

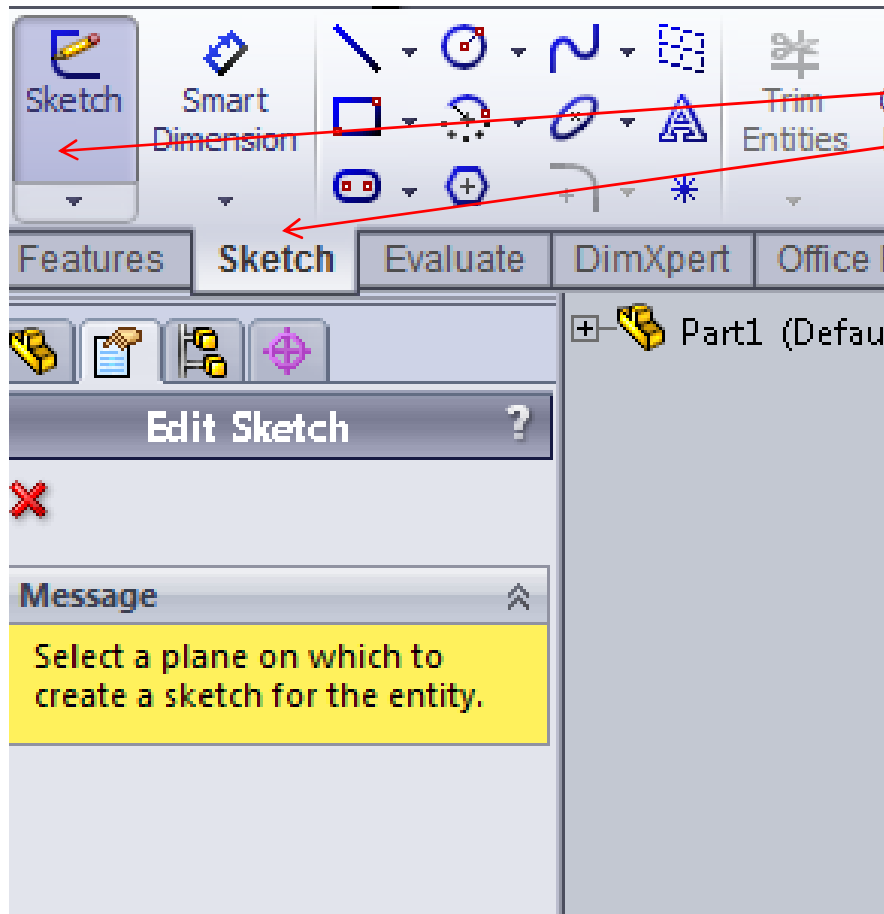
SolidWorks: Extruded Cuts, Fillet, and Patterns

Introduction to Robotics

Let's Design a Cover Part

- Choose FILE, then NEW PART
- Right click on your part in feature manger and choose DOCUMENT PROPERTIES.
- Change the UNITS to MMGS
- Save part and call it COVER PART

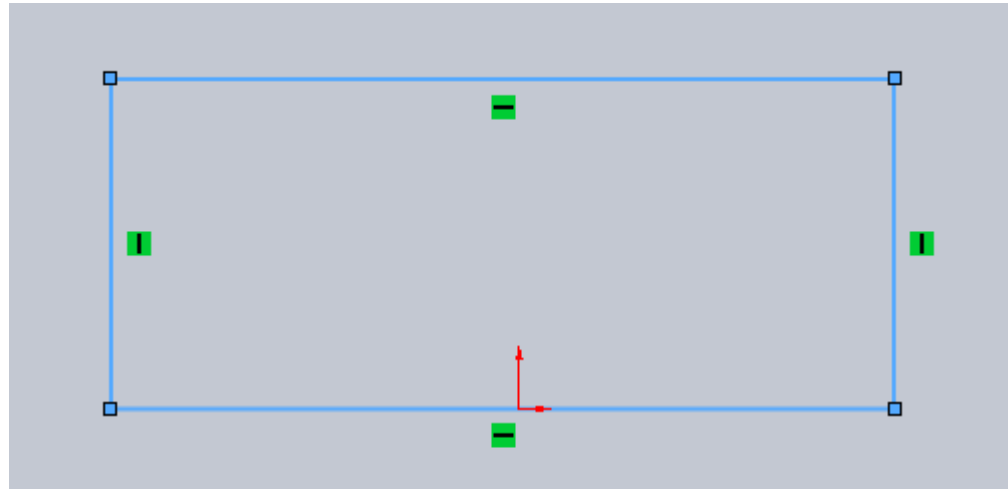
Cover Part



Let's begin by choosing the SKETCH tab and then SKETCH.

CHOOSE THE **FRONT PLANE** when asked to select a plane.

Cover



Using the rectangle option make a corner rectangle with the lower horizontal line touching the origin.

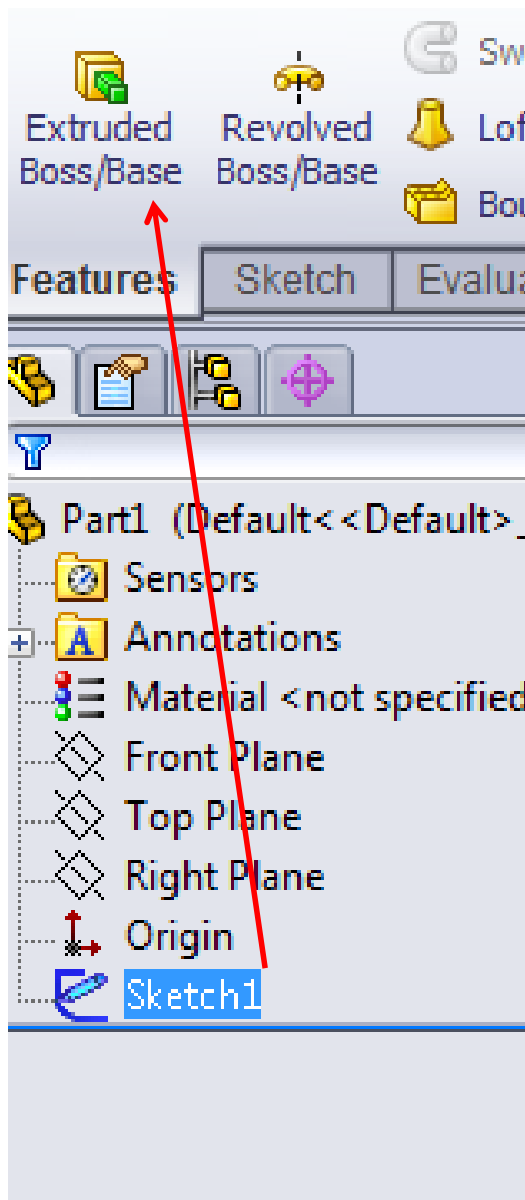
Using the CTRL key, select the MIDPOINT of the line and the origin and add a **COINCIDENT RELATION**.

Cover Part



Dimension the rectangle with the horizontal being 67.6 mm and the vertical being 12.6 mm. You should notice that the sketch is now **FULLY DEFINED**. Exit the sketch!

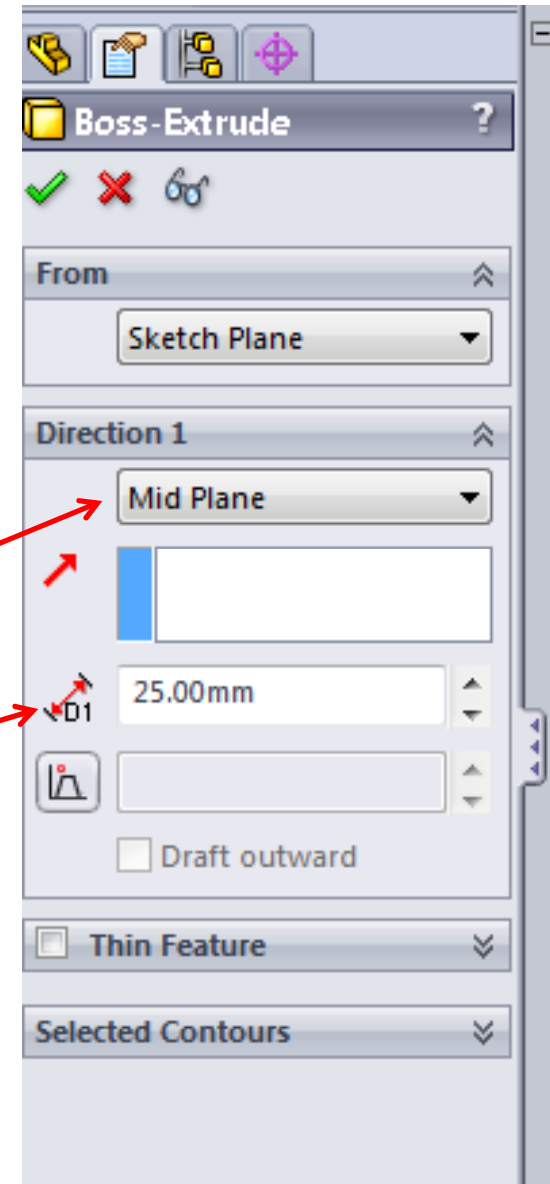
Cover Part



Click on your sketch in feature manager then choose **EXTRUDED BOSS/BASE** from the features tab.

Property manager will open. For direction choose **MIDPLANE** and for the distance enter **25.0 mm**

Click the green check!

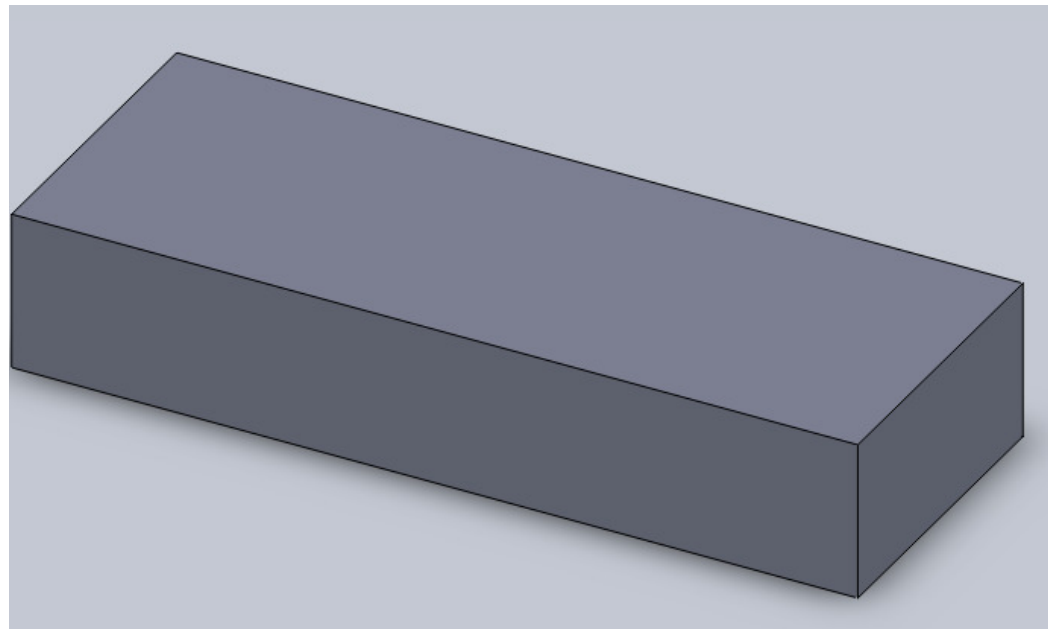


Cover Part



Choose **VIEW ORIENTATION** at the top of the design field and choose **ISOMETRIC**.

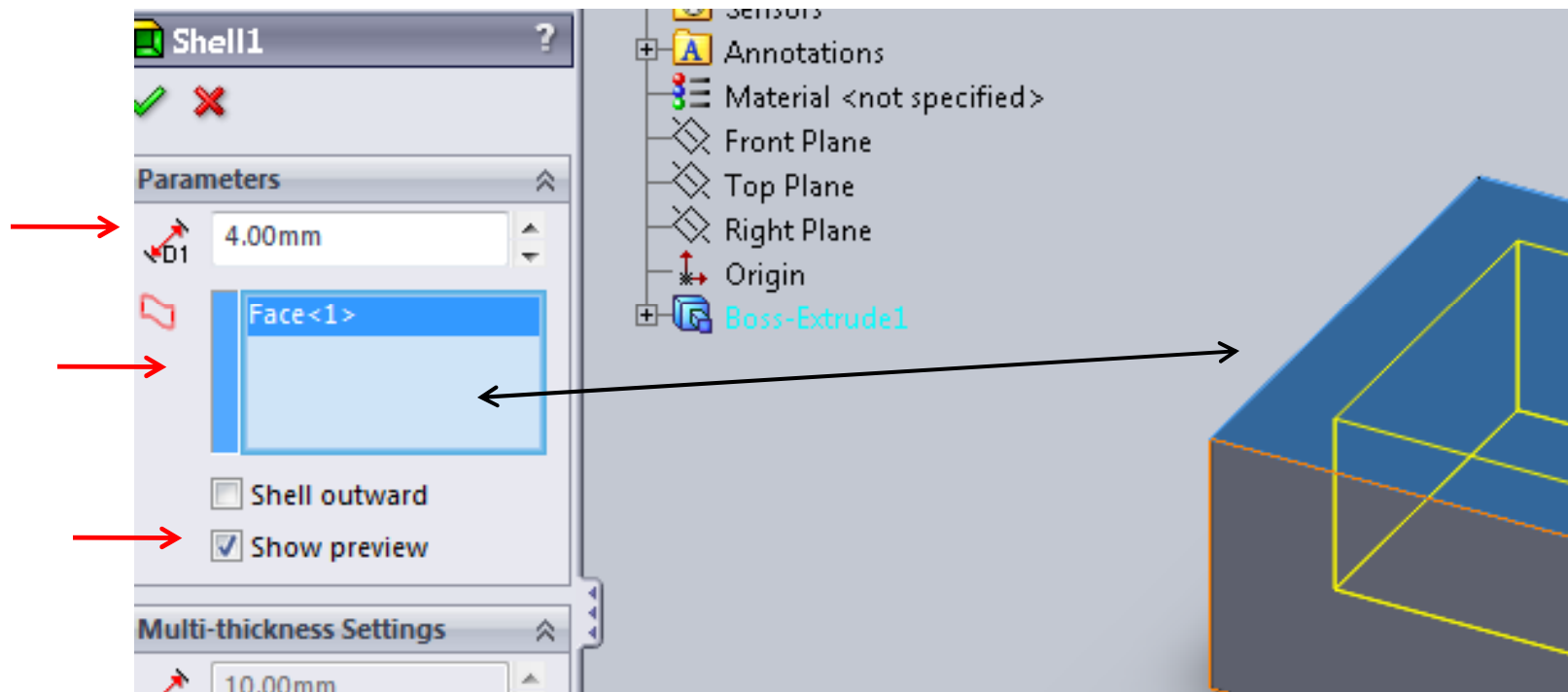
Save your file!



Shell Feature

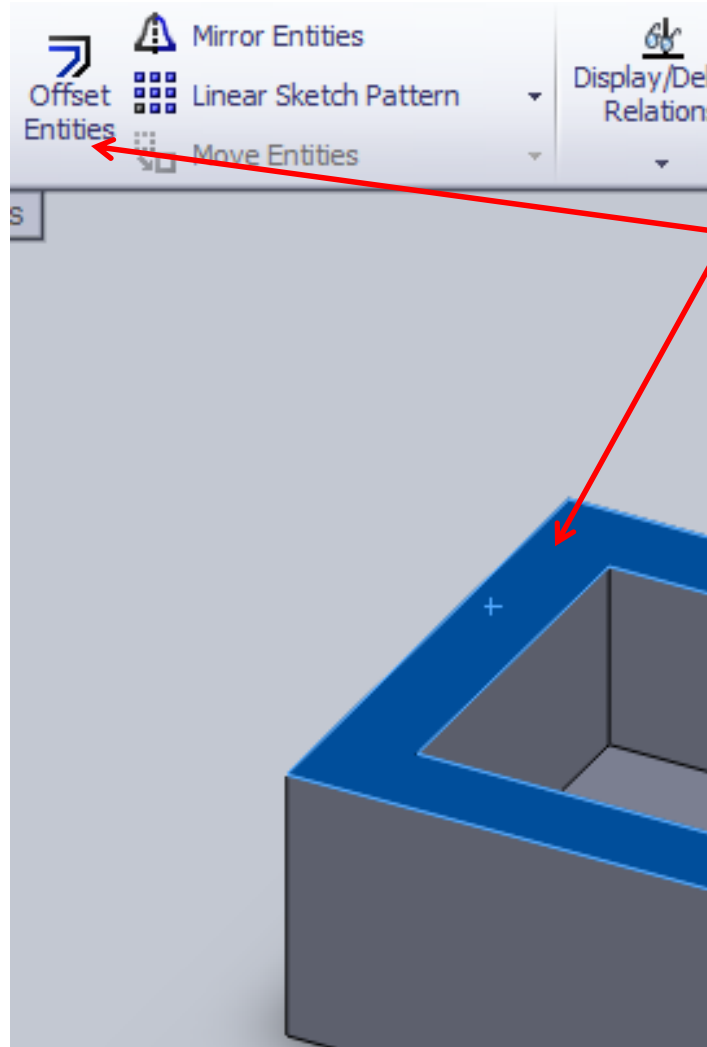


Under the FEATURES tab choose SHELL.



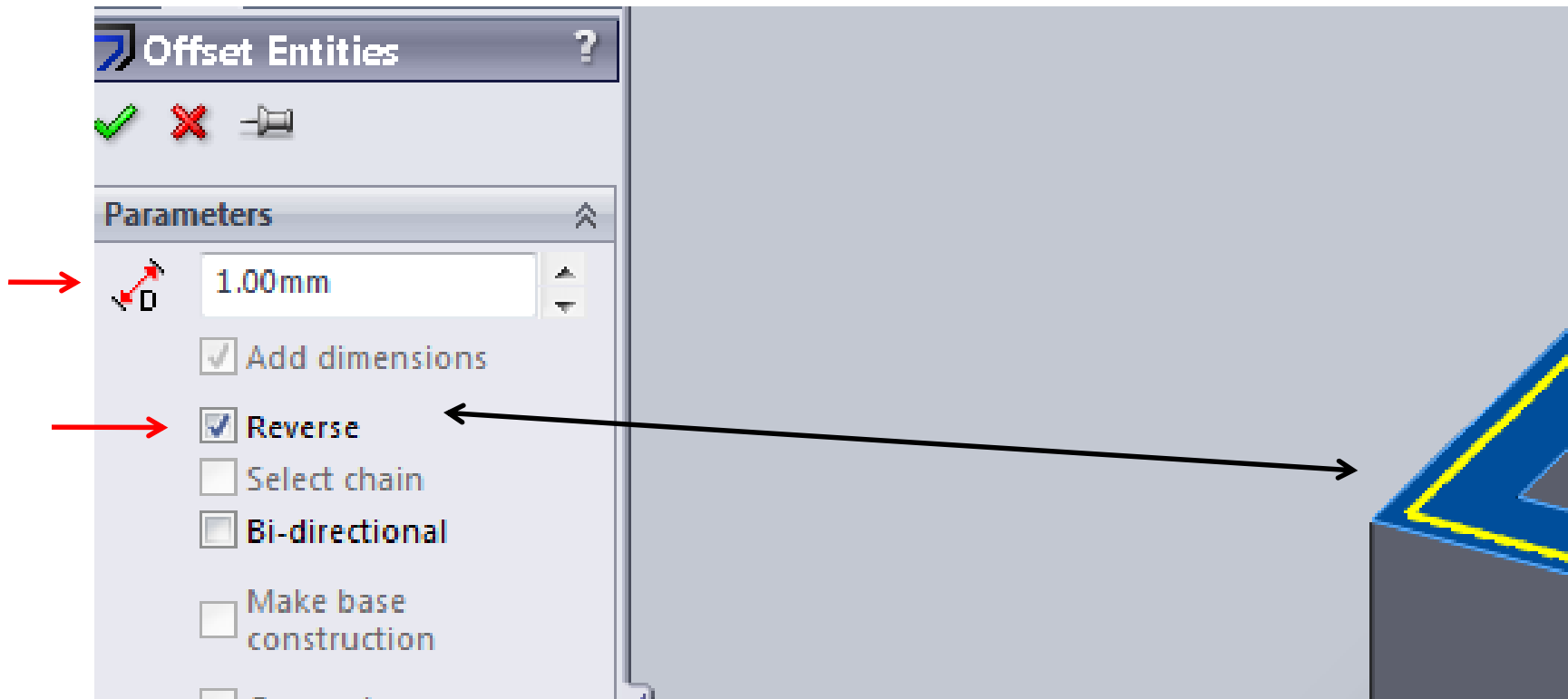
For the distance enter **4.0 mm**. Then select the **TOP FACE** of your part. This will tell SolidWorks which plane you want to shell out. Click the box **SHOW PREVIEW** to see the shell feature. **Click the green check. SAVE YOUR FILE.**

Extruded Cut - Offset



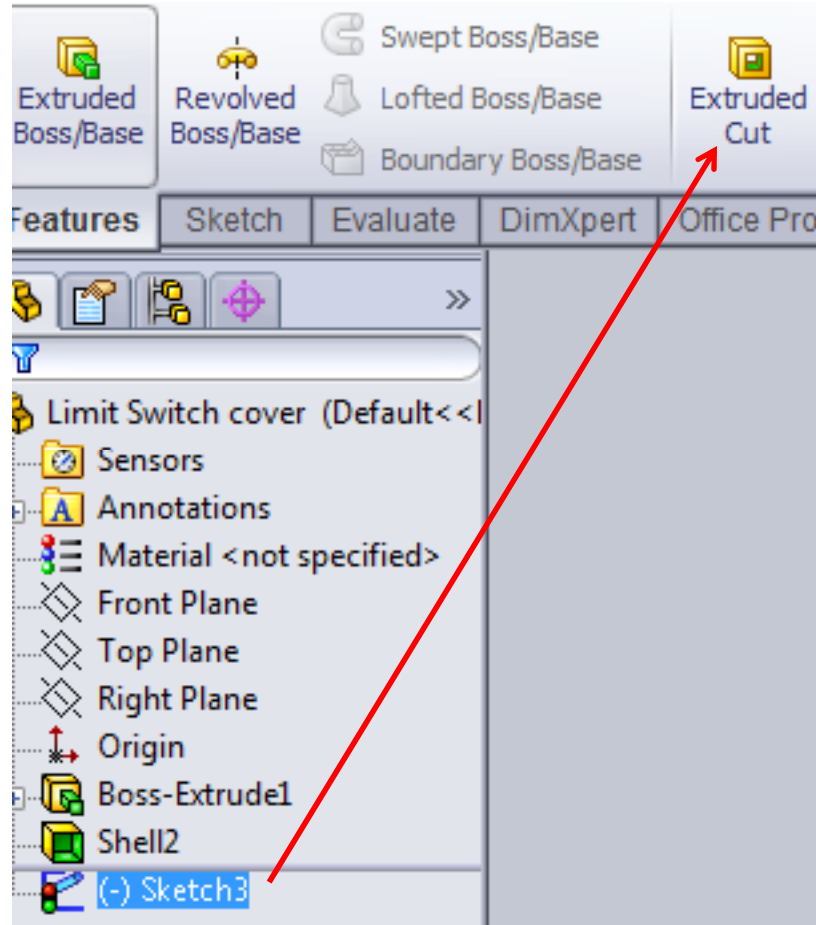
Right click on the top face of your part and find the SKETCH button. Click this and then the OFFSET ENTITIES button on the tool bar above.

Extruded Cut - Offset



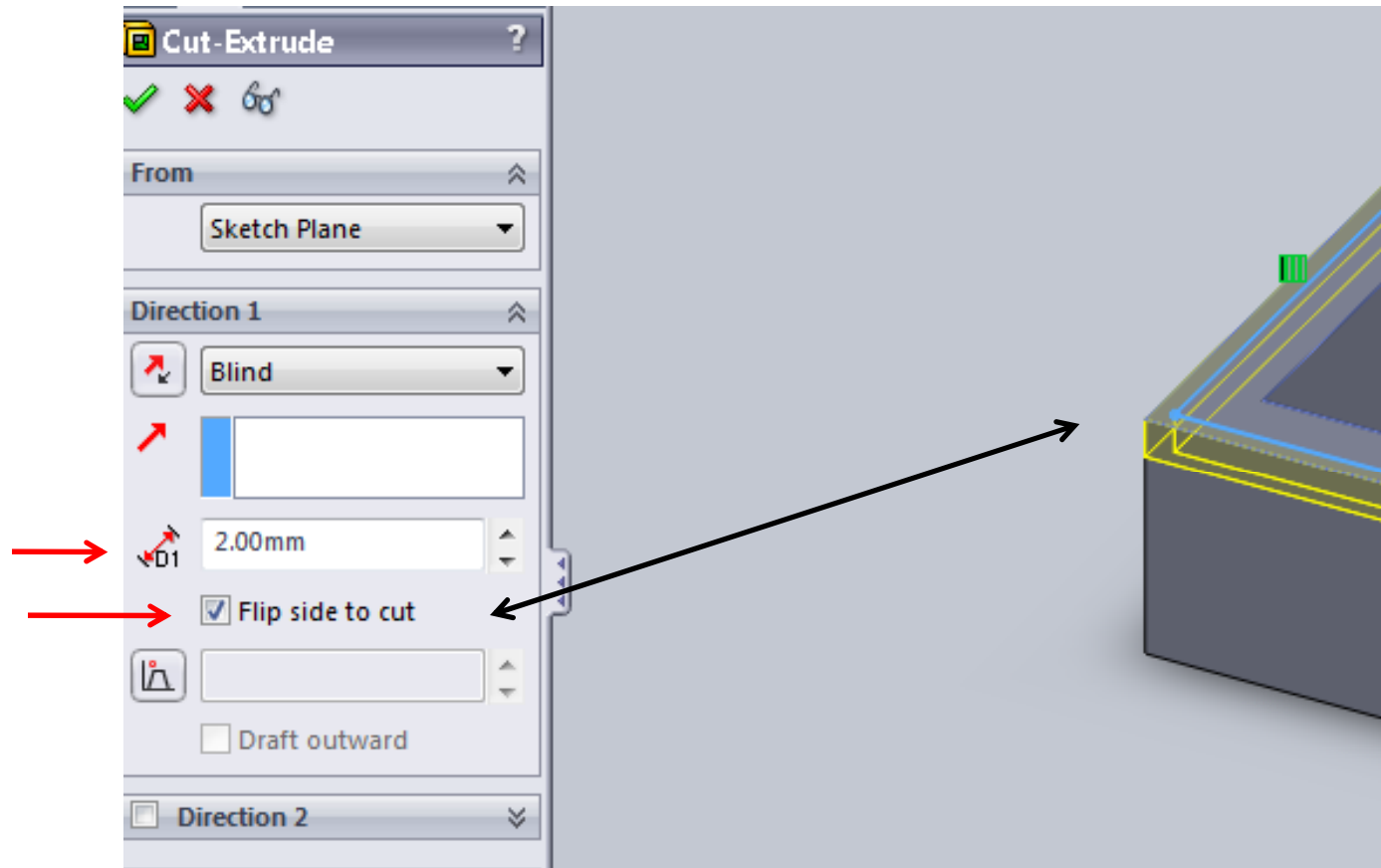
Enter **1.0 mm** for the distance and click the **REVERSE** box. You should now see an outline around the edge of your part. **Click the green check and save.**

Extruded Cut - Offset



Click on your new sketch in feature manager then select EXTRUDED CUT from the toolbar above.

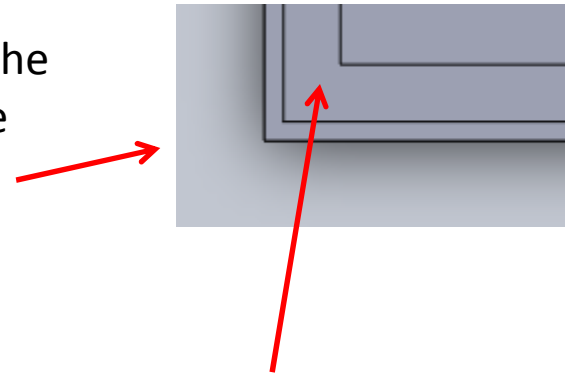
Extruded Cut - Offset



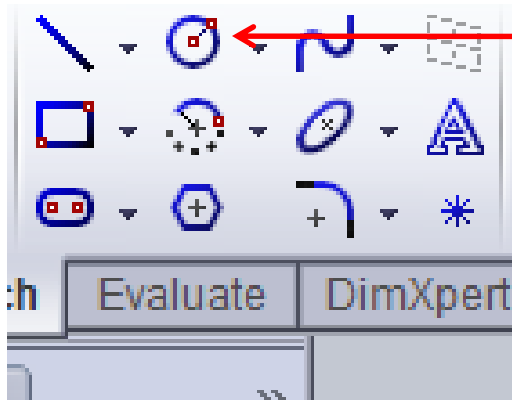
Enter a depth of **2.0 mm** and then check the box **FLIP SIDE TO CUT**. You should notice it cutting around the outside edge. Click the green check.

Making a circular hole

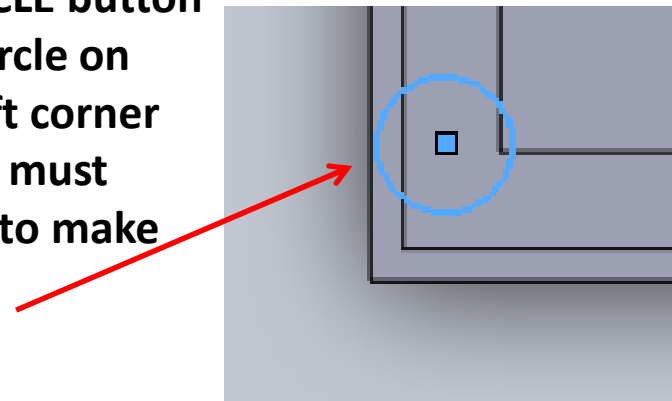
Choose the VIEW ORIENTATION button on the design plane above your part and select the **TOP VIEW**.



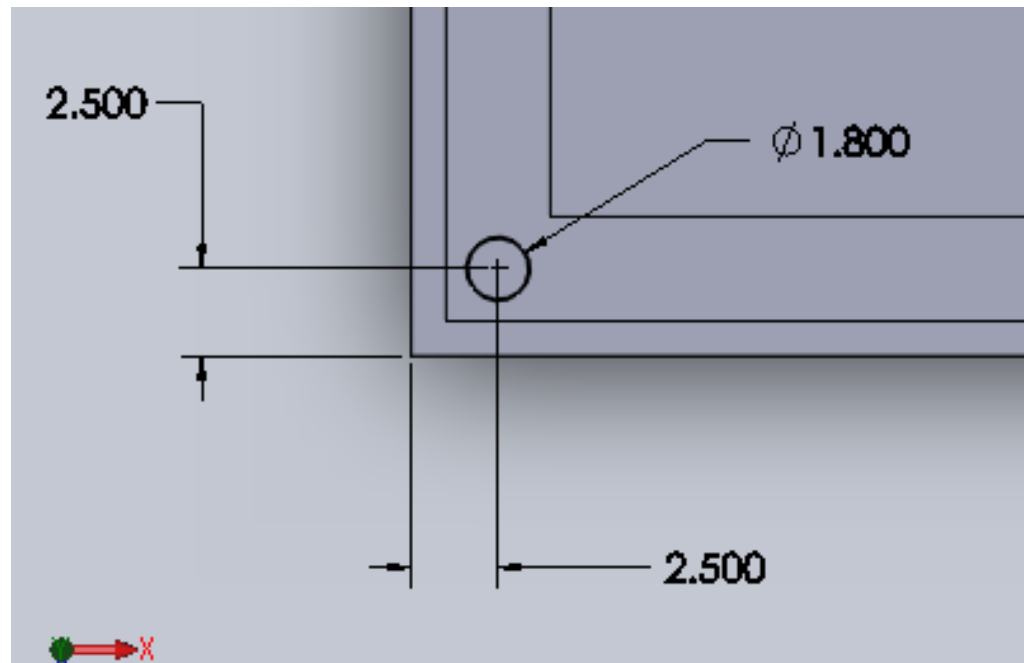
Select **SKETCH** from the Sketch tab and select the top plane of your part.



Select the **CIRCLE** button then draw a circle on the bottom left corner as shown. You must click and hold to make the circle.

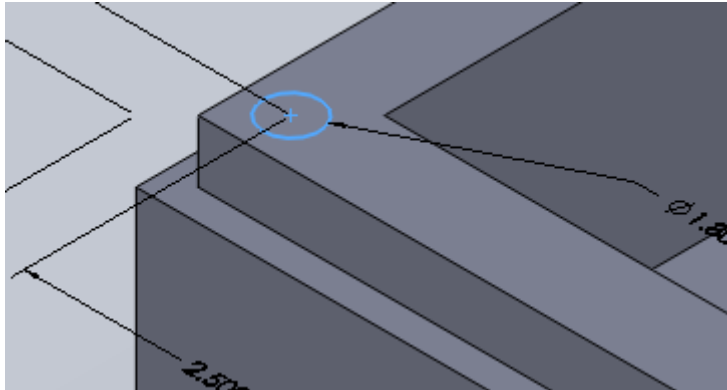


Making a circular hole



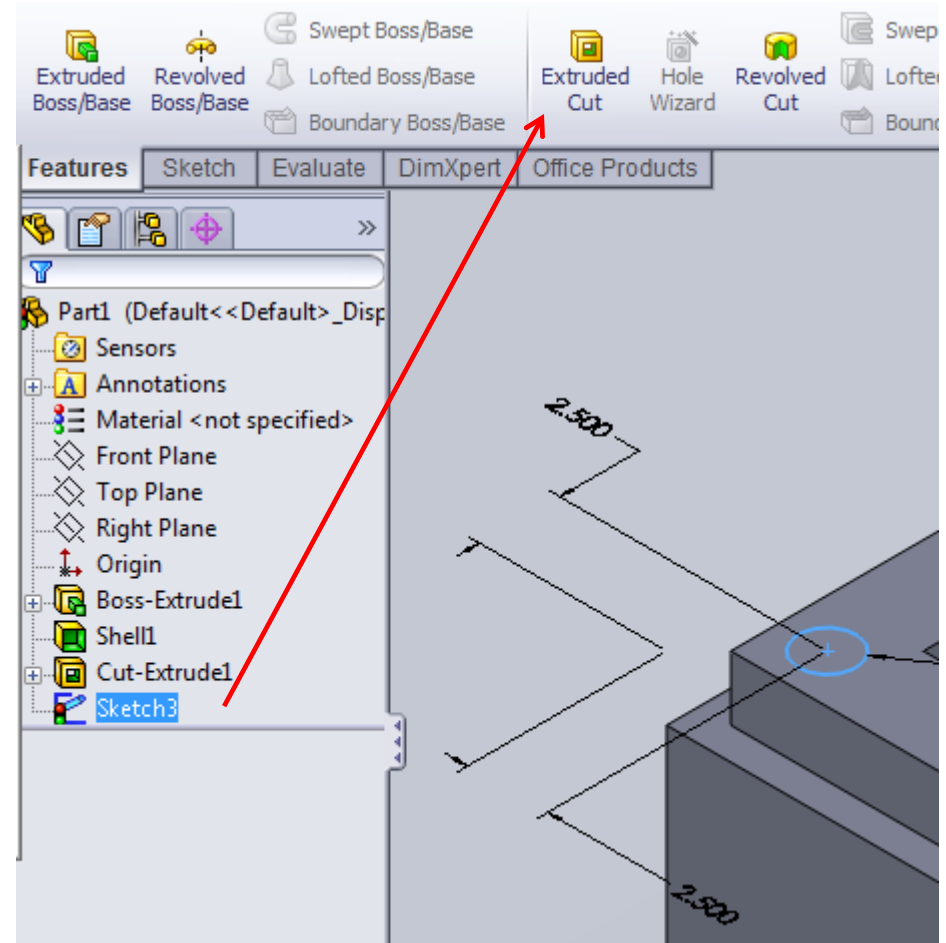
Using smart dimension, click on the outside circumference of the circle. Enter the dimension as 1.800 mm , which is its diameter. Dimension the CENTER of the circle with each OUTSIDE edge to 2.50 mm.

Making a circular hole



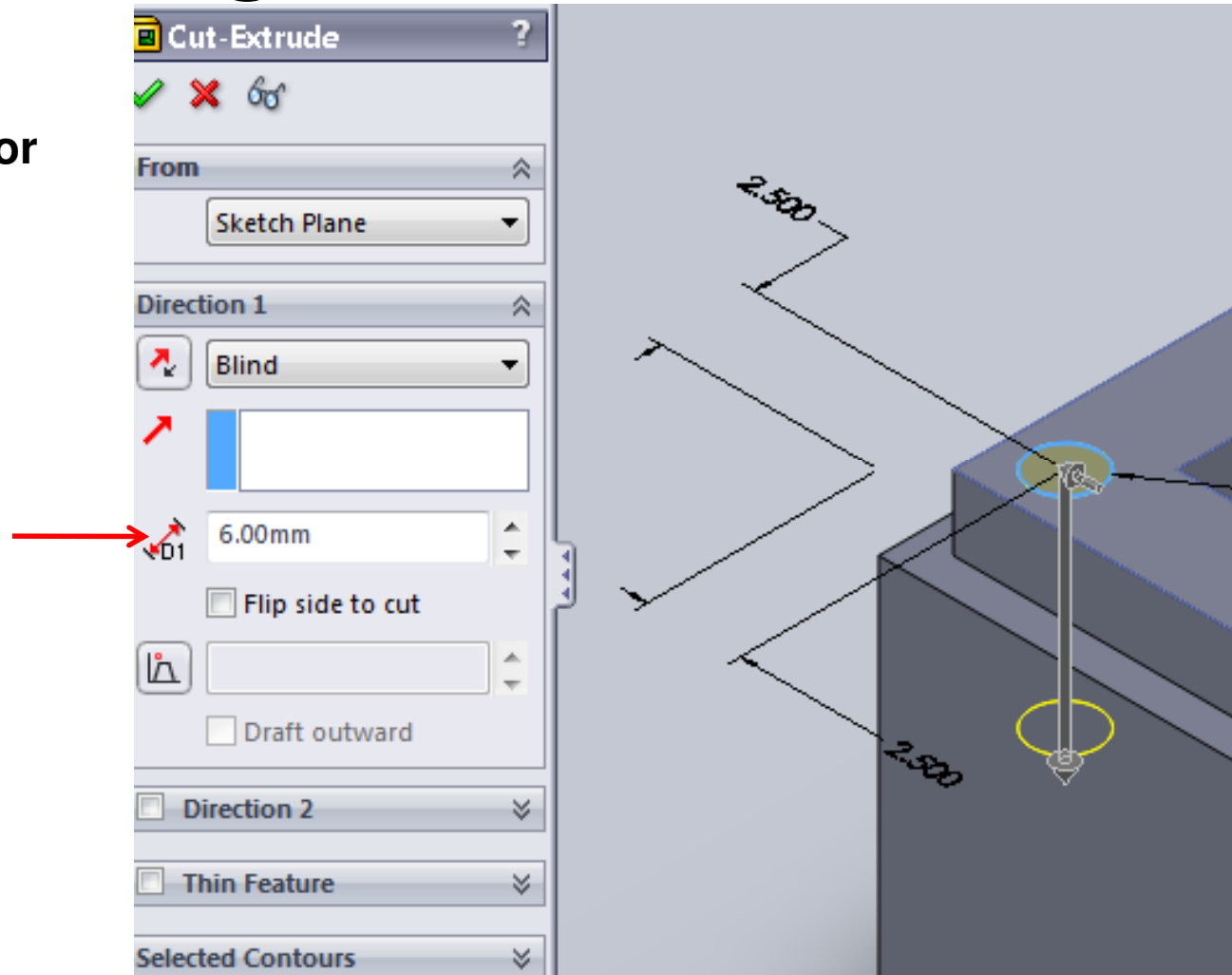
Make sure your sketch is highlighted in feature manager and select **EXTRUDED CUT** from the toolbar.

Switch your orientation to isometric.



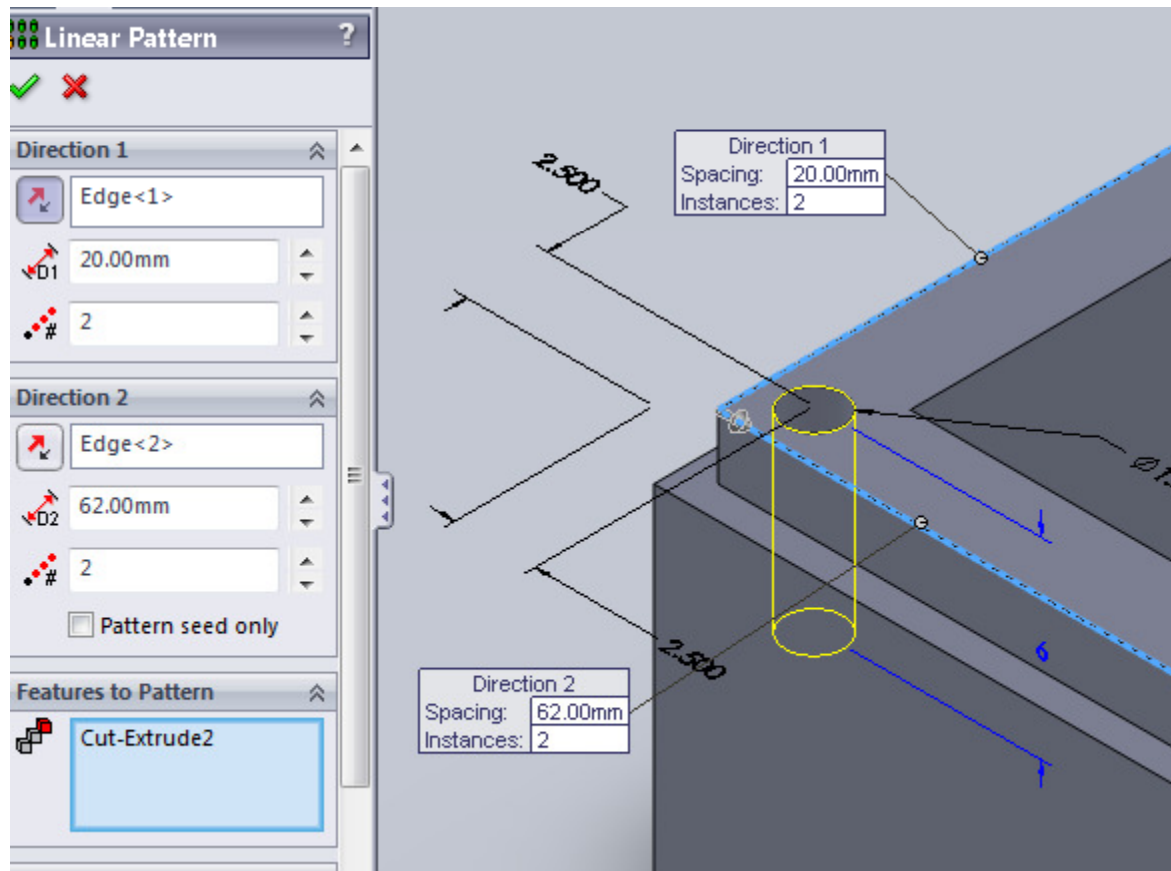
Making a circular hole

Choose **6 mm** for the depth and click the green check



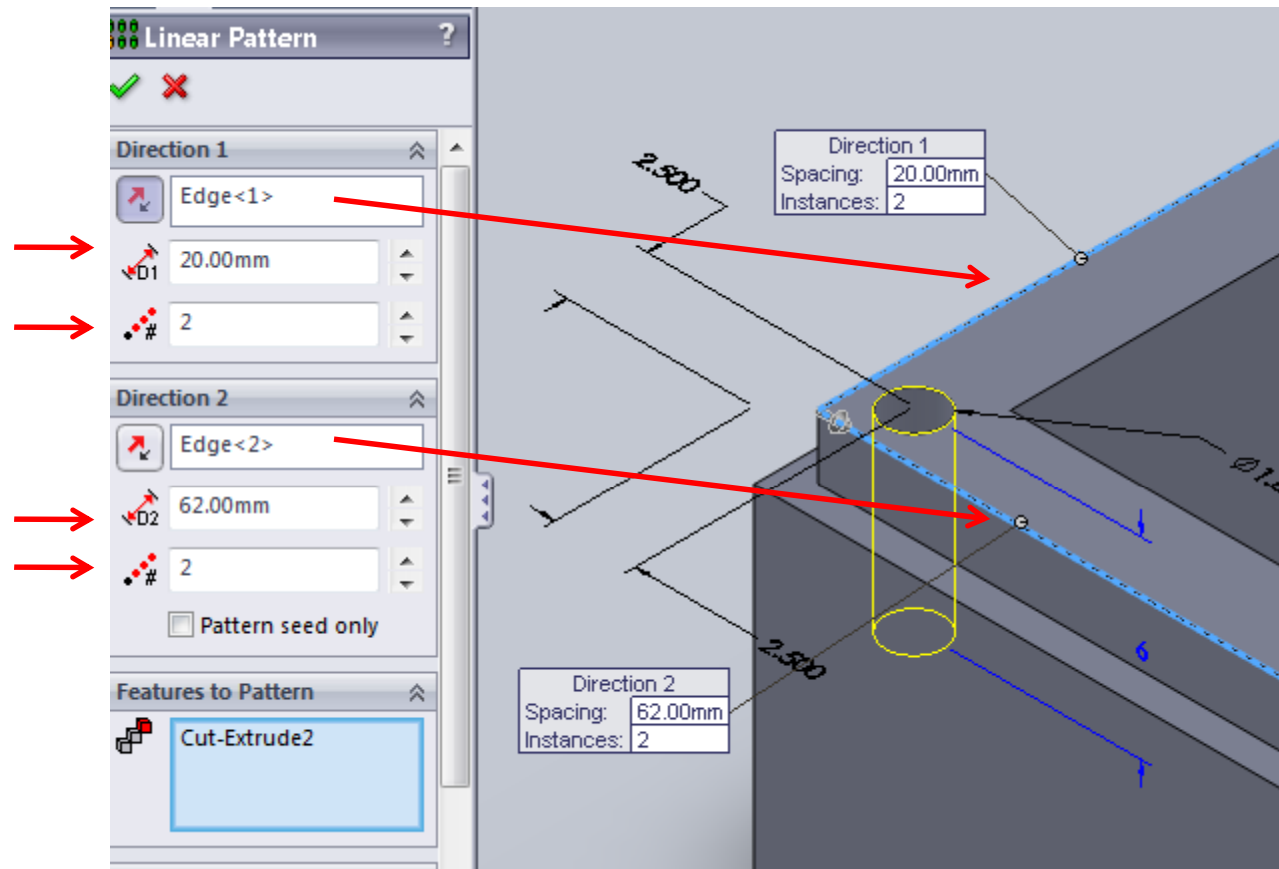
Using a Linear Pattern

Make sure the extruded hole is highlighted in feature manager then select linear pattern from the tool bar.



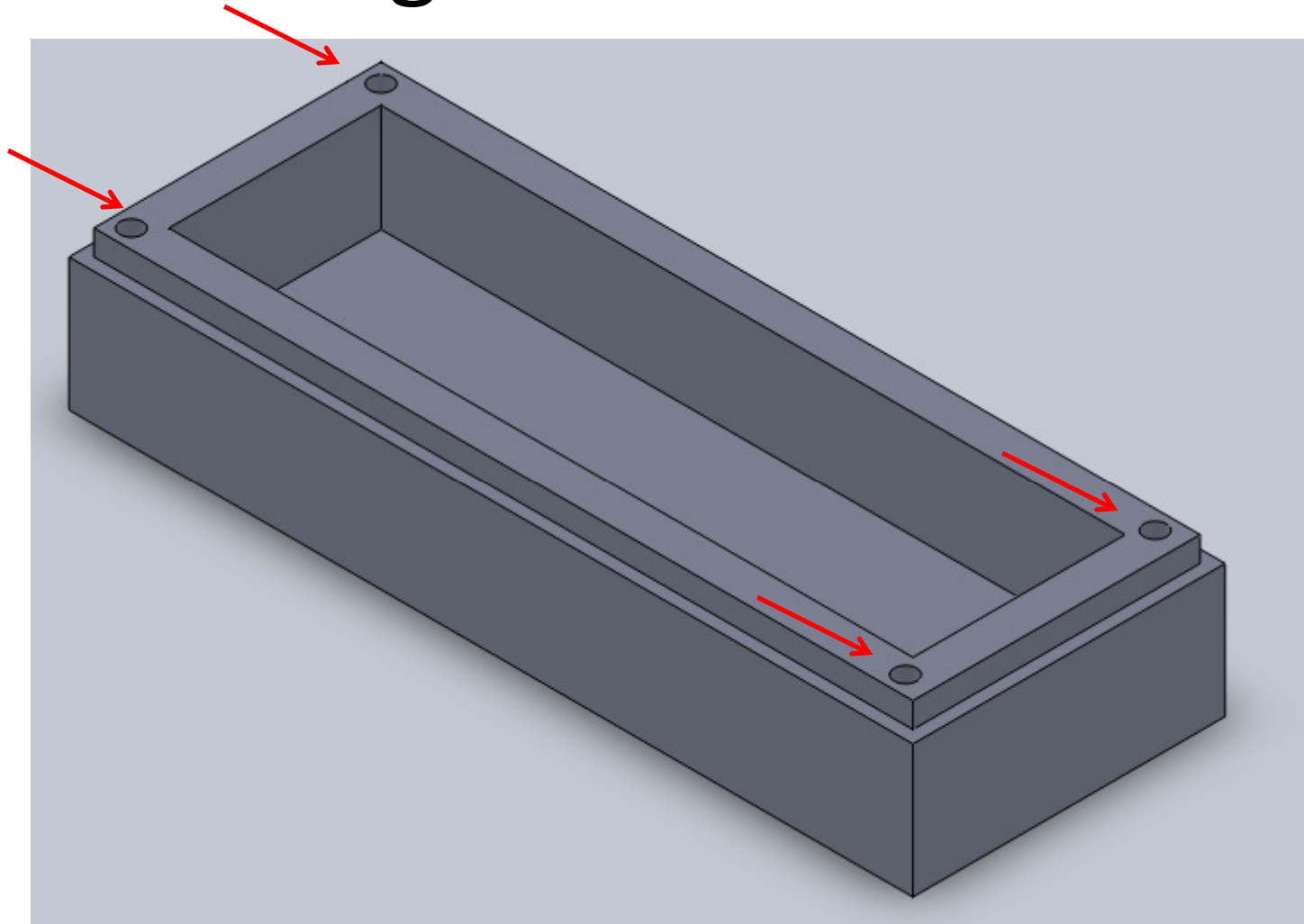
Using a Linear Pattern

For Direction 1 select the LEFT EDGE next to your hole. Enter in 20 mm for the distance and make sure the number “2” for the # of instances.



For Direction 2 select the BOTTOM EDGE next to your hole. Enter in 62 mm for the distance and make sure the number “2” for the # of instances. Click the green check!

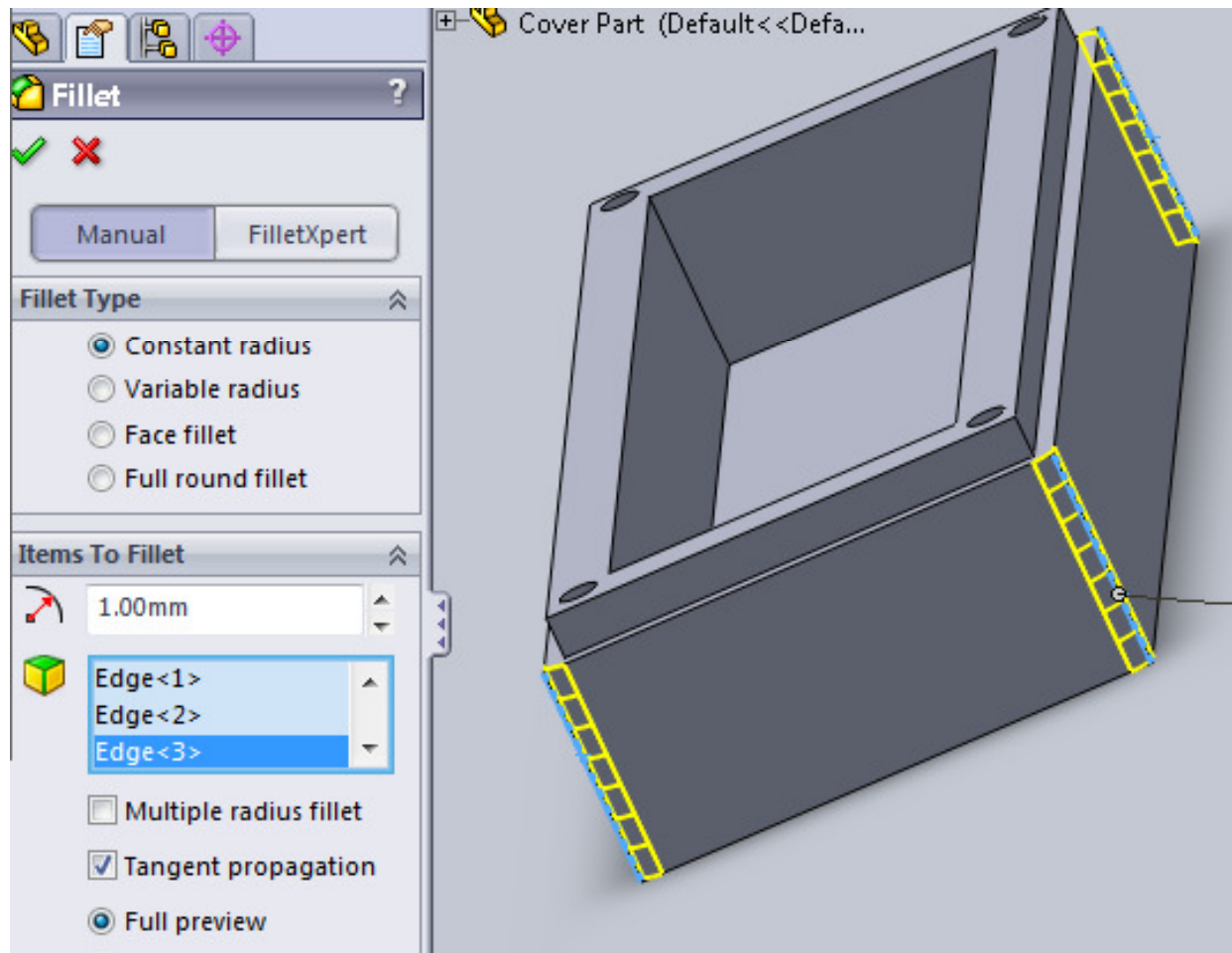
Using a Linear Pattern



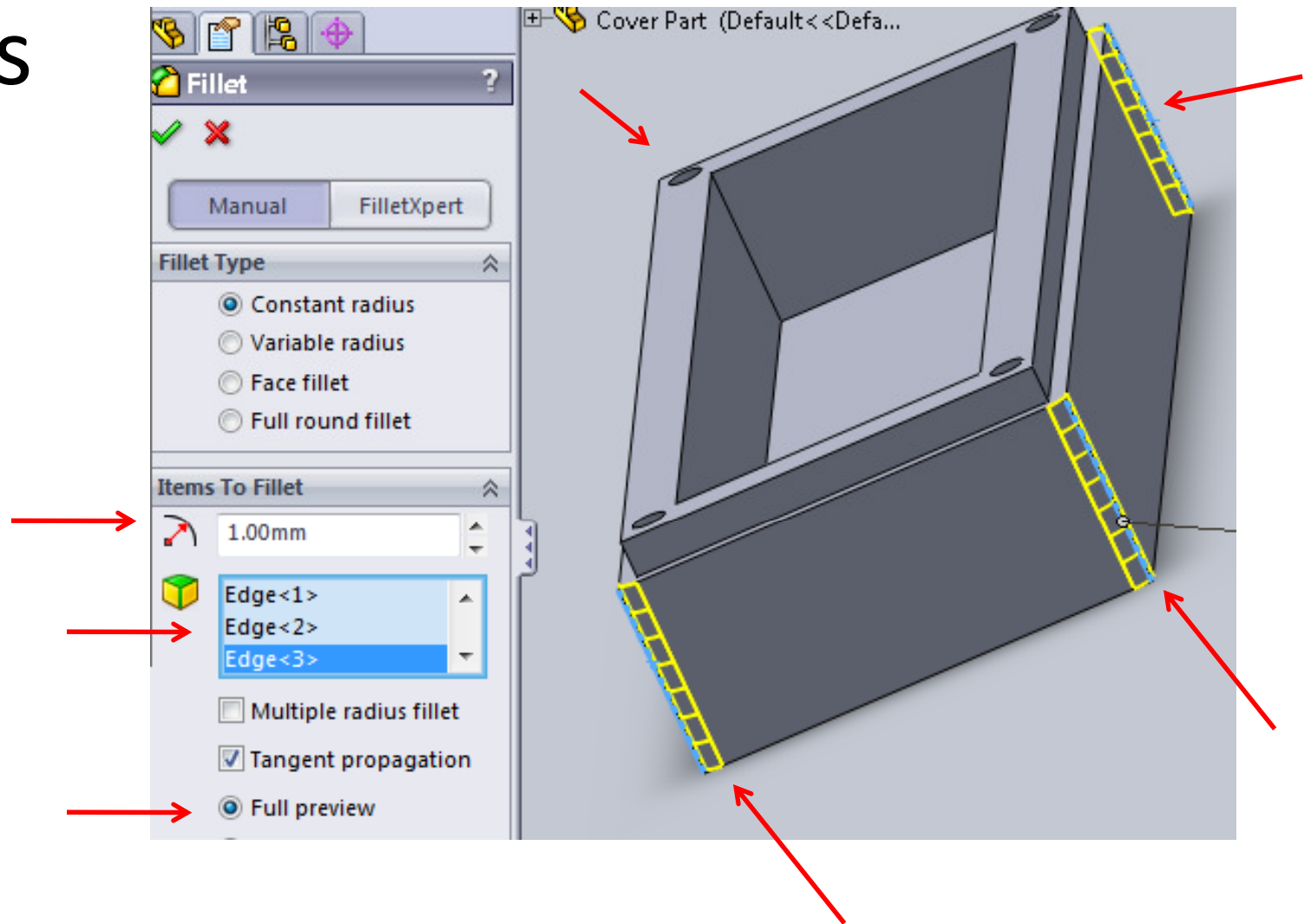
As you can see, linear pattern saves us the hassle of make the hole 3 more times.

Fillets

Choose FILLET from the toolbar.



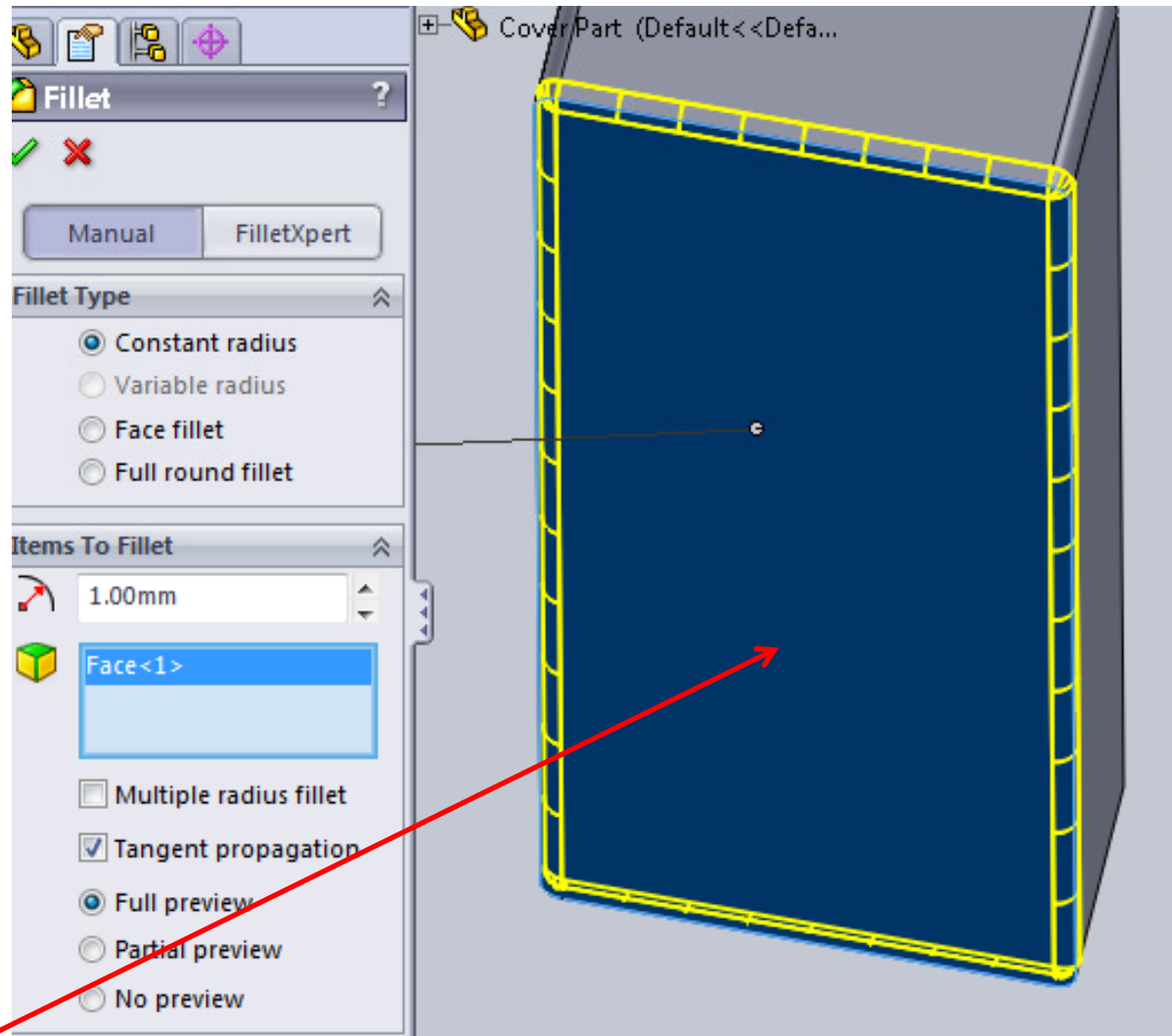
Fillets



For the radius of the fillet enter in 1 mm. Select the FOUR edges around the part and make sure FULL PREVIEW is selected so you can see the wireframe. Then click the green check.

Fillet

Click FILLET again.
Enter 1 mm for the
radius.



Select the bottom face. This will fillet the complete outside edge. Click the green check.

Cover Switch Drawing

Make a SolidWorks drawing for the cover part. Make sure each view, except isometric, has dimensions. Print this drawing out. Write your name in the title block and turn it in.