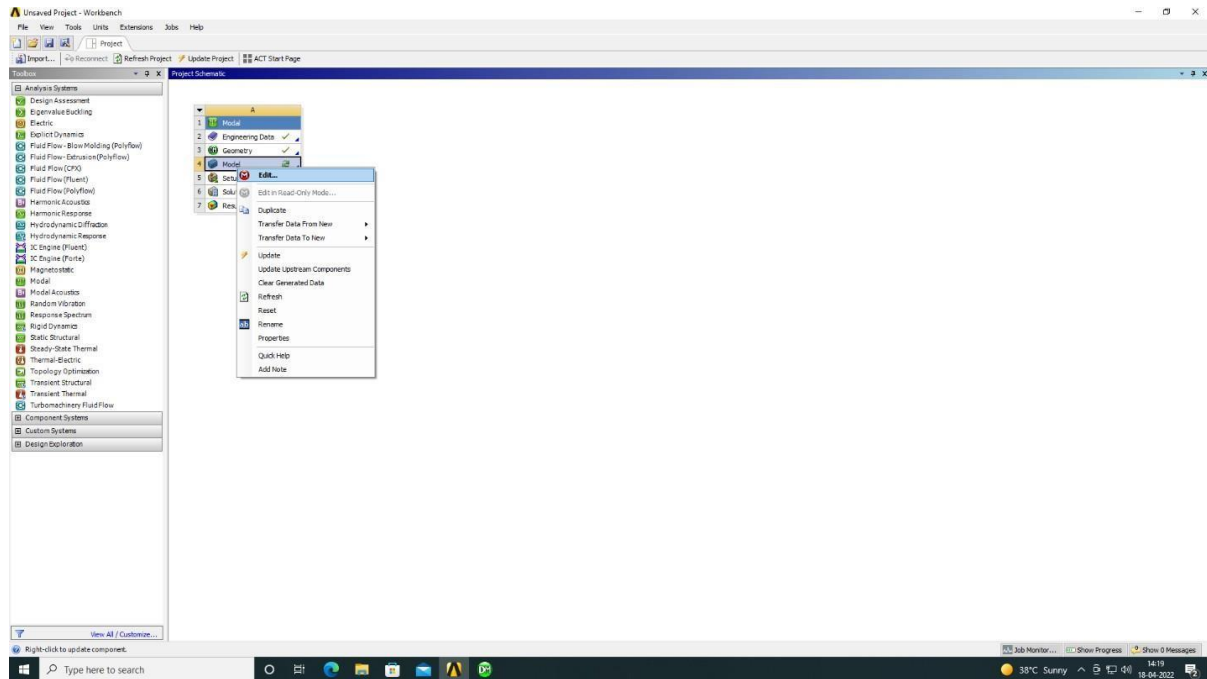
Step 11: Change dimensions from detail panel.



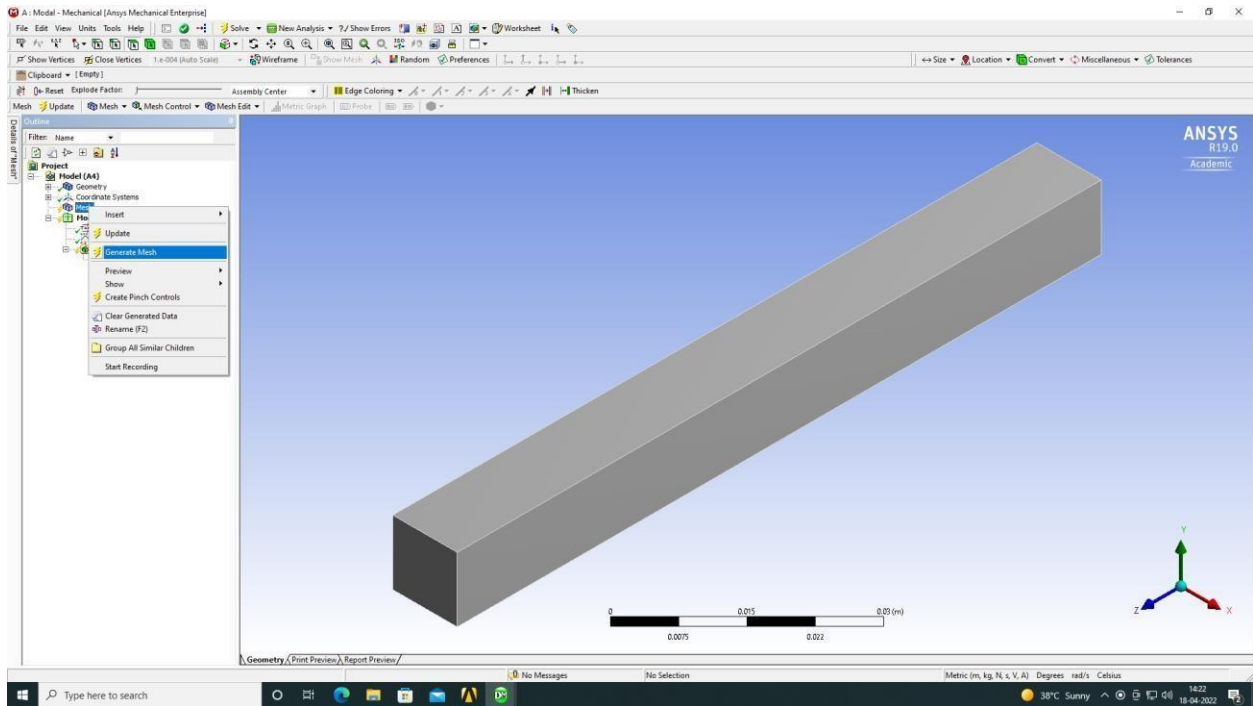
Step 12: Select Extrude from command panel > Select depth from detail panel > Generate



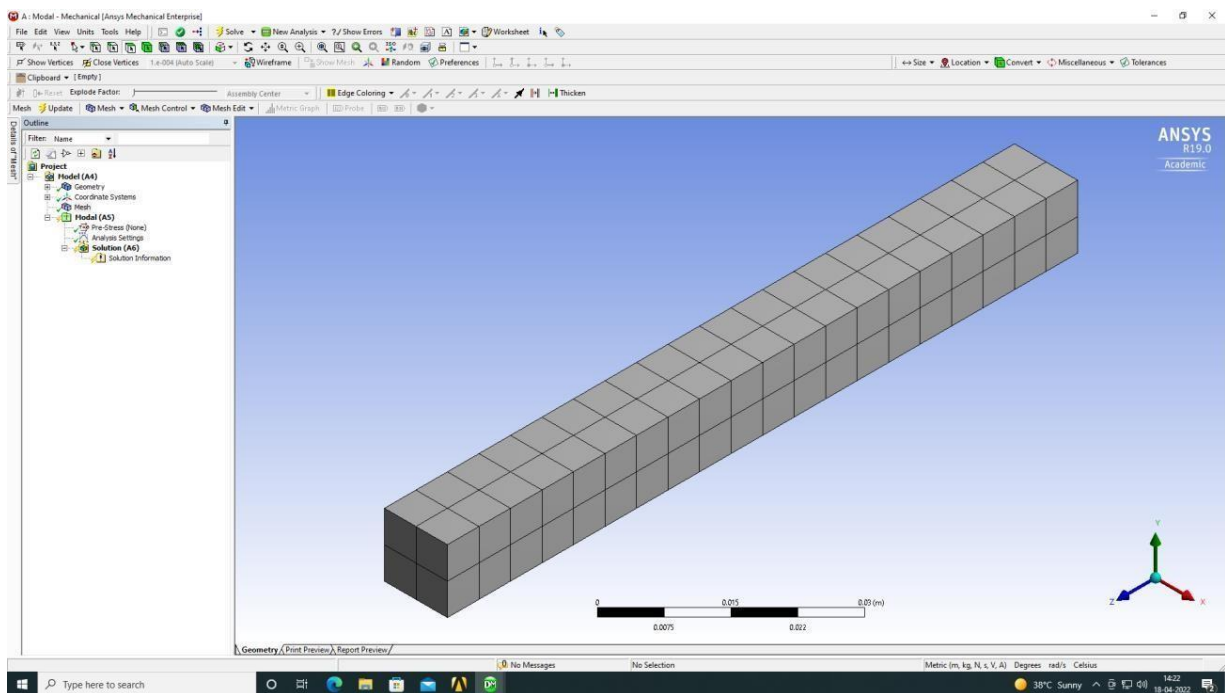
Step 13: Workbench > Modal > Edit > Ok.



Step 14: Mesh > Generate mesh > Ok.

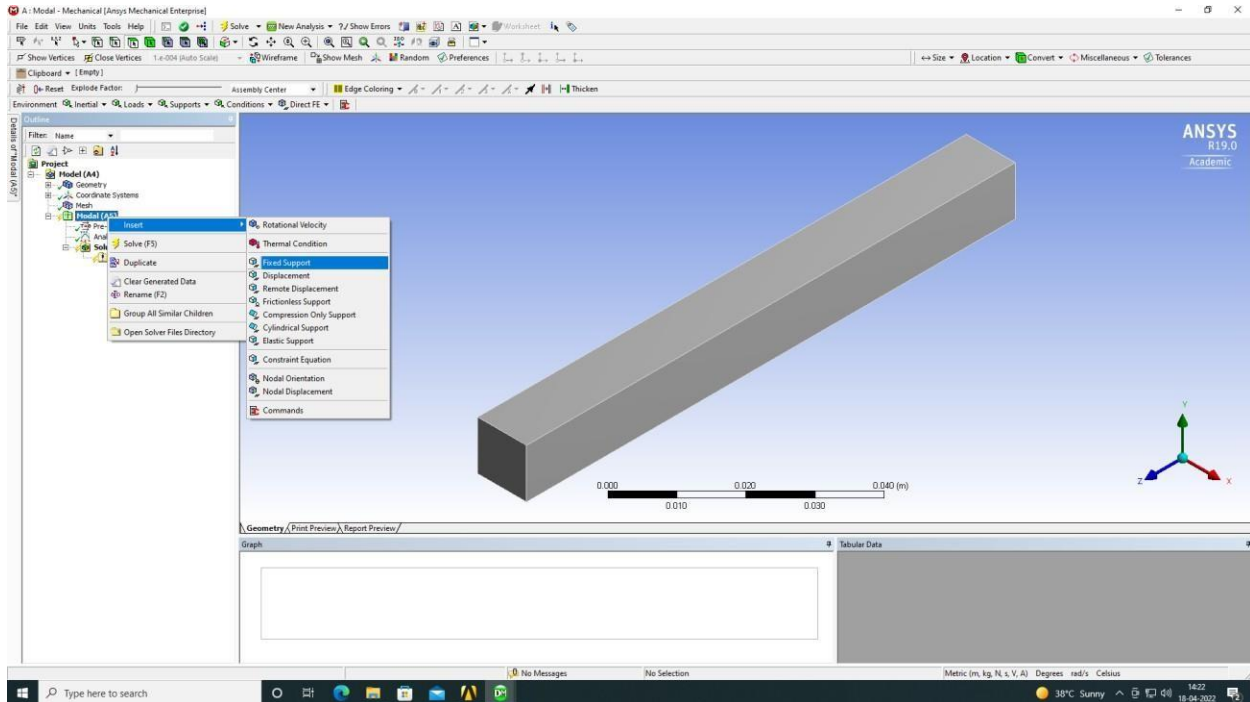


Step 15:

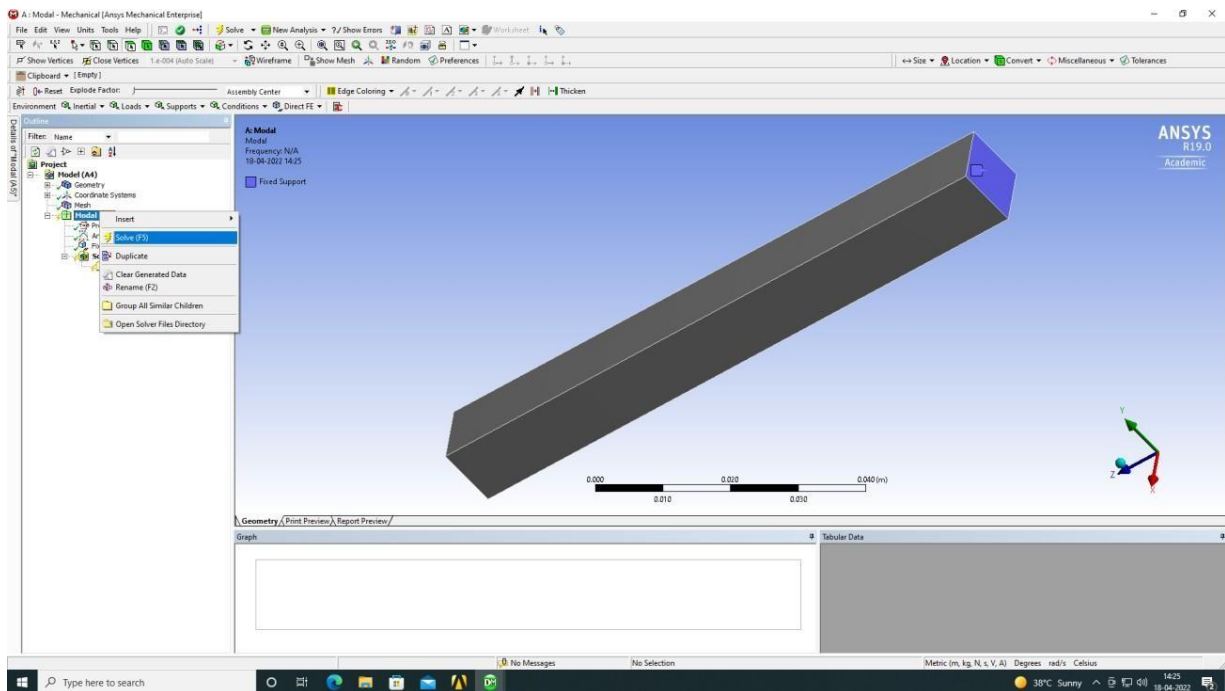




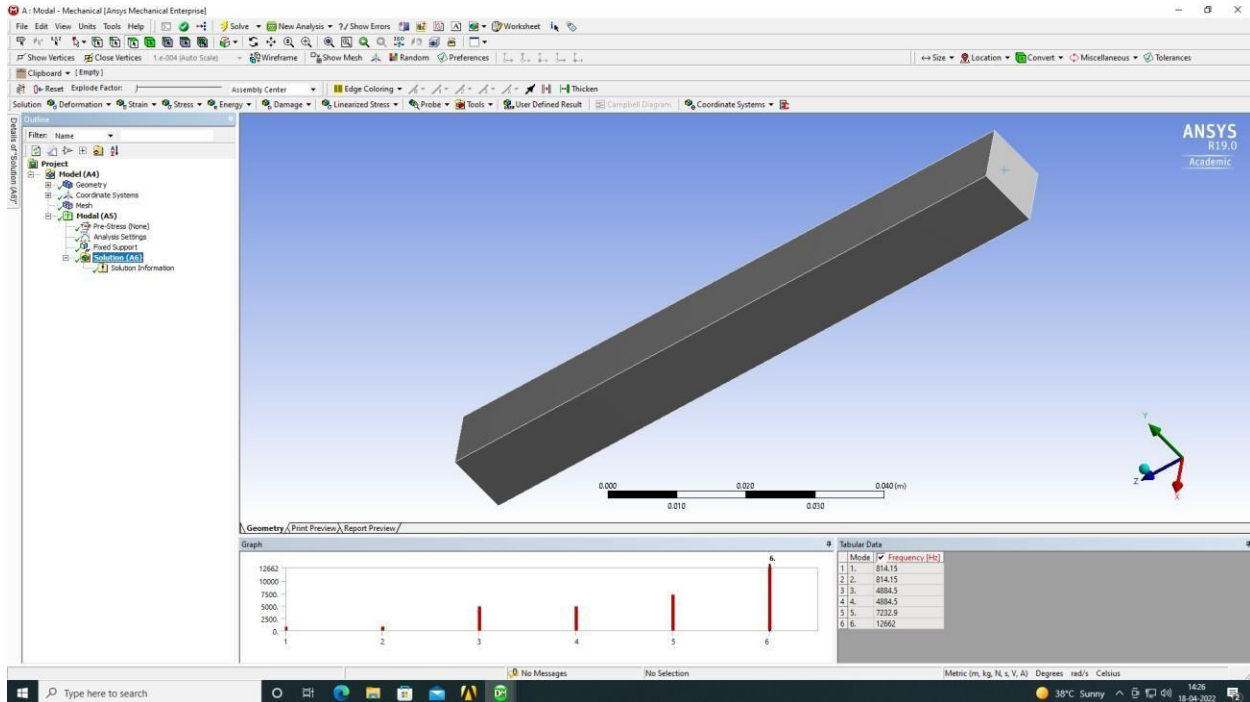
Step 16: Modal > Insert > Fixed support > Select one end of beam > click on apply detail panel.



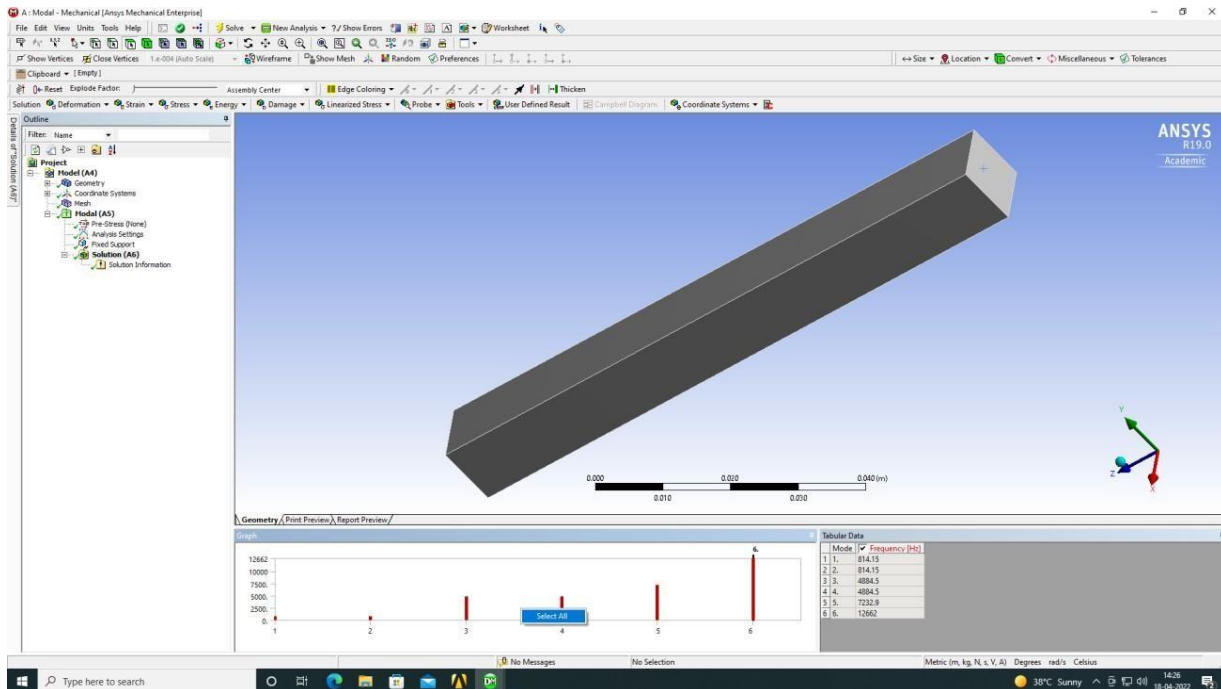
Step 17: Modal > Solve > Ok.



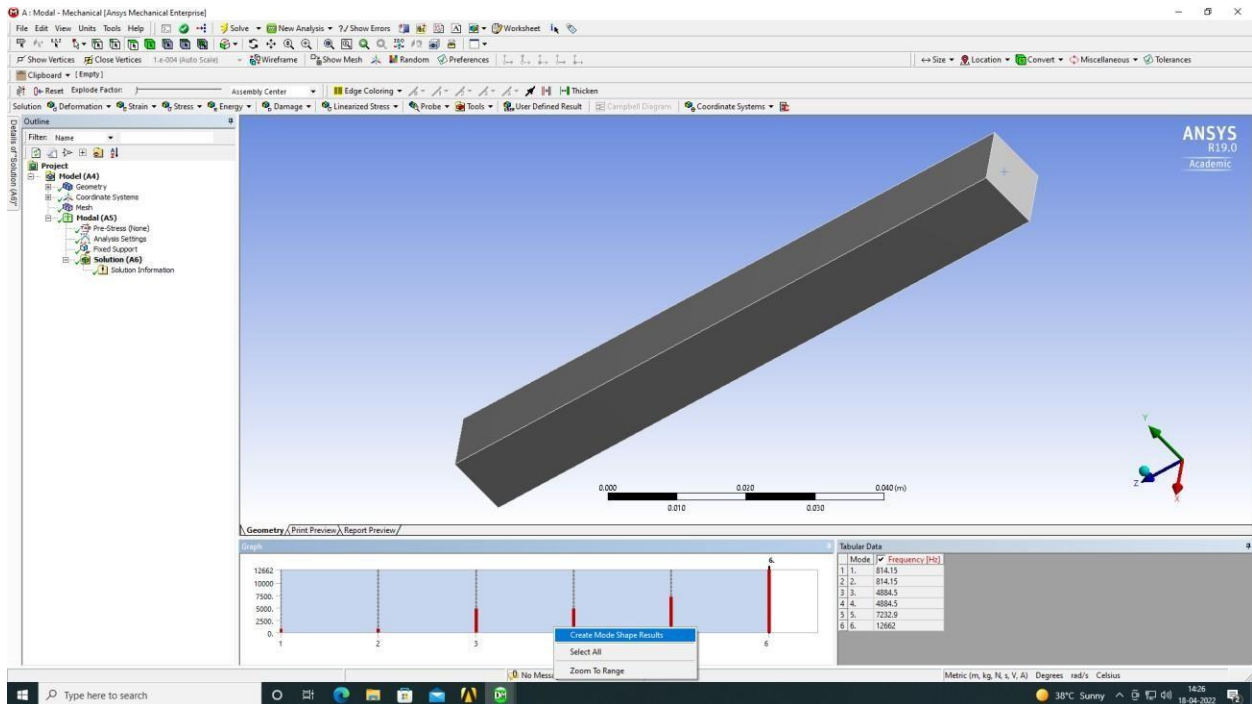
Step 18: Click on solution.



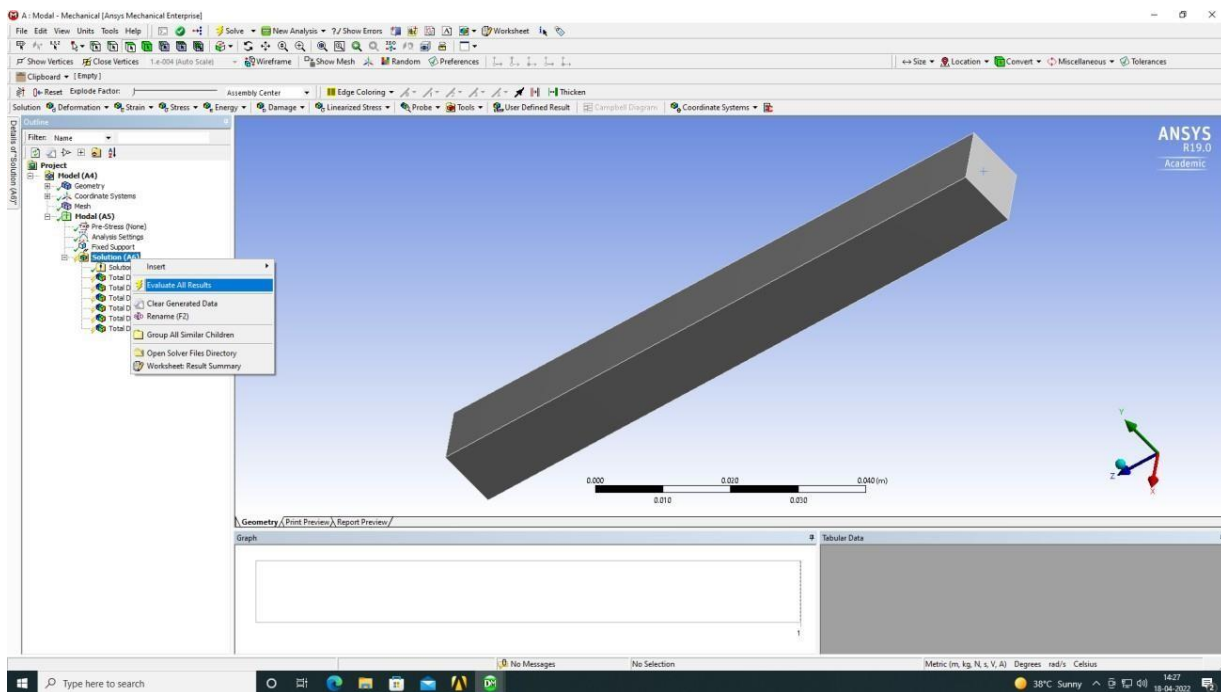
Step 19: Click on graph > click on select all.



Step 20: Click on graph > click on create mode shape results.



Step 21: Solution > Evaluate all result > Ok.



## Step 22: Solution >Total deformation.

