

# ME345: Modeling and Simulation

## Analysis of Plate with a hole by using SimulationXpress

Stevens Institute of Technology

Professor Frank Fisher

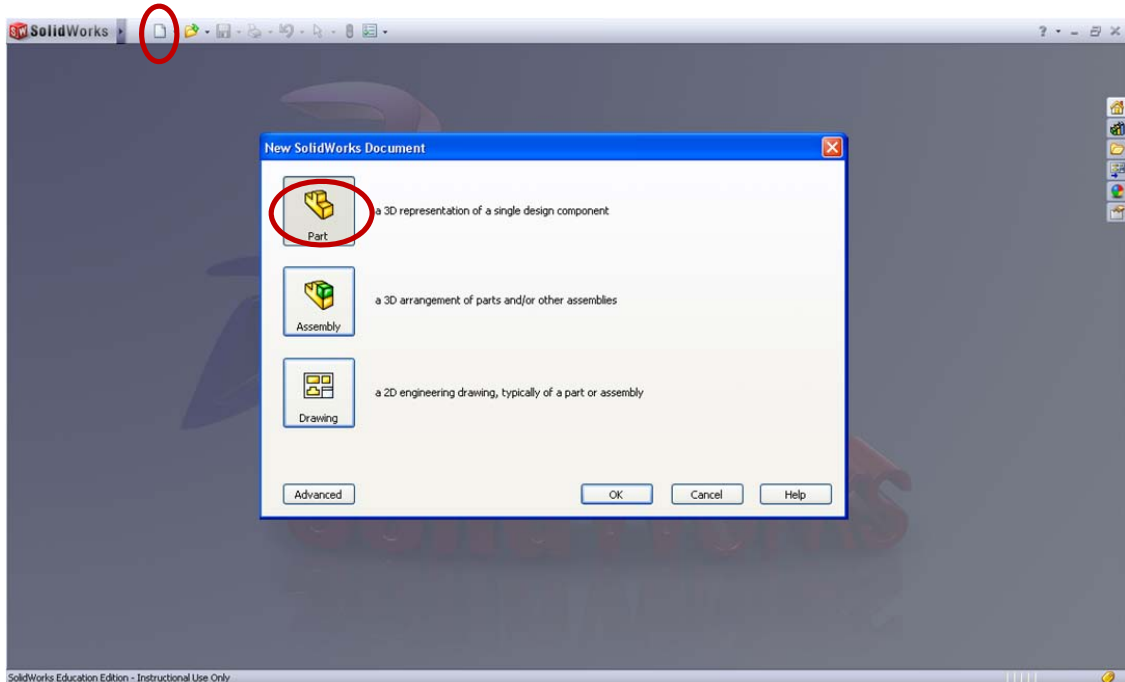
Last updated: July 5<sup>th</sup>, 2011

Written by Lin Dong

Software versions used in the tutorial: SolidWorks 2011

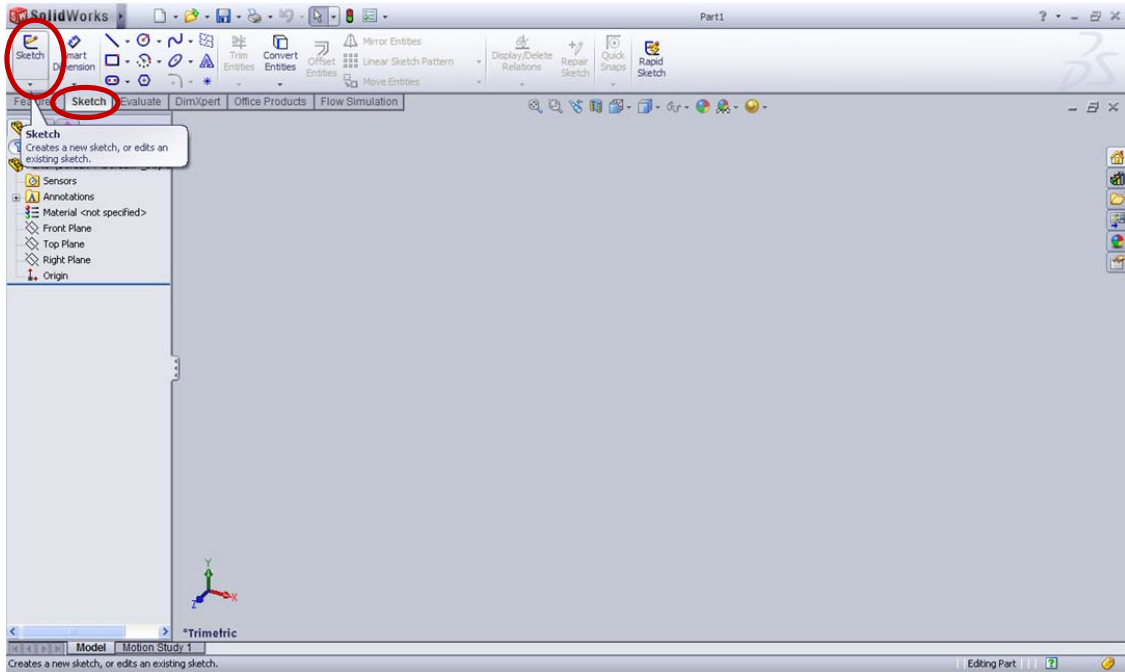
### 1. Start SolidWorks

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

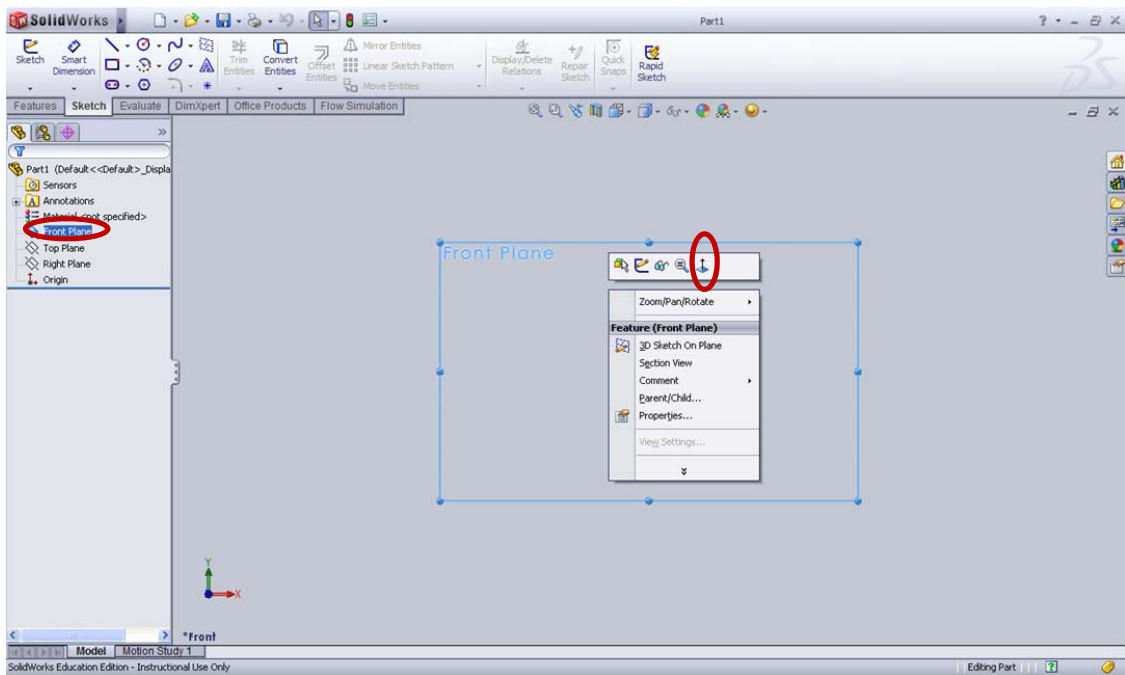


## 2. Create the plate sketch

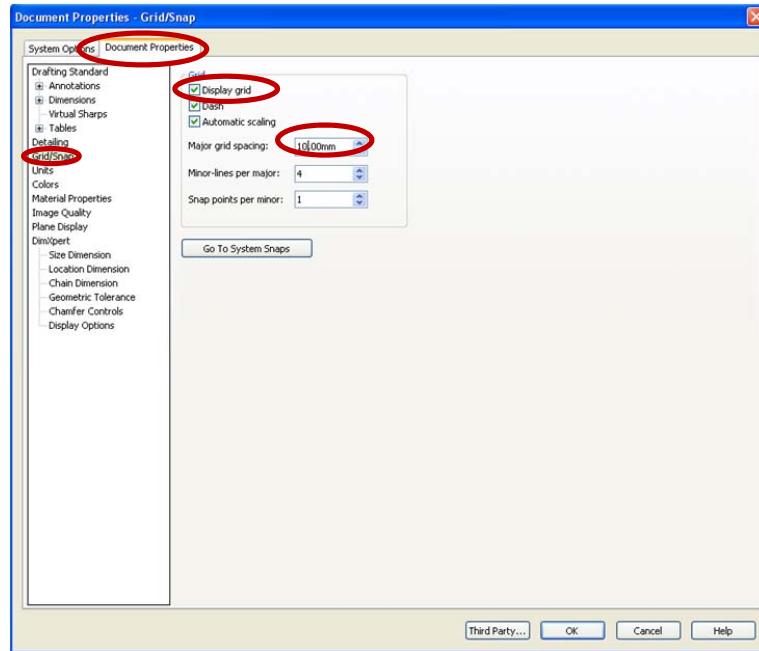
Click on the **Sketch** command and **Sketch** tab to start.



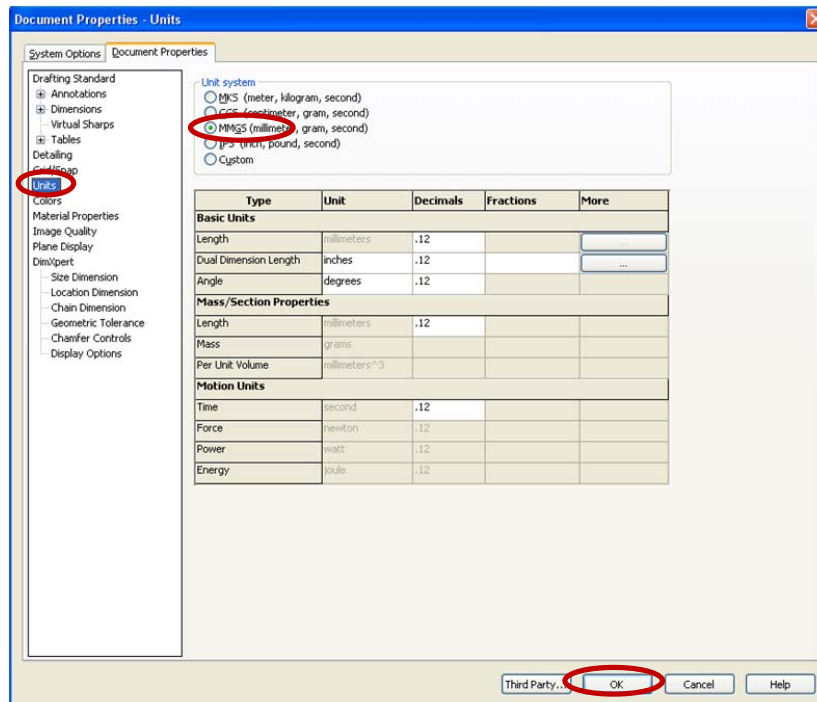
Select the **Front Plane** as the sketch plane then it automatically goes to orient a model normal to the nearest global XYZ coordinates. If not, right click to select **Normal to** command.



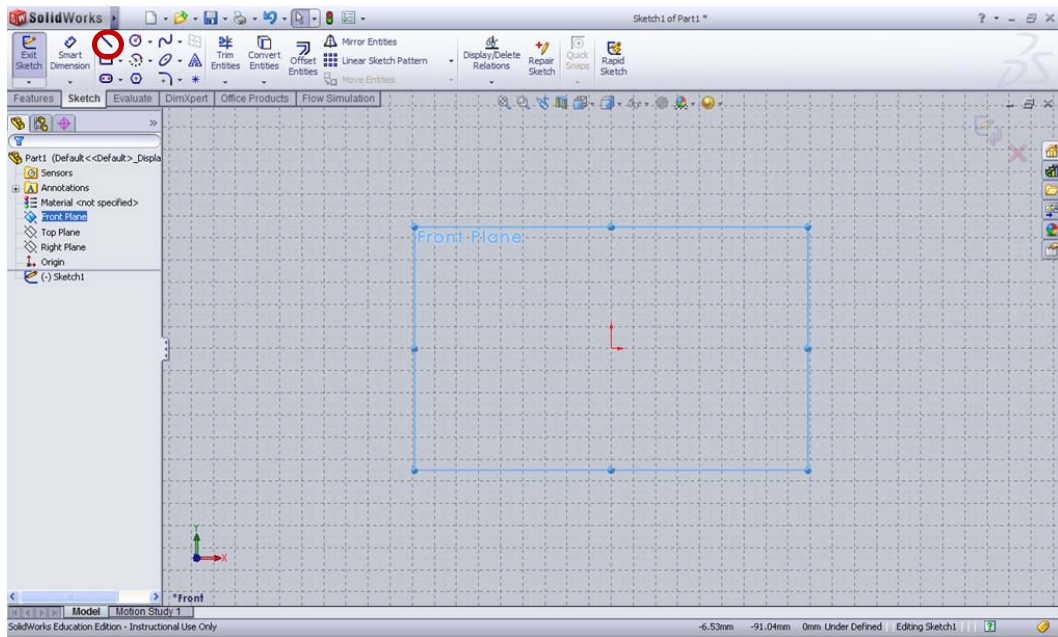
To display a sketch grid in an active sketch and set options for the grid display and snap functionality, go to the **Tools > Options > Document Properties > Grid/Snap > Check Display Grid > change grid spacing to 10mm**



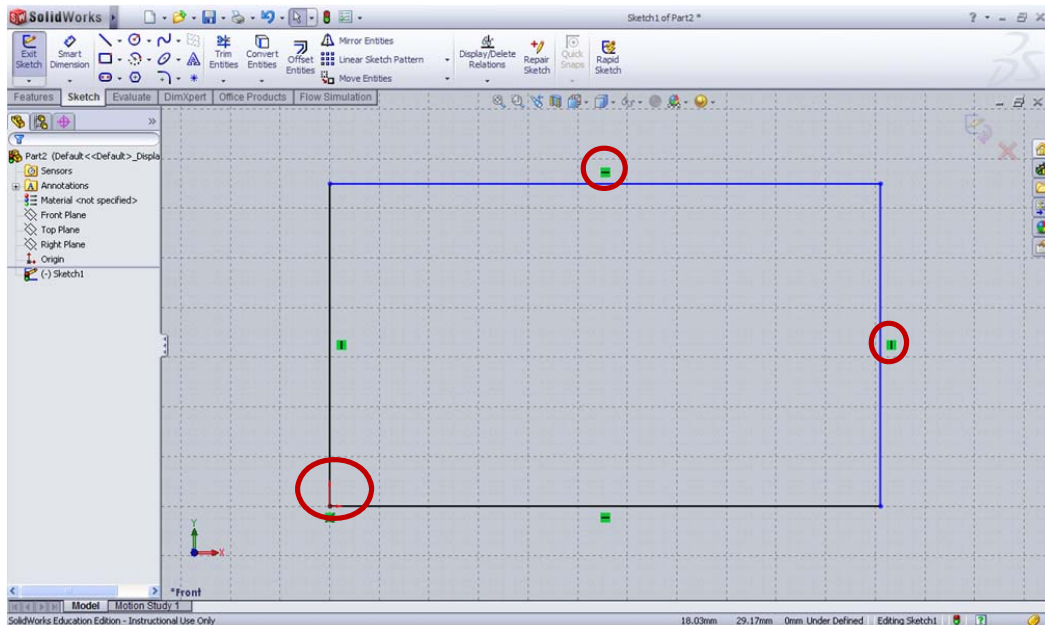
If the units are not in MMGS automatically, go to **Units > check MMGS**



Click on the **Line** command and then the screen will appear like below

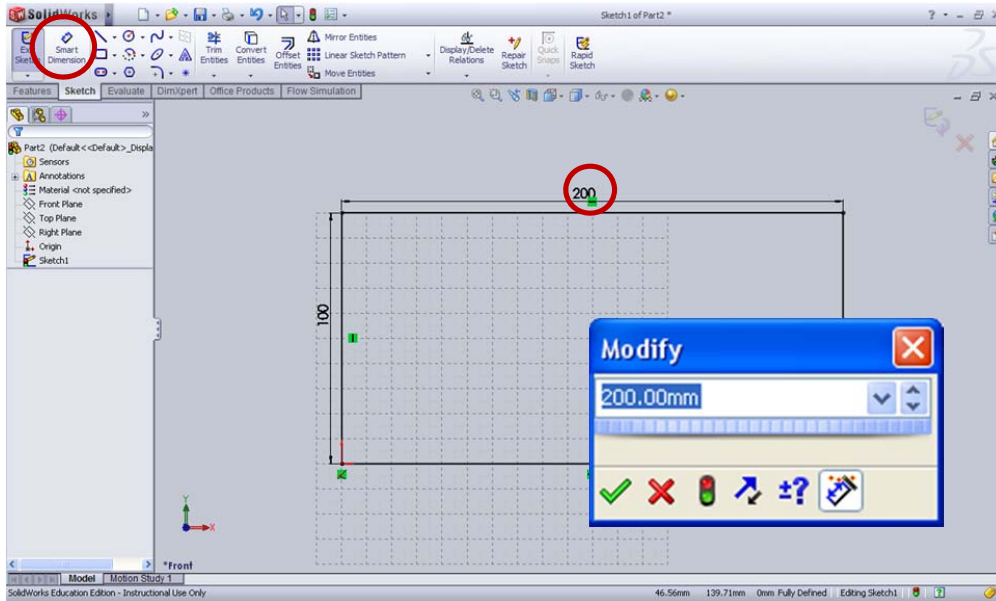


Complete a rectangle sketch by clicking on the **Line** command and starting at the **Origin**. Do not worry about the lengths of the structure, they can be defined later, but make sure the lines are horizontal or vertical. It can be implied by a green square with the line inside.



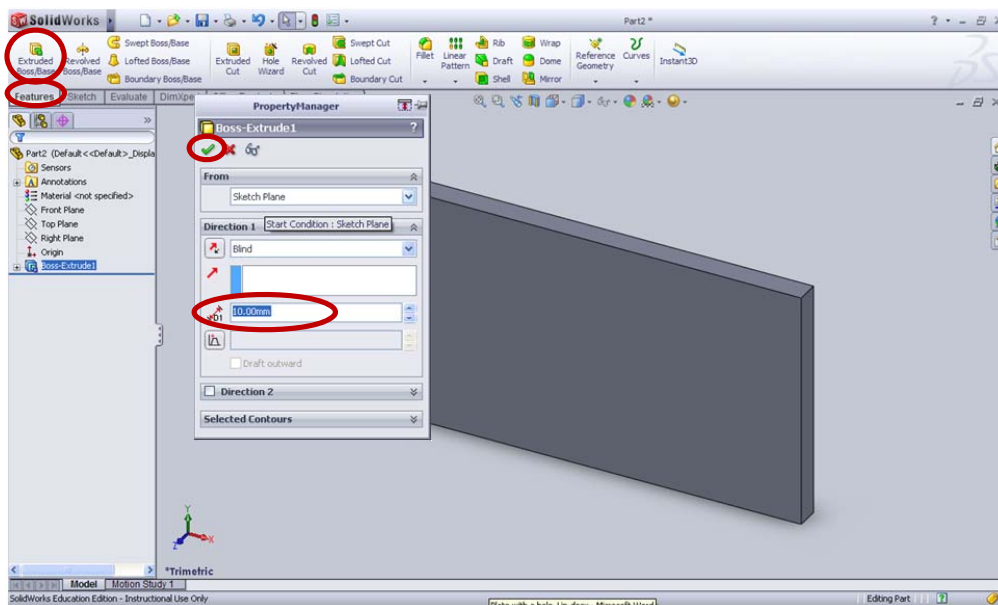
Then go to **Smart Dimensions** to define the length of the rectangle. Click on **Smart Dimension** >

Select the sketch of the line > edit the value



### 3. Extrude the plate

Click on the **Features Tab** > **Extruded Boss/ Base**. In the **PropertyManager** window, change the dimension of **D1** to 10.00mm.





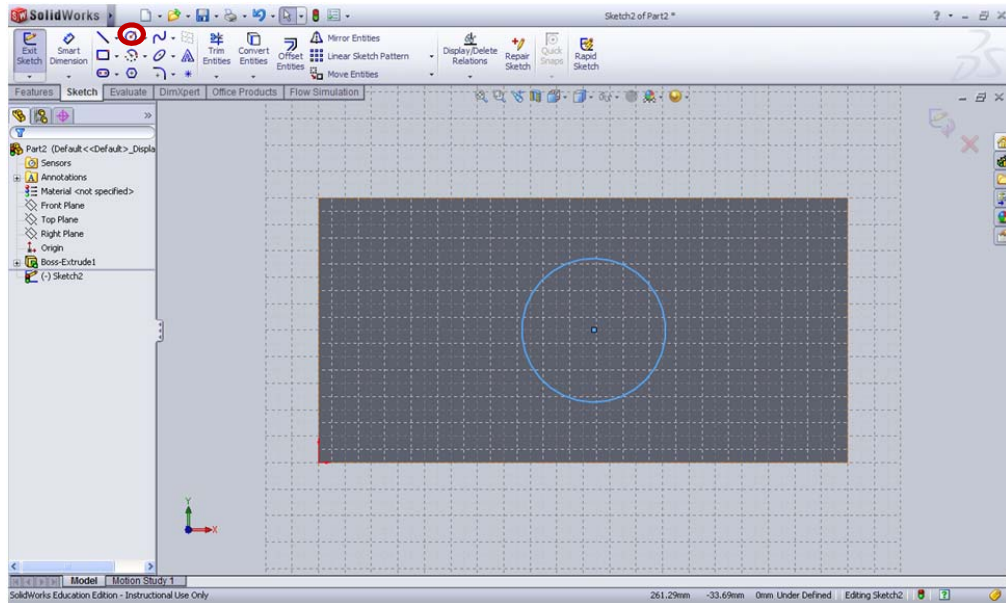
#### 4. Create a hole

Similarly, to create a hole in the plate, first create a circular sketch of the hole and then

**Extruded Cut** the material. Click the base of the plate, and then right click to select **Normal to**

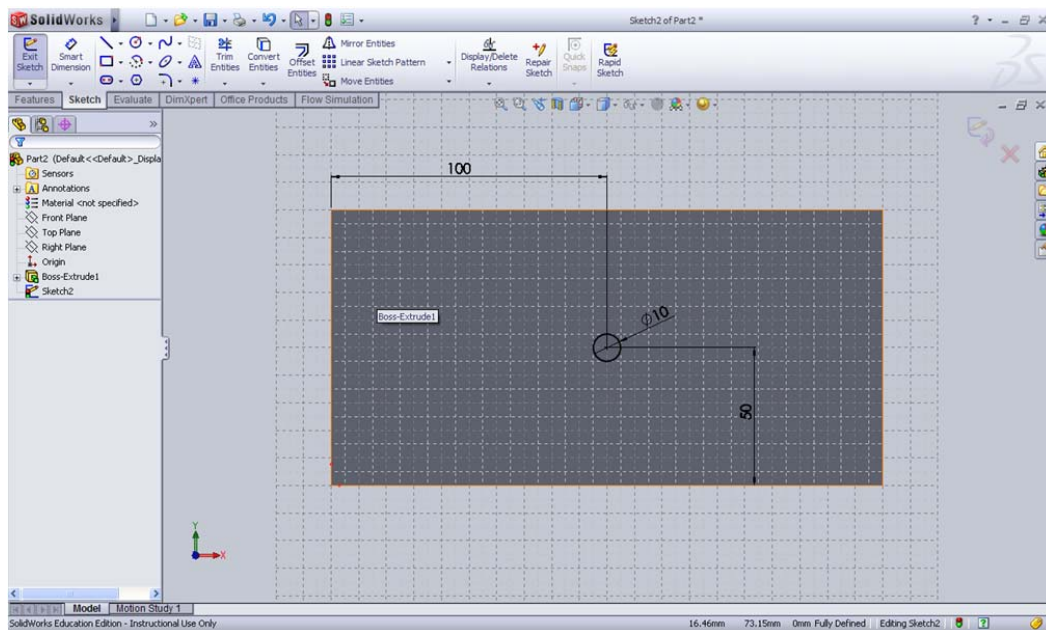


command to orient the base plane of the model. Click on **Sketch** tab to draw by using **Circle**.

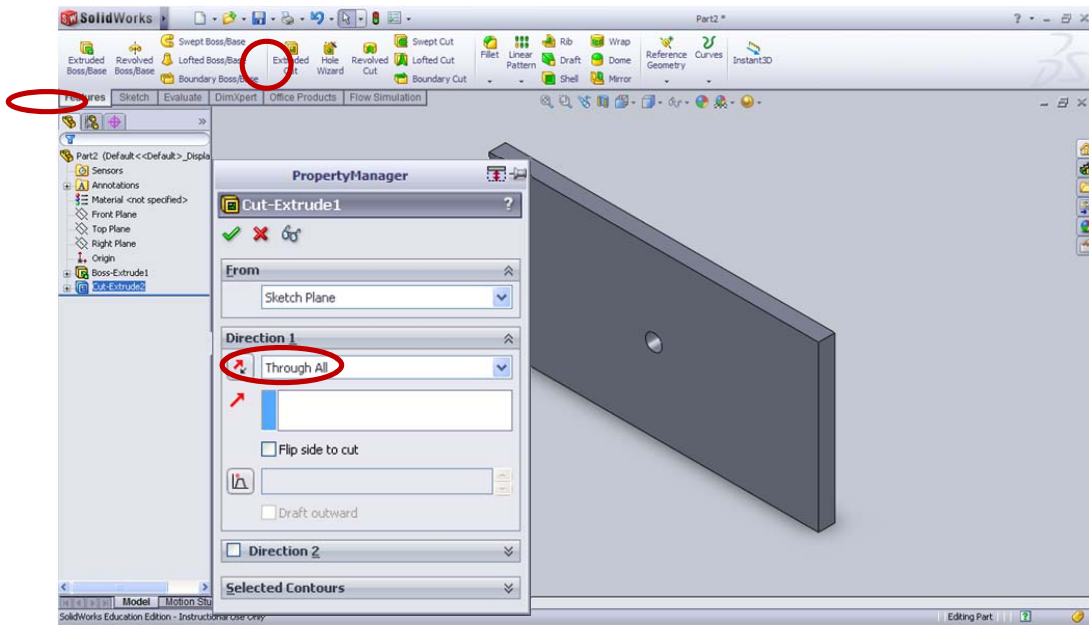


Edit the dimension of the circle as Diameter=10mm and it is also in the center of the plate. Click on

**Smart Dimension** to define the following dimensions.

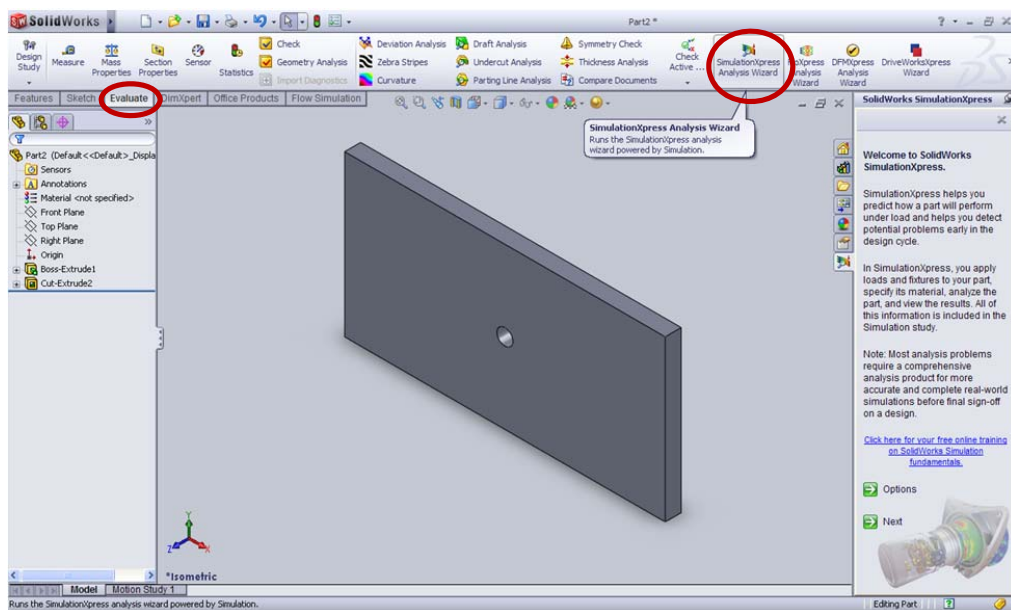


**Exit Sketch** to create a hole by using the features of **Extruded Cut**. Go to **Features > Extruded Cut > Select the circle sketch > Choose Through All** . Now the model of a plate with a hole has been built.

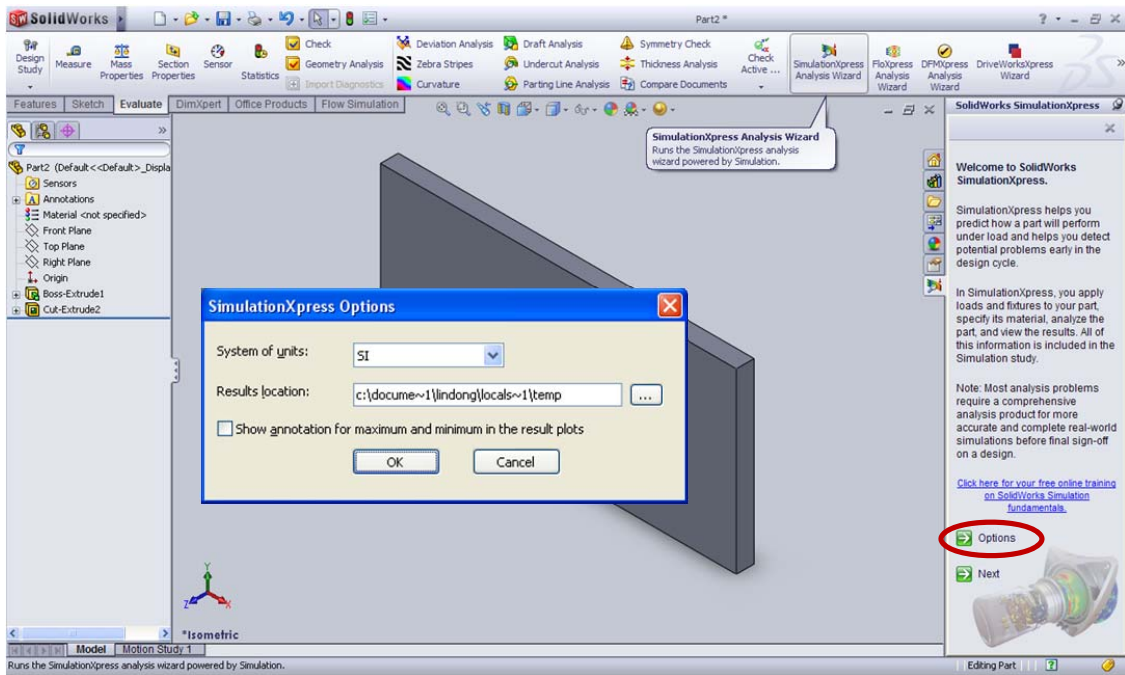


## 5. Start SimulationXpress

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

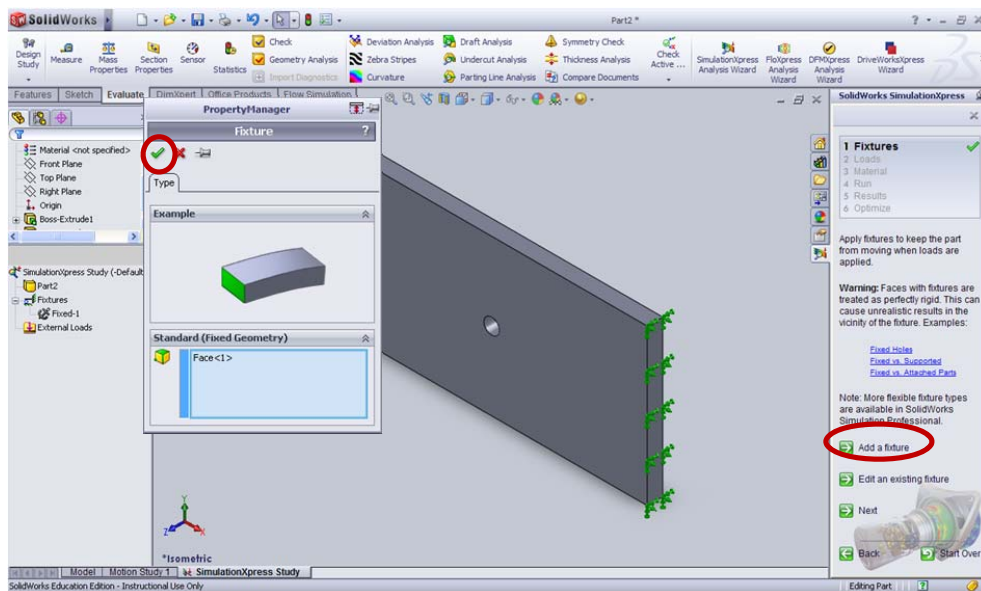


Click on the **Options** to change the **units** to SI.



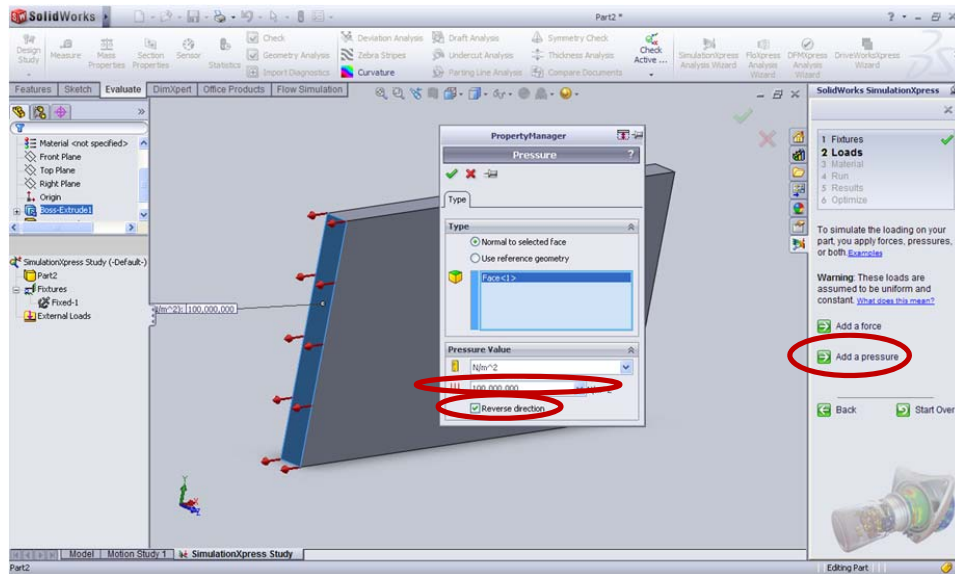
## 6. Run simulations

First, click on **Next** to **Add a fixture**. Select the side face of the plate to restraint from moving then you will see a green fixture there.

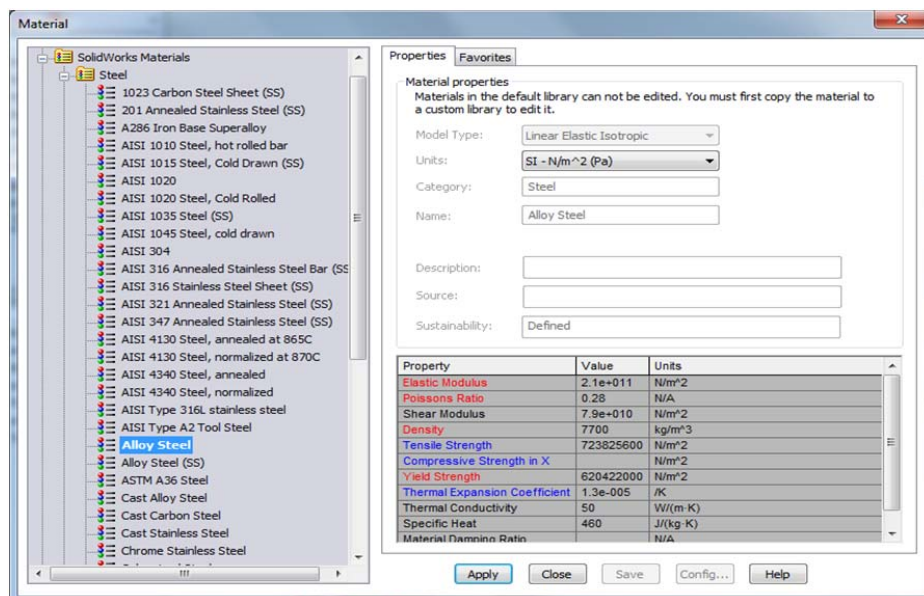




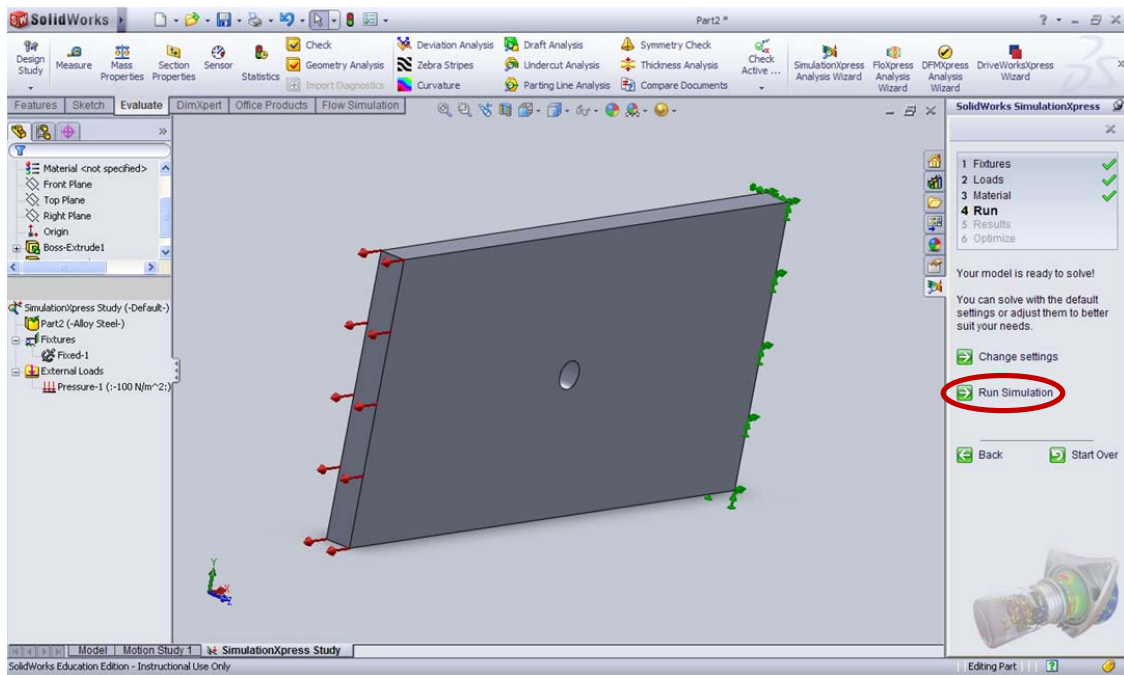
Second, click **Next** to define the **Loads**. In this tutorial, we will define a pressure on the other side face of the plate. Click on **Add a pressure**, then select the side face, the red pressure will appear on the plate. Then add the value of 100,000,000 N/m<sup>2</sup> to the plate, also check the box of **Reverse Direction**.



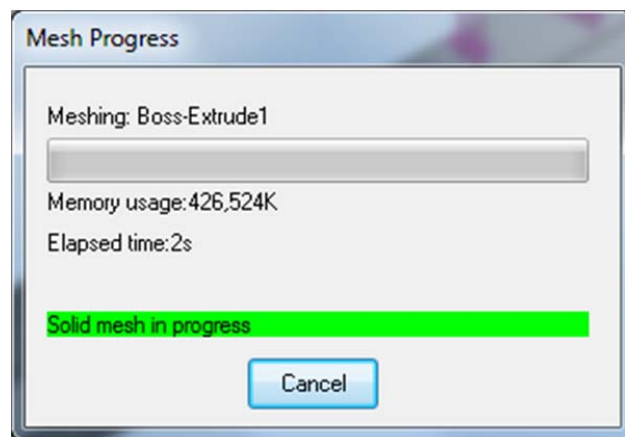
Third, click **Next** to apply the **Material**. Choose the material of Alloy Steel from the library. Click on **Apply** then close this window.



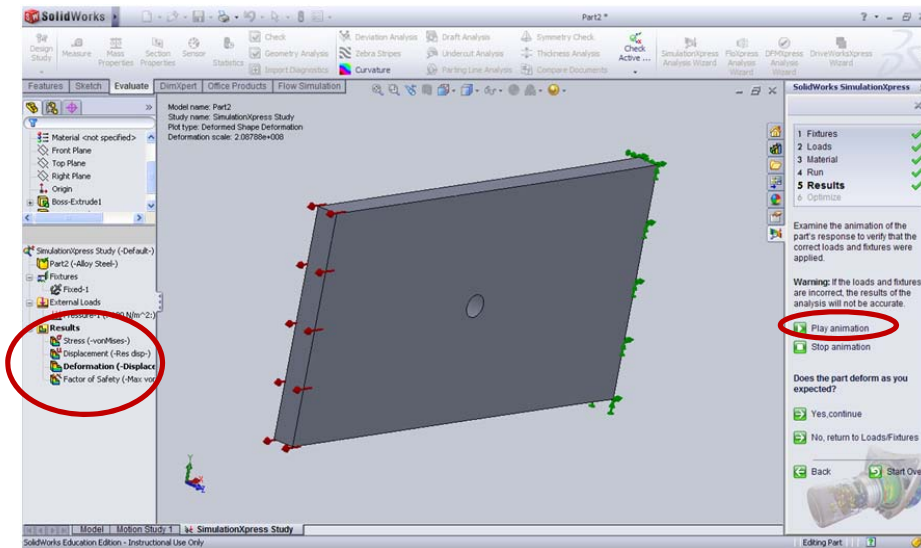
After the **Fixture, Loads, Material** are fully defined; click **Run the simulation**.



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

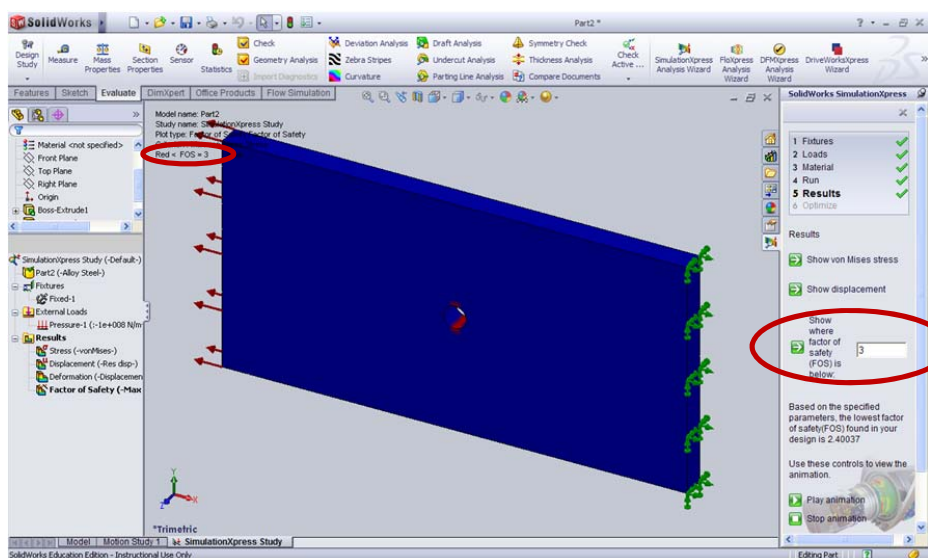


Once the simulation is completed you will see the **Results** icon on the left. To see how this simulation works, click **Play animation**.

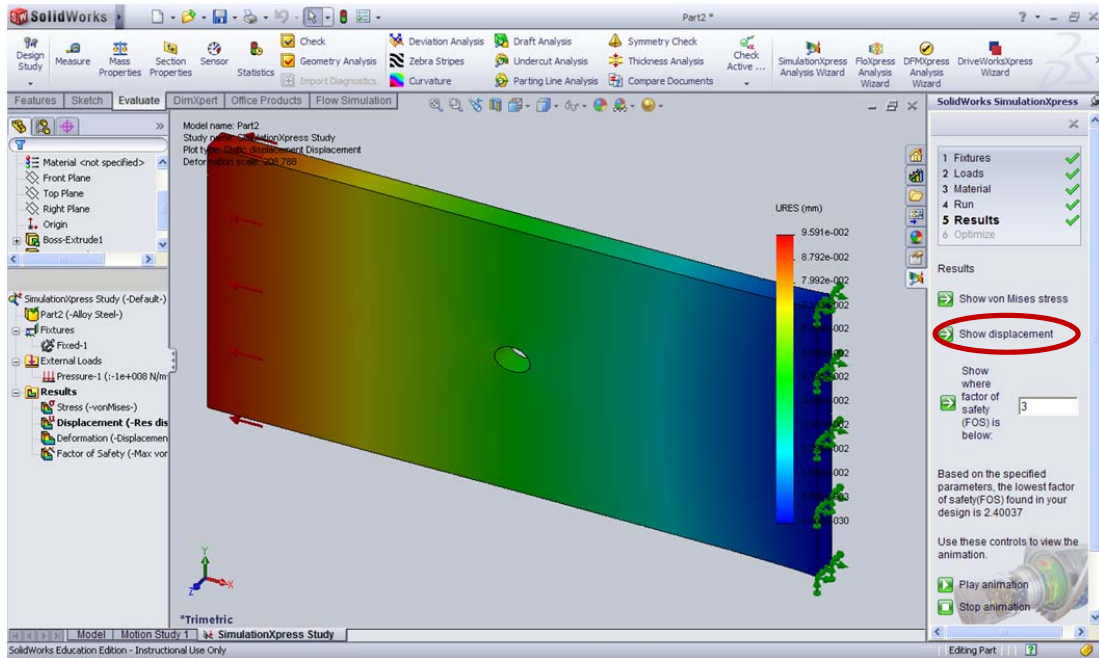


## 7. Results Analysis

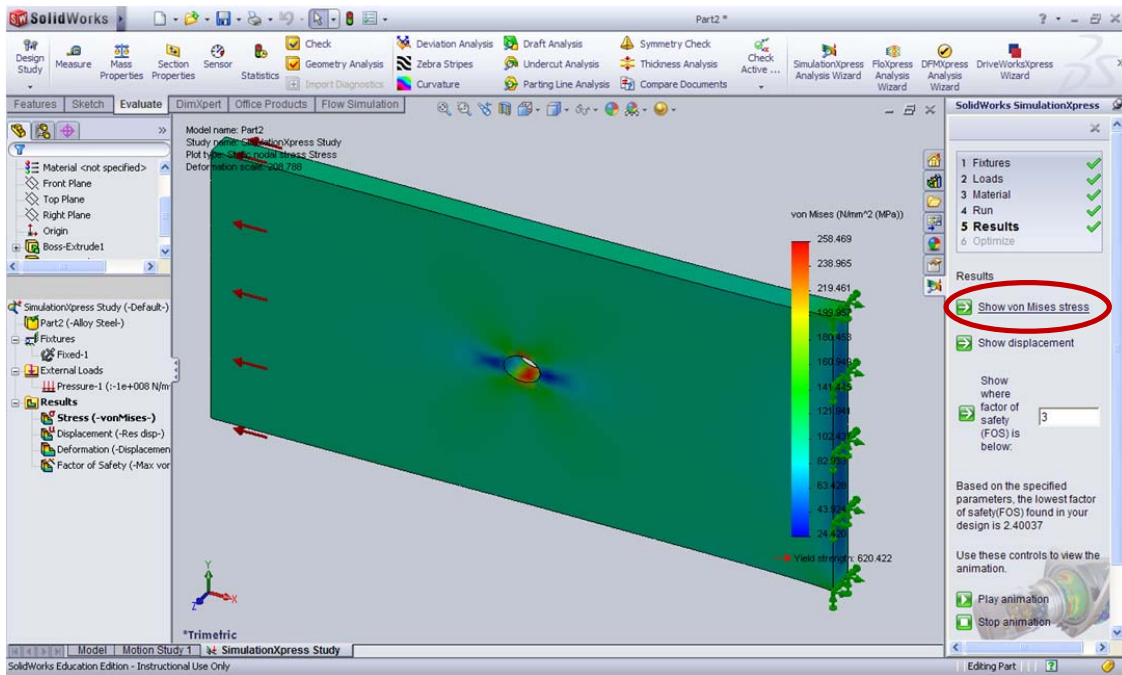
By default, SimulationXpress starts the analysis in terms of **FOS** (factor of safety). Based on the model, the simulation results of FOS in this design is 2.40037. In order to see the critical areas under a certain FOS, for example, here enter a value of 3, then you will see the results that **RED**  $\leq$  **FOS=3**  $\leq$  **BLUE**.



In order to see the other results, for example, **Show displacement**, and then it gives you an intuitionistic view of the displacement.



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 you can generate your report that provides details about your design, including the information of material, load, constrains, and meshing and simulation results.

