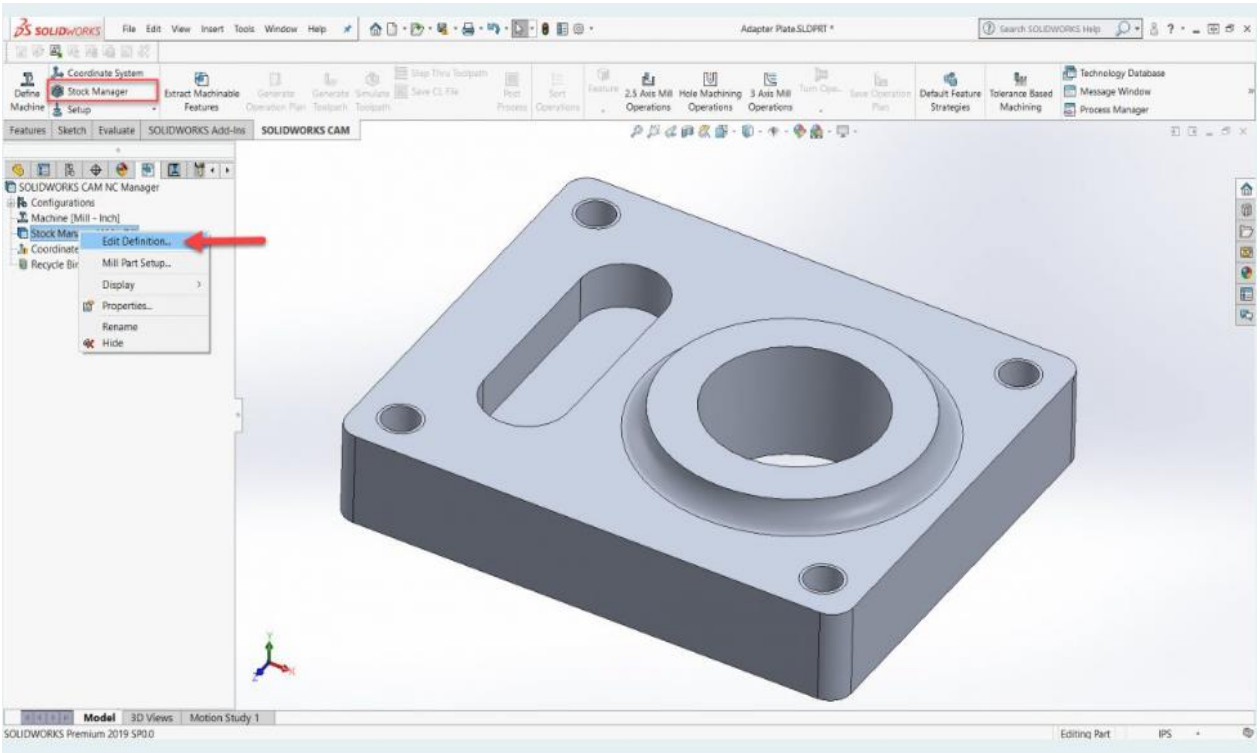
3.0 Defining the Stock

In the SOLIDWORKS CAM Feature Tree we can define our Stock using any of the following methods:

1. By clicking on the Stock Manager Icon in the Command Manager
2. Select the Stock Manager in the CAM Feature Tree, click your right-mouse button and select Edit Definition
3. Double click on Stock Manager in the CAM Feature Tree.



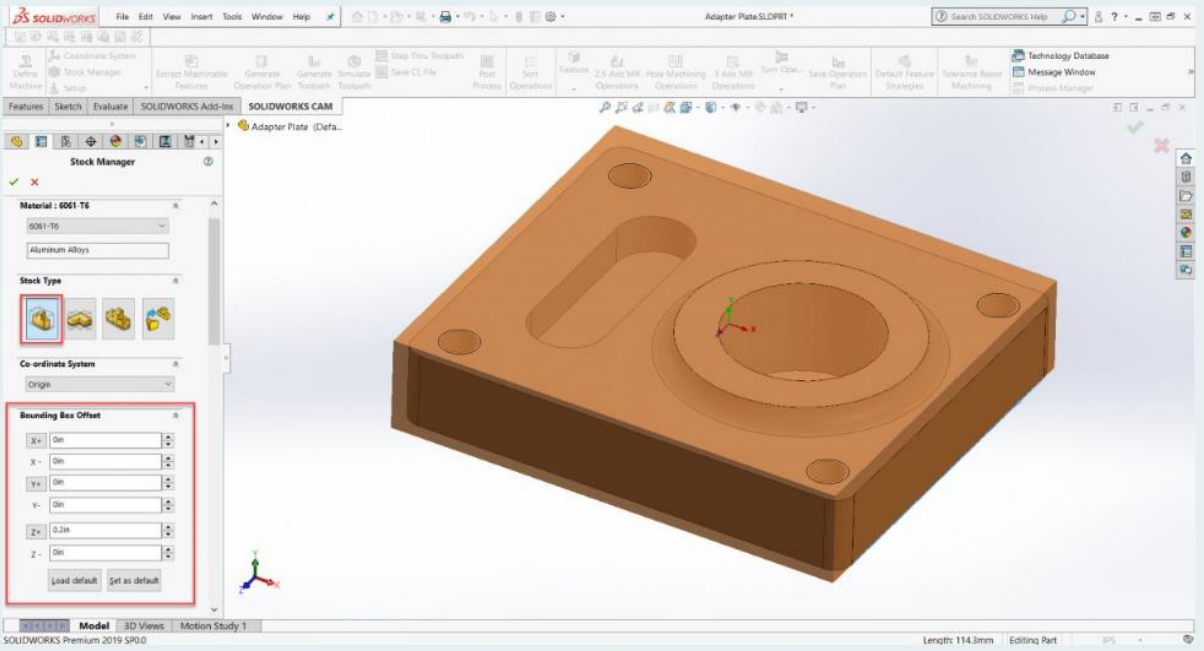
Stock Manager Definition

Once the Stock Manager is active, the material is imported from the model and can be changed using the pull down. These selections reference the technology database which can assign speed and feed rates for selected materials.

With our Material set we have 4 options for defining our stock. They are:

1. Part Bounding Box

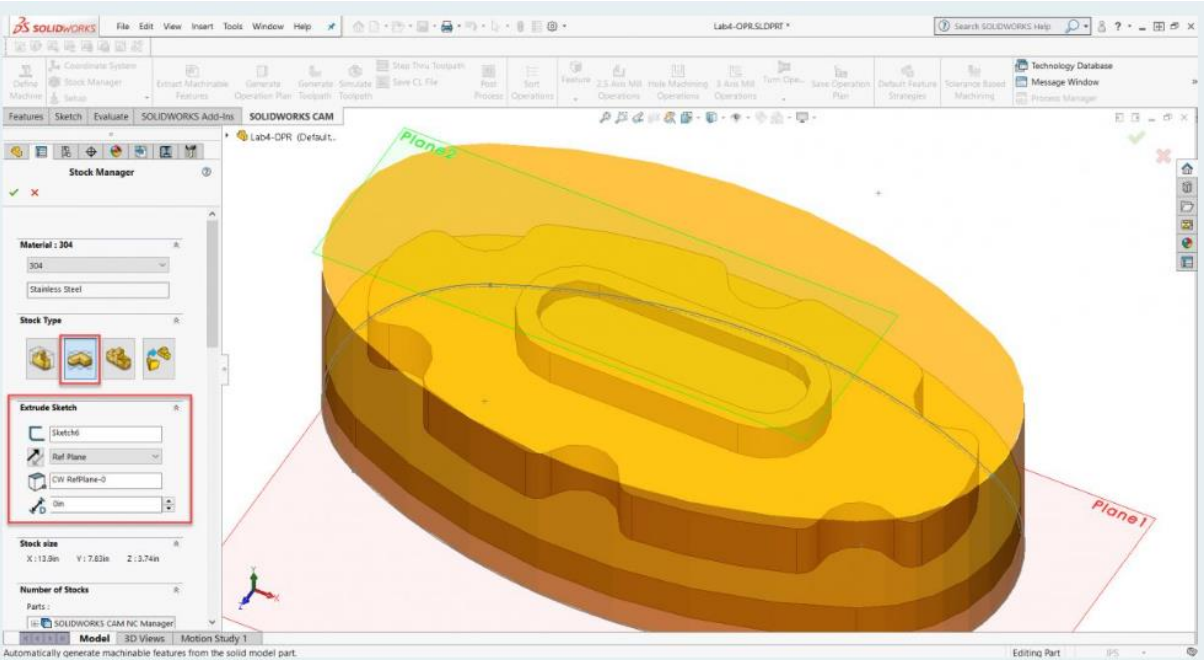
The first option is Part Bounding Box. It generates the smallest box/area that your geometry can fit inside. You can then add material to any side of the bounding box using the controls in the Feature Manager. If the parts dimensions should change the bounding box will update to the new size and maintain the settings in the Bounding Box Offsets. This method is the fastest way to define stock for rectangular or square shapes.



Part Bounding Box

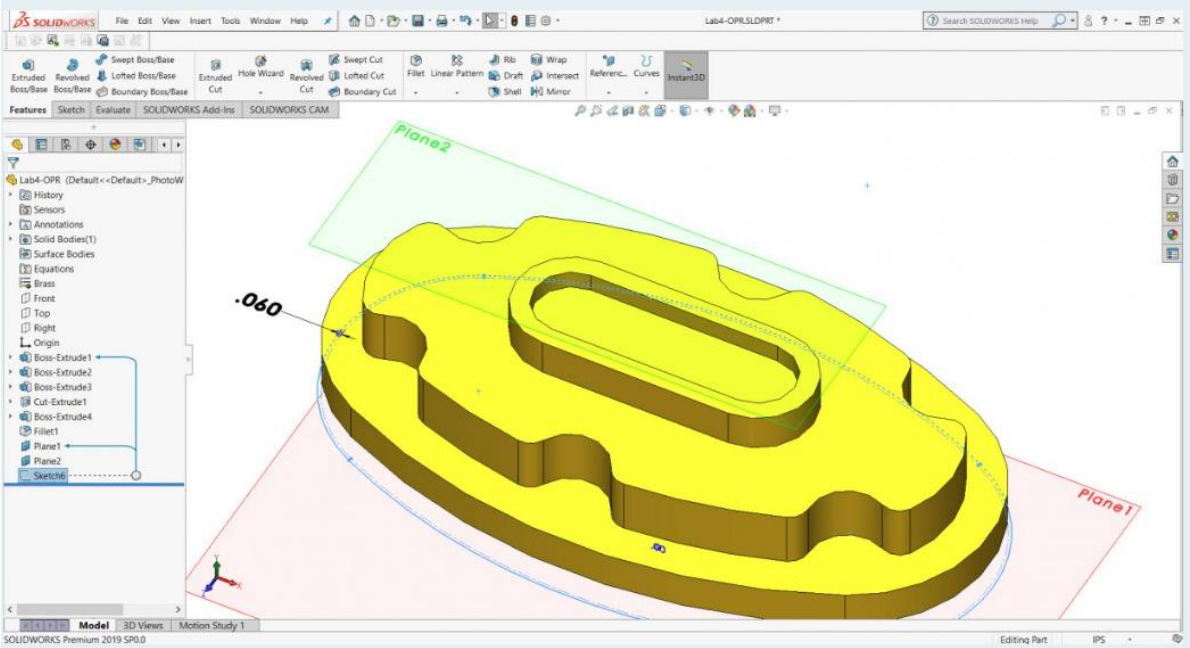
2. Use a Sketch

The second method is to use a sketch. This is very useful if your part is being machined from a laser or plasma cut blank. First, we create a plane referencing the bottom of the part (Plane 1). Offset the plane by the amount of material to be added on the bottom face. Create another plane for the material to be added on the top face (Plane 2). Place a sketch on Plane 1 and convert the entities of the part perimeter. If we need to machine the sides, offsetting the part perimeter will be required to account for the extra material, 2 mm this case.



Extrude Sketch

Extrude the sketch using Ref Plane and selecting the plane created from the top face (Plane 2). This ensures that the stock will update if the part geometry changes because our Plane and Sketch offsets control the amount of material added. Other commands like Blind extrude must be manually updated if part geometry changes.

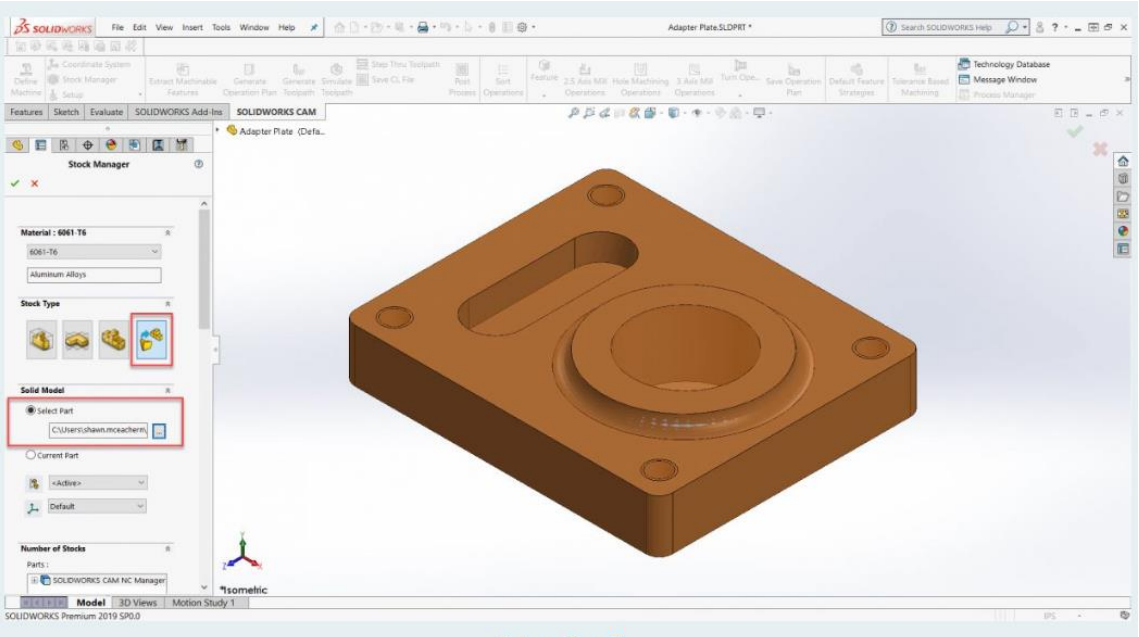


Use a Sketch

3. Use an STL file

The third method is to use an STL file. This is great for machining castings, 3D prints, parts from previous machining operations or any part were only few features require machining. In our case, we just need to machine the large diameter and drill/tap the holes. Simply locate the file to be inserted.

4. Use a Solidworks part file



Select a Part File