

Lecture 5

Finite Element Analysis

Finite element analysis will be performed for the Computer designs that were created in the previous lectures.

Finite element analysis procedures

Finite element analysis involves dividing the structure into a set of attached elements. Each element has a simple shape such as a line, a triangle or a rectangle, and is connected to other elements by sharing nodes.

The combination of CAD package with powerful FEA software is the base of any simulation work.

Steps to perform the analysis

- ✓ Geometry
- ✓ Study type
- ✓ Set the material
- ✓ Boundary conditions
- ✓ Mesh
- ✓ Run
- ✓ View the results

The stress, strain, and displacement will be calculated for the steel structure shown in figure (1)

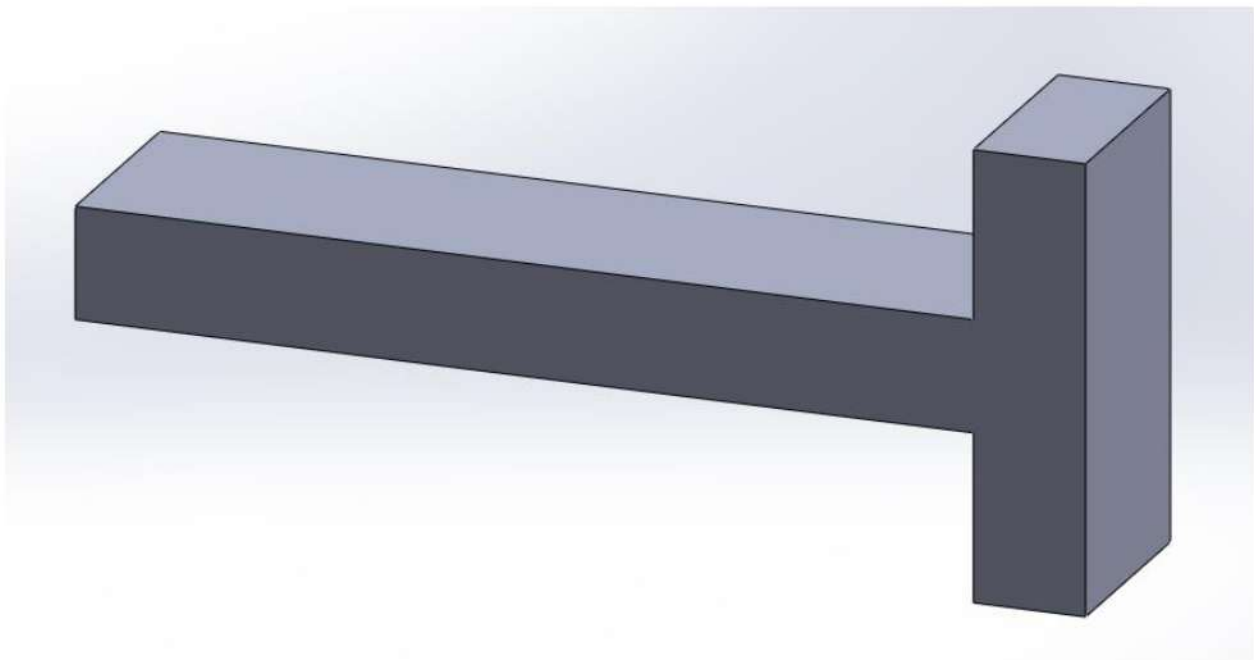


Figure (1)

⇒ Sketch the geometry as shown in figure (2)

⇒ Select extrude from features toolbar and extend for (20 mm)

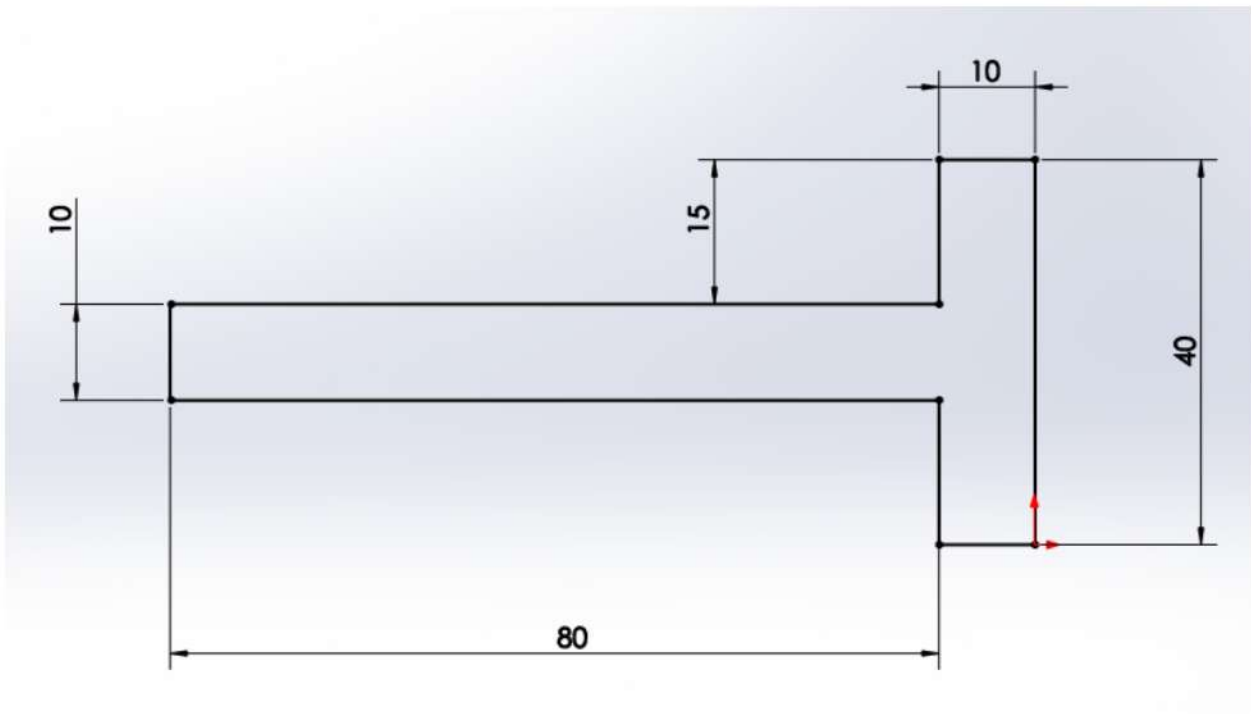


Figure (2)

⇒ From Simulation select New study as shown in figure (3)

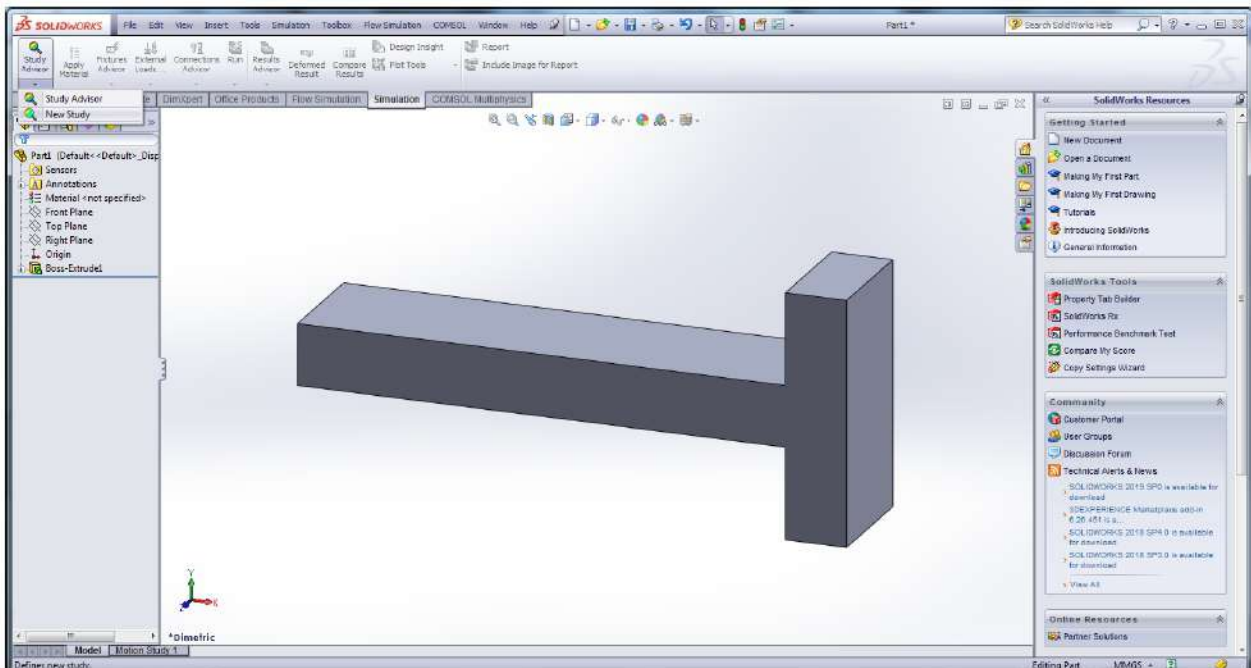


Figure (3)

⇒ Select Static and click Ok

⇒ Right click on the part and select Apply/Edit material, as shown in figure (4)

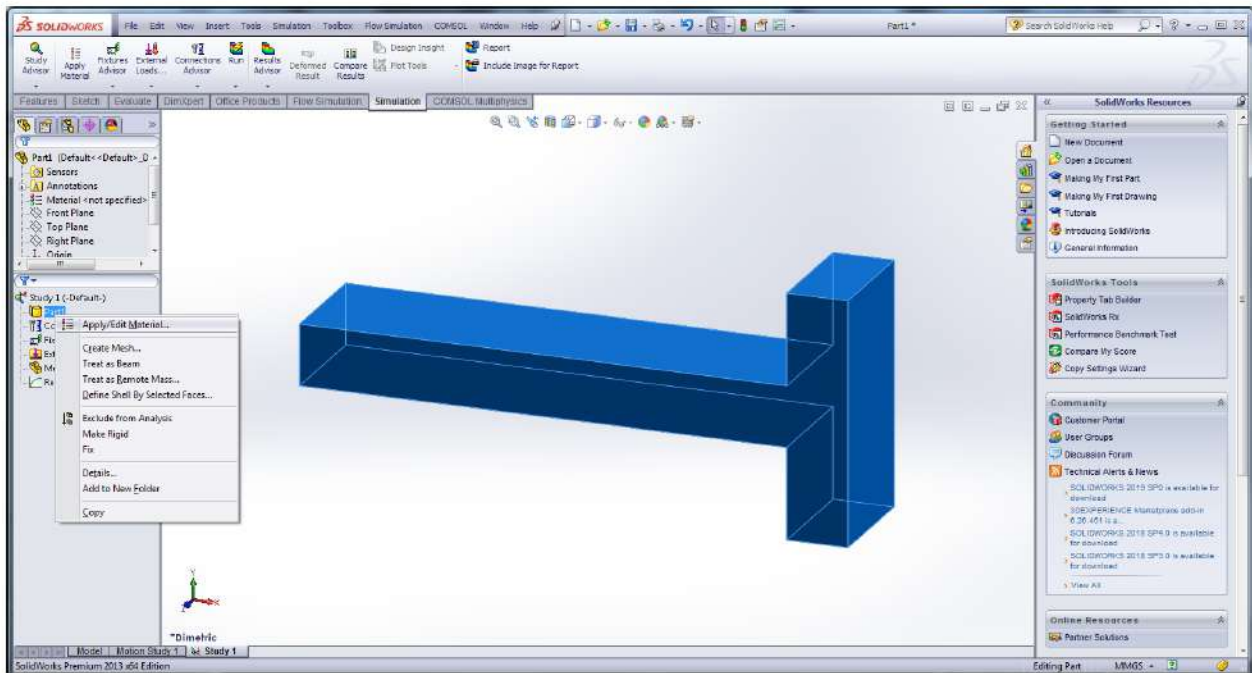


Figure (4)

- ⇒ Select Alloy Steel and click Apply, then close the dialog box
- ⇒ Right click Figures and select Fixed Geometry
- ⇒ Set the selection as shown in figure (5)
- ⇒ Click Ok

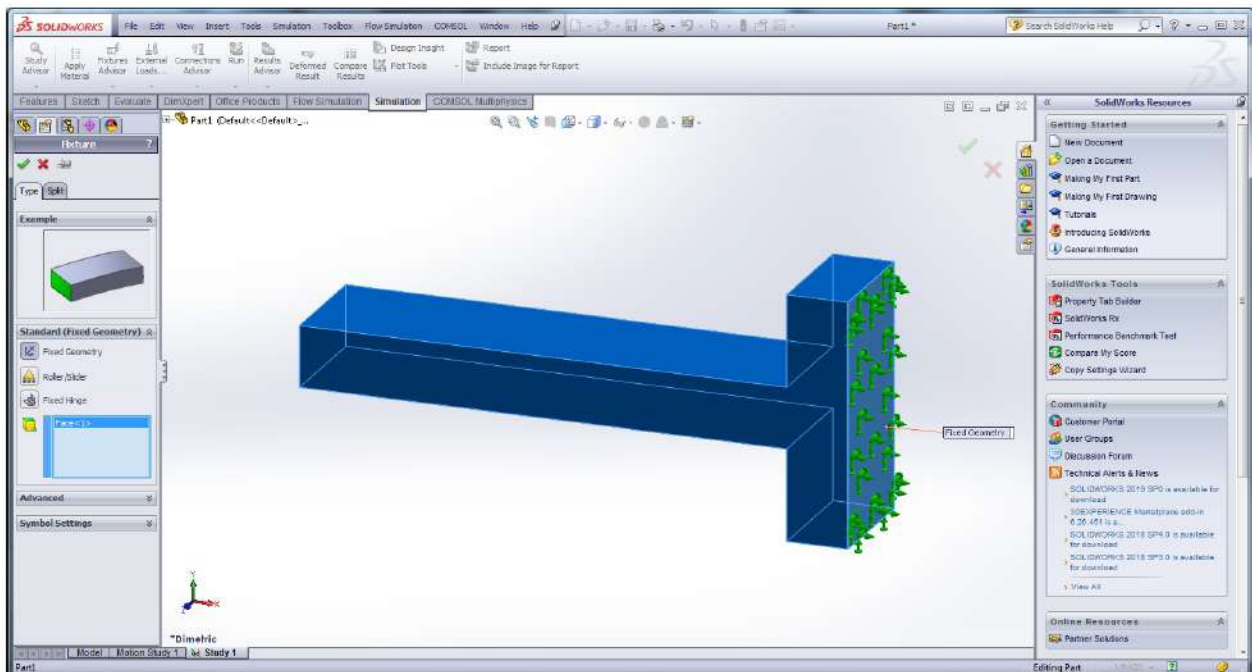


Figure (5)

- ⇒ Right click on External loads and select Force
- ⇒ Select the top left edge
- ⇒ Click select direction
- ⇒ Select the front vertical edge, this will set the direction of the force
- ⇒ Your window should be as figure (6)

- ⇒ Set the force as (100 N)
- ⇒ Click OK

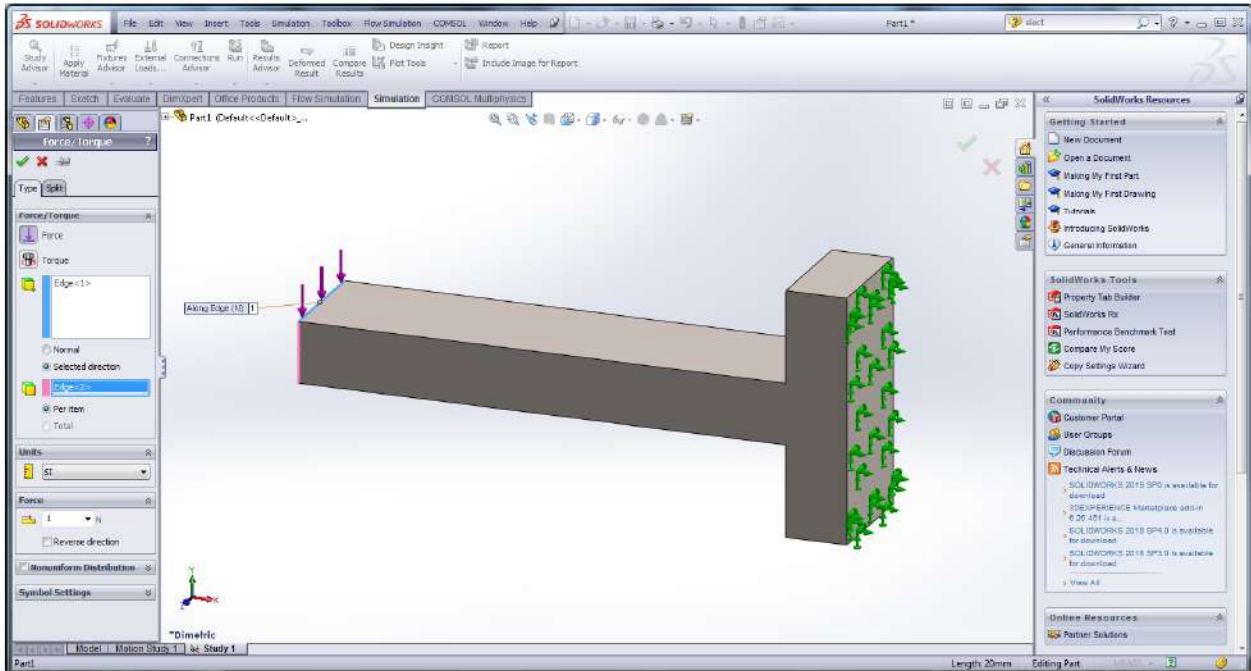


Figure (6)

- ⇒ Right click on Mesh and select create mesh
- ⇒ Keep the mesh density and click OK, figure (7) is an example of how the result should be.

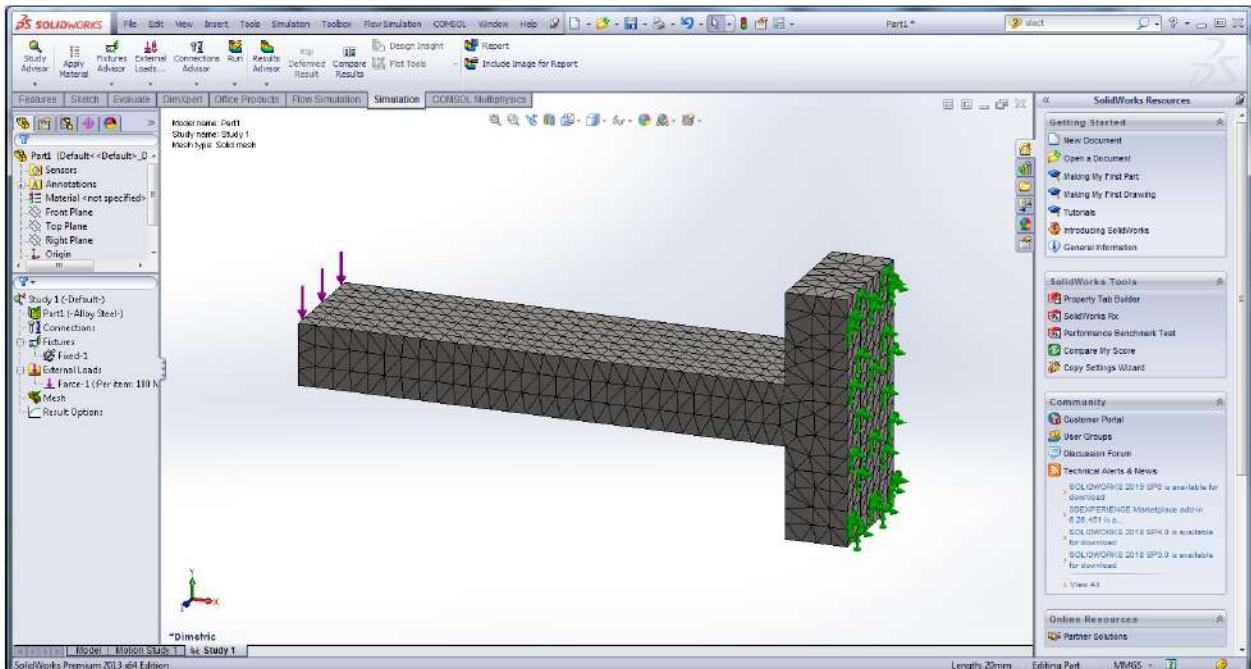


Figure (7)

- ⇒ Right click on the study name and select Run as shown in figure (8)

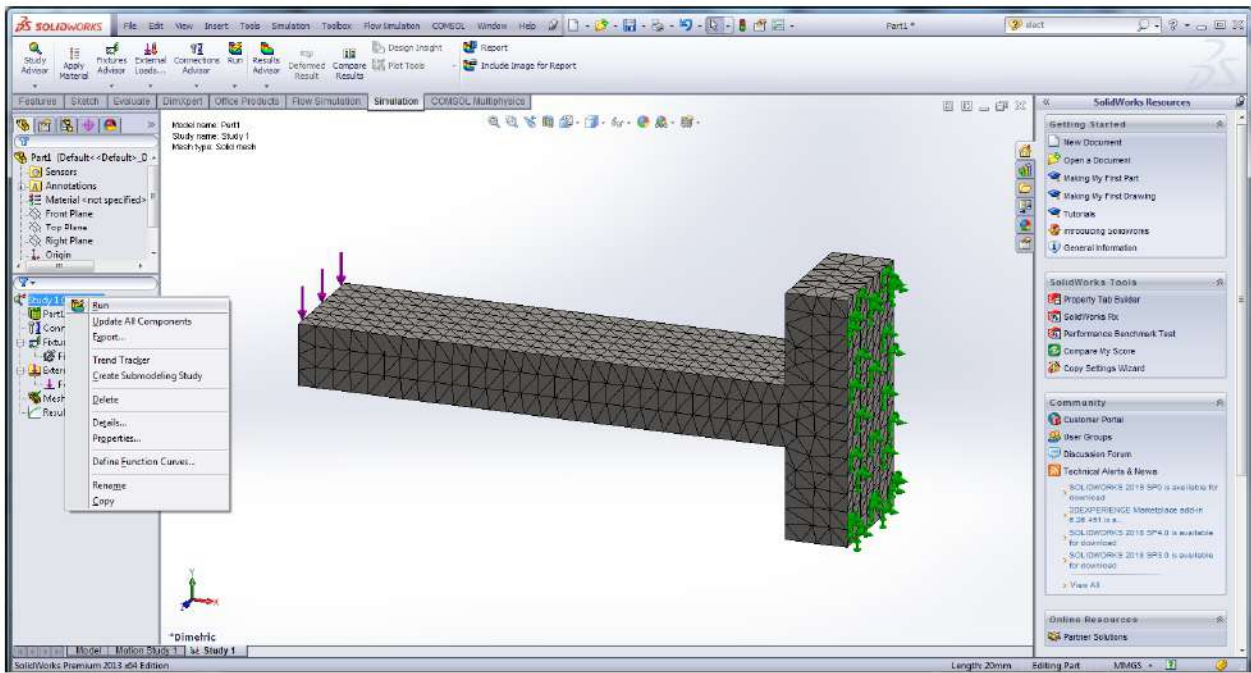


Figure (8)

Figure (9) is an example for the stress results

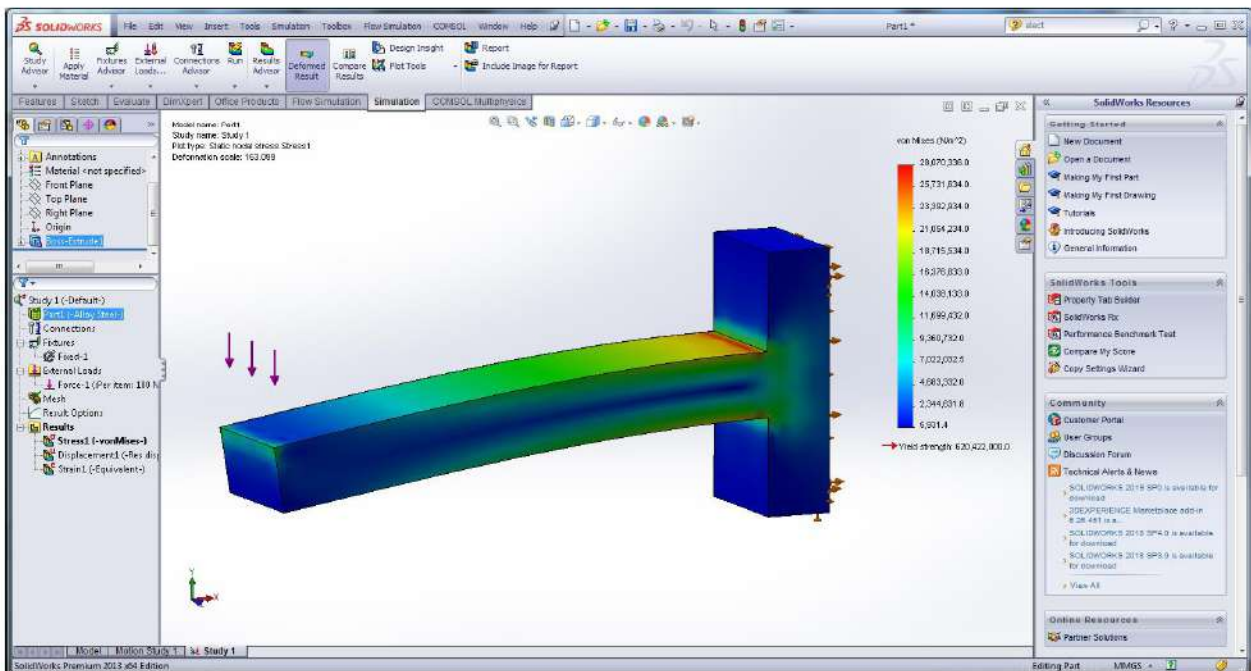


Figure (9)

⇒ Right click on the stress and select Animate as shown in figure (10)

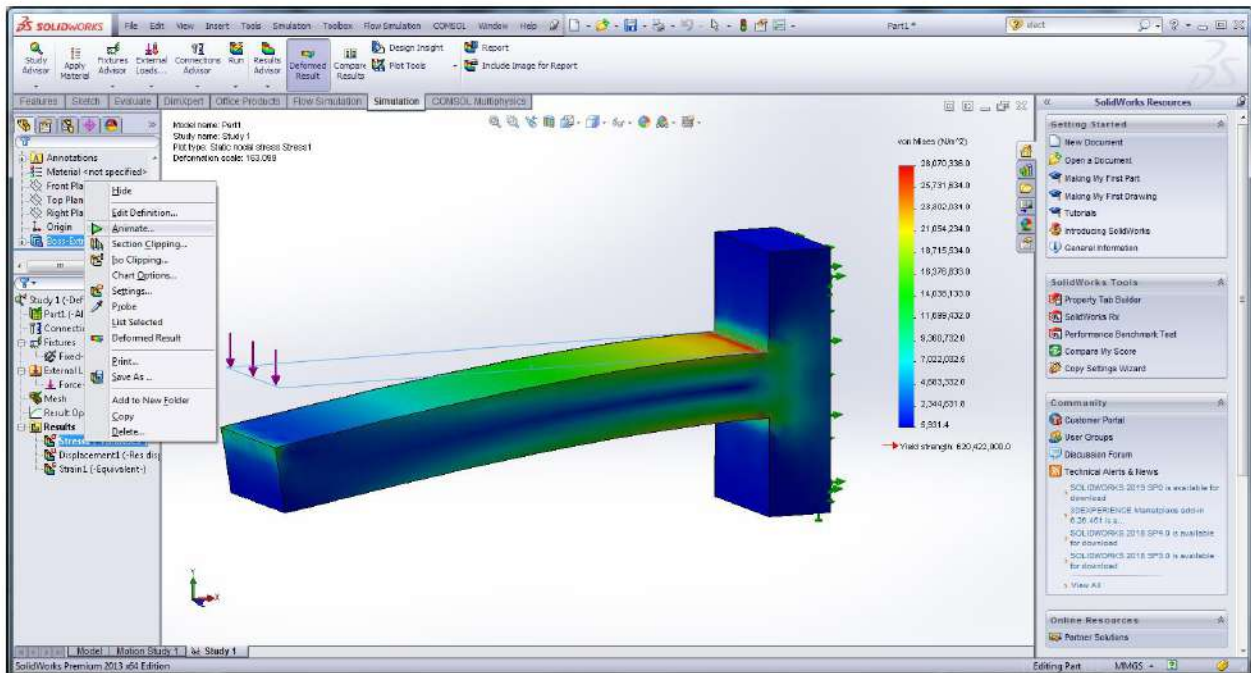
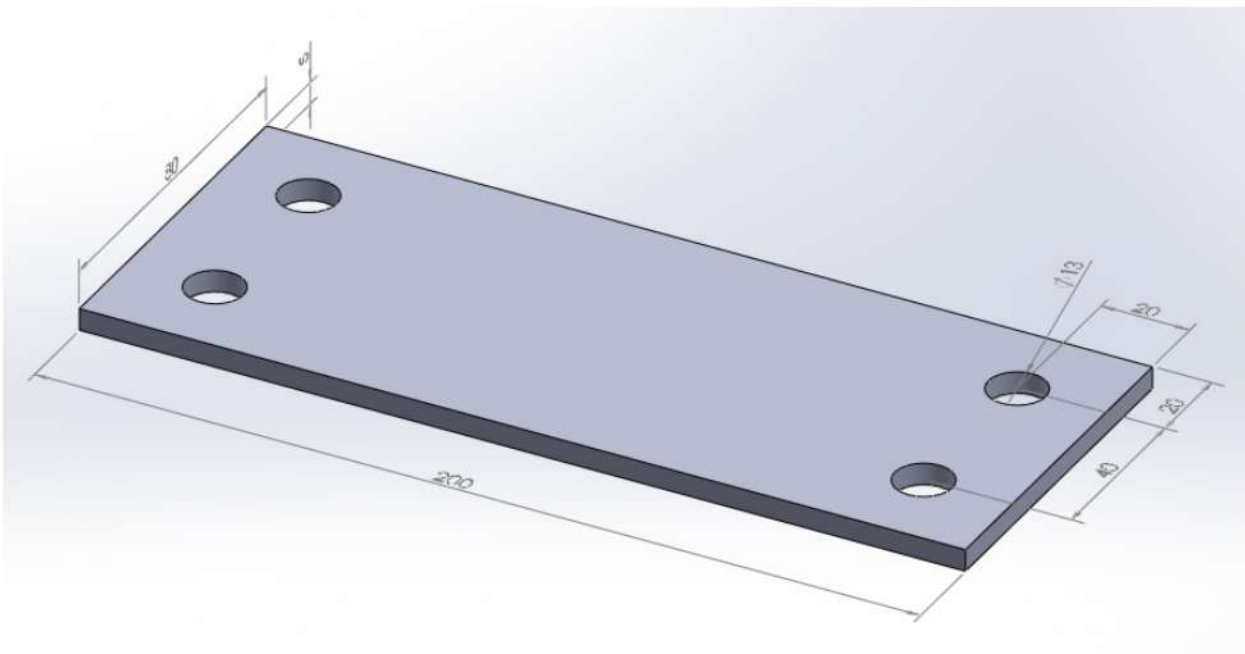


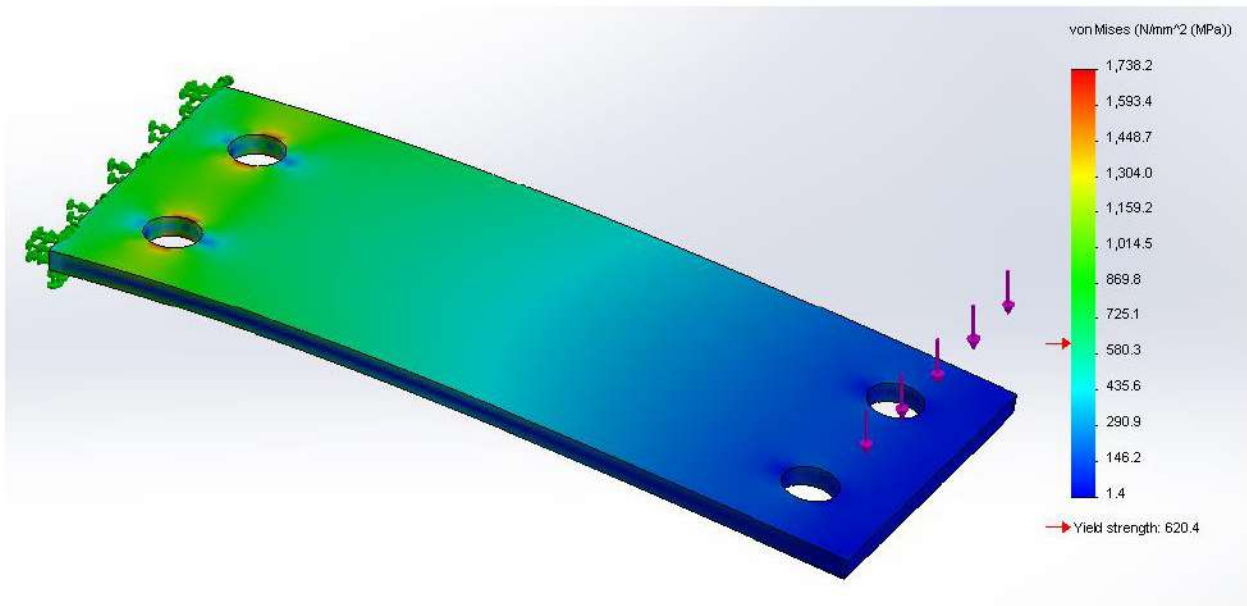
Figure (10)

Note you can stop to change the number of frames and the speed

HW

Get the stress plot for the geometry given as shown.





The force is 1500 N