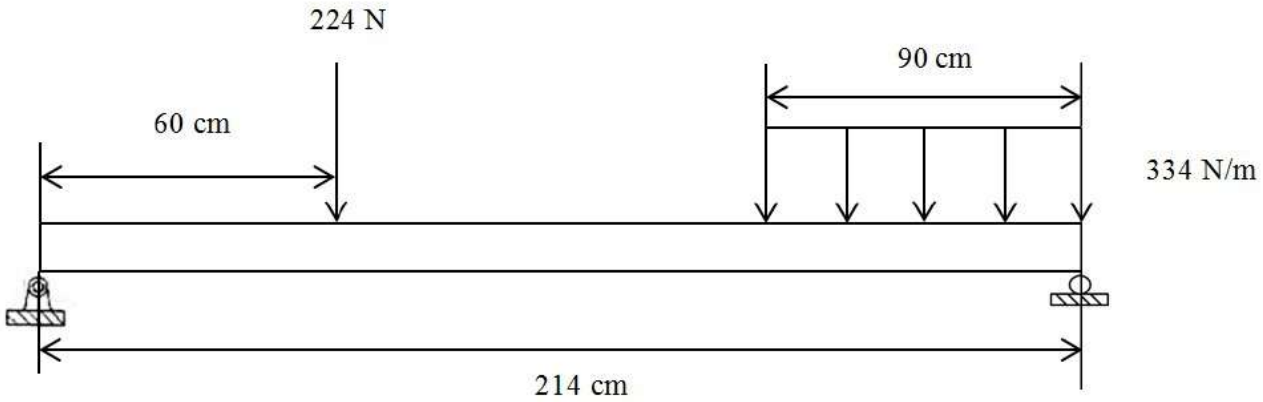
## Beam under distributed load

Beams are used extensively in engineering. As the members are rigidly connected, the members at a joint transmit not only axial loads but also bending and shear. Furthermore, beams are often designed to carry loads both at the joints and along the lengths of the members. In this lecture we will calculate the maximum stress that loading produce in the steel member shown in figure (1).



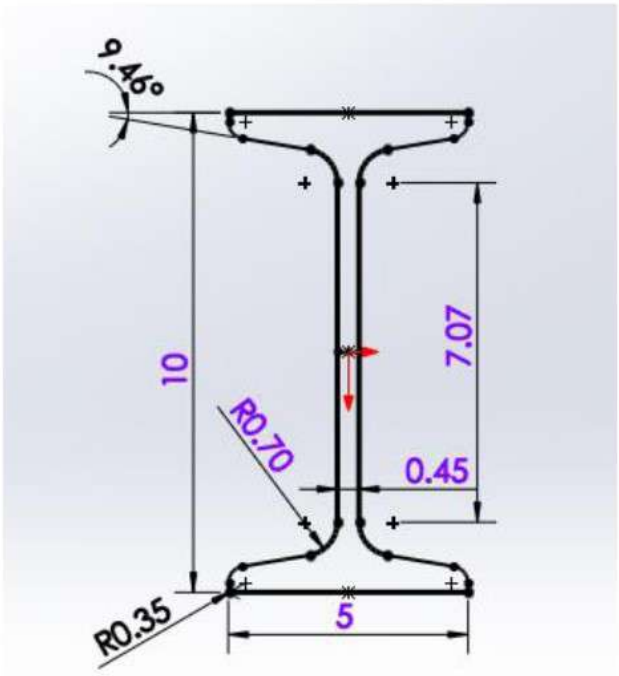




Figure (1)

- ⇨ On the front plane sketch the two lines as shown in figure (2)
- ⇨ Sketch the point as shown in figure (3)
- ⇨ Note: when sketch the point , the coincident relation must be select as shown in figure (4)
- ⇨ Select exit sketch

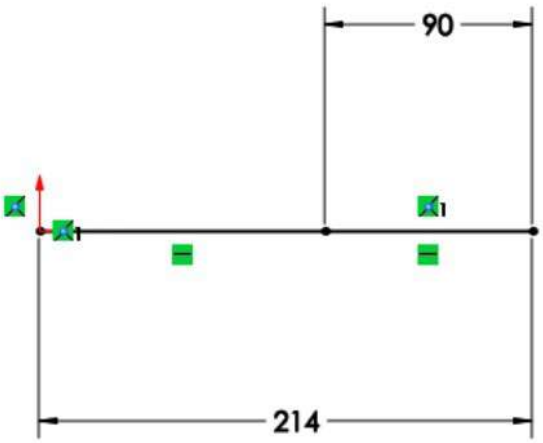



Figure (2)

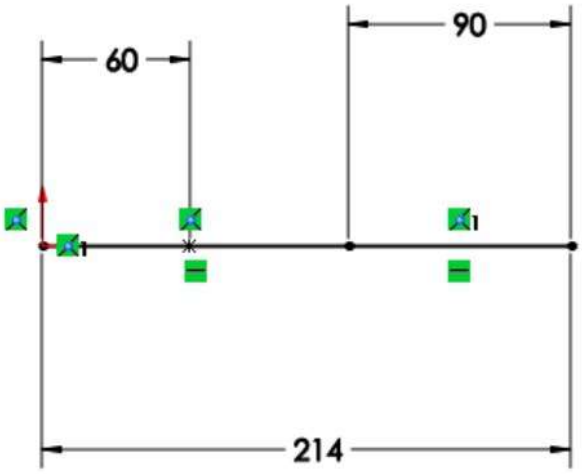



Figure (3)

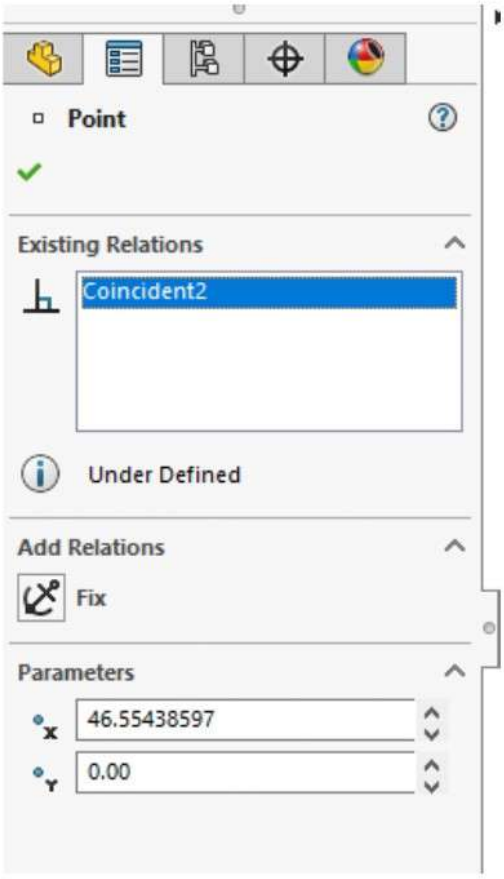



Figure (4)

- ⇨ From insert, Weldments, select Structural Member, figure (5)
- ⇨ Set the type as sb beam
- ⇨ Set the size as 100×8
- ⇨ Select the two lines you just created, figure (6)
- ⇨ Change the angle into 90 deg, figure (7)
- ⇨ Click OK

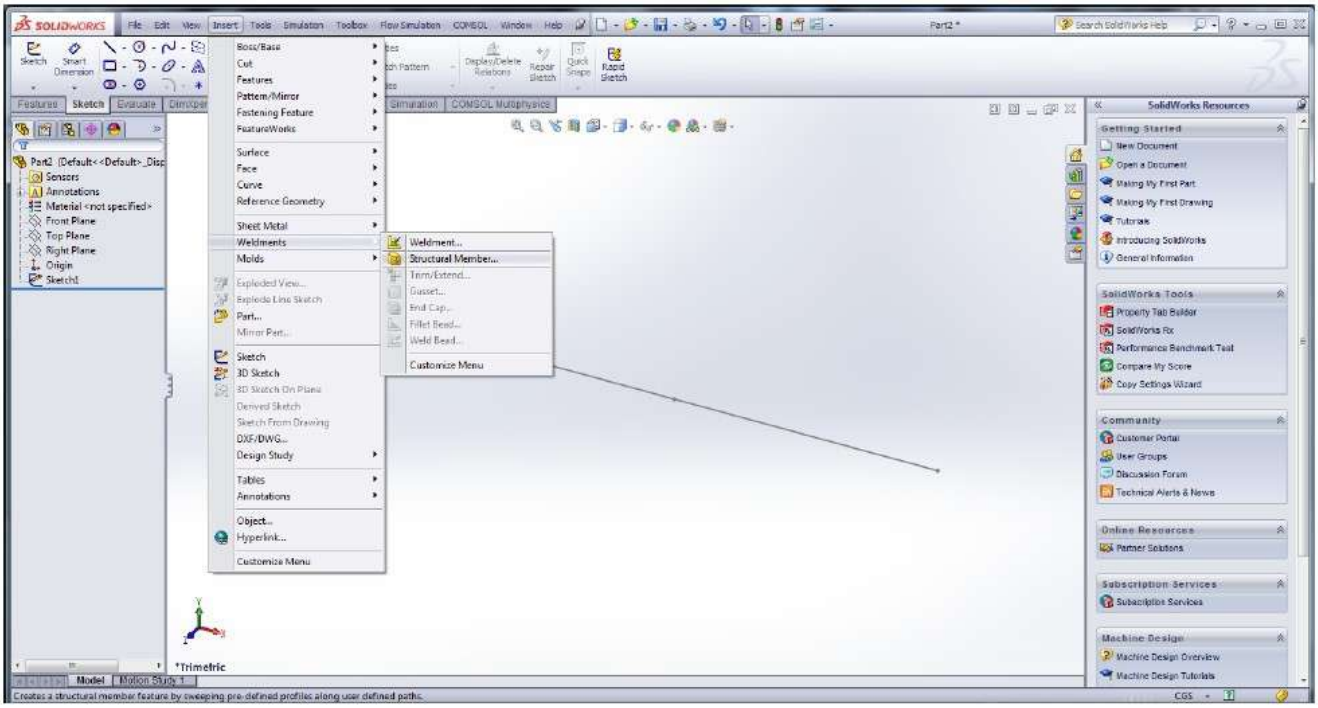



Figure (5)

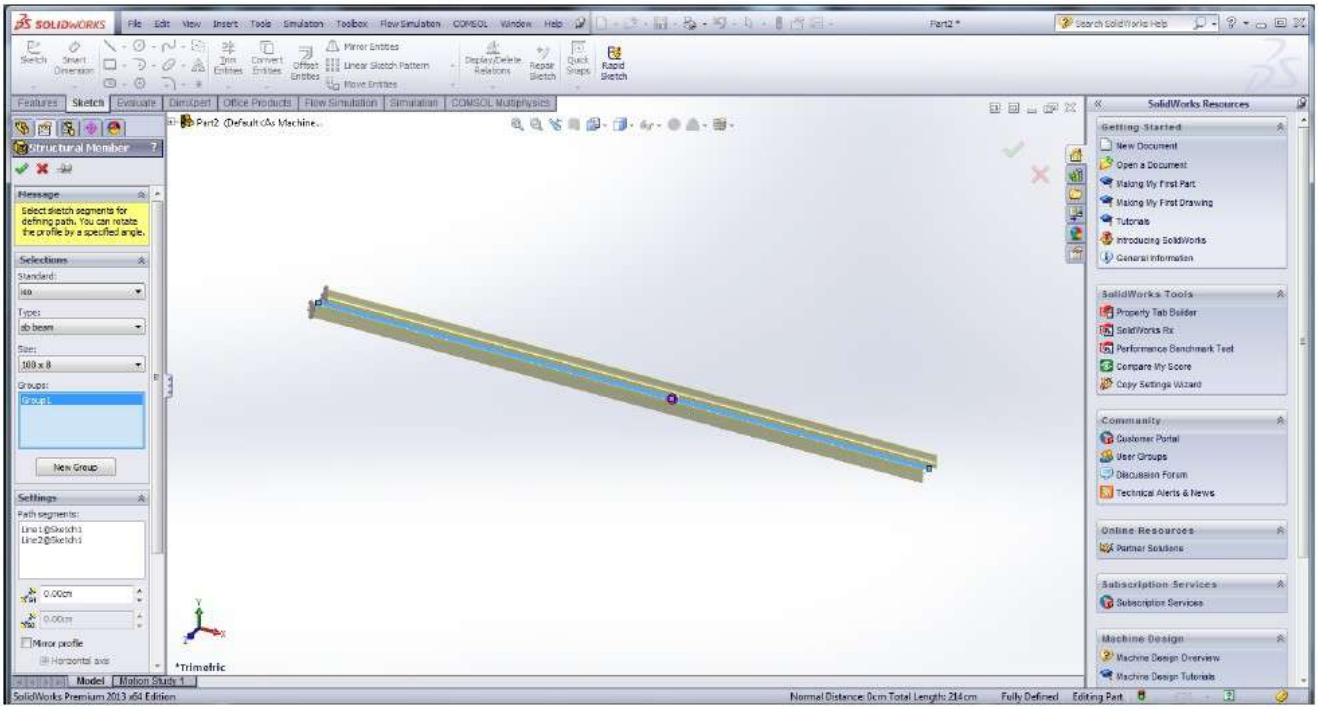



Figure (6)

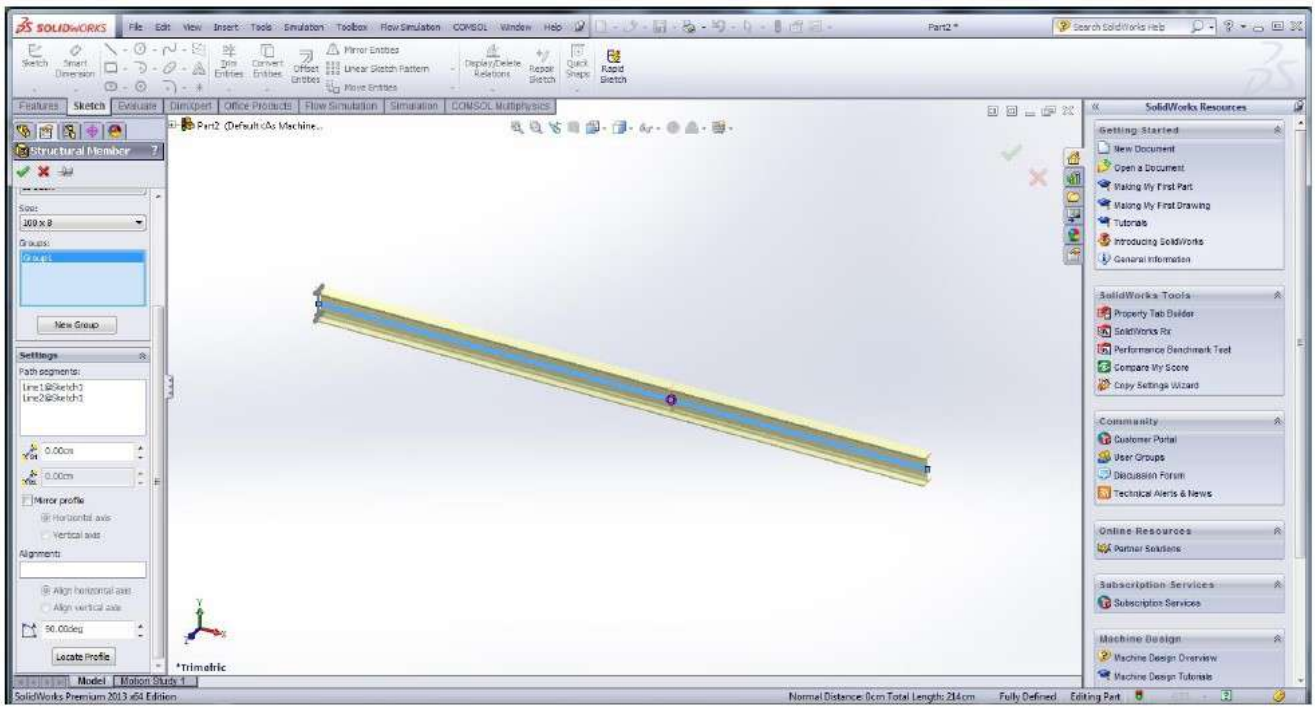


Figure (7)

- ⇨ From features tool bar, reference geometry select Point
- ⇨ Select the point from the sketch, and the top surface as reference entities
- ⇨ Select projection
- ⇨ Click OK, figure (8)

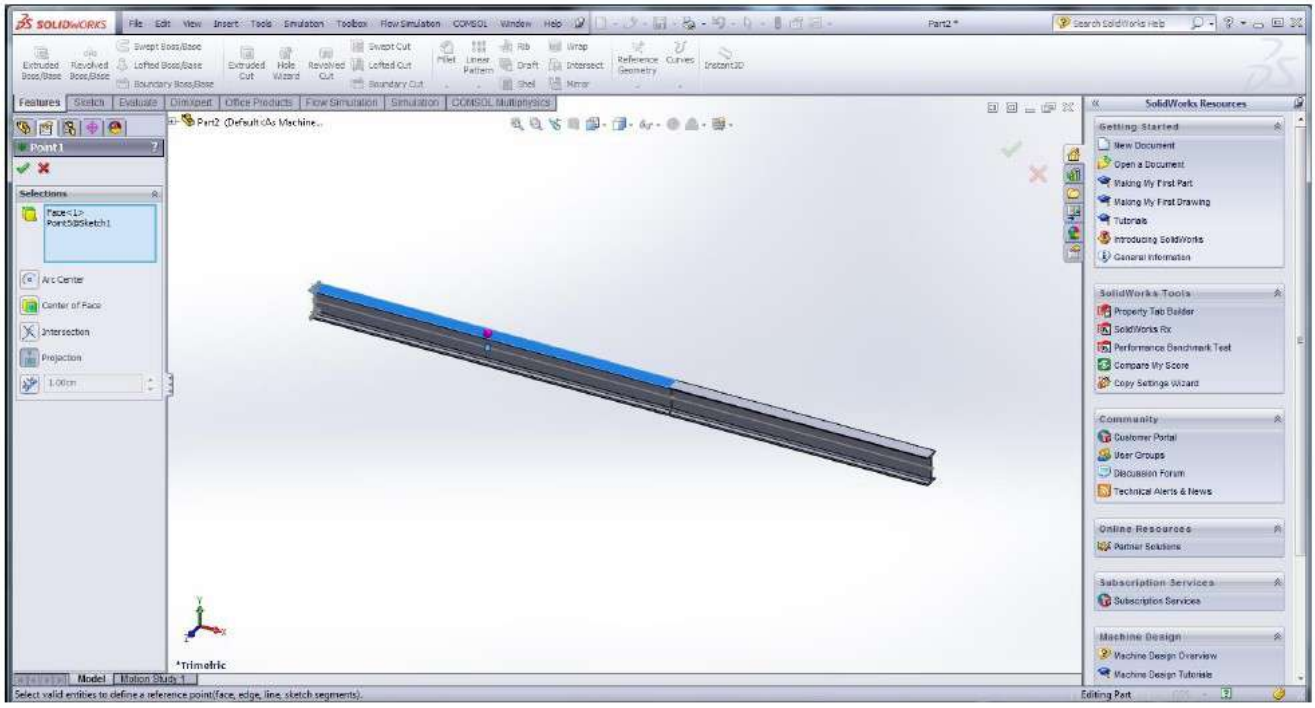


Figure (8)

- ⇨ From Simulation toolbar select New study
- ⇨ Set it as static study type
- ⇨ Set the material as Alloy steel
- ⇨ From fixtures select fixed geometry
- ⇨ Select the left node as immovable (No translation), as shown in figure (9)

⇨ Click OK

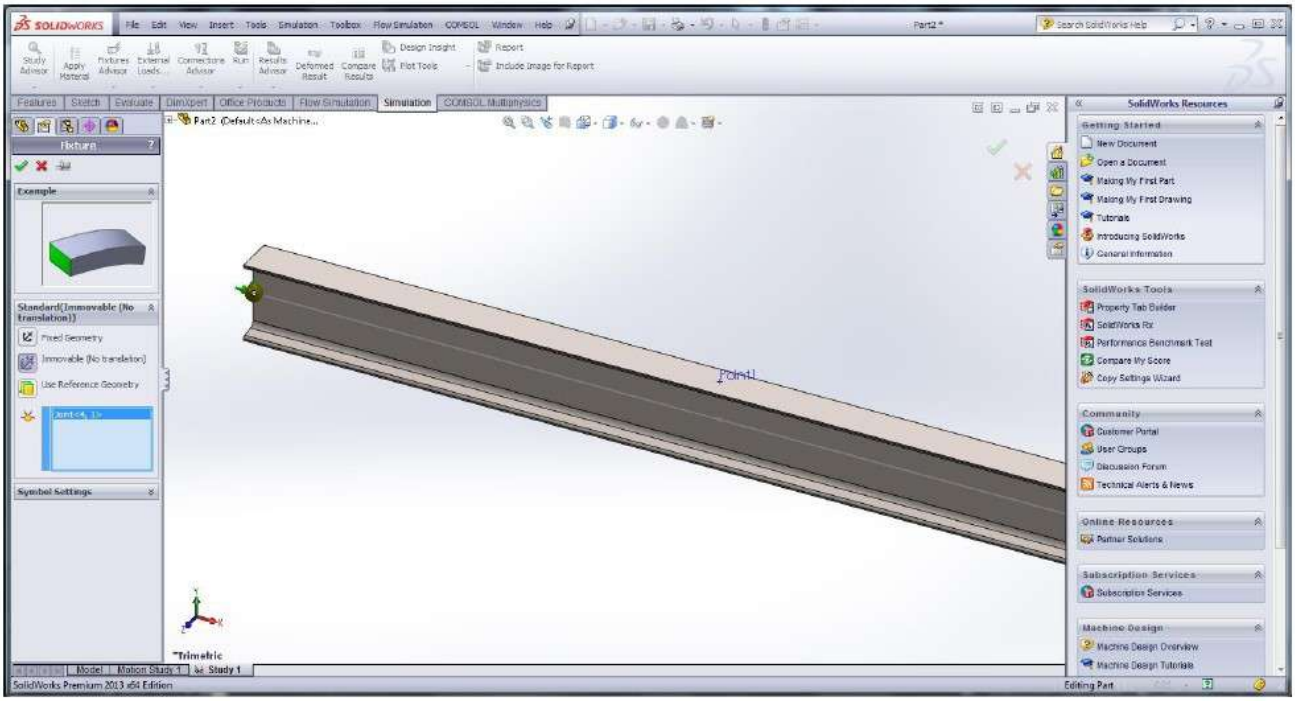


Figure (9)

⇨ Select fixed geometry once again
⇨ Select the right node
⇨ Select use reference geometry
⇨ Select the front plane as the reference for direction, as shown in figure (10)
⇨ Set the translations and rotations as shown in figure (11)
⇨ Click OK

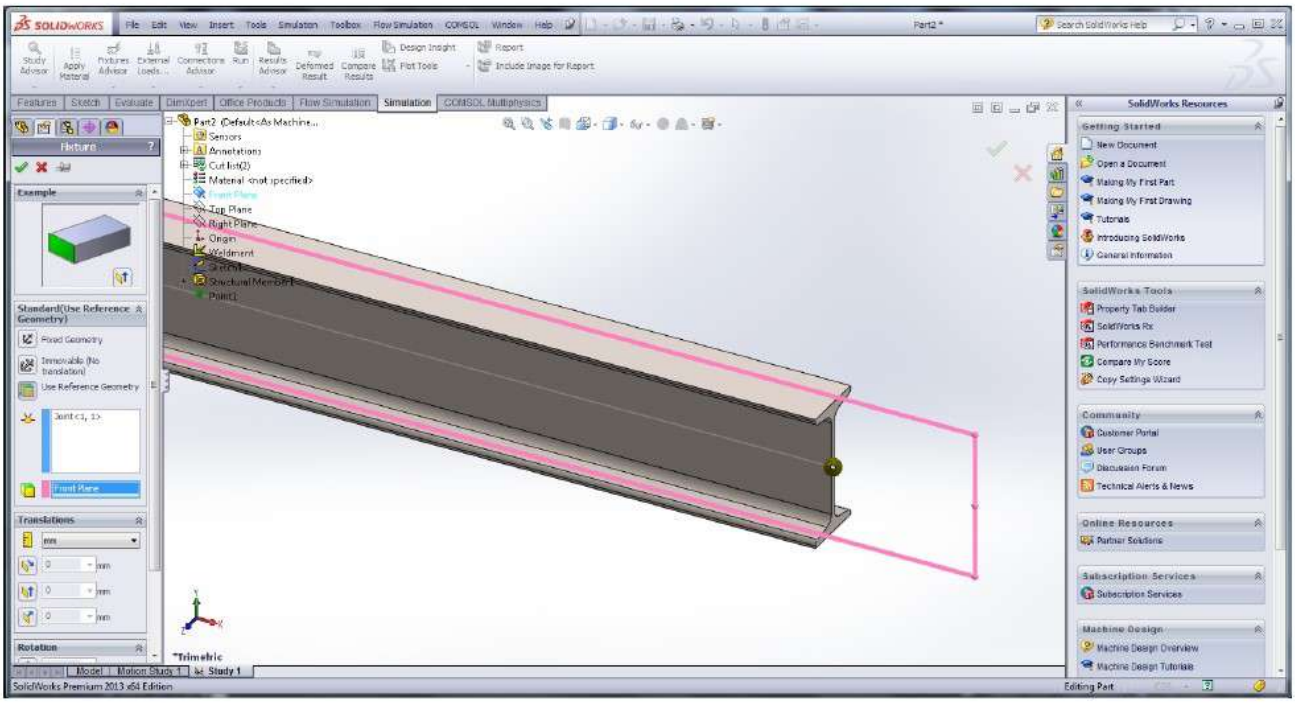


Figure (10)

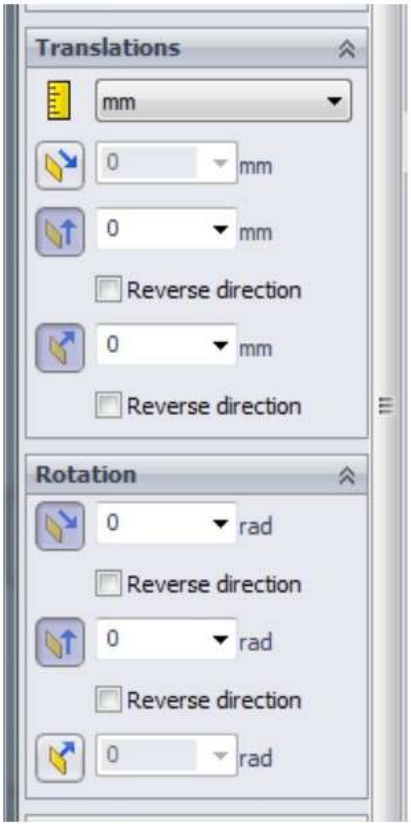



Figure (11)

The translations and the rotations in the directions who are values set to be Zero are not moving in that direction.

The other ones are free to move, which will be the point of freedom for this beam.

- ⇨ Right click on the external loads and select force
- ⇨ Select the point
- ⇨ Select the front plane as the reference direction
- ⇨ Select the vertical direction
- ⇨ Set the value as 224 N
- ⇨ Click OK, figure (12)

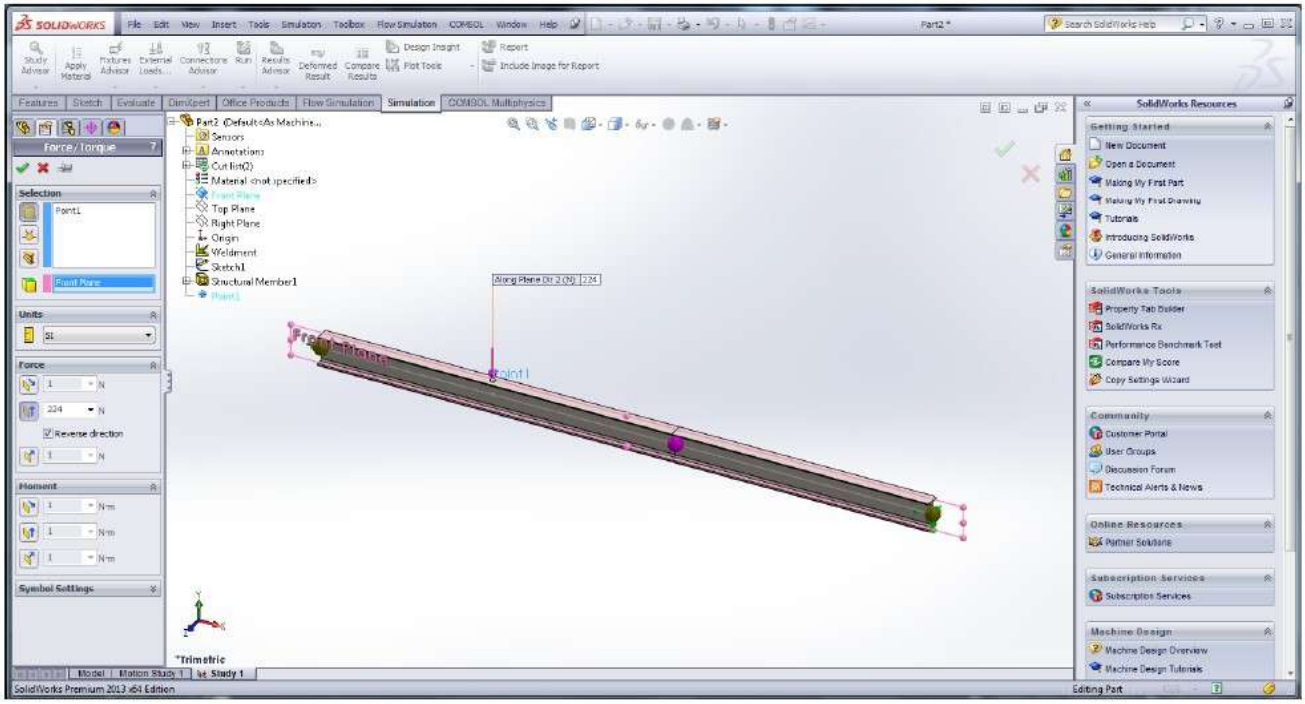


Figure (12)

- ⇨ Select force once again
- ⇨ Select Beams
- ⇨ Select the (90 cm) structure member
- ⇨ Select the front plane for the reference direction
- ⇨ On the unites click per unit length
- ⇨ Set the value on the vertical direction as (334 N)
- ⇨ Click OK, figure (13)

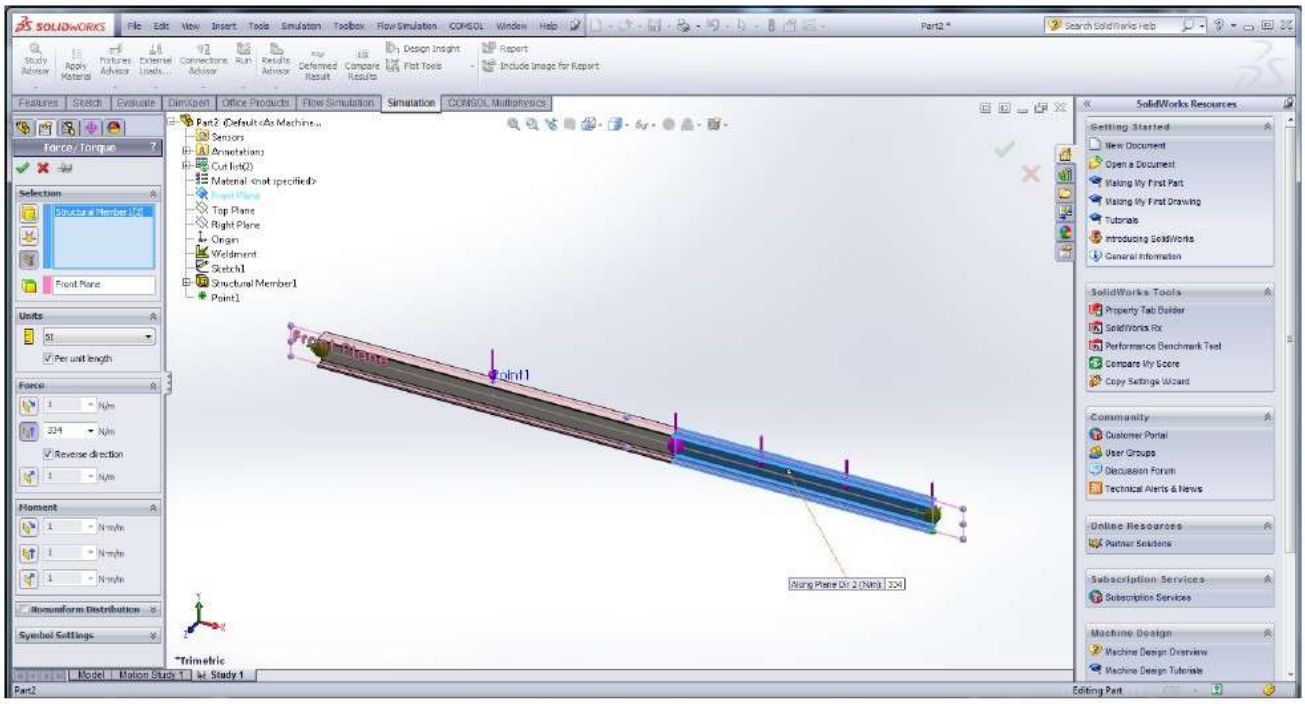


Figure (13)

- ⇨ Right click on mesh and select Mesh and Run

The stress should be as plotted in figure (14), and the bending values should be something similar depending on you mesh.

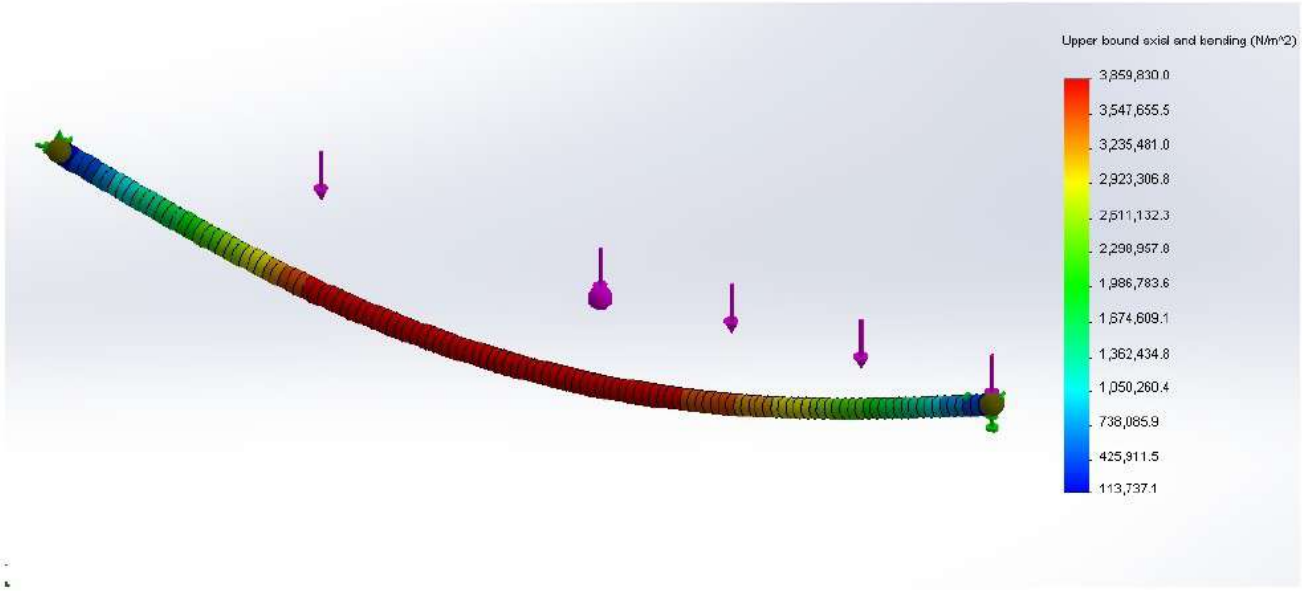



Figure (14)

- ⇨ Right click on the results and select define beam diagrams
- ⇨ Select shear force in direction 1 and click OK, figure (15)

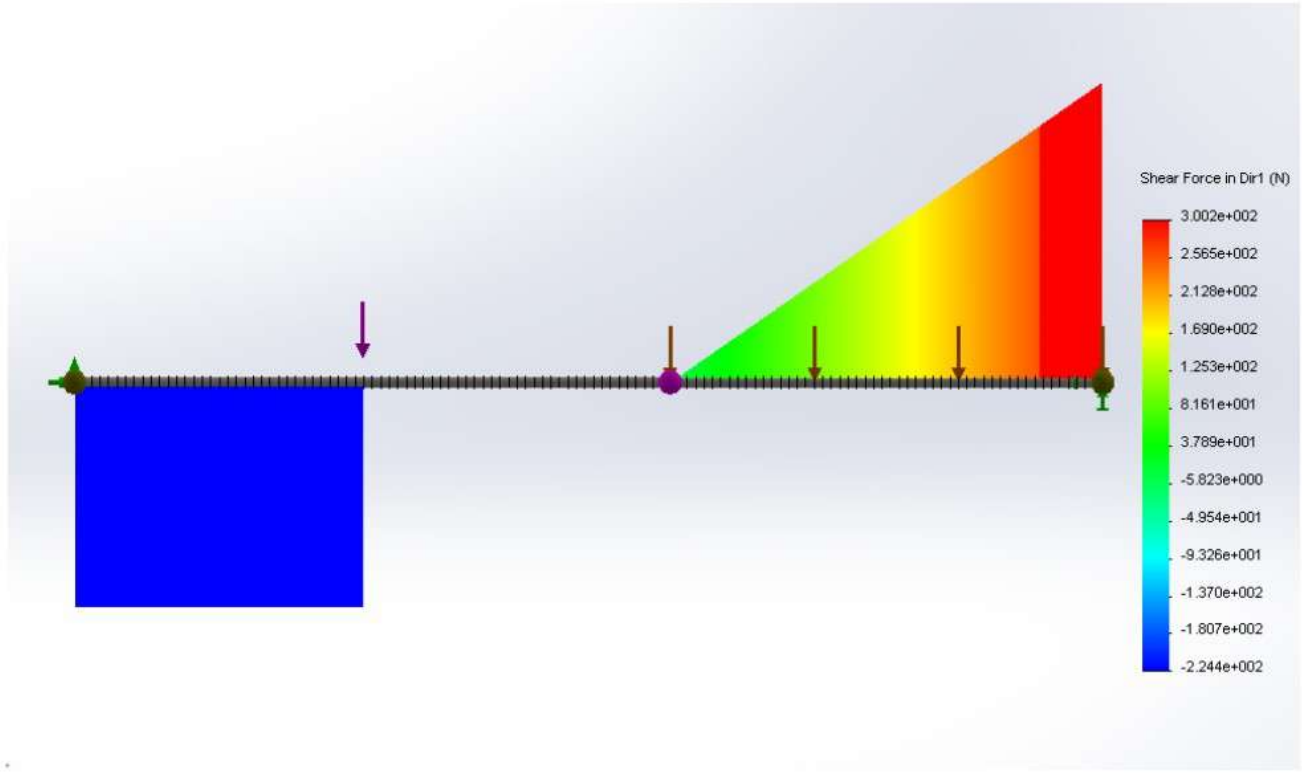



Figure (15)

- ⇨ Right click on the results and select define beam diagrams
- ⇨ Select momentum in direction 2 and click OK, figure (16)

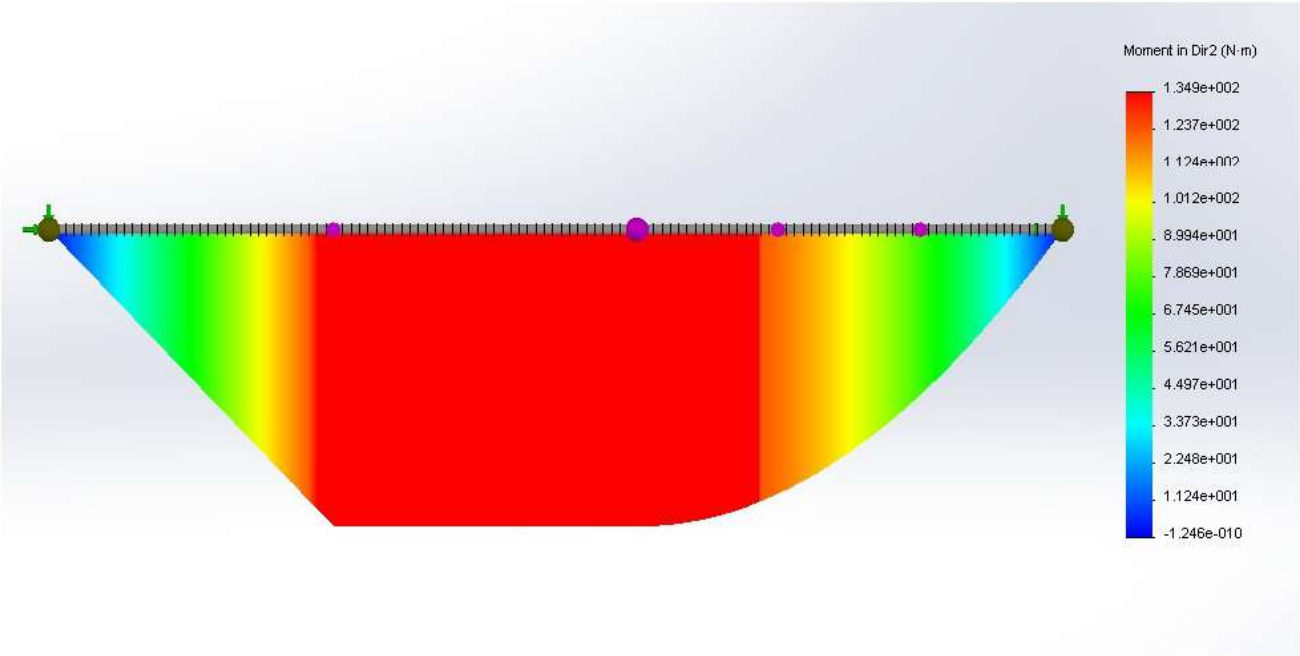
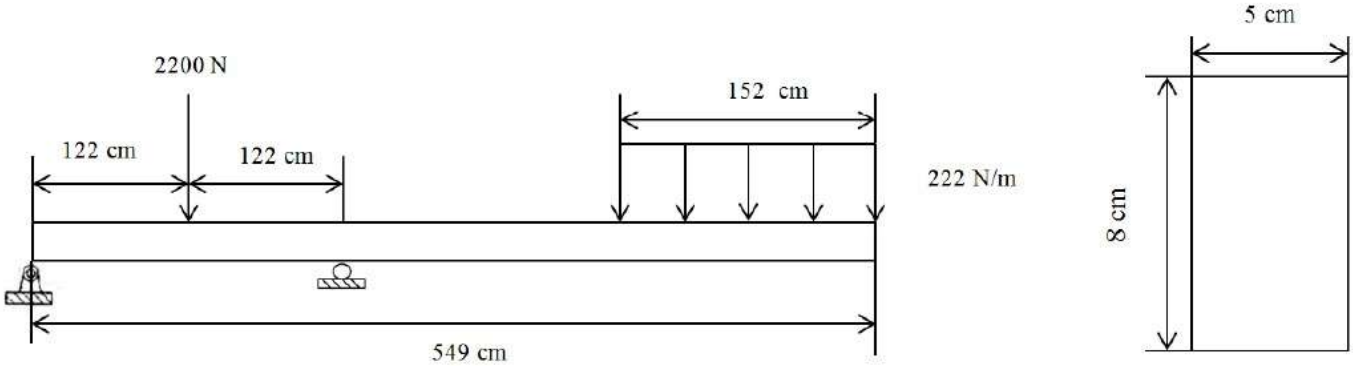


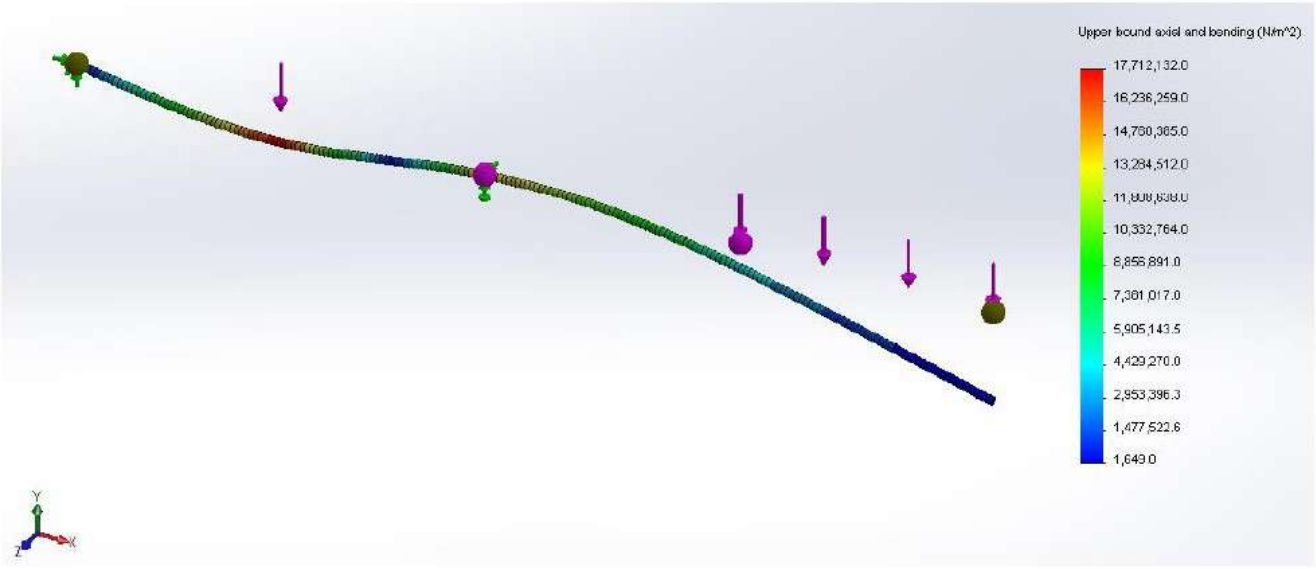

Figure (16)

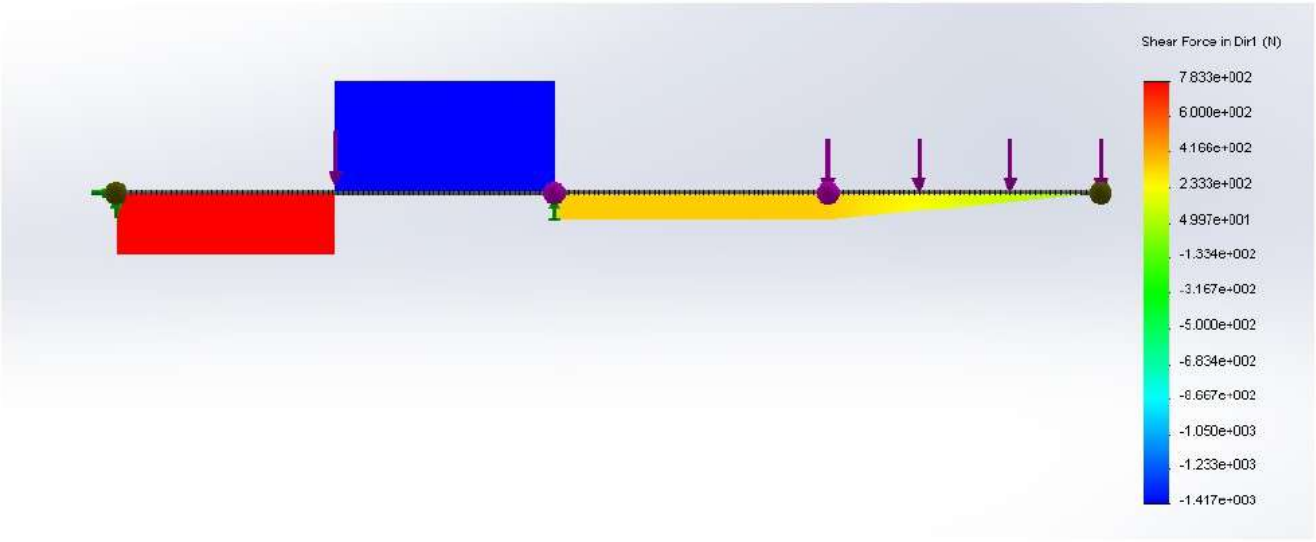
## HW

Get the force and the momentum diagrams for this beam




The results should be something like this


Upper bound axial and bending (N/m^2)
17,712,132.0
16,236,259.0
14,760,385.0
13,284,512.0
11,808,638.0
10,332,764.0
8,856,891.0
7,381,017.0
5,905,143.5
4,429,270.0
2,953,396.3
1,477,522.6
1,649.0


Shear Force in Dir1 (N)
7.833e+002
6.000e+002
4.166e+002
2.333e+002
4.997e+001
-1.334e+002
-3.167e+002
-5.000e+002
-6.834e+002
-8.667e+002
-1.050e+003
-1.233e+003
-1.417e+003