

ME345: Modeling and Simulation

Analysis of I-cantilever beam by using SimulationXpress

Stevens Institute of Technology

Professor Frank Fisher

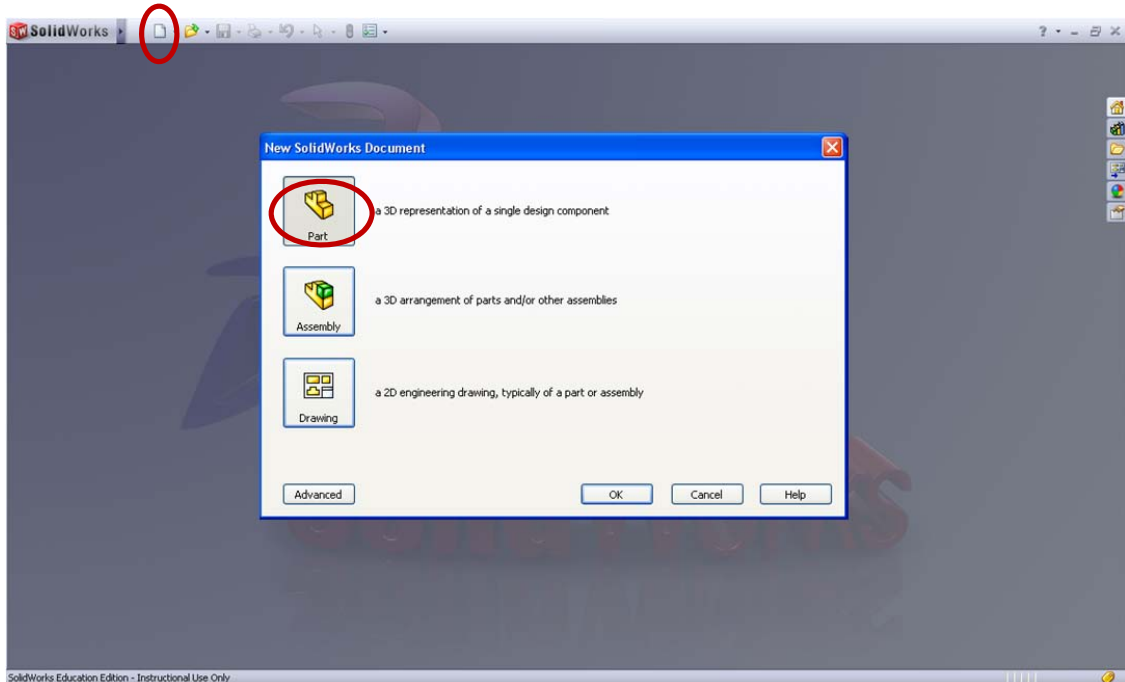
Last updated: July 5th, 2011

Written by Lin Dong

Software versions used in the tutorial: SolidWorks 2011

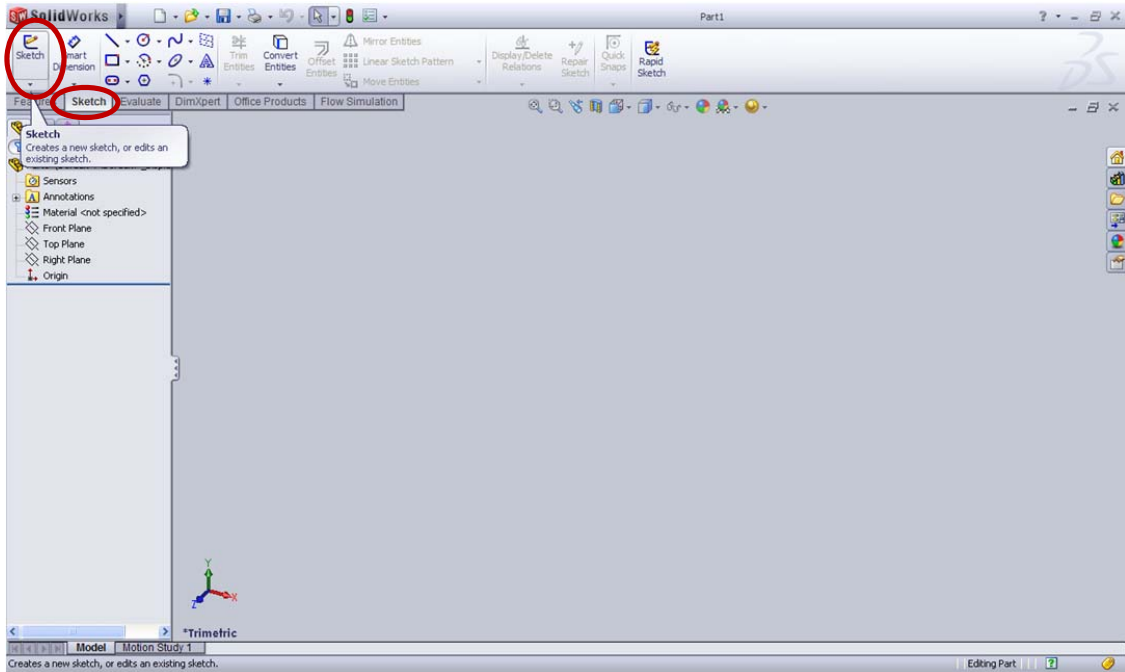
1. Start SolidWorks

Click on the **New** tab on the top left corner of the screen and then create a new **Part**.

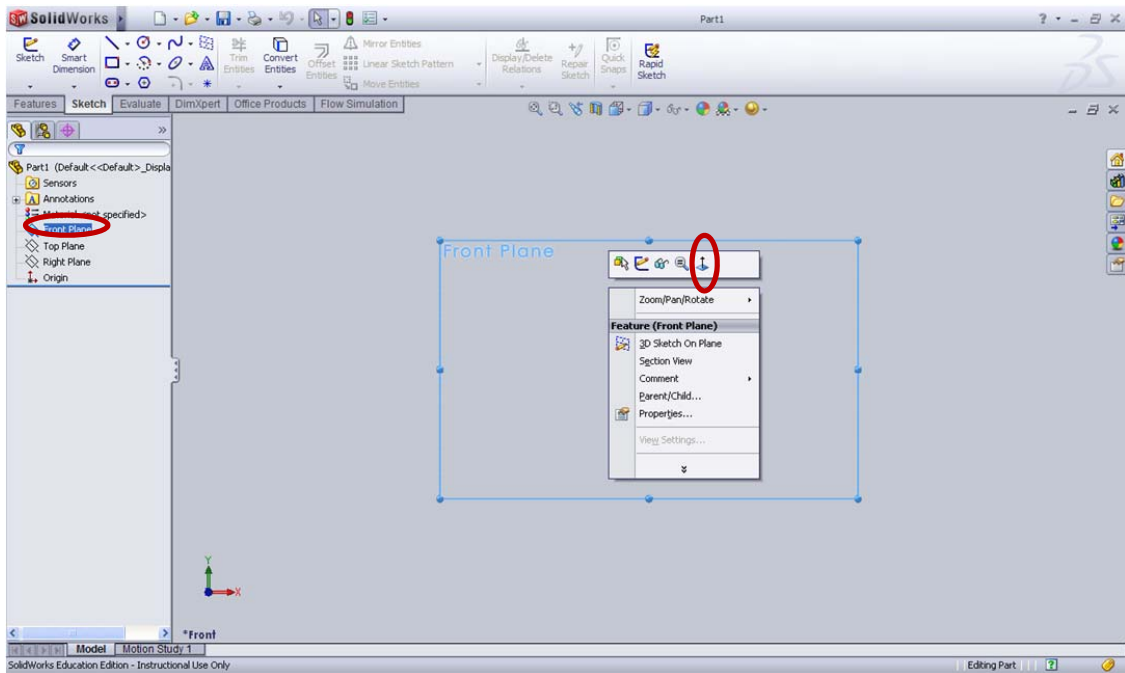


2. Create a sketch

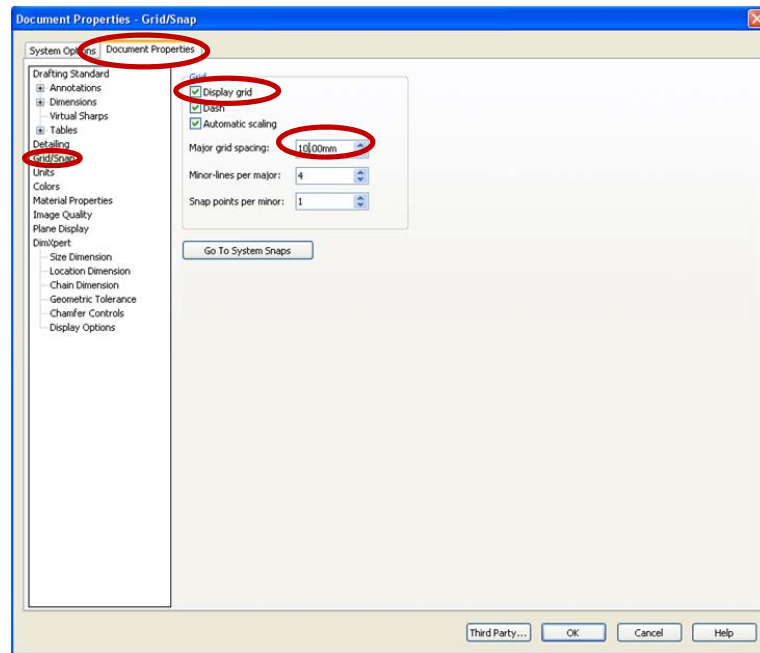
Click on the **Sketch** command and **Sketch** tab to start.



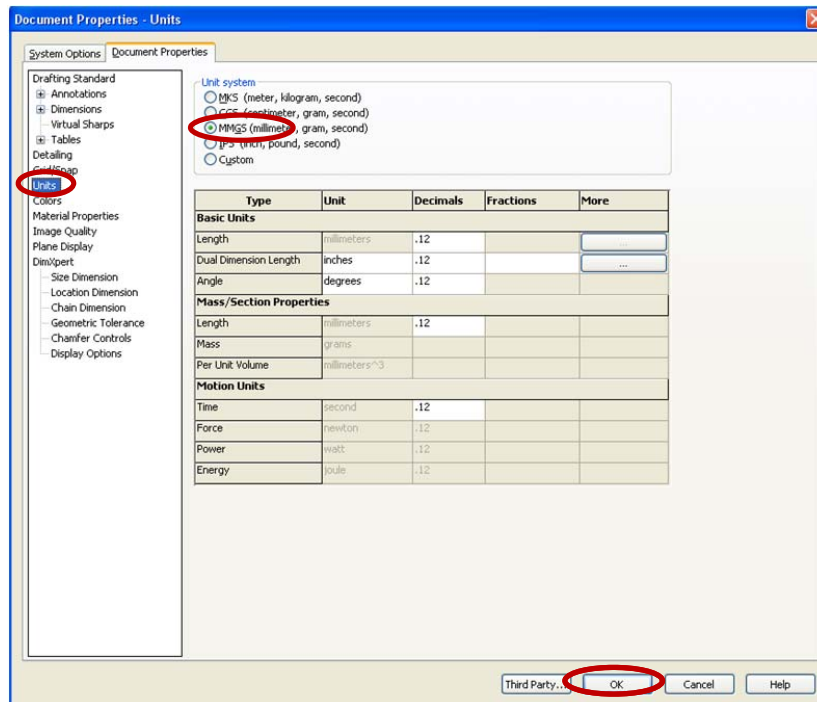
Select the **Front Plane** as the sketch plane and then it automatically goes to orient a model normal to the nearest global XYZ coordinates. If not, right click to select **Normal to** command.



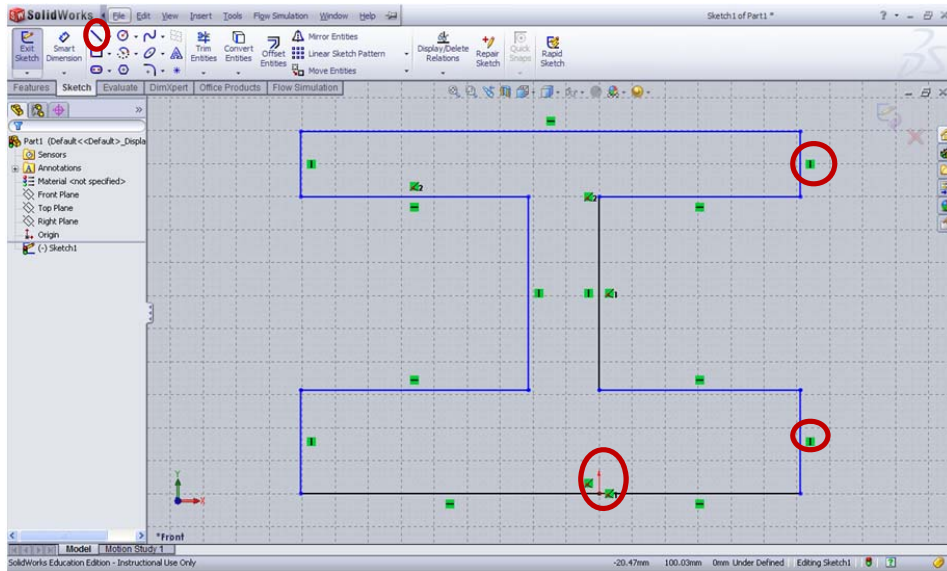
To display a sketch grid in an active sketch and set options for the grid display and snap functionality, go to the **Tools > Options > Document Properties > Grid/Snap > Check Display Grid > change grid spacing to 10mm**




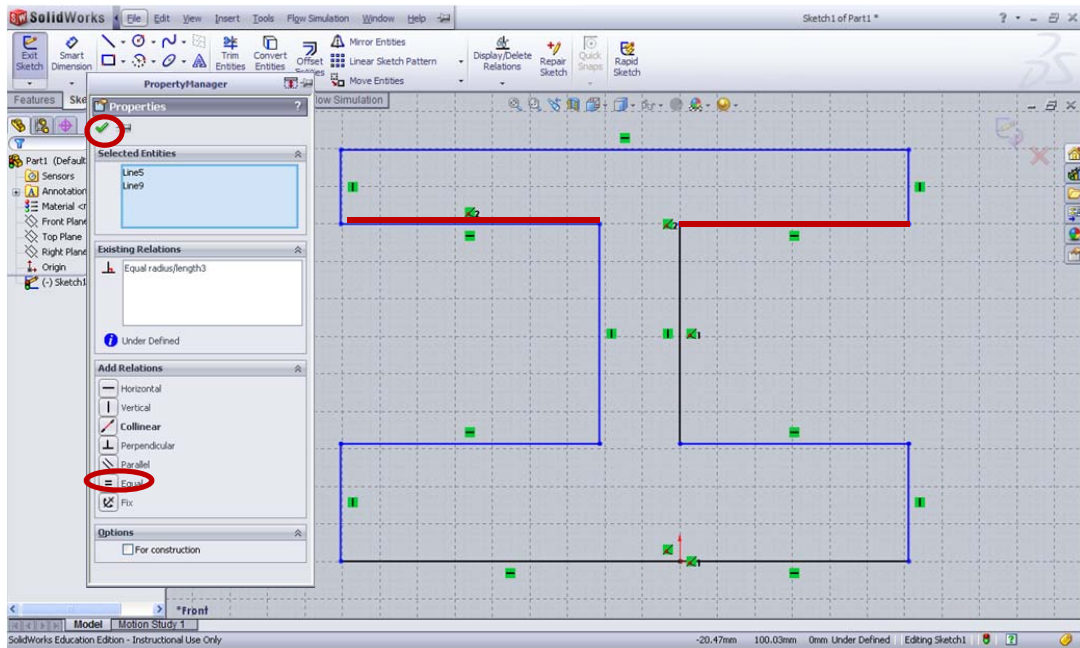
If the units are not in MMGS automatically, go to **Units > check MMGS**



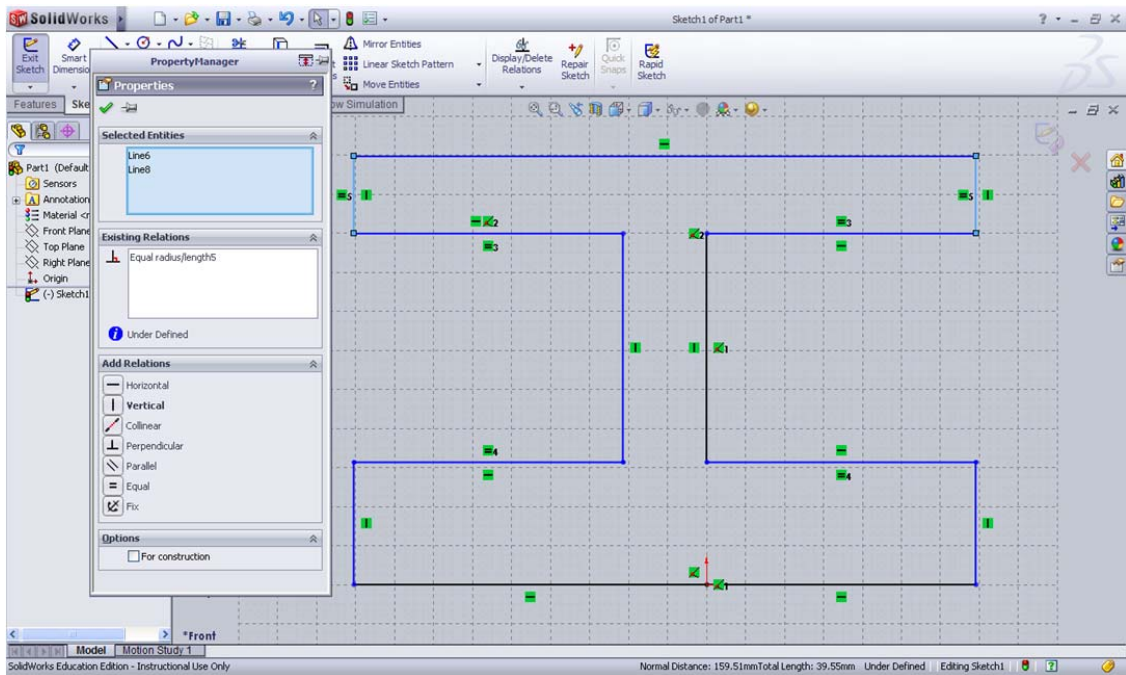
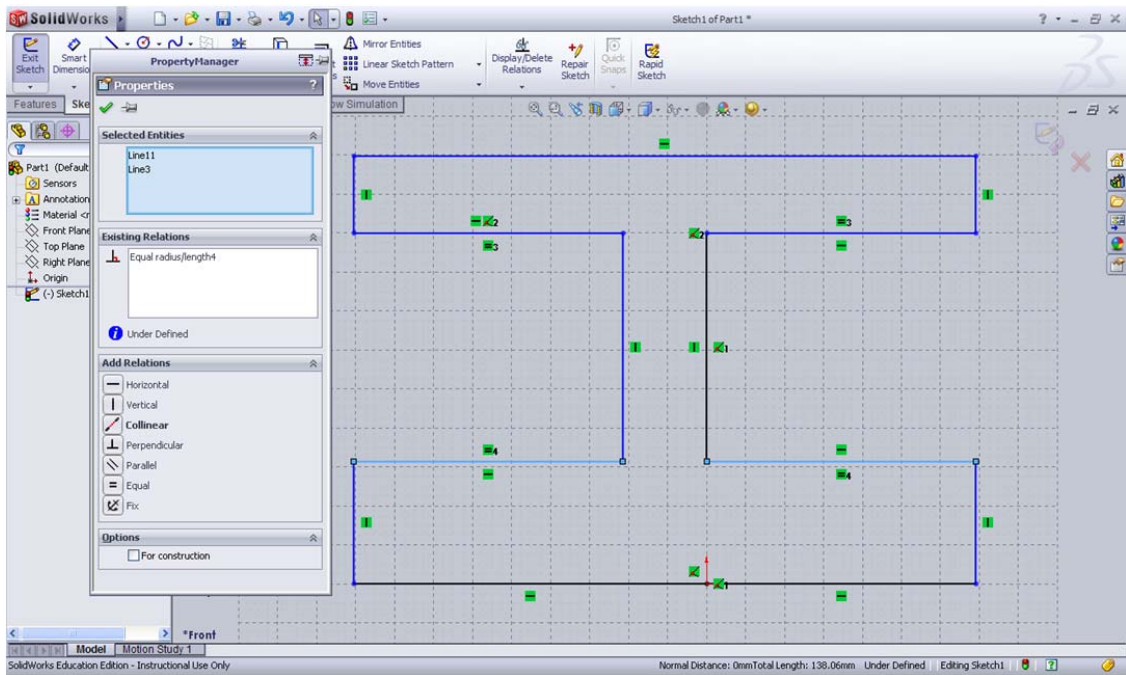
Complete the following sketch by clicking on the **Line** command and starting at the **Origin**. Do not worry about the lengths of the structure, they can be defined later, but make sure the lines are horizontal or vertical. It can be implied by a green square with the line inside.

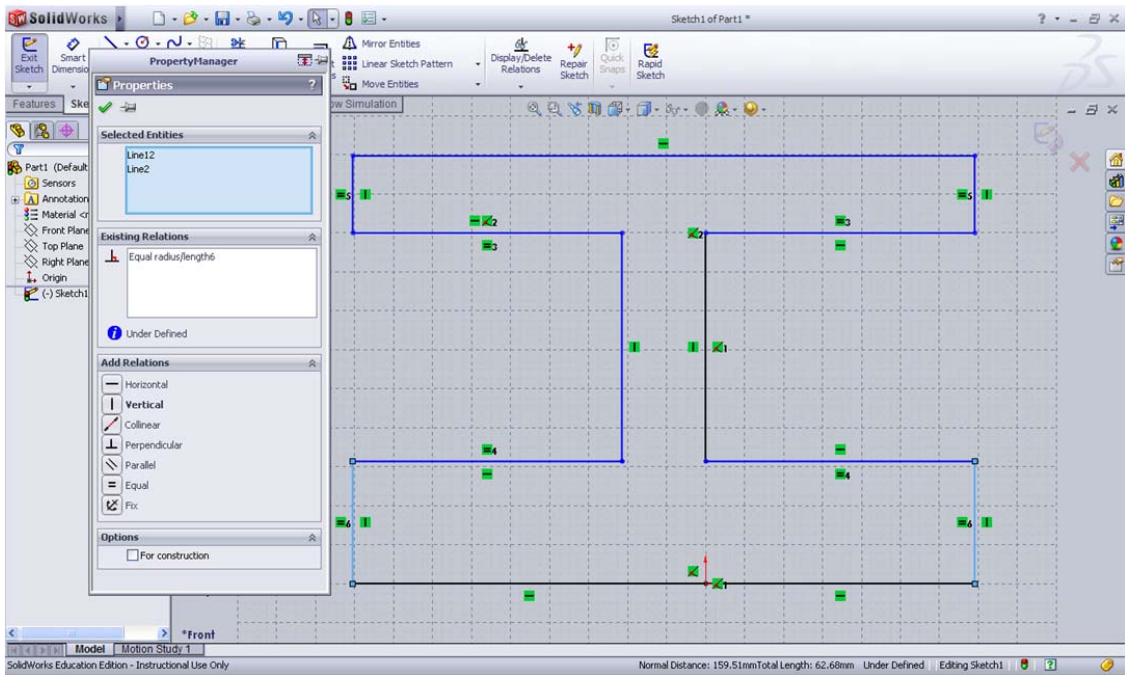


In order to make the I-cantilever beam symmetric, add relation of the lines. Click **Shift** and selected two lines > Add relations of **Equal** > Click  to close



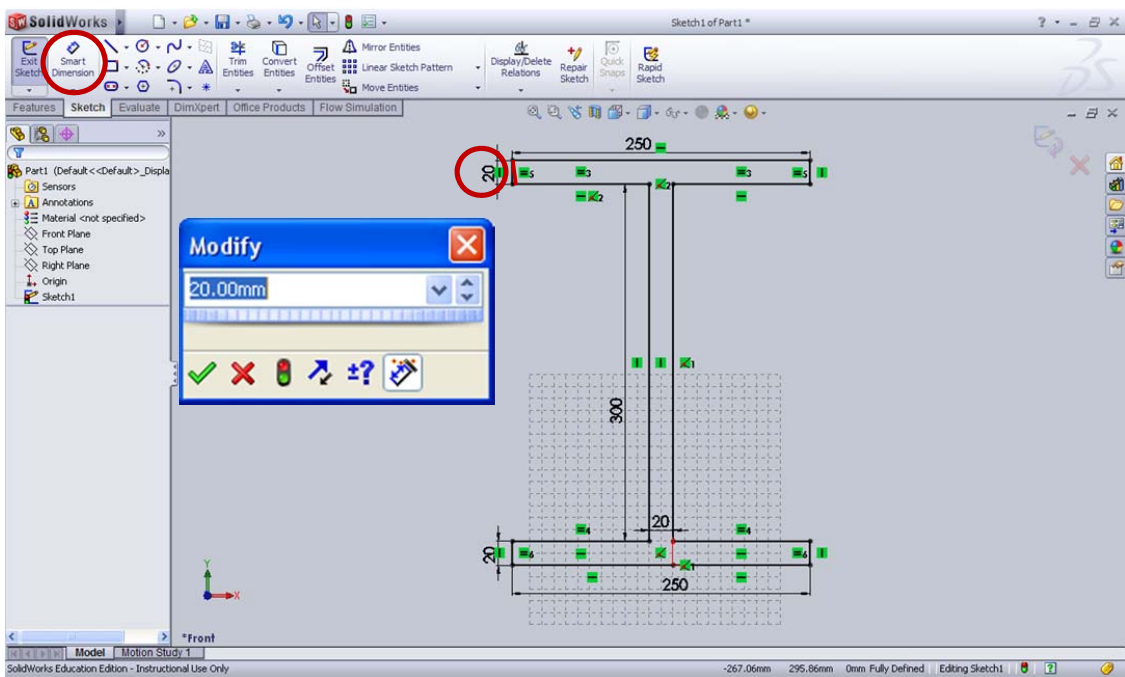
Repeat the same procedure to define the **equal** relation of the other three groups of lines.





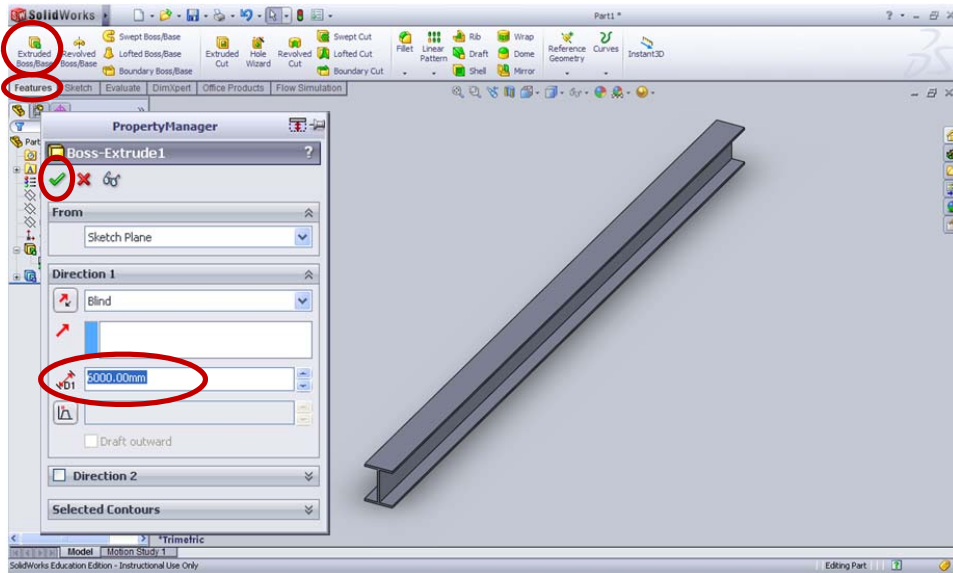
Then go to smart dimensions to define the length of the structure. Click on **Smart Dimension** >

Select the sketch of the line > edit the value



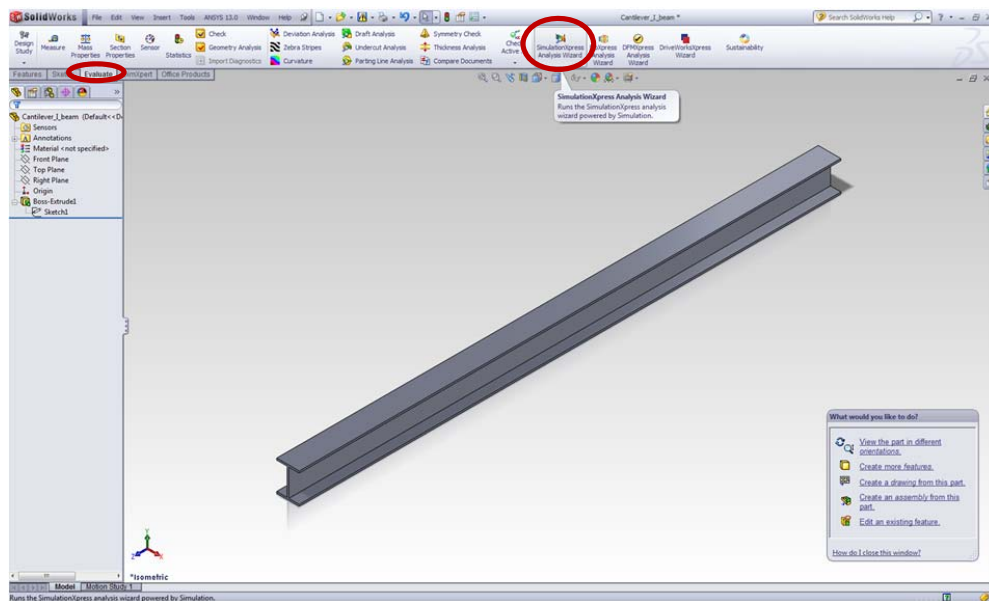
3. Extrude the sketch to create a solid in 3D dimension

Click on the **Features** Tab > **Extruded Boss/ Base**. In the **PropertyManager** window, change the dimension of **D1** to 6000.00mm.

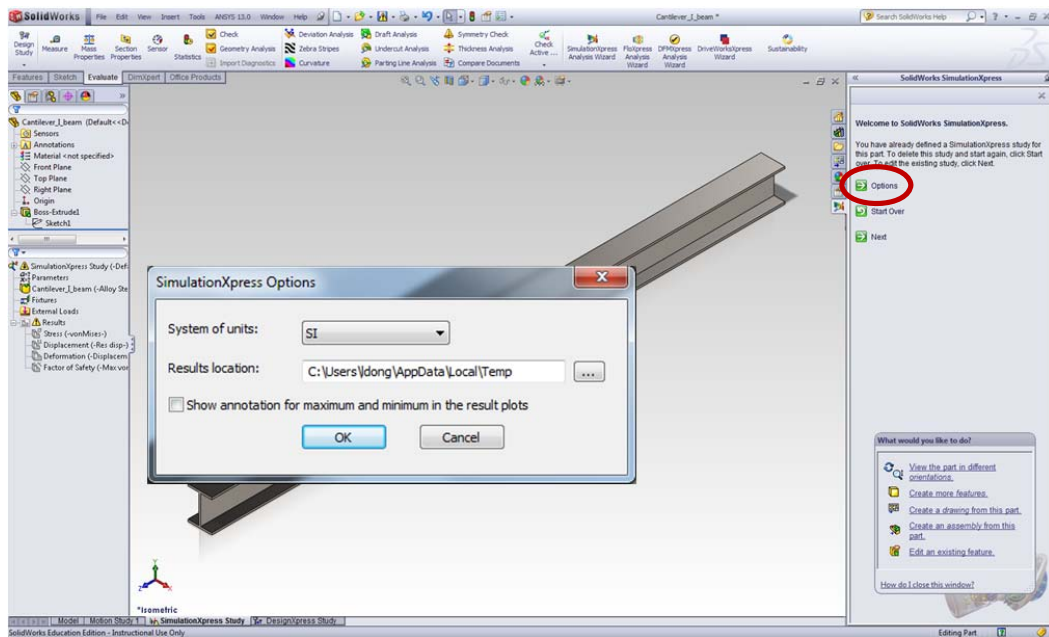


4. Start SimulationXpress

Go to Tools, add-ins and remove **Simulation** from the checked list. Then go to **Evaluate** Tab and click on **SimulationXpress** Command.

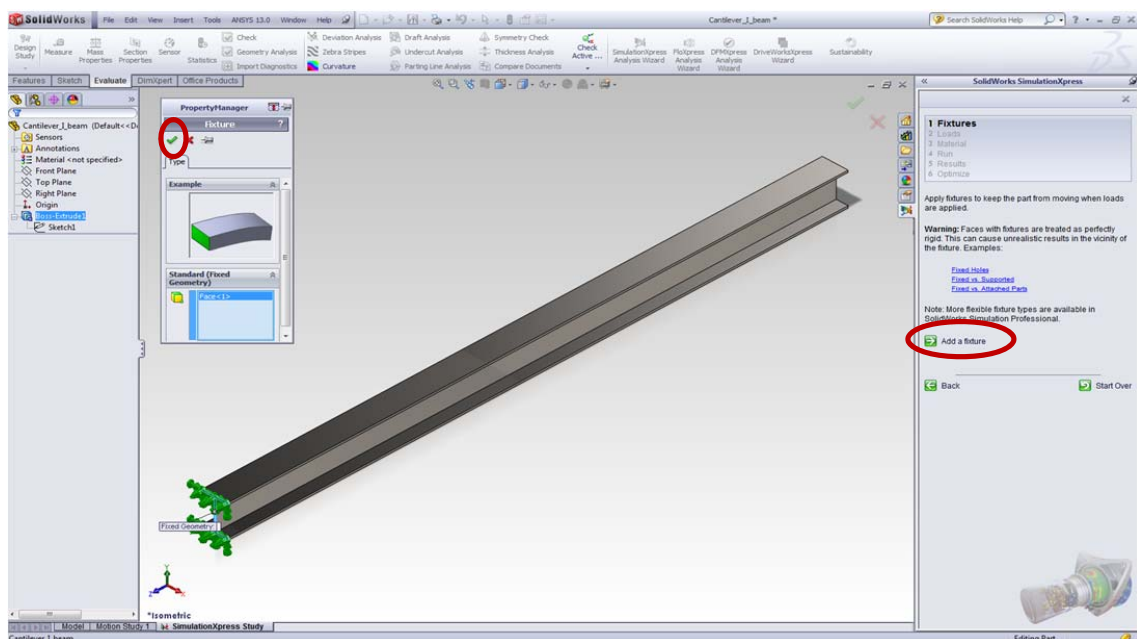


Click on the **Options** to change the **units** to SI.

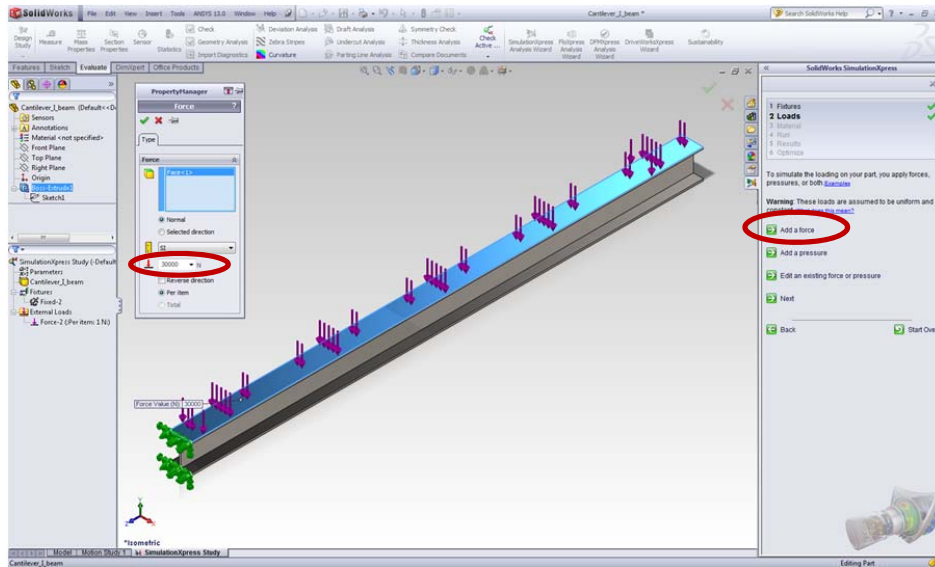


5. Run simulations

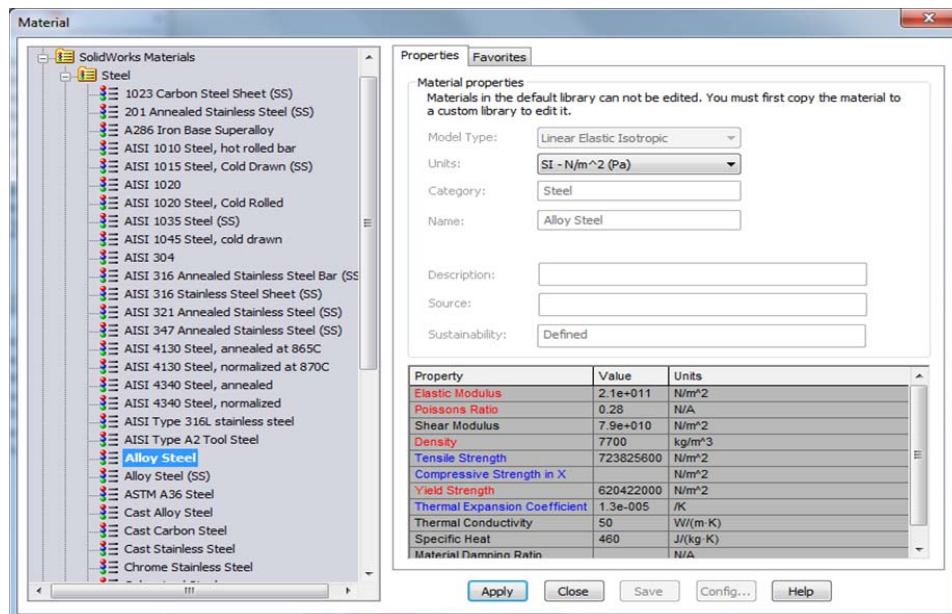
First, click on **Next** to **Add a fixture**. Select the I-shaped face to restrain the face from moving then you will see a green fixture there.



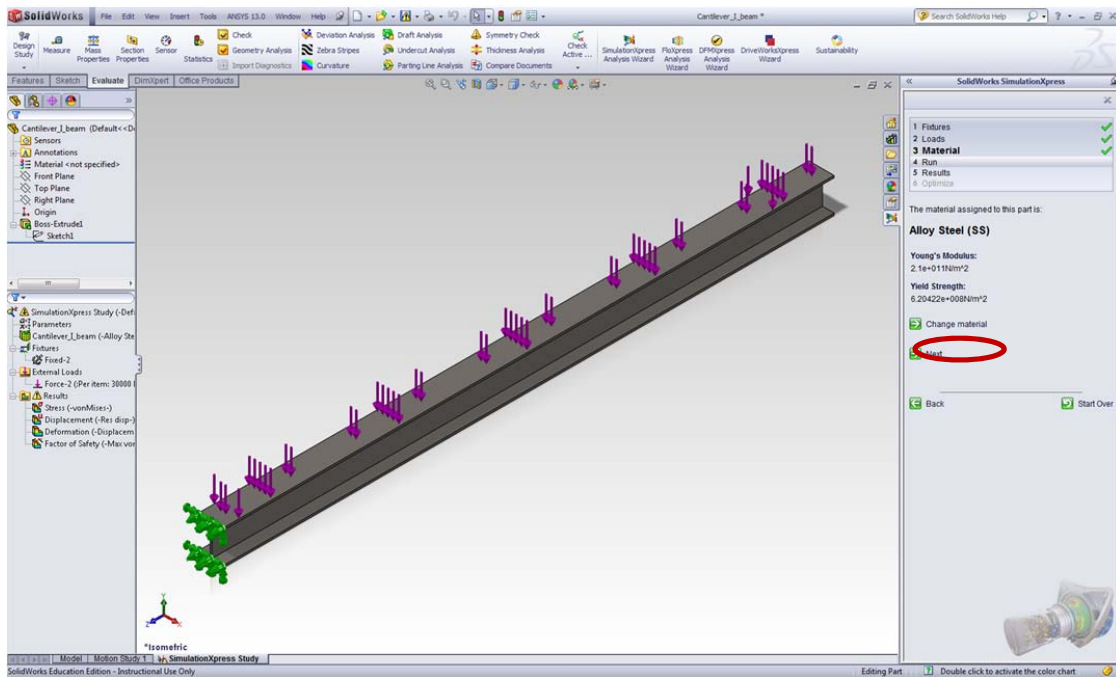
Second, click **Next** to define the **Loads**. In this tutorial, we will define a force on the top face of the beam. Click on **Add a force**, then select the top face, the pink force will appear on the solid. Then add the value of 30000N to the beam, this distributed load is applied normal to the face and over the whole surface of the beam.



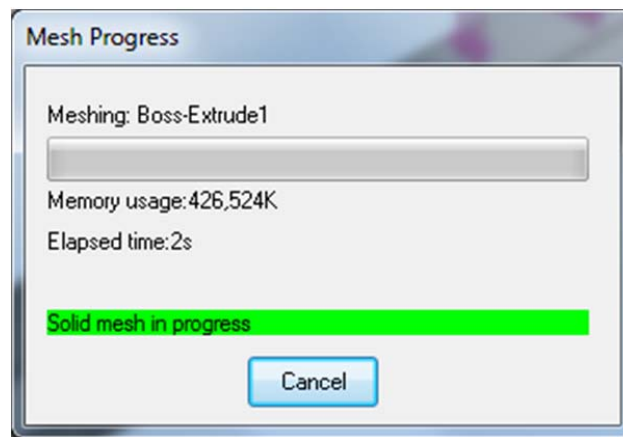
Third, click **Next** to apply the **Material**. Choose the material of Alloy Steel from the library. Click on Apply then close this window.



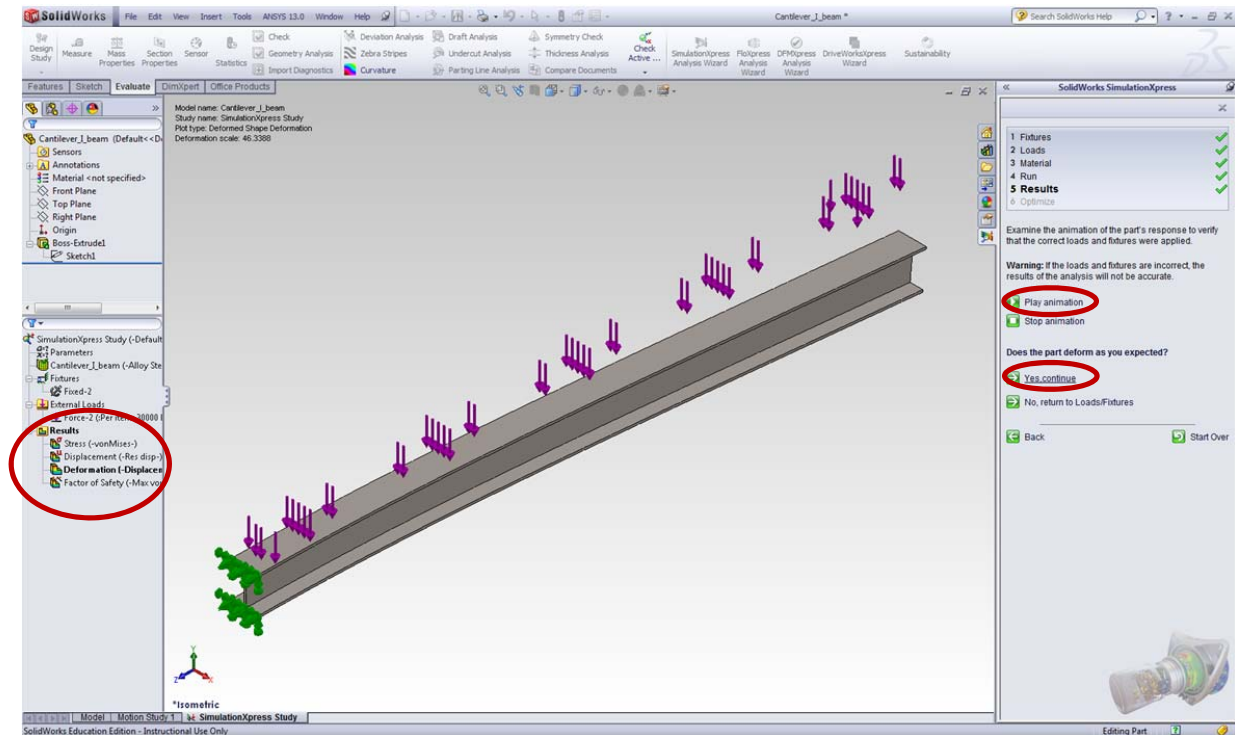
After the **Fixture**, **Loads**, **Material** are fully defined; click **Next** to **Run the simulation**.



Then you will see the screen below when it is running, solid mesh is in progress.



Once the simulation is completed you will see the **Results** icon on the left. To see how this simulation works, click **Play animation**. After then, click **Yes, continue**.



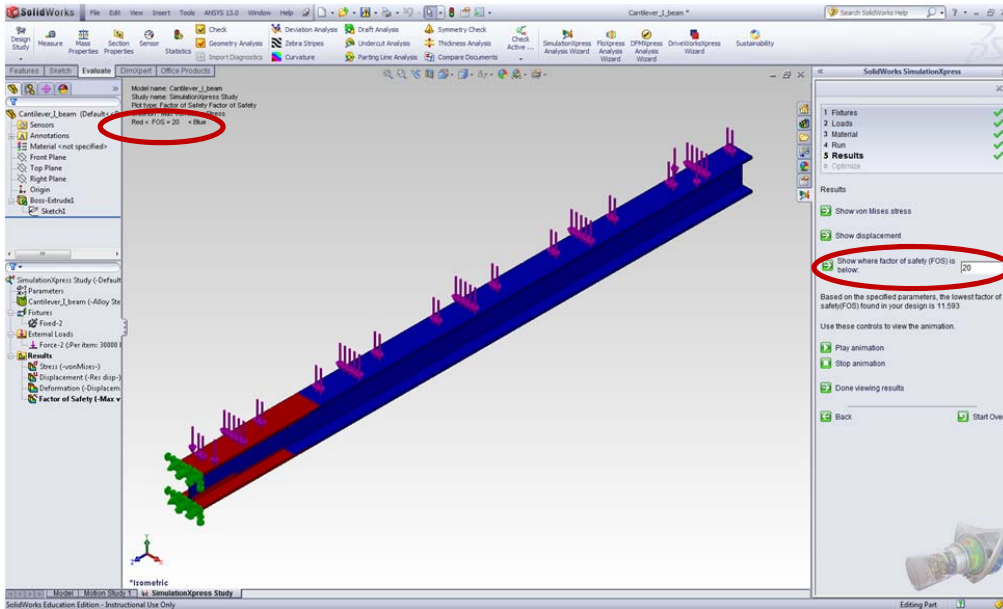
6. Results Analysis

By default, SimulationXpress starts the analysis in terms of **FOS** (factor of safety).

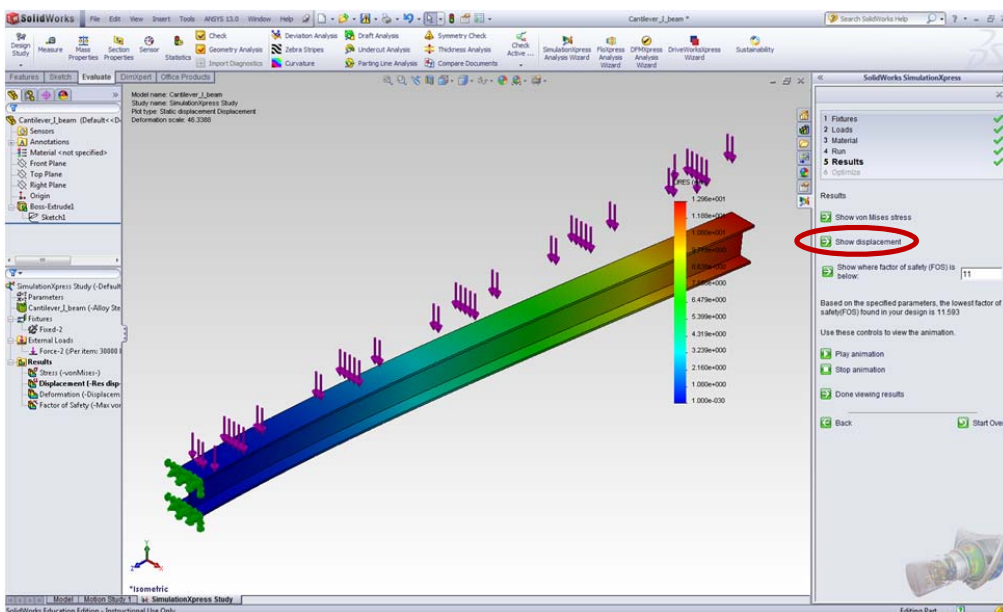
(It is a term describing the structural capacity of a system beyond the applied loads or actual loads.

Essentially, how much stronger the system is than it needs to be for a given load. So FOS tells you that the part will fail at locations where $FOS < 1$. To account for uncertainties in values, flaws in materials, etc., values for FOS are typically greater than 2. If a FOS is "too large", then the designer may consider changing the geometry of the part as the design may be over-conservative. However, there may be other reasons, for example, other loading conditions that put further restrictions on the geometry.)

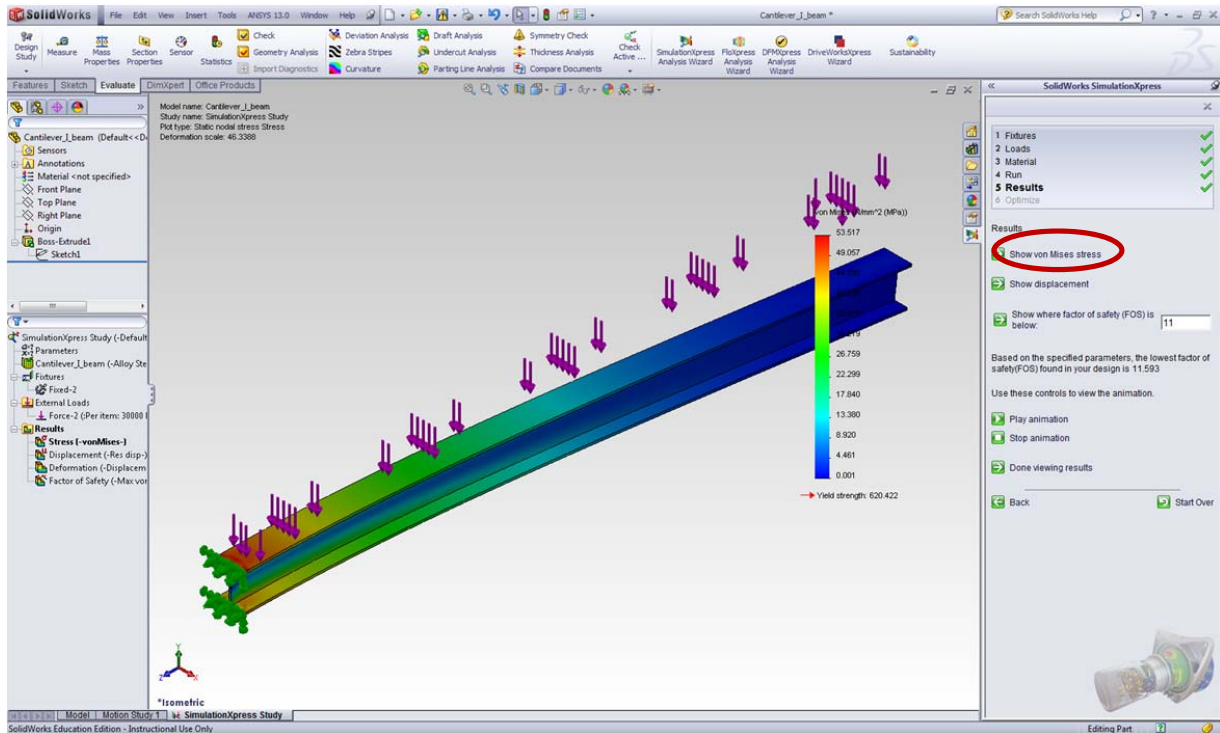
To see the results for the entire analysis, here enter a value greater than the default FOS=1, for example enter 20, then you will see the results that **RED** <= **FOS=20** <= **BLUE**.



Click on the other results, for example, **Show displacement**, and then it gives you an intuitionistic view of the displacement and as it is shown that the maximum displacement occurs at the end of the beam in red.



Repeat the same procedure to look at the stress distribution. Click **Show von Mises stress**



Note that all the variables of the **Fixture, Loads, and Material** can be redefined by simply going back to each step to change. Then run the simulation again to get new results.

Congratulations! You have completed this tutorial.

Now that you are done, can you try to...

- 1) What happens if you fix both ends of the beam (rather than just one end)? How does this effect the stresses and displacements? Does this make sense?
- 2) What happens if you change the material? How do the stresses in the structure change? How does the factor of safety (FOS) change? (Hint: do a Google search and compare the values of the Yield Stress for different materials.)
- 3) For the given initial geometry, calculate by hand the theoretical maximum force that can be applied given a Factor of Safety of 3.0. Apply this load within the software are verify that the stresses are the same.
- 4) What happens if you load the structure such that it acts like an 'H beam' rather than an I beam?
- 5) Change the cross-section of the beam so that you minimize weight but maintain a FOS of greater than 3 (keeping the load the same and maintaining a width of 250 mm and keeping the symmetry of the beam). Also allow the maximum deflection not to exceed 1.5 mm.