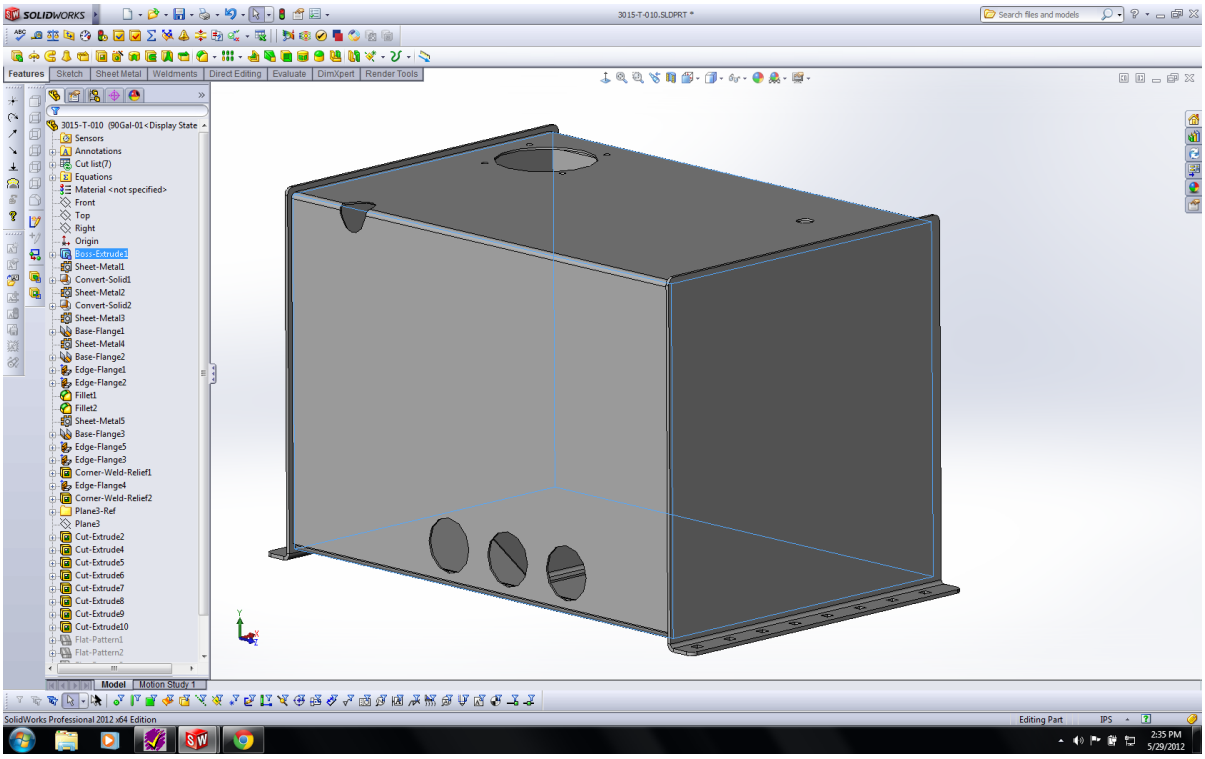
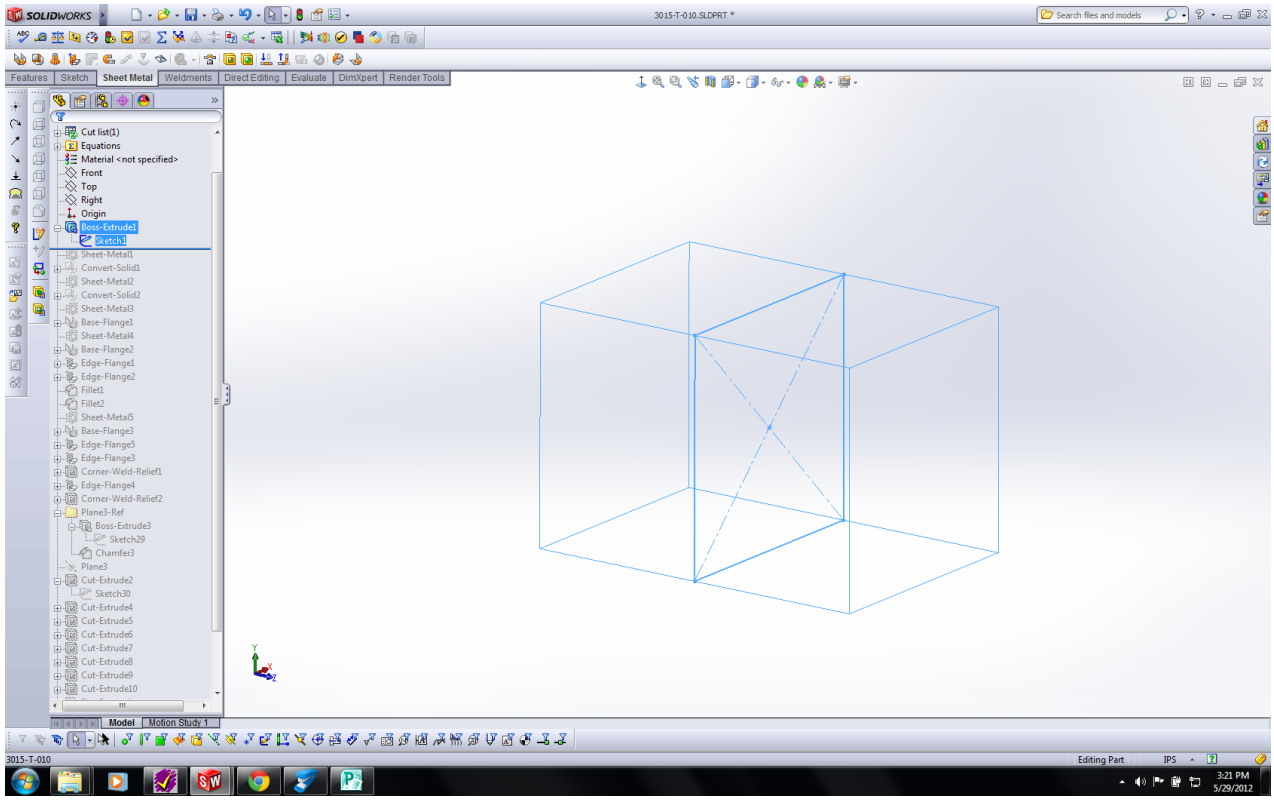
Hydraulic Reservoir

Multi-Body Sheet Metal Part / Table Driven Design



Summary: Use SolidWorks 2012 to design an easily configurable hydraulic reservoir. This method will allow for a variety of porting, capacity, and mounting options. This method has the following advantages over using an Assembly design approach; all flat patterns are contained within the part file, no time is spent mating parts, and the overall design is easier to control through design tables.

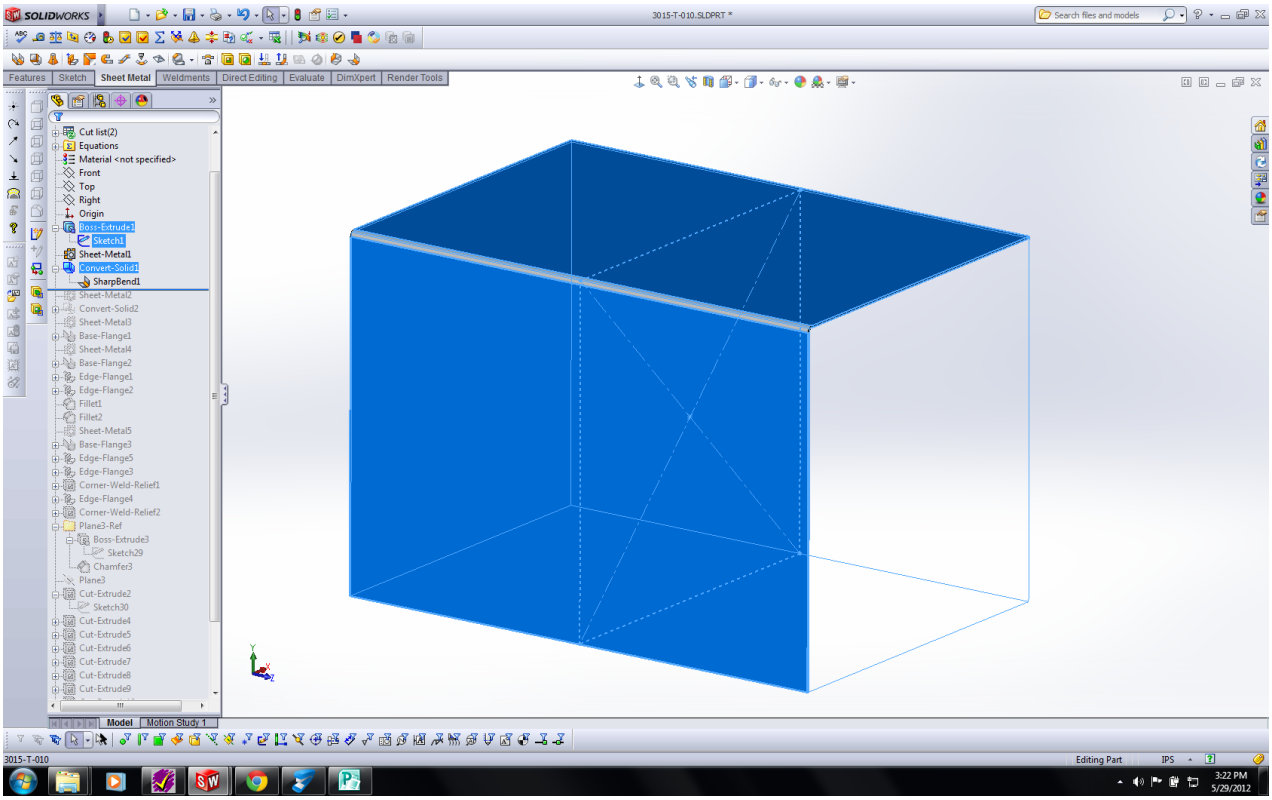
STEP 1:



⇒ Use one of the origin planes to sketch the cross section of your tank

⇒ Extrude the cross section to a depth that reflects a desired fluid volume

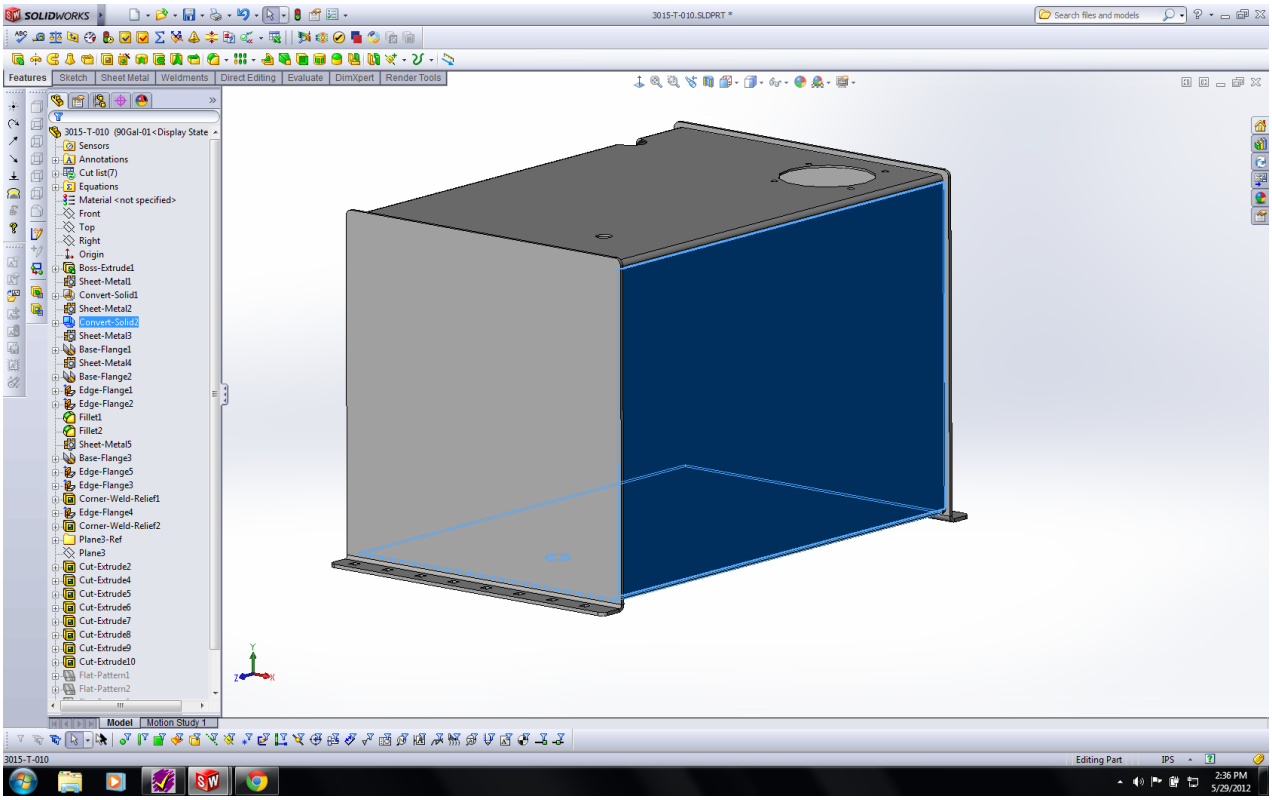
STEP 2:



⇒ Using the Covert to Sheet Metal function create the first half of the body skin. You will need to check the box “keep body” for use in the next steps

⇒ Select the desired sheet metal and radius settings

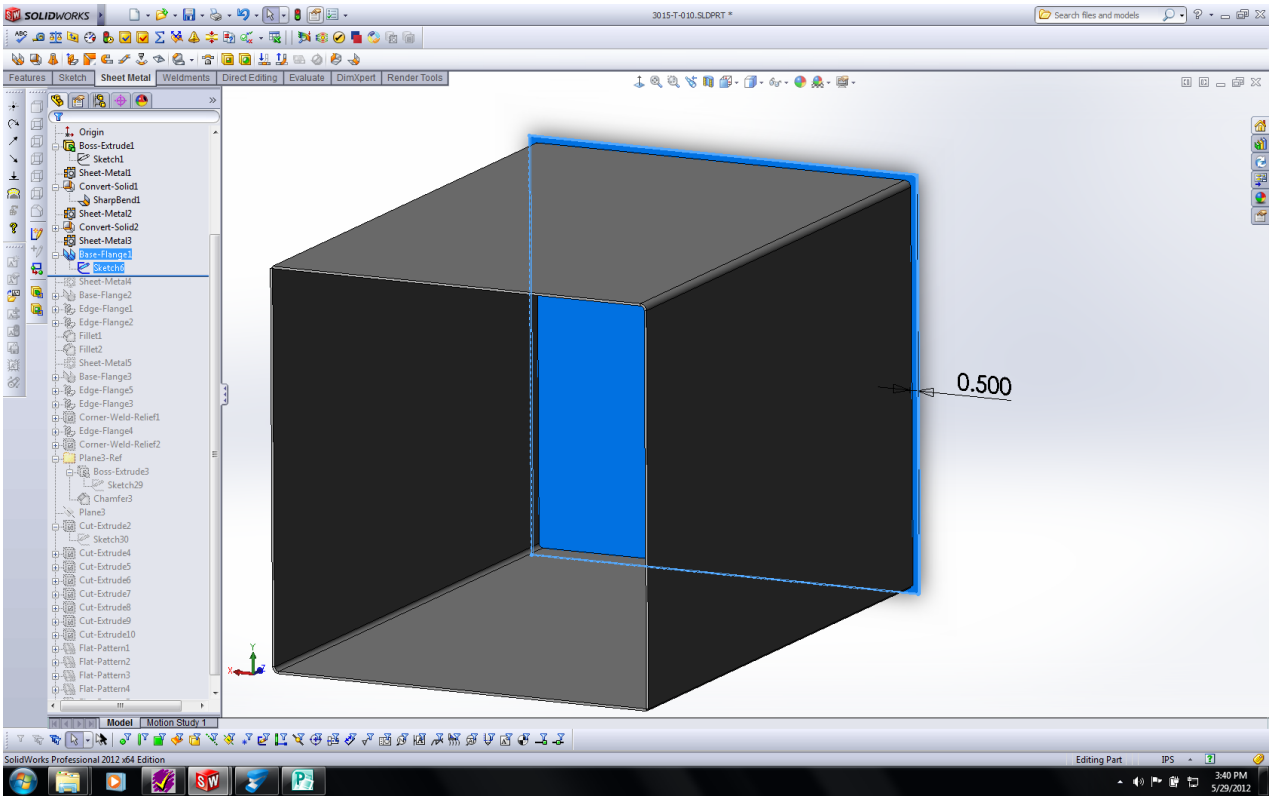
STEP 3:



⇒ Repeat Step 2 for the opposite half of the tank skin

⇒ Select the desired sheet metal and radius settings

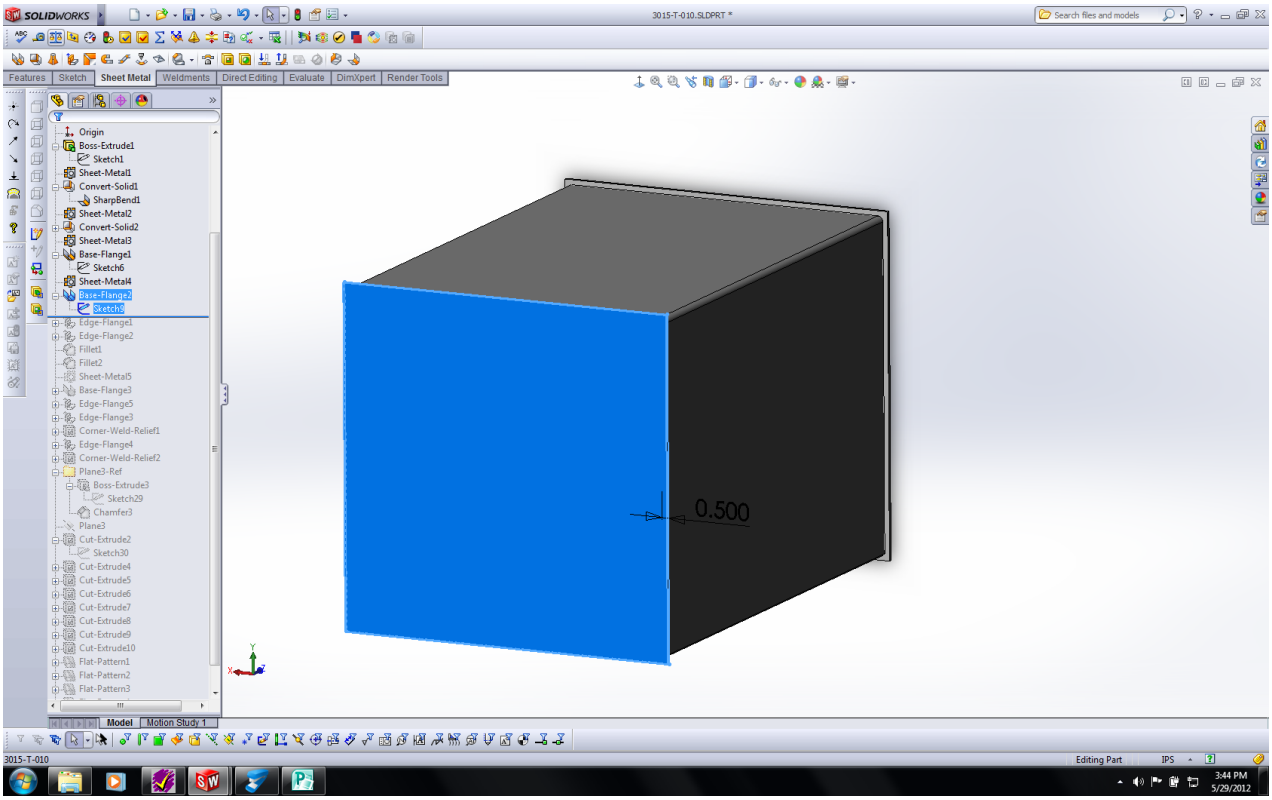
STEP 4:



⇒ On the end face of one of the tank body skins insert a sheet metal base flange.

⇒ For the sketch of the base flange use an offset sketch entity to ensure a 1/2” clearance around the perimeter for weld.

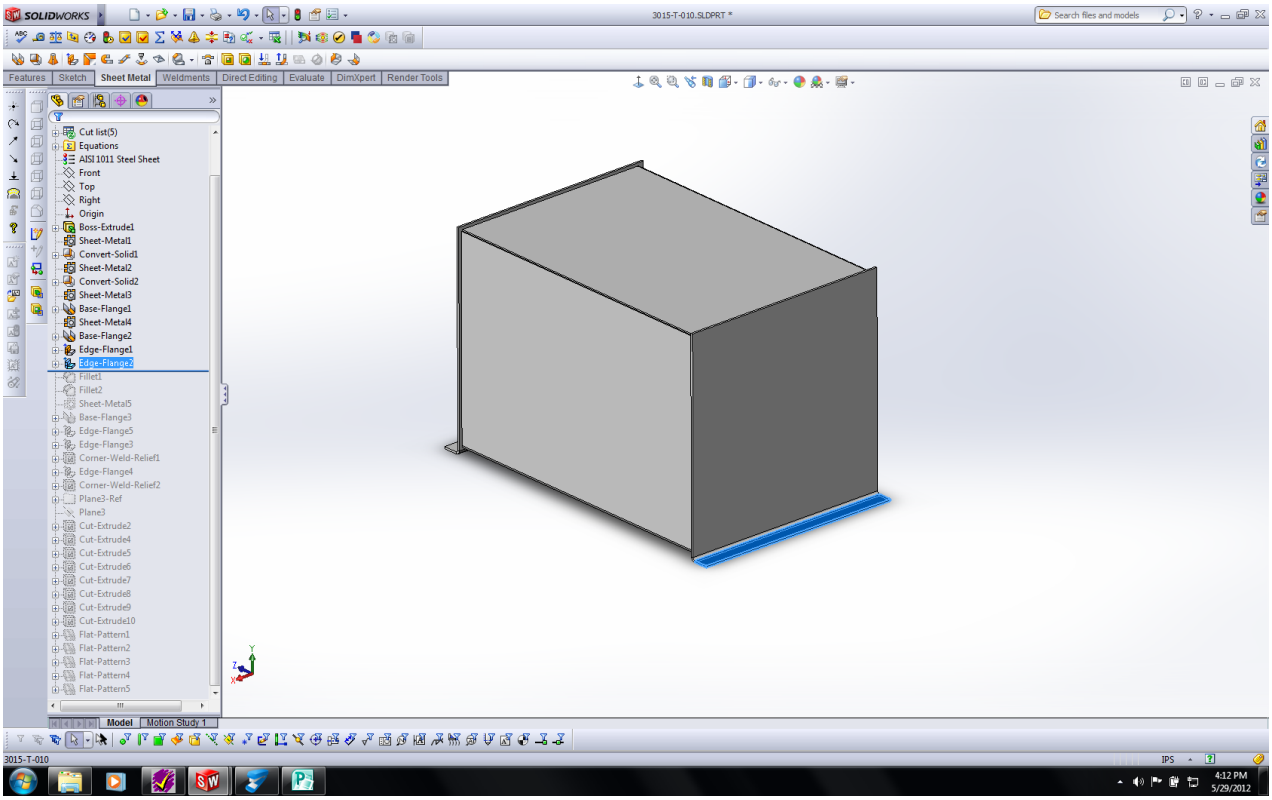
STEP 5:



⇒ Repeat Step 4, now on the opposite side of the tank. You should now have a fully enclosed sheet metal body.

⇒ In the next steps we will add weld flanges and mounting tabs to the new sheet metal components.

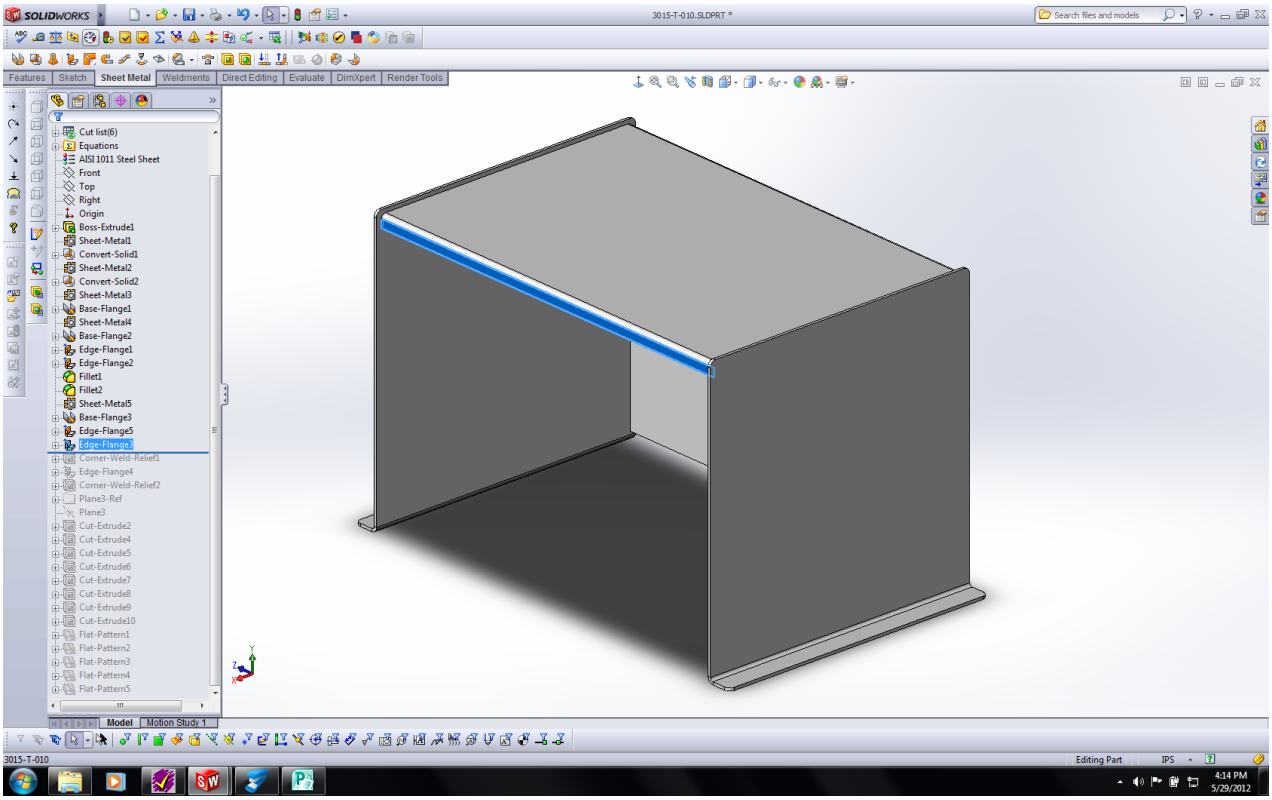
STEP 6:



⇒ Using the Edge Flange command, add a 2” mounting flange to the bottom of the tank.

⇒ Repeat this process for the opposite side.

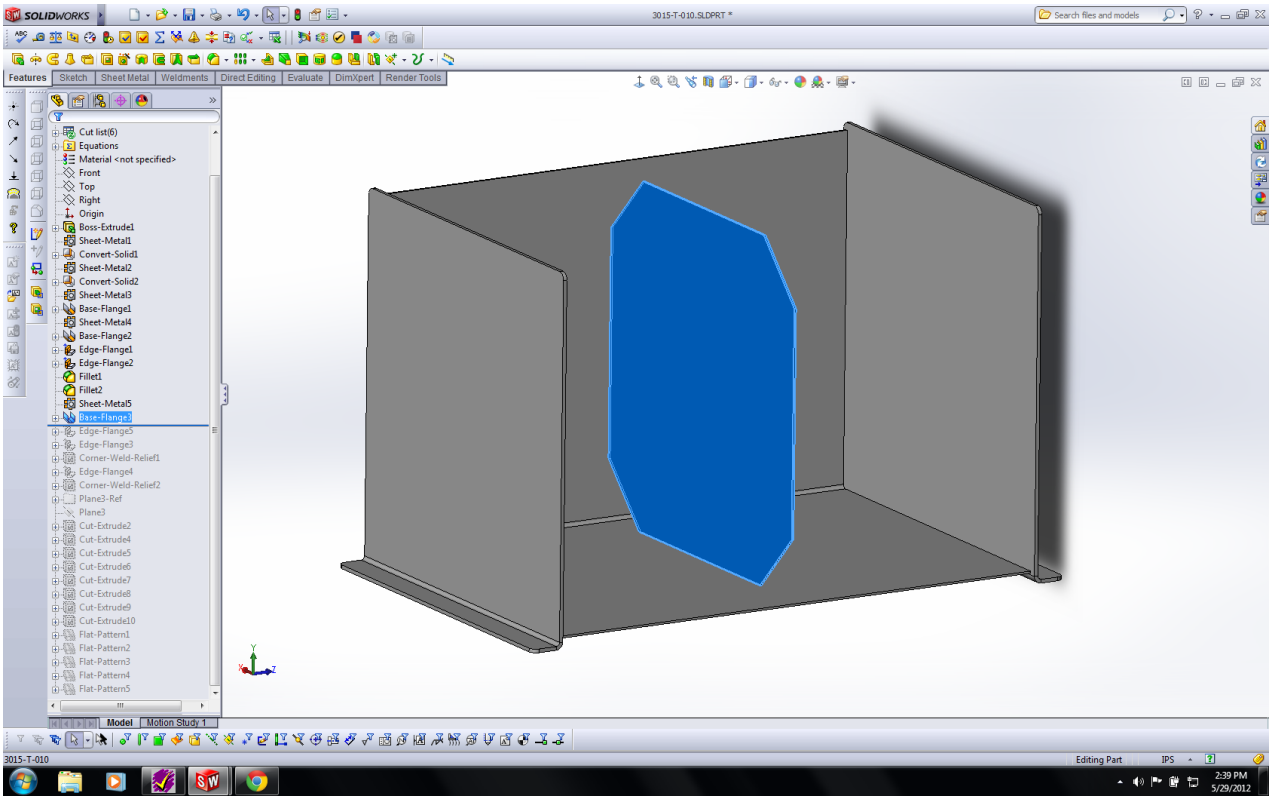
STEP 7:



⇒ In the Feature Manager -> Cut List, find the initial solid body used to model the tank and hide it.

⇒ Using the Edge Flange command add a 1” weld flange to the one edge of the tank skin. Repeat the process 180 degrees opposite on the opposing tank skin.

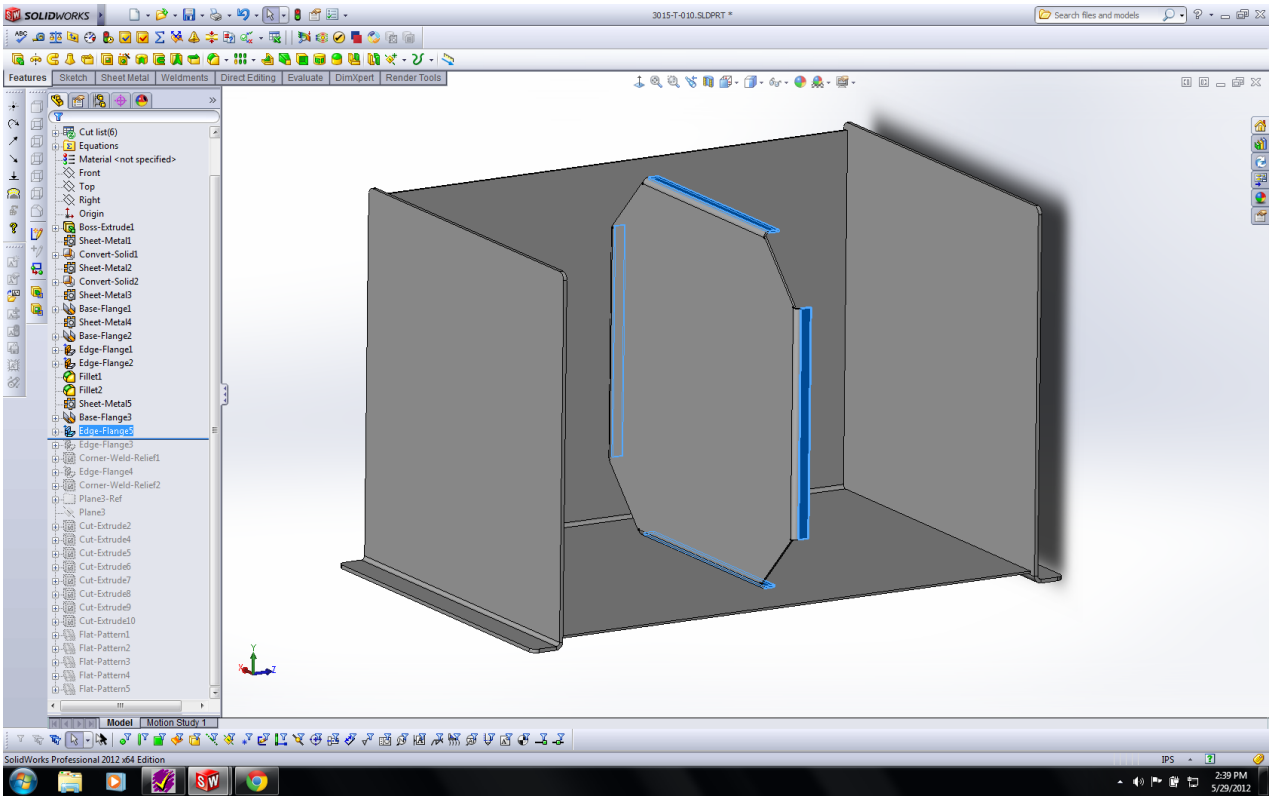
STEP 8:



⇒ Using the Front Plane, insert a base flange linked to the size of the original body. Chamfer the corners for fluid flow. This will become the internal fluid baffle.

⇒ Use collinear sketch relations between the baffle and the original body sketch to maintain associativity.

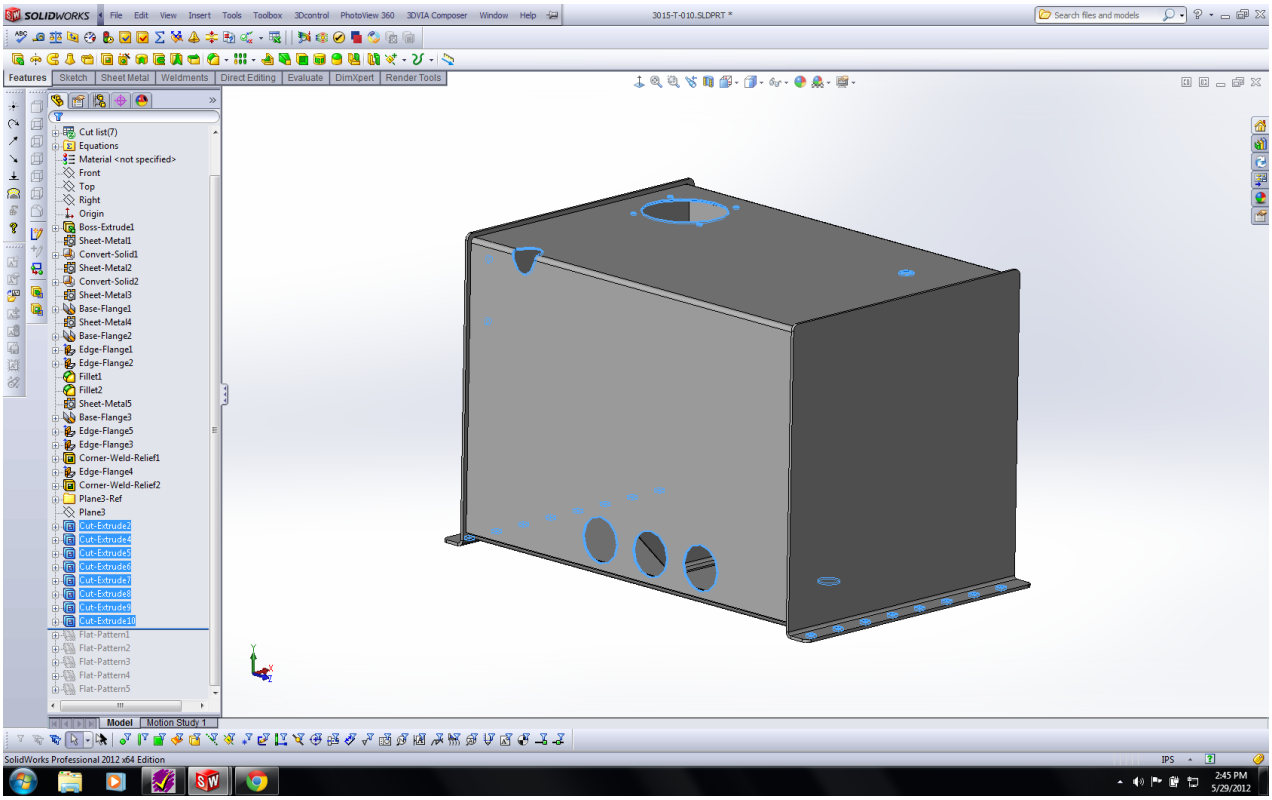
STEP 9:



⇒ Using the Edge Flange command insert weld flanges on the four long edges of the internal baffle.

⇒ Select flange profiles that will keep the flanges within the internal envelope of the tank.

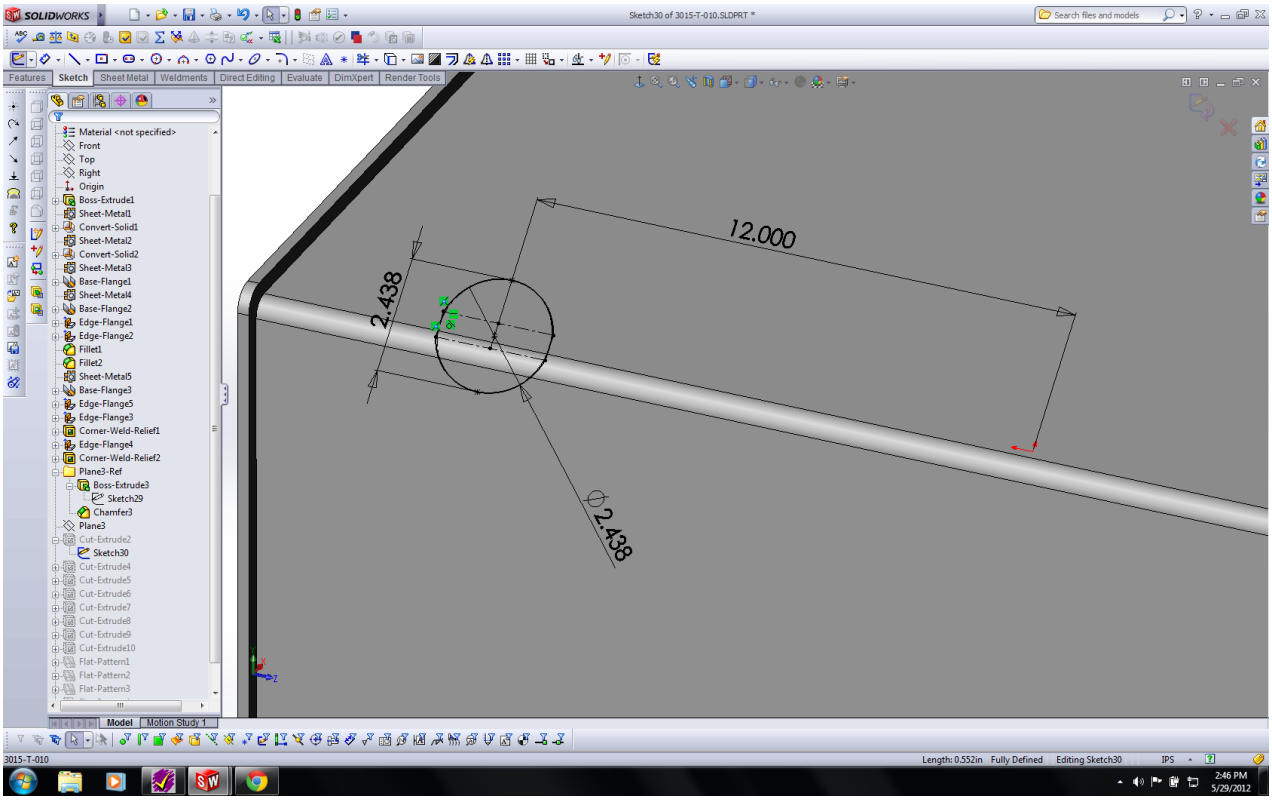
STEP 10:



⇒ Using Cut Extrude commands insert all the desired porting, filler, drain, and mounting holes in your tank.

⇒ If desired these can be easily suppressed for different configurations.

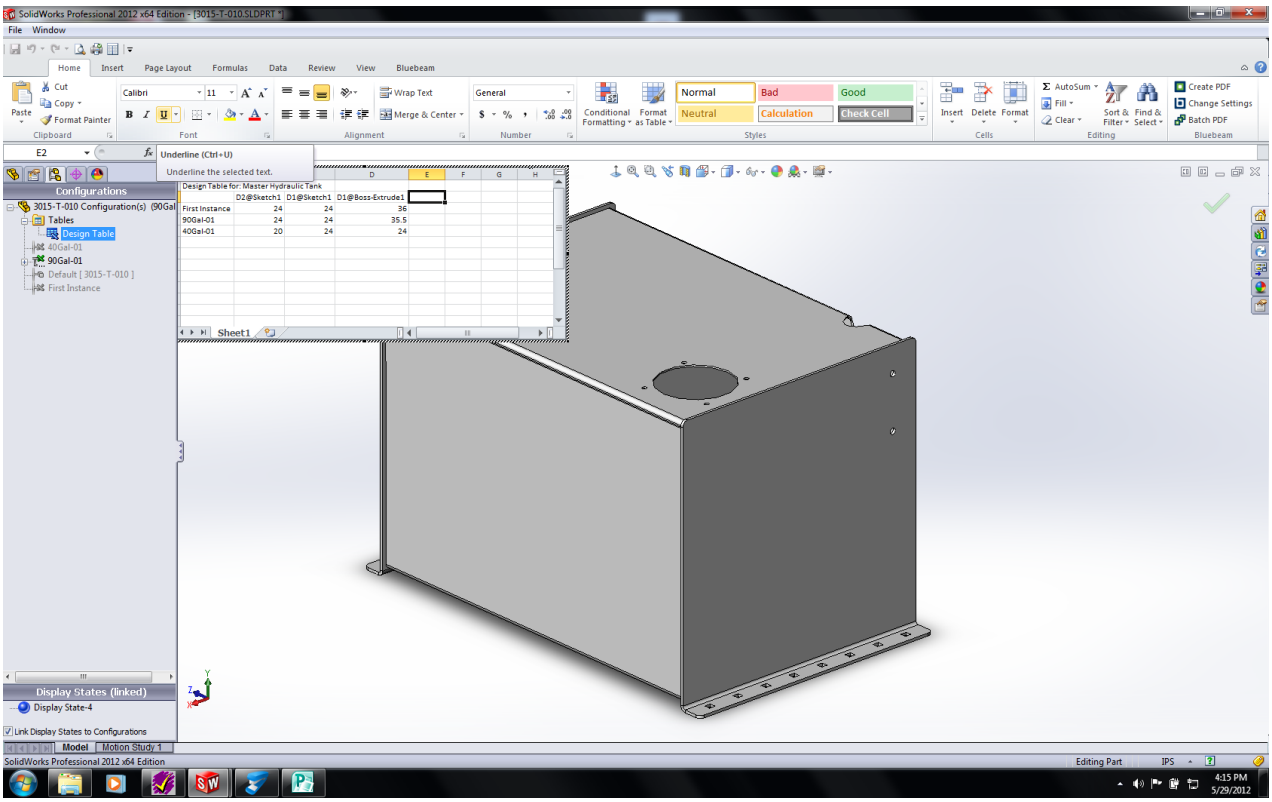
STEP 11:



⇒ If a corner mounted filler neck will be used, care will have to be taken in the cut extrude sketch.

⇒ SolidWorks 2012 cannot extrude a curved cut through a sheet metal bend. To create a circular hole for the filler neck, straight line elements will have to be used through the bend radius.

STEP 12:



⇒ Insert a Design Table and capture all of the desired sizing dimensions that control fluid capacity.

⇒ Create variations of capacity as necessary

⇒ Using Derived Configurations suppress unnecessary porting, filler, drain, or mounting holes.