

PCET's & NMVPM's

# Nutan College of Engineering & Research (NCER)

(Affiliated to Dr. Babasaheb Ambedkar Technological University, Lonere)

## CAD/CAM Assignment No: 04 on **Structural Analysis Problem to Solve using a CAE Software**

By

Mr. Chaudhry Sufiyan Ahmad Imtiyaz Ahmad

( PRN: 50641920181162511002 )

Guided By

**Prof. P.V Mohite**



Department of Mechanical Engineering  
NCER, Talegaon Dabhade

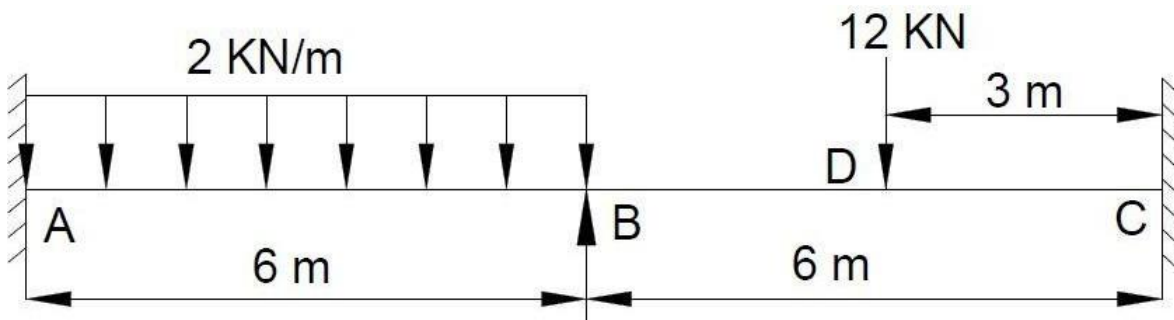
(2021-2022)

## Assignment No: 04

### Structural Analysis Problem to Solve using a CAE Software Like Ansys, Hyper-works Etc.

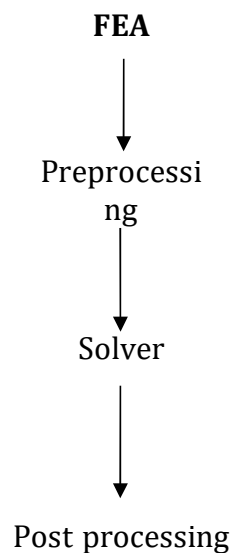
#### Question No: 01 –

Consider a beam of uniform square c/s 5cm X 5cm subjected to point load and uniformly distributed load. Young's Modulus is  $2 \times 10^{11}$  N/m<sup>2</sup> and Poisson's ratio is 0.3. Determine deflection and slopes.



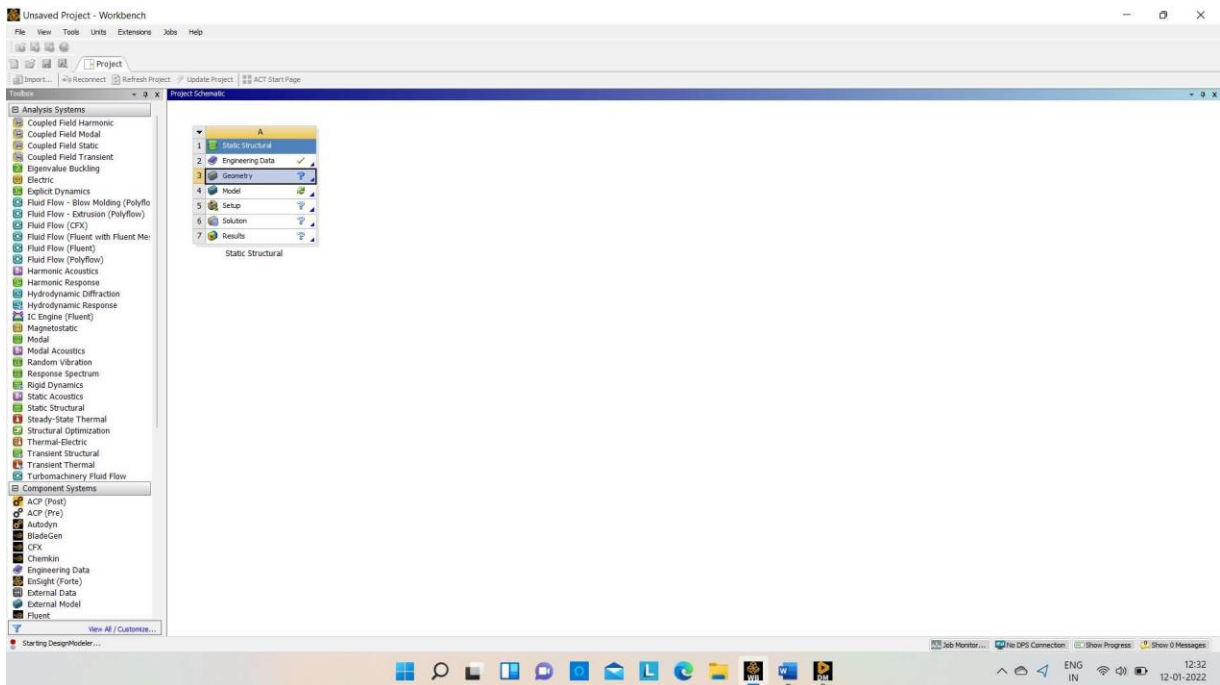
**Answer:**

**Steps in FEA:-**



## 2. Preprocessing- first point in preprocessing is to give analysis type – Analysis type – New Analysis – Static

- in the first process we have to open ANSYS workbench from stat window
- then we have to select the type of problem that we have to solve
- there are five steps are required to solve the problem



### Modeling:-

#### a. Generation of Nodes:-

Preprocessor –Geometry – new design modeler – Point– coordinatesystem-  
in modeling we have to select the design modeler  
in design modeler there have a option to insert a point called aspoint  
then click on point input the coordinate in x,y,z coordinate  
then click on line option and choose the point then click on applygeometry  
the next step is to generate the line between points  
then give the cross section are as circular

<b>Sr. No.</b>	<b>Key point No.</b>	<b>Co-ordinates (x,y) in cm</b>
1	1	1,0
2	2	7,0
3	3	10,0
4	4	13,0

Generation of Nodes

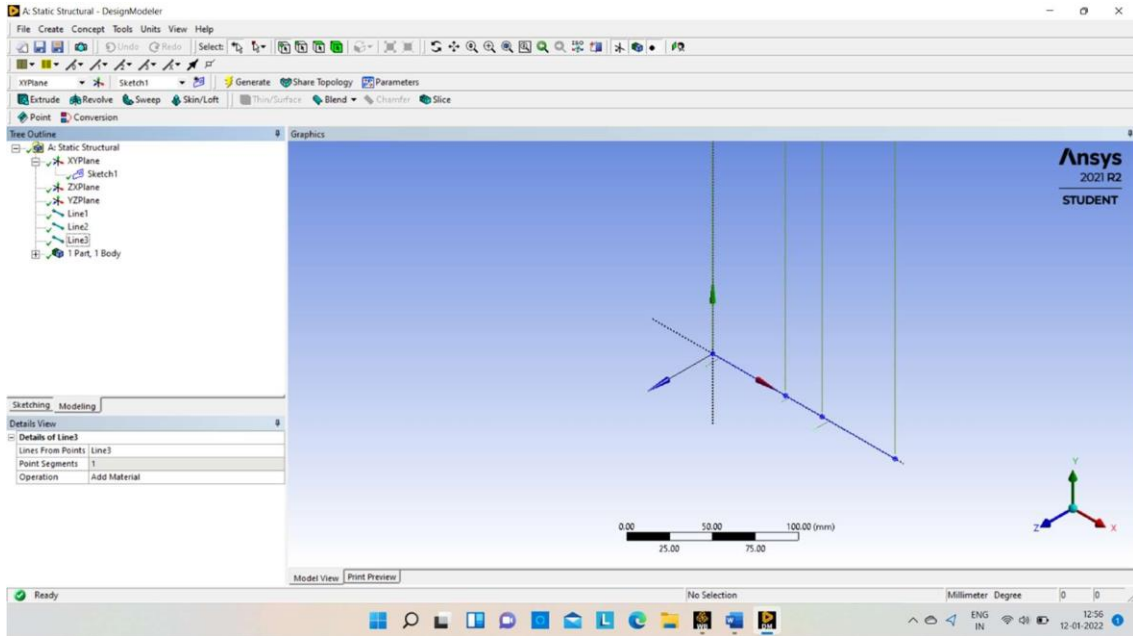


Fig.No.3 Generation of Line

Assign the Element or apply material properties

Preprocessor – Element type – Add – 3D finit stn 180

Preprocessor – engineering data –engineering data source– general material –structure steel -properties outline –Isotropic-

Young's Modulus =  $200 \times 10^3$  Poisson's Ratio = 0.3

next step is to assign the material as per problem you can do it fromtwo ways

1) at main page there is an option called as engineering data fromthat you can add the material

2) in design modeler there was an option to add the material.

in engineering data click on engineering data source select general material and click on structure steel and add the valves of young's modulus and poisons ratio

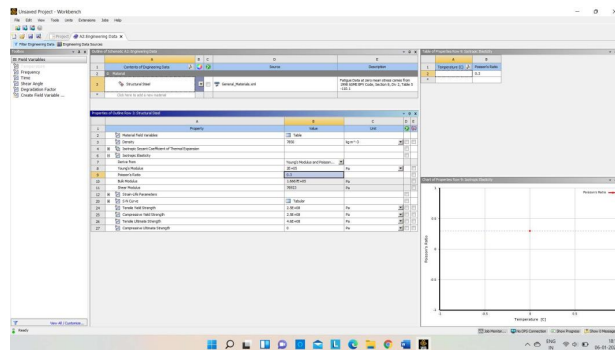


Fig No 4 engineering data

**mechanical modeling** : In this step the solution type and loads are defined.

Define Loads (Boundary Conditions)

**Apply fixed point**

Static structure – insert -fixed point -point 1 and point 4

3 has force

Apply force

-Solution – Define Loads – Apply – structural – force –FY= -2000N on element 1

-Solution – Define Loads – Apply – structural – force –FY= -1200 N on Node NO.3

-for Applying loads and Boundary conditions open the mechanicalmodeling.

In mechanical modeling firstly click on the mesh.

Next step is to go to static structure which is in the flow chart.

After that right click on the static structure to insert fixed point afterthat click on point on which you want to apply Fixed point

again right click on line force for UDL then click on the element onwhich you want to apply the force

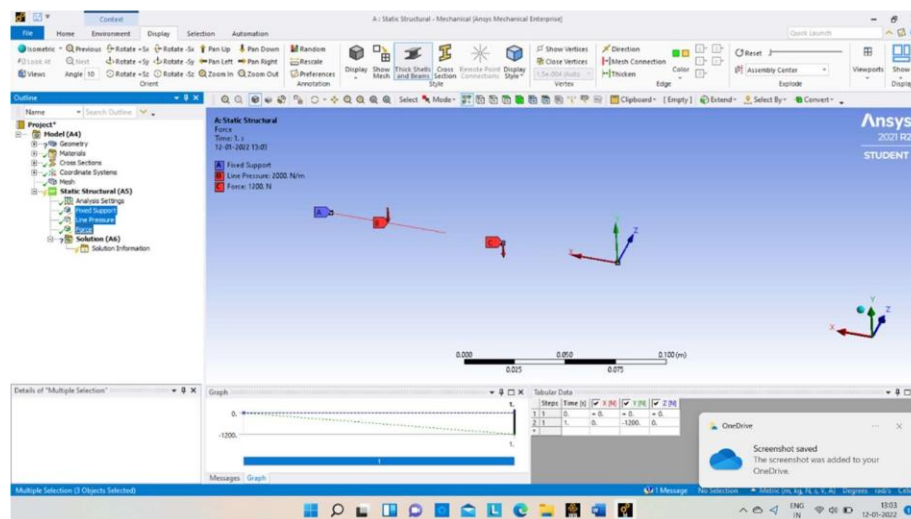


Fig. No.4 Applying the Boundary Conditions

Solve the FEA Model

Solution – Solv-total displacement-stress-reaction at fixed support

next process is to give command to solve

for that click on solution which is in the flow chart.

Right click on solution add the different type of solution you want to solve(displacement, stress, reaction, bending)

Fig 5 add stress, deformation, reaction

**Post processing:-** In the post processing step the Deflection and stress plot are taken and also the reaction values are taken.  
maximum stress

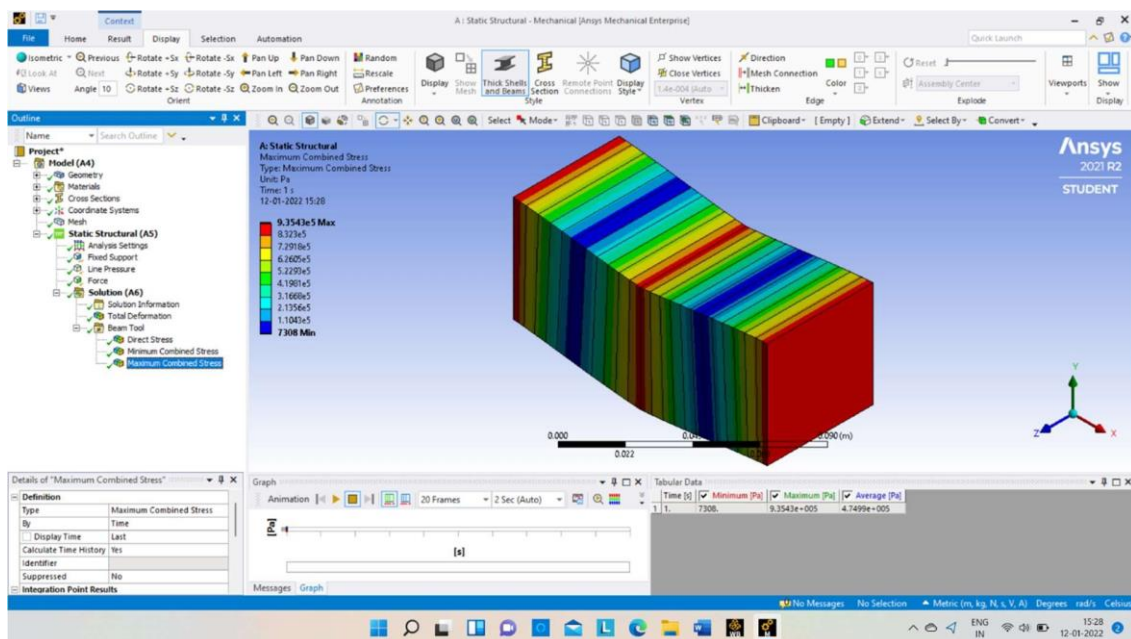


Fig. No.5 maximum stress

Maximum stress: 9.354e5  
total deformation

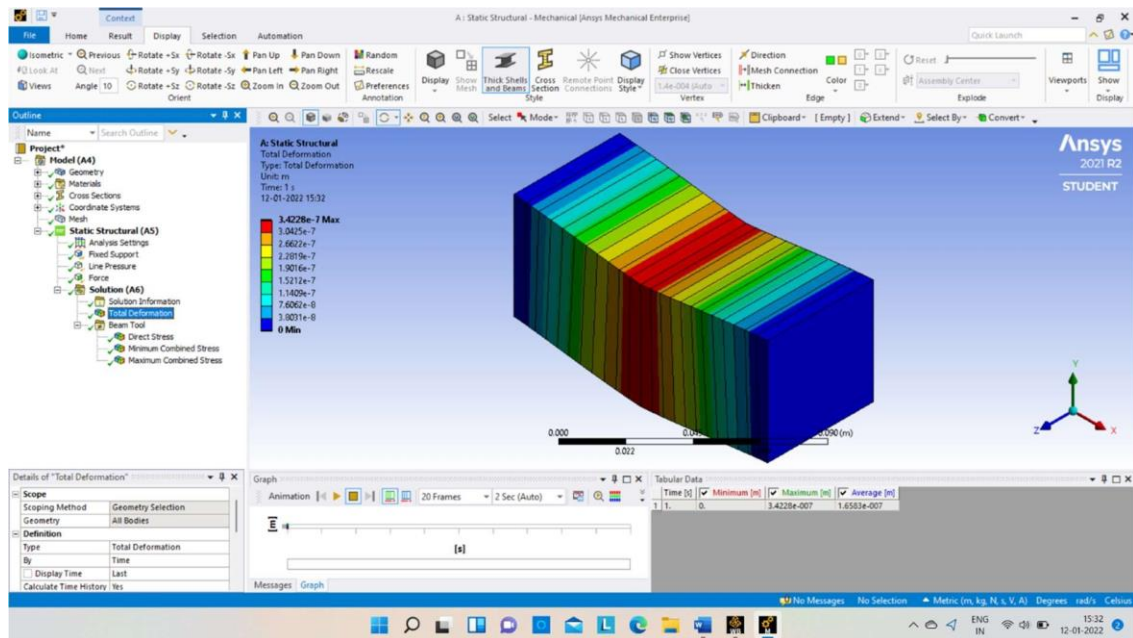


Fig. No.6 total deformation

Total deformation is in the truss element 3.422e-5

### Conclusion:-

Stress and Deflection analysis of beam is carried out with Finite Element Package (ANSYS 17.0). Maximum Stress and Displacement results from ANSYS are 5.3635e6 N/mm<sup>2</sup> 3.422e-5 mm respectively.



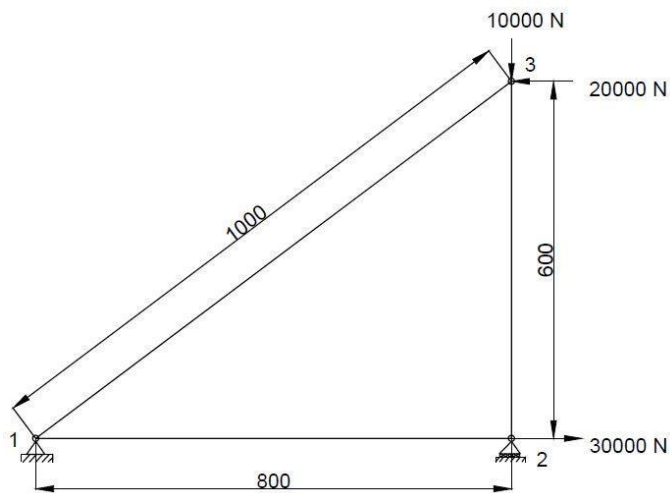
## Question No: 02 –

The three bar truss made of steel ( $E=200 \times 10^3 \text{ N/mm}^2$ ) is subjected to the horizontal forces of 30 KN and 20 KN and vertical force of 10 KN as shown in figure. The c/s areas is  $300 \text{ mm}^2$  for each element using the finite element method. Determine:

Nodal displacements

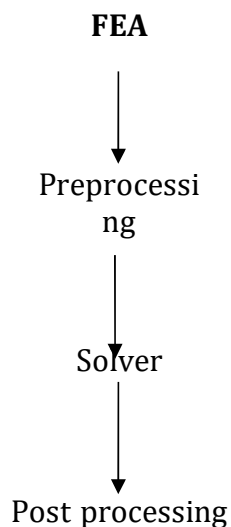
The stresses in each element

The reaction forces at the supports



Answer:

### Steps in FEA:-



Preprocessing- first point in preprocessing is to give analysis type  
– Analysis type – New Analysis – Static

in the first process we have to open ansys workbench from statwindow  
then we have to select the type of problem that we have to solve  
there are five steps are required to solve the problem

Modeling:-

a. Generation of Nodes:-

Preprocessor –Geometry – new design modeler – Point– coordinatesystem-  
in modeling we have to select the design modeler  
in design modeler there have a option to insert a point called aspoint  
then click on point input the coordinate in x,y,z coordinate  
then click on line option and choose the point then click on applygeometry  
the next step is to generate the line between points  
then give the cross section are as circular

Sr. No.	Key point No.	Co-ordinates (x,y)
1	1	10,10
2	2	810,10
3	3	810,610

Fig.No.2 Generation of Nodes

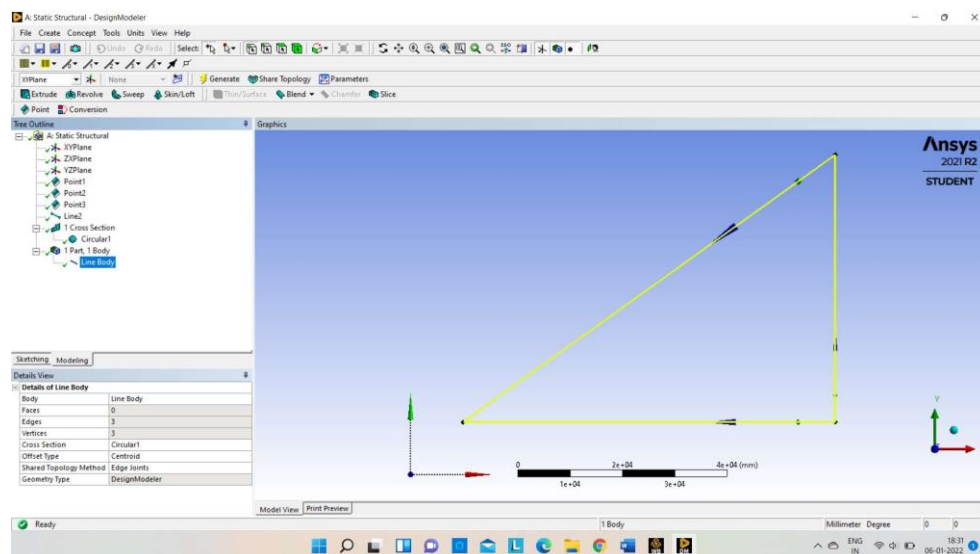


Fig.No.3 Generation of Line

Assign the Element or apply material properties

Preprocessor – Element type – Add – 3D finit stn 180

Preprocessor – engineering data –engineering data source– general material –structure steel -properties outline –Isotropic-  
Young's Modulus = 200e3 Poisson's Ratio = 0.3

next step is to assign the material as per problem you can do it from two ways

1) at main page there is an option called as engineering data from that you can add the material

2) in design modeler there was an option to add the material.

in engineering data click on engineering data source select general material and click on structure steel and add the values of young's modulus and poisons ratio

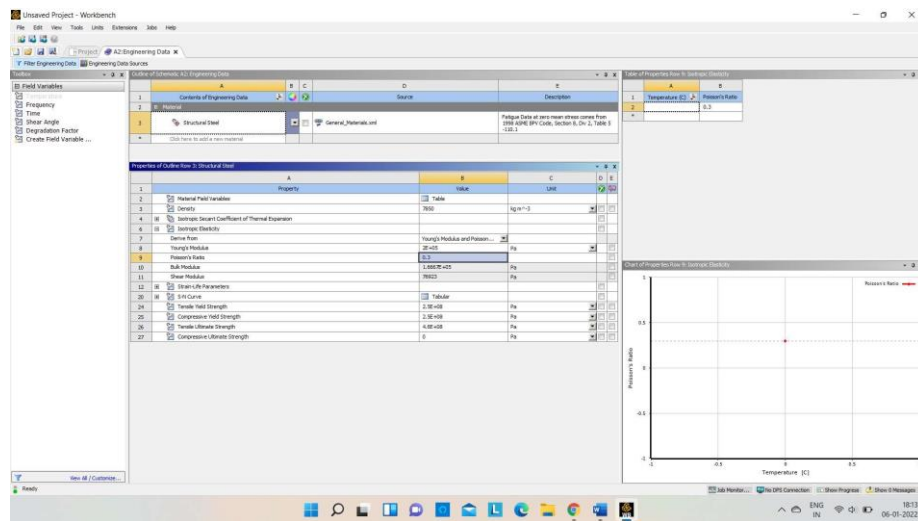


Fig No 4 engineering data

**mechanical modeling** : In this step the solution type and loads are defined.

Define Loads (Boundary Conditions)

**Apply fixed point**

Static structure – insert -fixed point -point 1

– Define Loads – Apply – structural – Displacement No.2 (It has roller support)

Apply force

-Solution – Define Loads – Apply – structural – force –FX= 30000 N on Node NO.2

-Solution – Define Loads – Apply – structural – force –FX= 20000 N on Node NO.3

-Solution – Define Loads – Apply – structural – force –FY= -10000 N on Node NO.3  
for Applying loads and Boundary conditions open the mechanical modeling.

In mechanical modeling firstly click on the mesh.

Next step is to go to static structure which is in the flow chart.

After that right click on the static structure to insert fixed point afterthat click on point on which you want to apply Fixed point  
again right click on force then click on the point on which you wantto apply the force

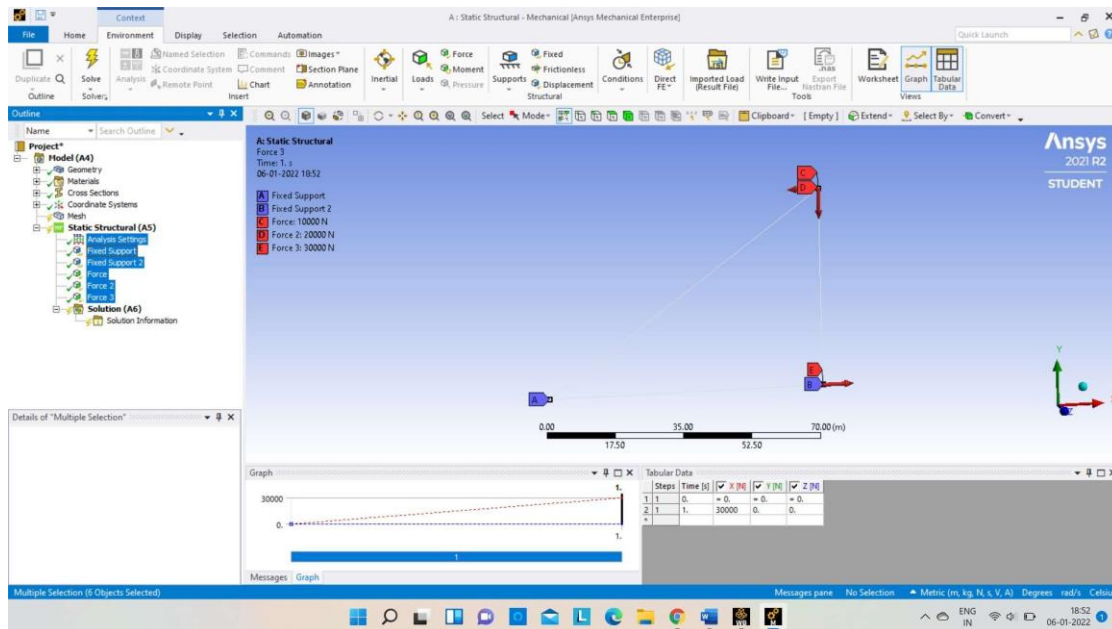


Fig. No.4 Applying the Boundary Conditions

Solve the FEA Model

Solution – Solv-total displacement-stress-reaction at fixed support

next process is to give command to solve

for that click on solution which is in the flow chart.

Right click on solution add the different type of solution you want to solve(displacement, stress, reaction, bending)

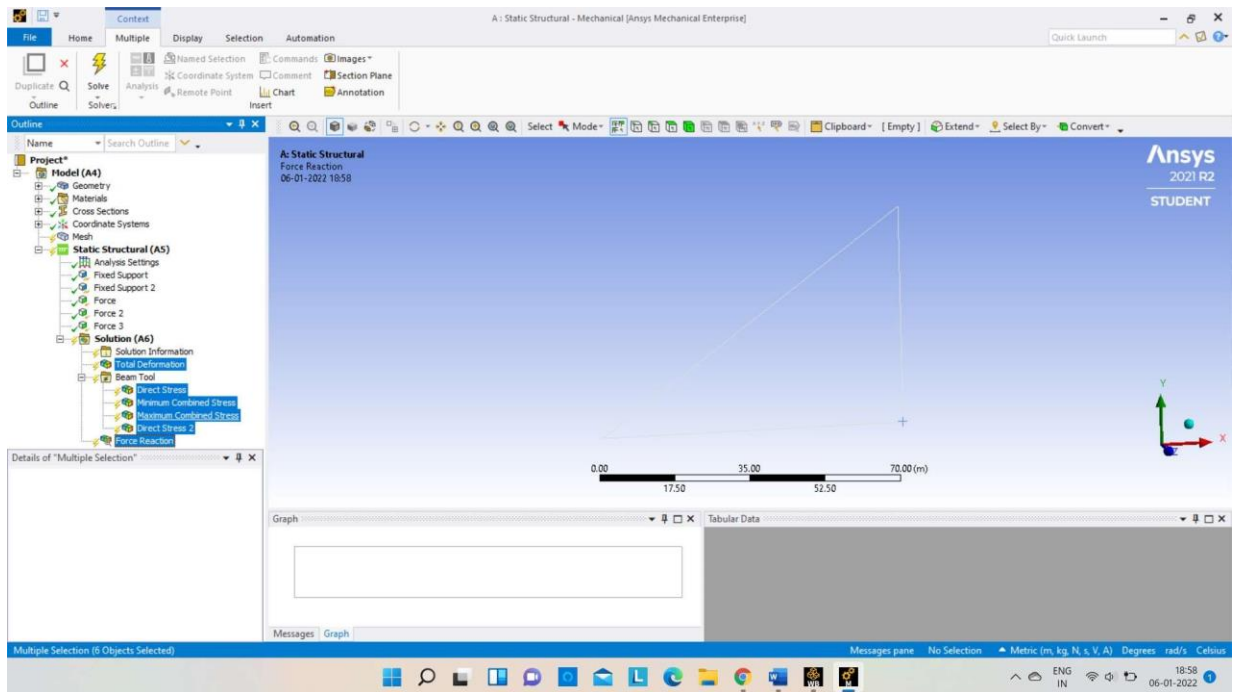


Fig 5 add stress, deformation, reaction

**Post processing:-** In the post processing step the Deflection and stress plot are taken and also the reaction values are taken.  
maximum stress

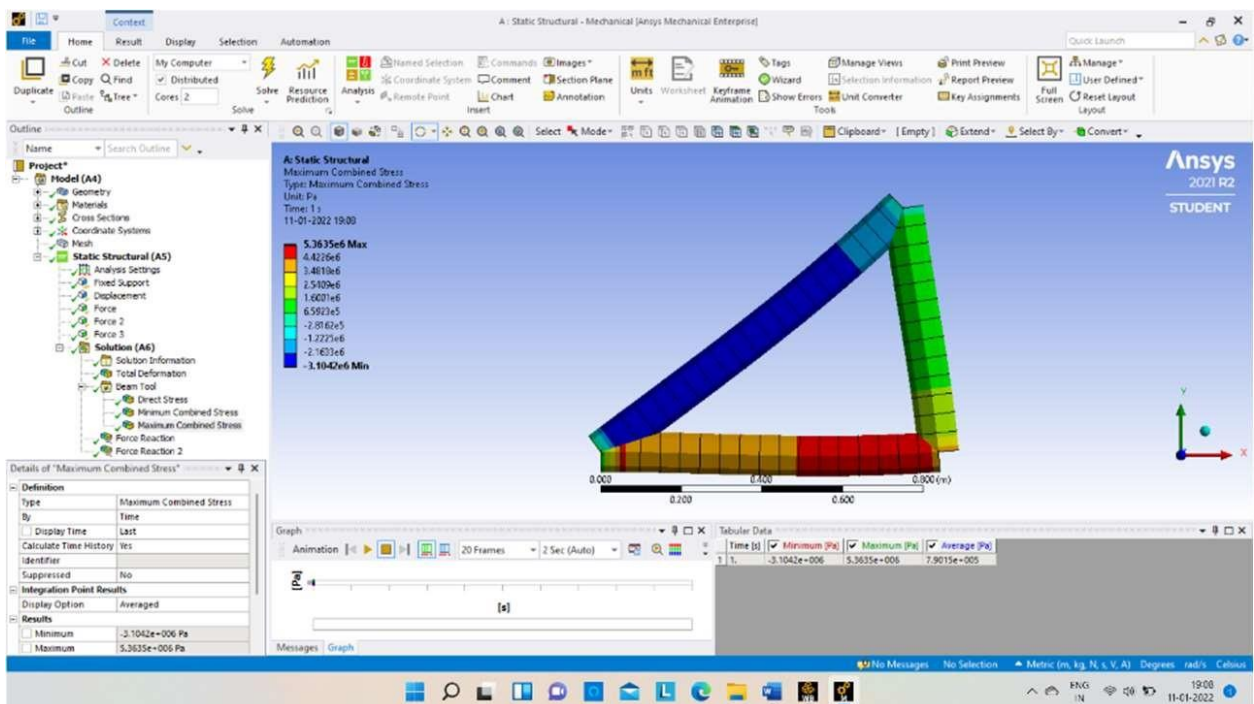


Fig. No.5 maximum stress  
Maximum stress: 5.3635e6  
total deformation

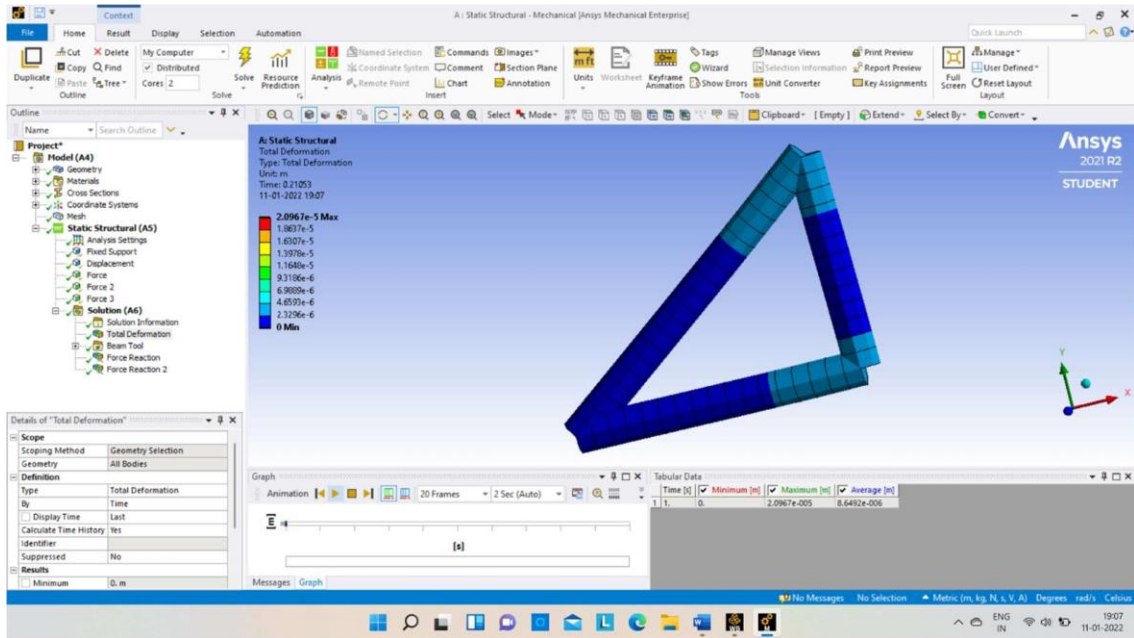


Fig. No.6 Total deformation

Total deformation is in the truss element 2.0967e-5

**c. Reaction Value plot:-**

Reactions at Node	FX	FY
1	-10000 N	15127 N
2	0 N	-5127 N



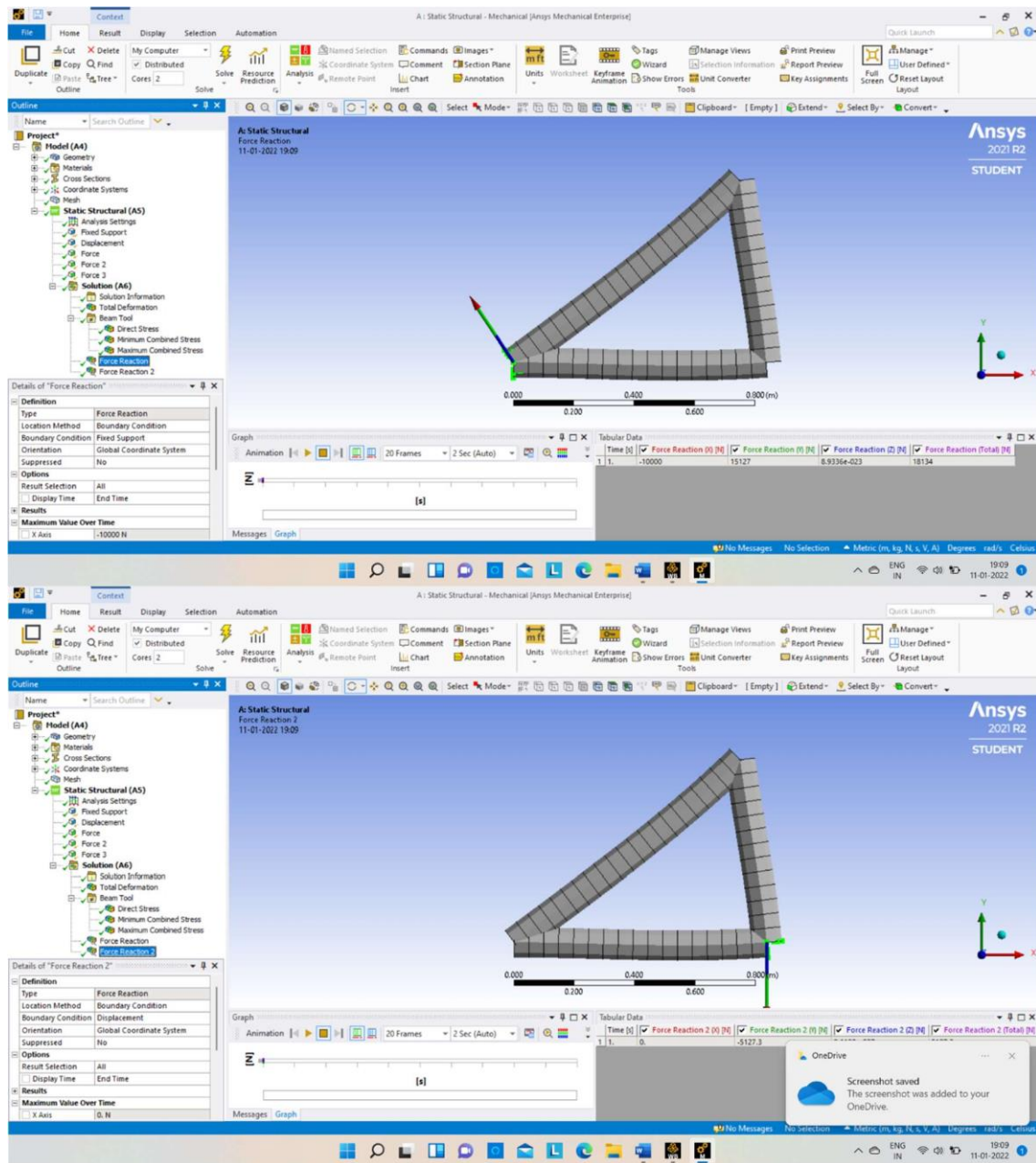


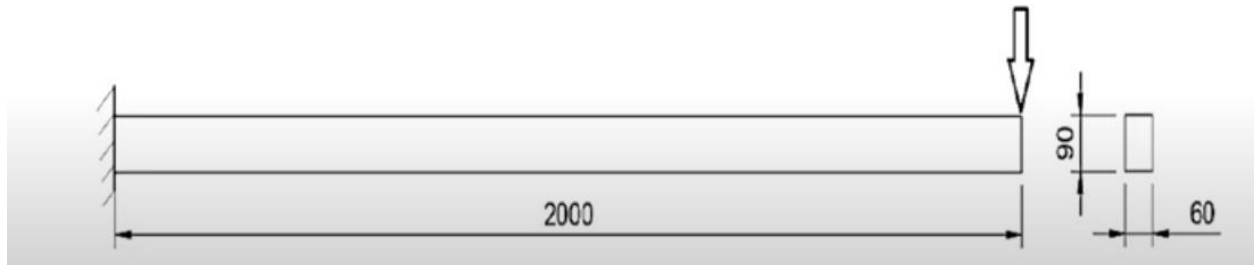
Fig. No.7 Reaction values

## **Conclusion: -**

Stress and Deflection analysis of two-dimensional trusses is carried out with Finite Element Package (ANSYS 17.0). Maximum Stress and Displacement results from ANSYS are  $5.3635 \times 10^6 \text{ N/mm}^2$  and  $2.0967 \times 10^{-5} \text{ mm}$  respectively.

### Question No: 3 –

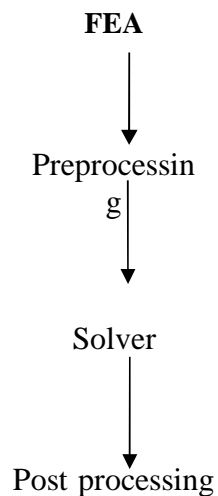
A Rectangular cantilever beam with rectangular section of 60x90mm is subjected to concentrated load of 5kN at free end of beam. Determine the deflection and maximum stresses because of bending. Shear force & bending



moment. Assume  $E=200\text{GPa}$ .

Answer:

#### Steps in FEA:-



Preprocessing- first point in preprocessing is to give analysis type  
– Analysis type – New Analysis – Static

in the first process we have to open ansys workbench from statwindow  
then we have to select the type of problem that we have to solve  
there are five steps are required to solve the problem

Modeling: -

a. Generation of Nodes: -

Preprocessor –Geometry – new design modeler – Point– coordinatesystem-



in modeling we have to select the design modeler  
 in design modeler there have a option to insert a point called as point  
 then click on point input the coordinate in x,y,z coordinate  
 then click on line option and choose the point then click on apply geometry  
 the next step is to generate the line between points  
 then give the cross section are as circular

Sr. No.	Key point No.	Co-ordinates (x,y)
1	1	10,0
2	2	2000,0

Fig.No.2 Generation of Nodes

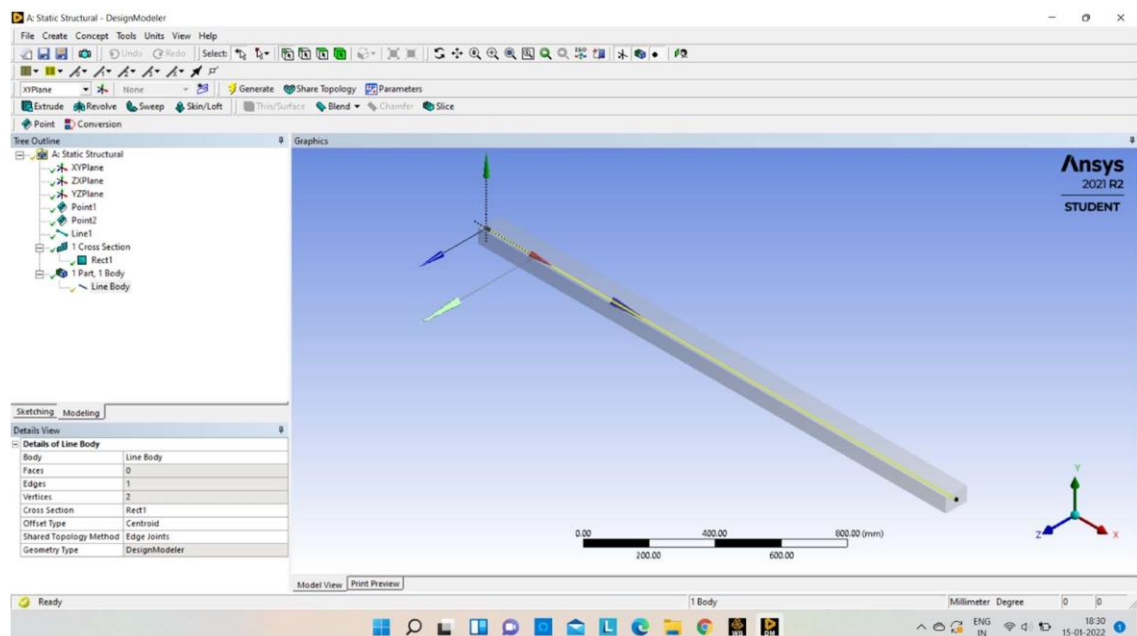


Fig.No.3 Generation of Line

Assign the Element or apply material properties

Preprocessor – Element type – Add – 3D finit stn 180

Preprocessor – engineering data –engineering data source– general material –structure steel -properties outline –Isotropic-

next step is to assign the material as per problem you can do it from two ways

1) at main page there is an option called as engineering data from that you can add the material

2) in design modeler there was an option to add the material.  
in engineering data click on engineering data source select general material and click on structure steel and add the values of young's modulus and poisons ratio

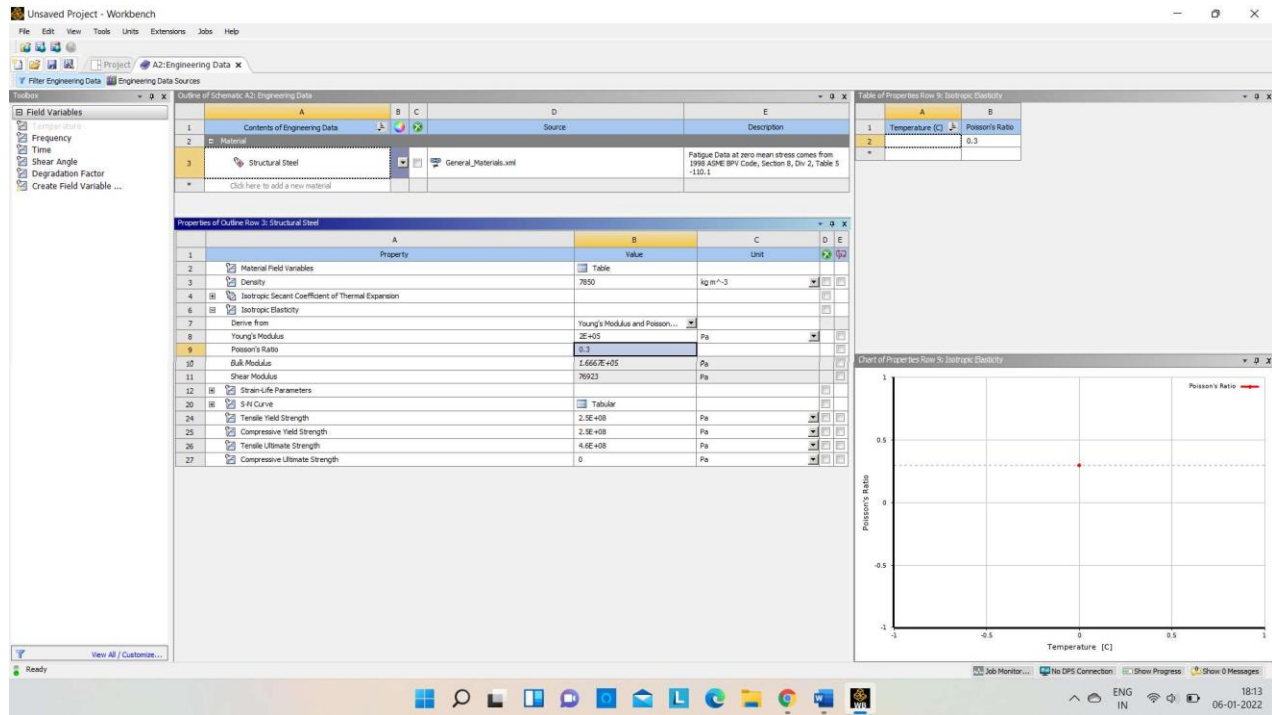


Fig No 4-engineering data

**mechanical modeling :** In this step the solution type and loads are defined.

Define Loads (Boundary Conditions)

**Apply fixed point**

Static structure – insert -fixed point -point 1

Apply force

FY:-5000 on node no 2

for Applying loads and Boundary conditions open the mechanical modeling.

In mechanical modeling firstly click on the mesh.

Next step is to go to static structure which is in the flow chart.

After that right click on the static structure to insert fixed point afterthat click on point on which you want to apply Fixed point

again, right click on force then click on the point on which you want to apply the force

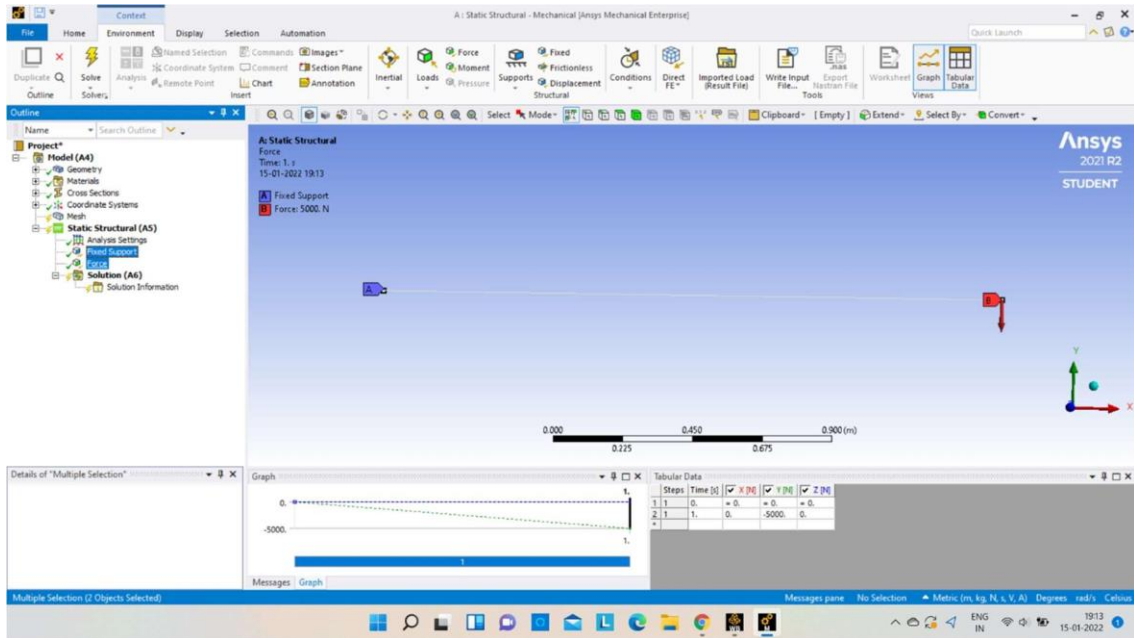


Fig. No.4 Applying the Boundary Conditions

Solve the FEA Model

Solution – Solv-total displacement-stress-reaction at fixed support

next process is to give command to solve

for that click on solution which is in the flow chart.

Right click on solution add the different type of solution you want to solve(displacement, stress, reaction, bending)

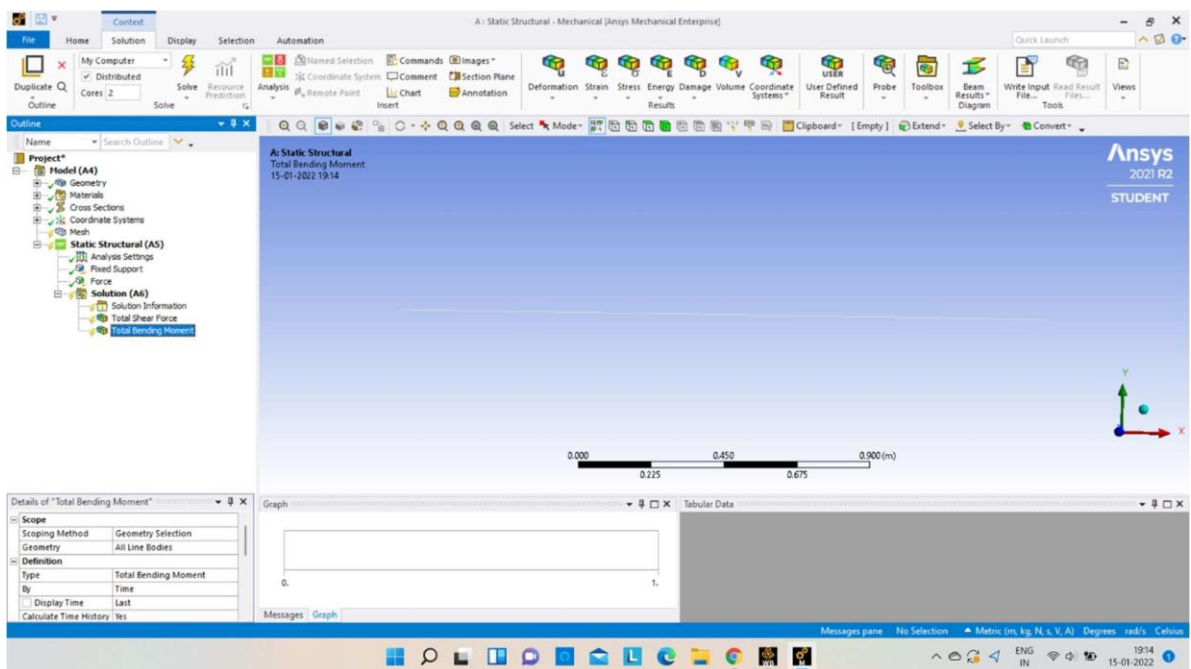


Fig 5 add stress, deformation, reaction

**Post processing:-** In the post processing step the Deflection and stress plot are taken and also the reaction values are taken.  
maximum stress

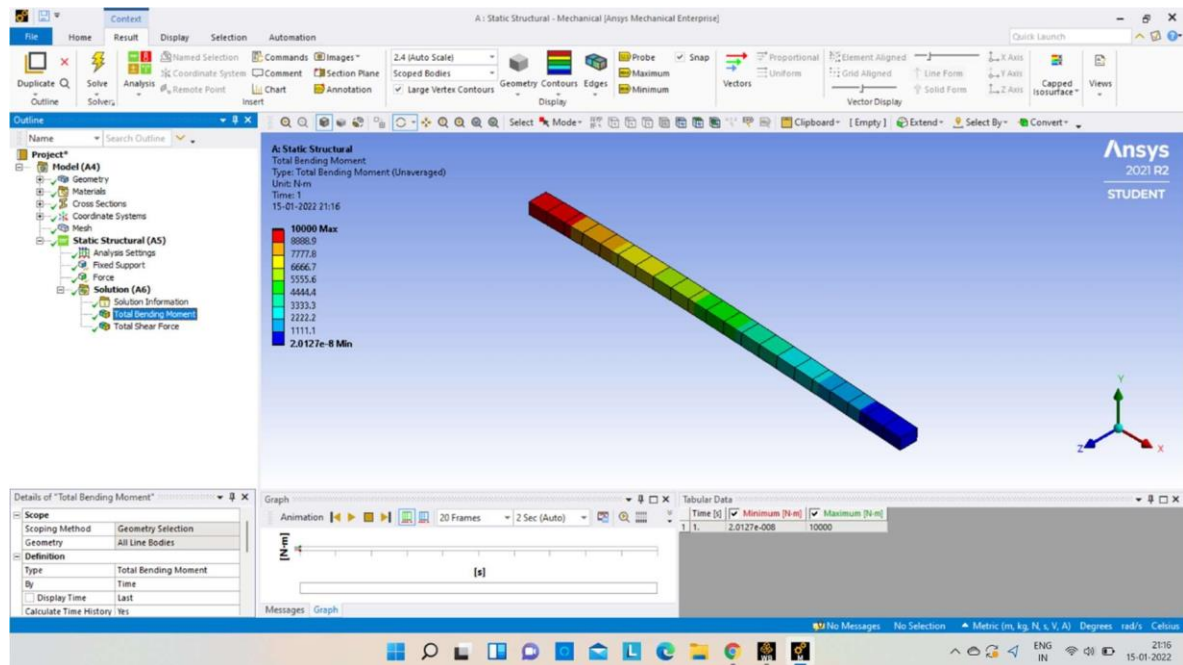


Fig. No.5 bending moment Bending moment :  $10 \times 10^3$  N-m

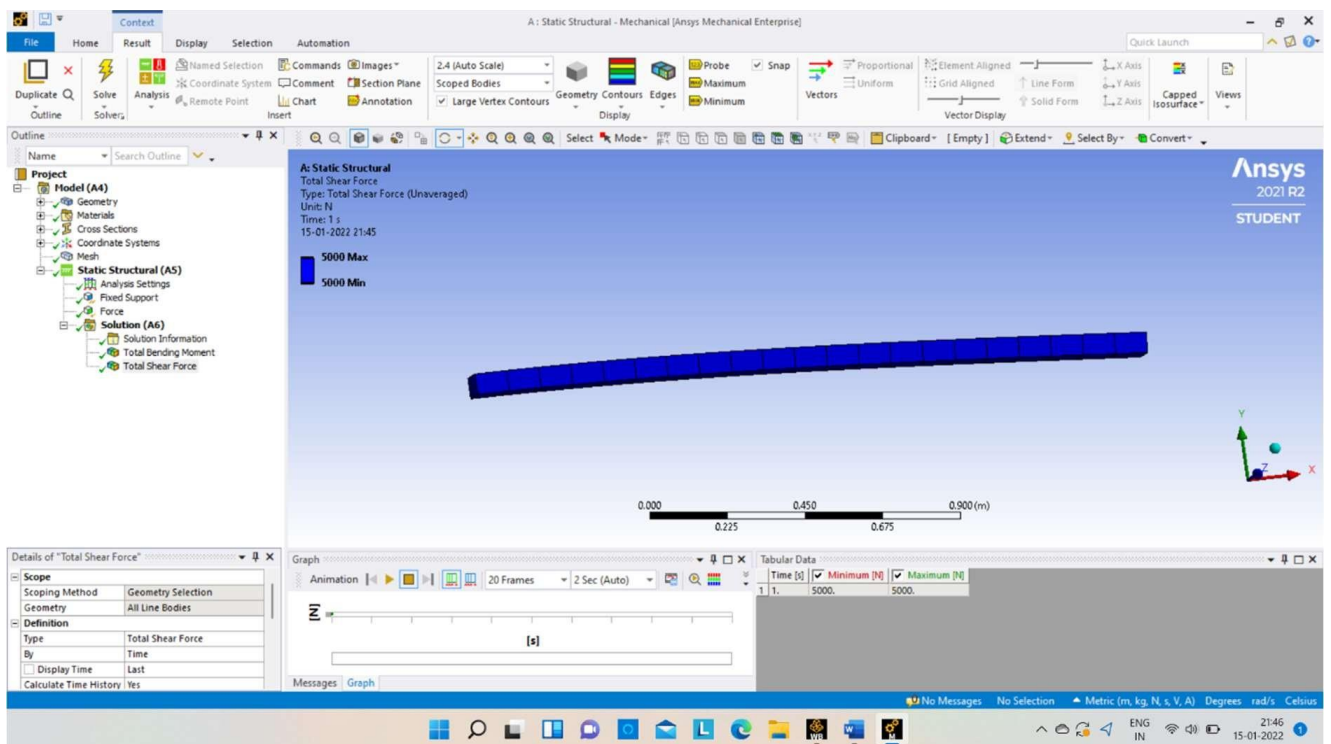


Fig. No. shear stress

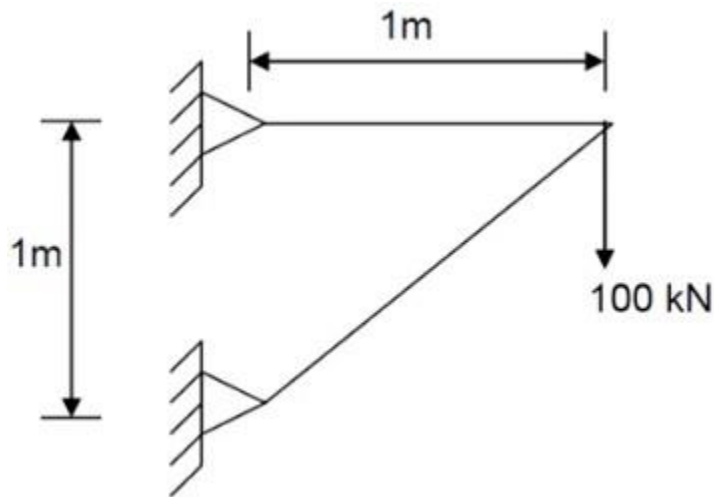
Shear stress:  $5 \times 10^3$

**Conclusion:-**

Stress and bending moment is carried out with Finite Element Package (ANSYS 17.0).  
Maximum Stress and bending moment results from ANSYS are  $5 \times 10^3 \text{ N/m}$  and  $10 \times 10^3 \text{ N-m}$  respectively.

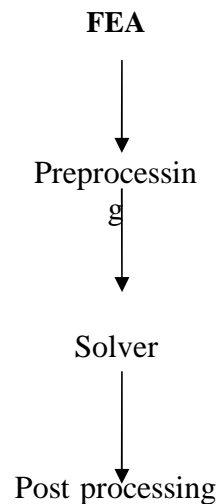
#### Question No: 4 –

Two bar Truss element is shown in figure. Determine the nodal displacement and stresses in each member. Assume  $E=200\text{GPa}$ ,  $A=0.01\text{m}^2$ .



Answer:

#### Steps in FEA:-



Preprocessing- first point in preprocessing is to give analysis type  
– Analysis type – New Analysis – Static

in the first process we have to open ansys workbench from statwindow  
then we have to select the type of problem that we have to solve  
there are five steps are required to solve the problem

## Modeling:-

### a. Generation of Nodes:-

Preprocessor –Geometry – new design modeler – Point– coordinatesystem-  
in modeling we have to select the design modeler

in design modeler there have a option to insert a point called as point

then click on point input the coordinate in x,y,z coordinate

then click on line option and choose the point then click on apply geometry

the next step is to generate the line between points

then give the cross section are as circular

Sr. No.	Key point No.	Co-ordinates (x,y)m
1	1	1,0
2	2	2,0
3	3	1,1

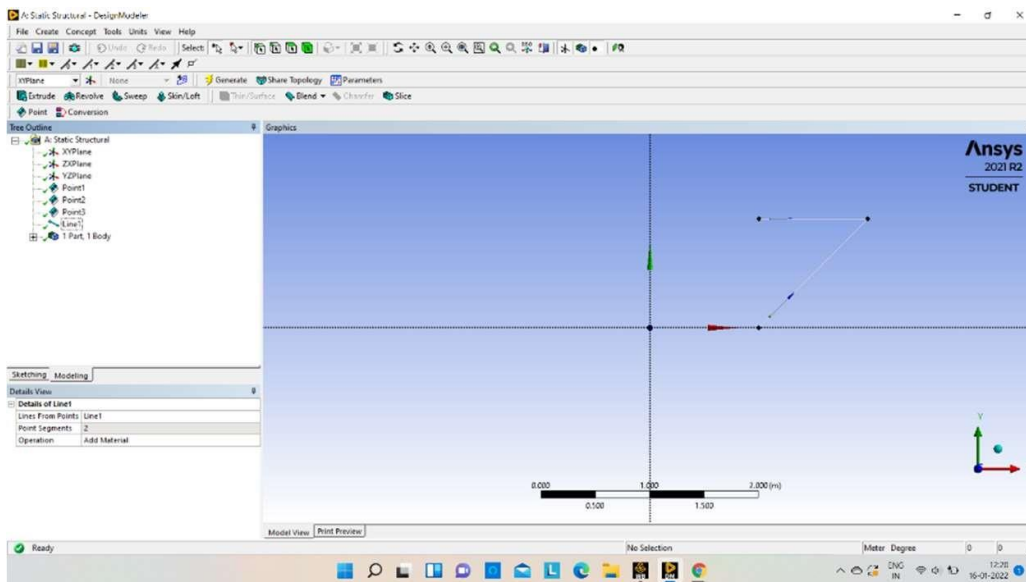


Fig.No.3 Generation of Line

Assign the Element or apply material properties

Preprocessor – Element type – Add – 3D finit stn 180

Preprocessor – engineering data –engineering data source– general material –structure steel -properties outline –Isotropic-

next step is to assign the material as per problem you can do it from two ways

1) at main page there is an option called as engineering data from that you can add the material

2) in design modeler there was an option to add the material.

in engineering data click on engineering data source select general material and click on structure steel and add the values of young's modulus and poisons ratio

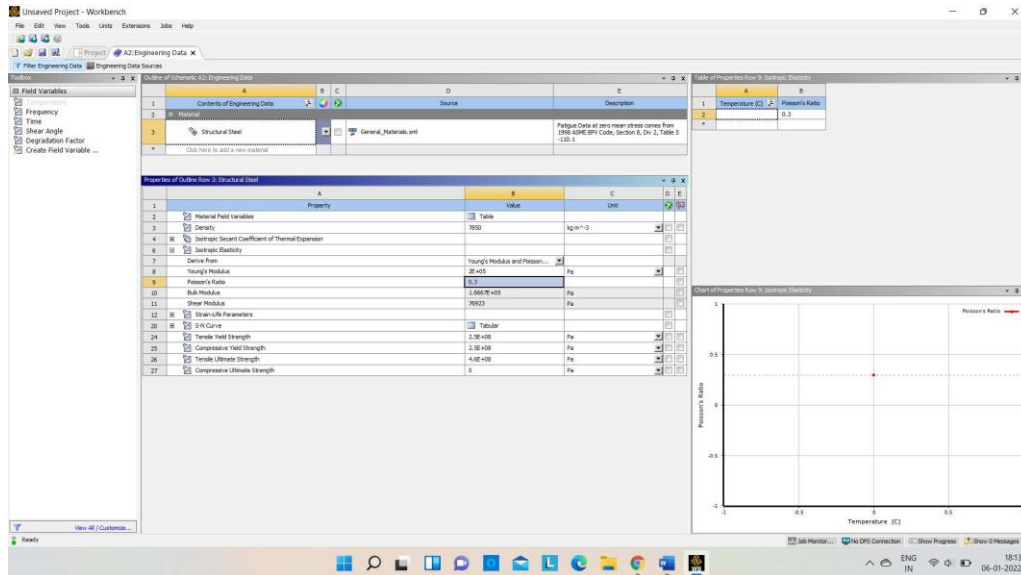


Fig No 4-engineering data

### **mechanical modeling:**

In this step the solution type and loads are defined.

Define Loads (Boundary Conditions)

**Apply fixed point**

Static structure – insert -fixed point -point 1

Apply force

FY:-5000 on node no 2

for Applying loads and Boundary conditions open the mechanical modeling.

In mechanical modeling firstly click on the mesh.

Next step is to go to static structure which is in the flow chart.

After that right click on the static structure to insert fixed point afterthat click on point on which you want to apply Fixed point

again right click on force then click on the point on which you want to apply the force



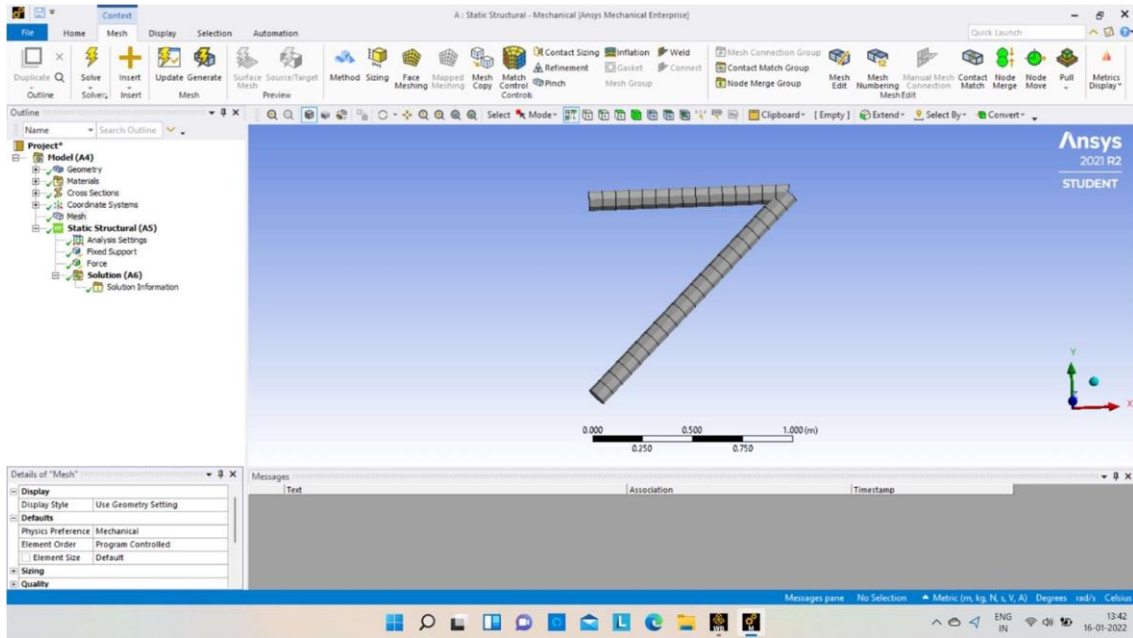


Fig. No.4 Applying the Boundary Condition

Solve the FEA Model

Solution – Solv-total displacement-stress-reaction at fixed support

next process is to give command to solve

for that click on solution which is in the flow chart.

Right click on solution add the different type of solution you want to solve(displacement, stress, reaction, bending)

**Post processing:-** In the post processing step the Deflection and stress plot are taken and also the reaction values are taken.

maximum stress

Total deformation  $5.06 \times 10^{-3}$

stress

Maximum stress:  $4.254 \times 10^3$

### **Conclusion:-**

Stress and deformation is carried out with Finite Element Package (ANSYS 17.0). Maximum Stress and deformation results from ANSYS are  $4.254 \times 10^3$  and  $5.06 \times 10^{-3}$  m respectively.

