

### **Overview of Instructable :**

- This Solidworks instructable part will help teach the basic concepts and understanding of the Solidworks program as well as 3D design. Creating this part will teach students how to use basic Solidworks functions that will carry with them in their future of their academic career in STEM or jobs. Understanding planes, origins, dimensioning practices, extrusions, fillets, and extruded cutting will aid students in creating not only this part, but other 3D parts in the future.

### **By the end of this lesson students will be able to (objectives) :**

- Create a part file in Solidworks
- Create a unique part in Solidworks
- Evaluate mass properties of a Solidworks part
- Have a basic understanding of 3D design and Solidworks commands

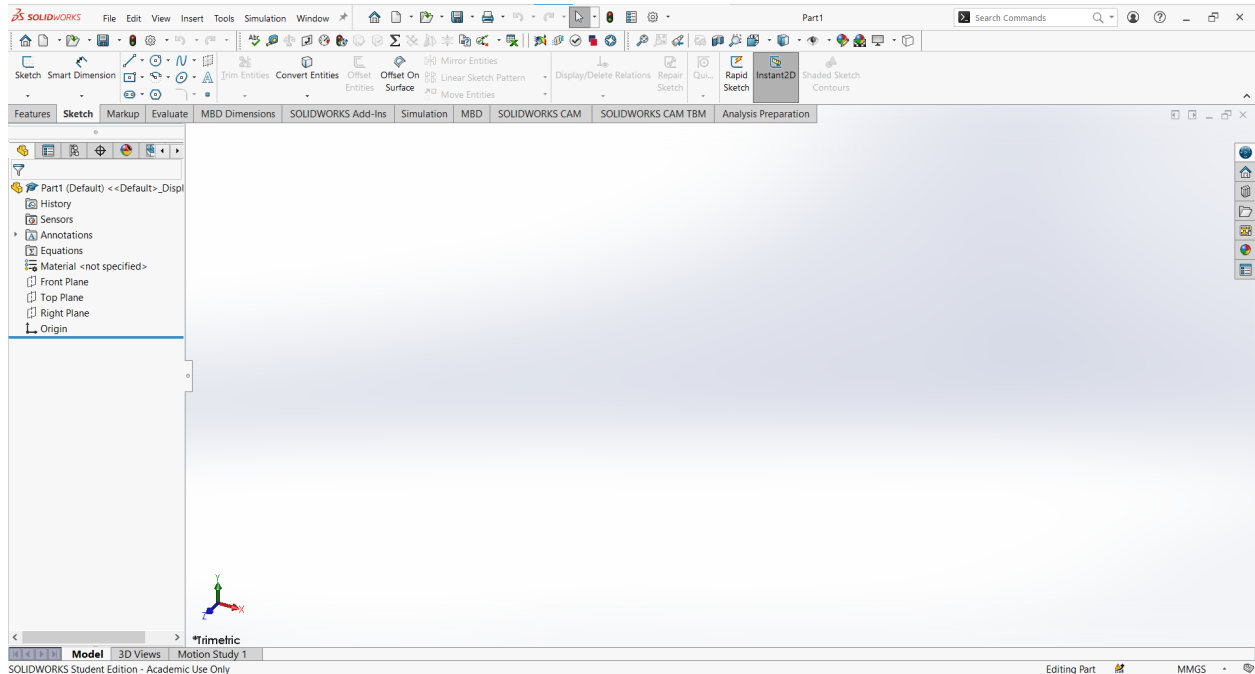
## **Standards :**

- The 8 core standards in technology and engineering education help students grasp how technology shapes our world, encouraging them to think critically and solve problems. Students will learn the basics of systems thinking and design processes, and they understand how society and history have influenced the technological process. By exploring the impacts of technology on our lives and the environment, they are prepared to create and maintain sustainable solutions, making informed decisions that contribute positively to our ever-evolving world.

**Age range (student level) :** 9th-12th grade students interested in STEM and/or graphic design.

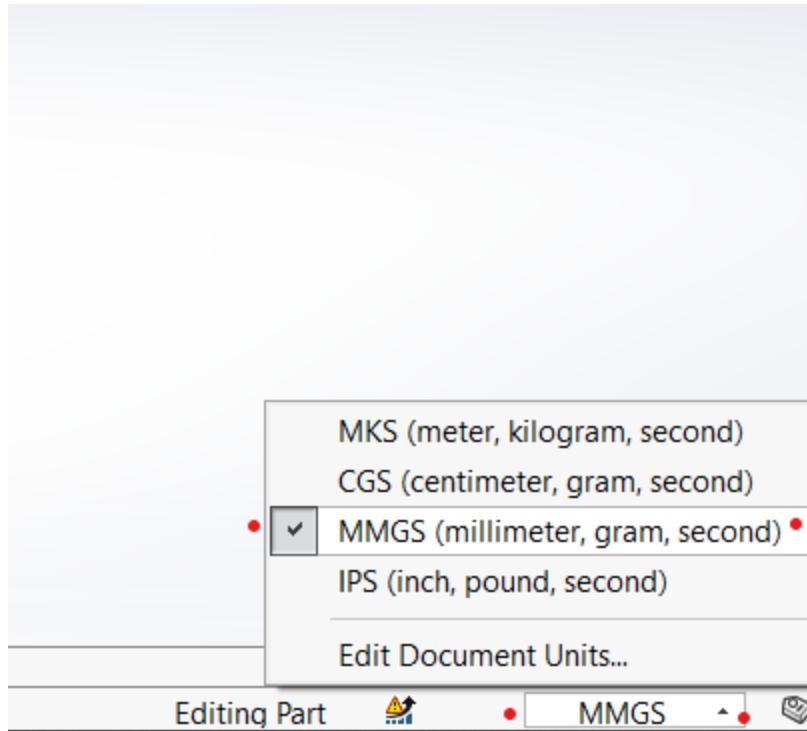
## **Step 1 : Open Solidworks and Create new part**

- Open Solidworks program, when opened a prompt may appear that will ask if you want to open a *part* file, assembly, or drawing. If the prompt appears, click *part*, if not, navigate to top left of screen, click *File < New < Part <* and *Ok*.
- Now you should be in a new workspace



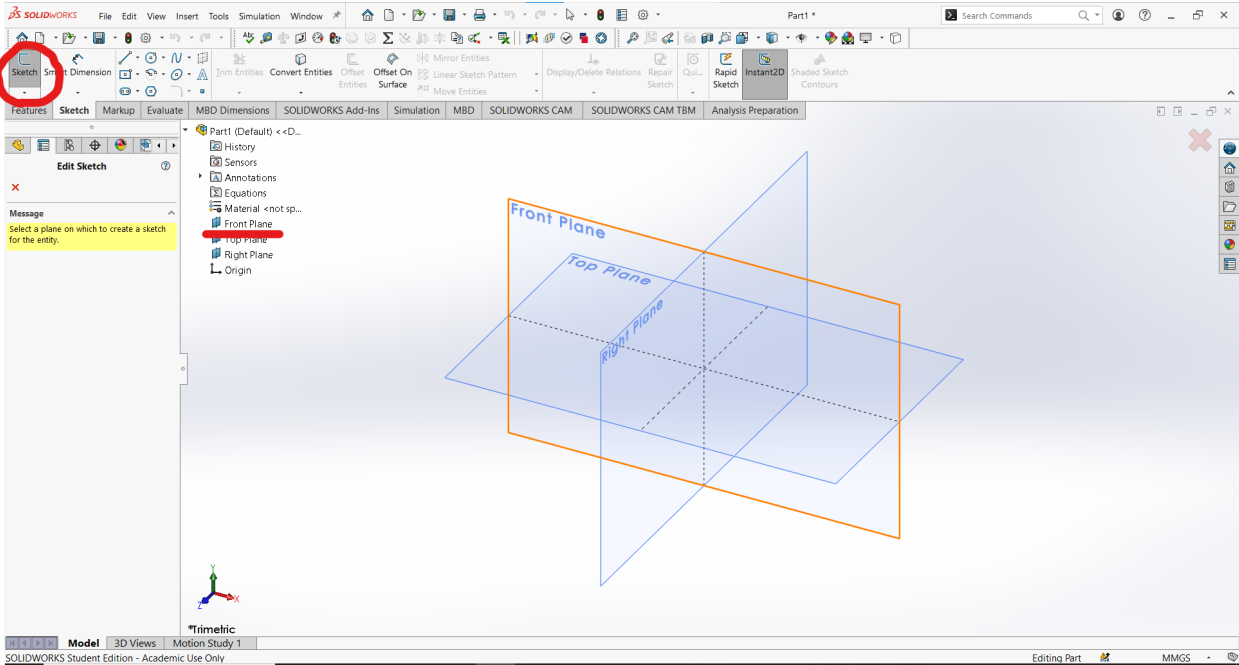
## Step 2 : Select units

- There are multiple ways to select units, this is something that should be done prior to starting any sketching to ensure accurate dimensions and mass properties. The easiest way to select units is to navigate to the bottom right corner, expand the *Unit System tab*, and select *MMGS (Millimeter, Gram, Second)*. Now your part will be created with the correct units (always be sure to check your units before making 3D parts).



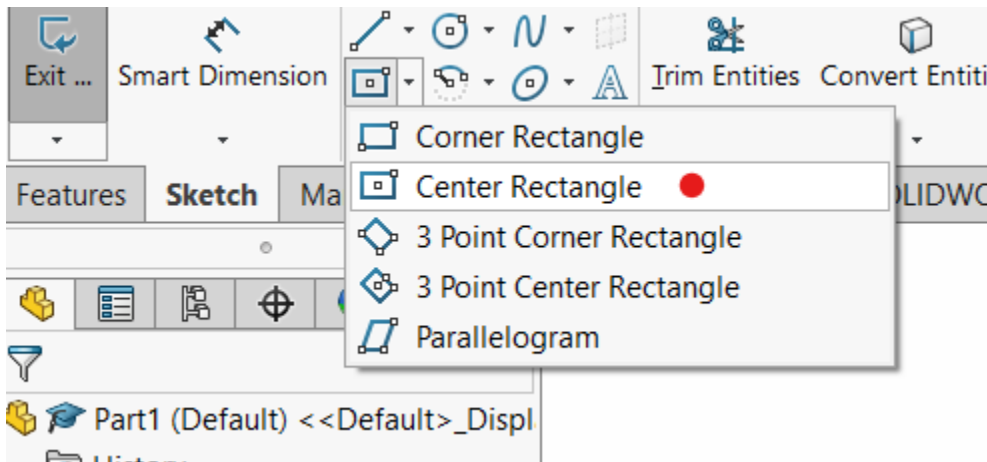
### Step 3 : Activate sketch on the front plane

- To create a new sketch, navigate to the top left corner, next to *Smart Dimension*, click *Sketch* to activate the sketch tool, then select the *Front Plane* either from the Part 1 pull down (underlined in red) or in the 3D model space (highlighted orange). Now you are ready to start sketching on the front plane.





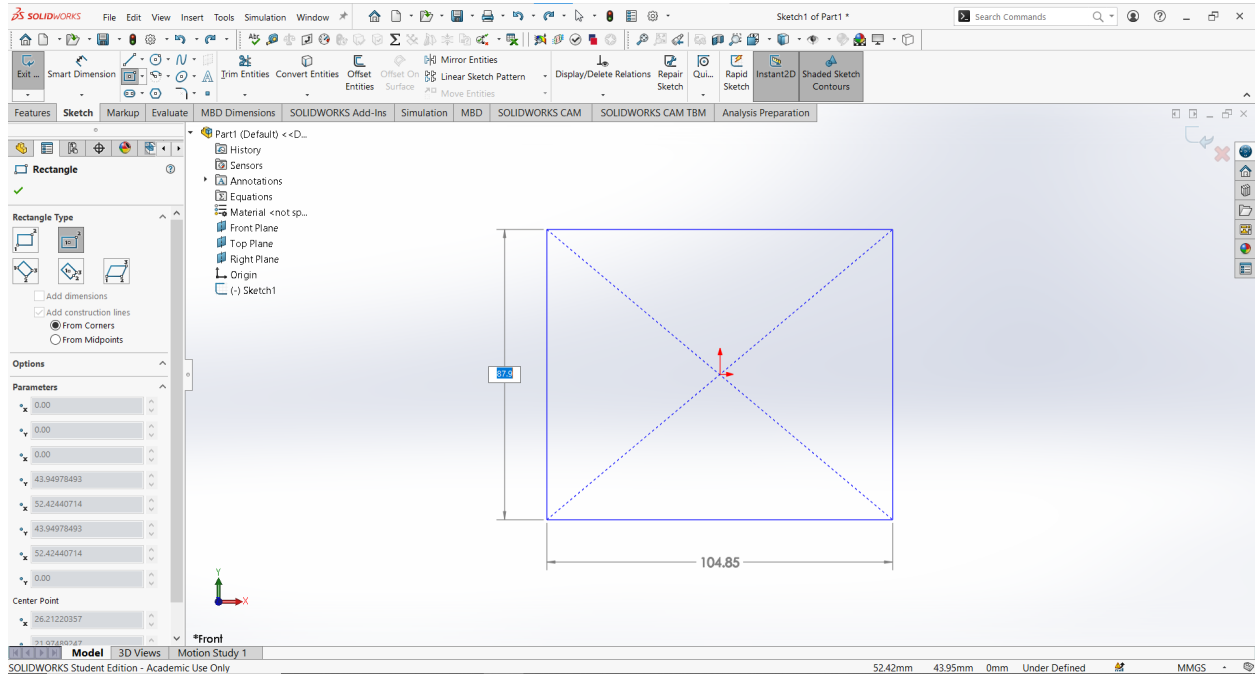
#### Step 4 : Create center rectangle at origin

- Next to *Smart Dimension*, and below the *line* tool, expand the rectangle command and select *Center Rectangle*.



- This will allow you to center the rectangle on the origin point without needing to add relations.

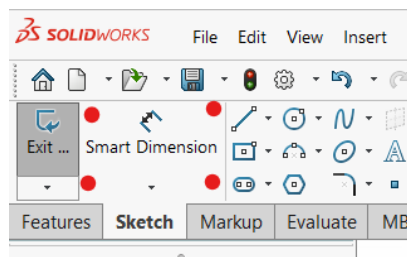
- With the *Center rectangle* command active, you will left click the origin point  one time, expand the shape to rough square, and left click again to close the sketch. Then either click *Escape* key on keyboard, or *Ok* button  on the property manager to close the command.





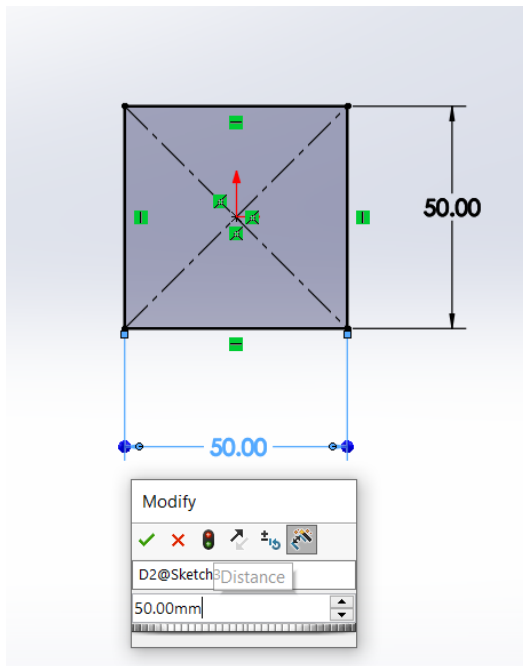
## Step 5 : Dimensioning the shape

- The next step is to add dimension to the shape. This process will fully define the sketch and prepare us to extrude the shape
- To add dimensions to the shape, navigate to the top left corner and click the *Smart Dimension* button

*Dimension* button

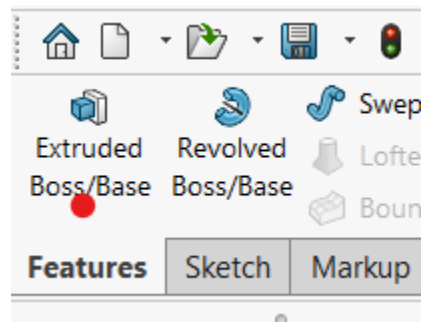


- With *Smart Dimension* now selected, left click one vertical line and drag the dimension away from the shape, left click again, add dimension of “50” into prompt, click ok  to finish the dimension.
- Repeat step above for one horizontal line of the shape. Dimension at “50” again.
- Click ok  to close command.

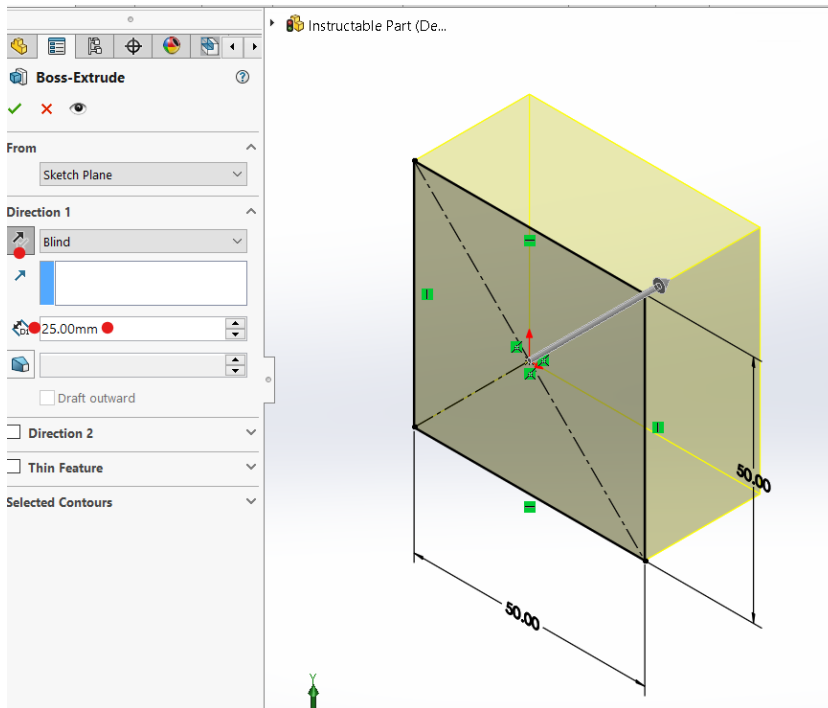



## Step 6 : Extruding the part into 3D shape

- Navigate to the top left, next to the *Sketch* tab select *Features* button, then select *Extruded Boss/Base* command.





- Within the property manager for the extrusion, select the *Reverse Direction* button, and add “25” into the depth box. This will be the distance you are extruding your part.

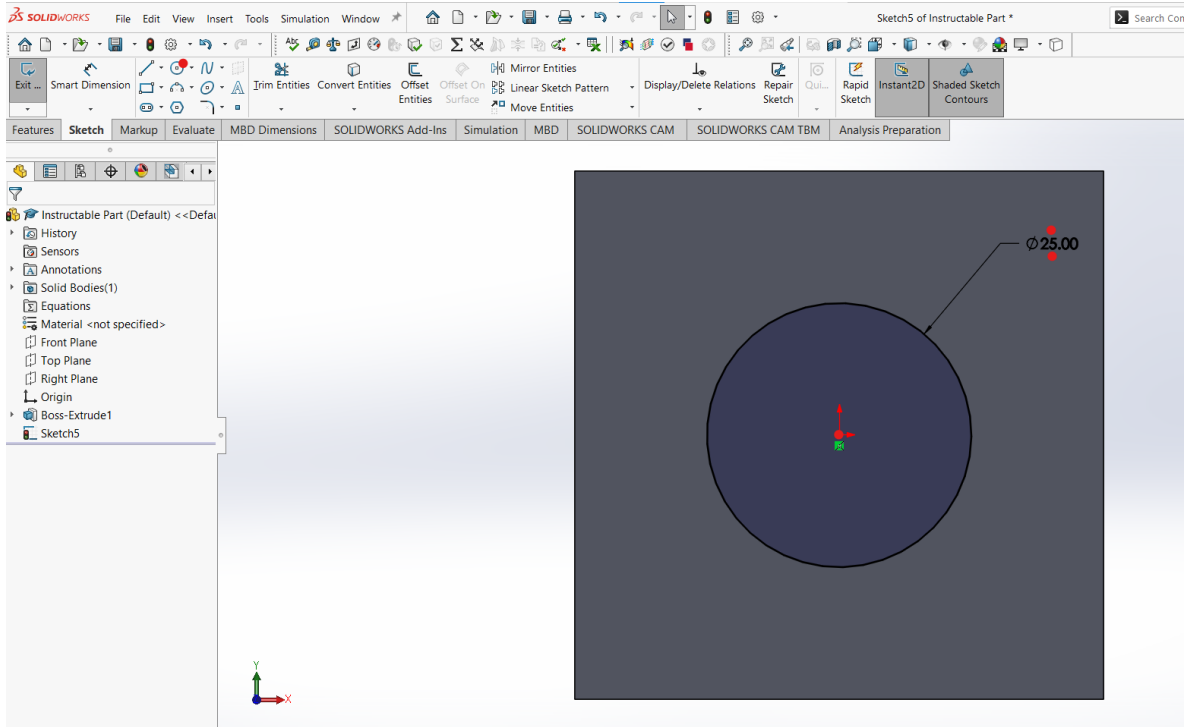


- Select *Ok*  to finish extrusion.

## Step 7 : Sketching the hole in the part

- First, select the *Sketch* command, then select the front face of the cube. Now you are set up to create a new sketch on the front face of the part.
- Next, select the *Circle* command, with circle command activated, left click the origin  point of the cube and expand the circle outward, left click again to create a sketch.
- Next *Smart Dimension* the circle to a diameter of “25”. Click *Ok* .

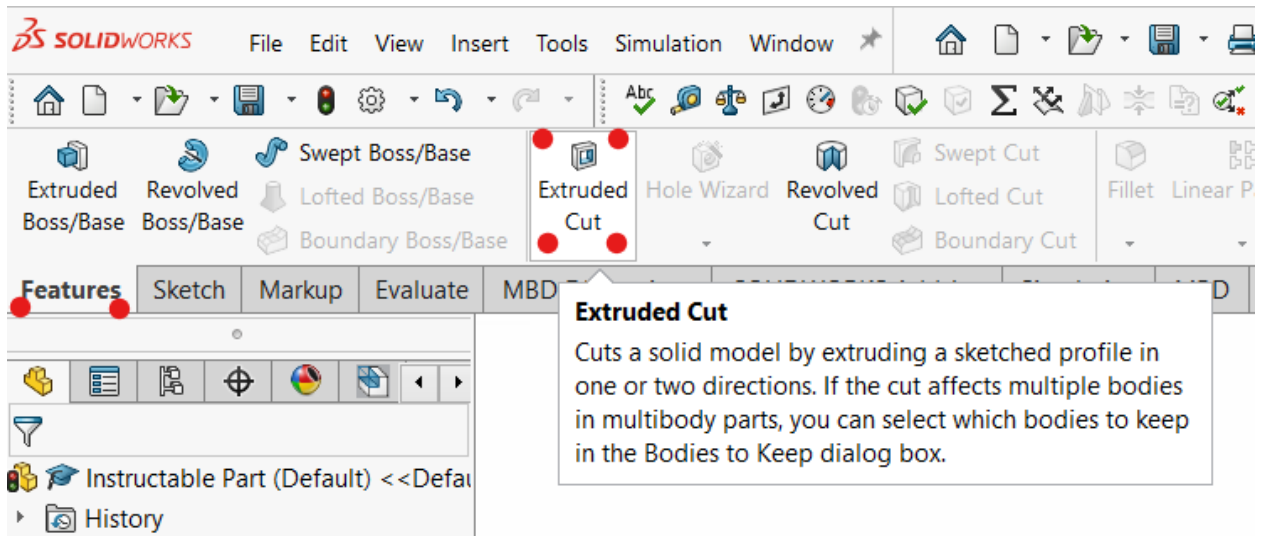




- DO NOT exit the sketch yet, we are not done.

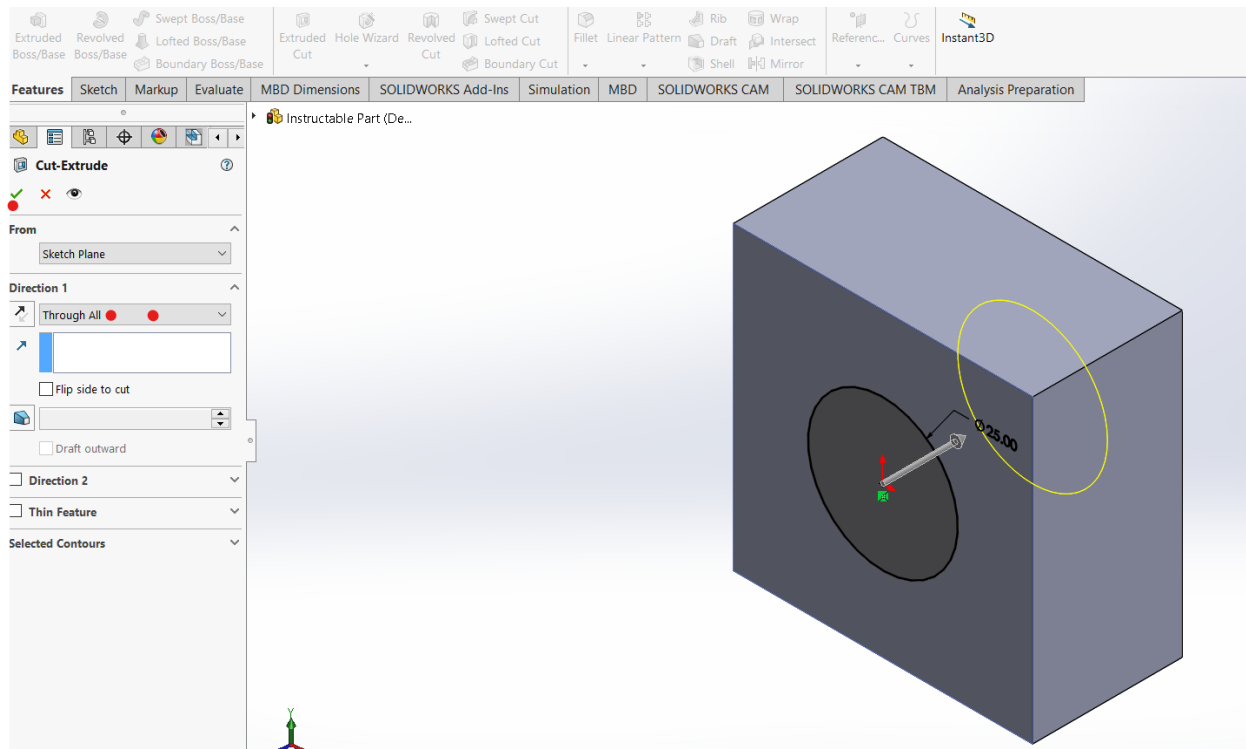
## Step 8 : Cutting the hole in the part

- First, select *Features*, which is next to *Sketch*, then select the *Extruded-Cut* command.




- With the *Extruded-Cut* command activated, left click the Direction 1 pulldown and select *Through All*.

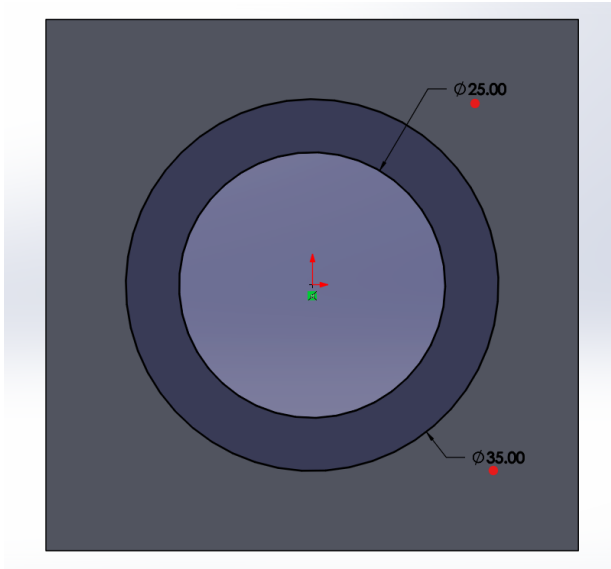
- Next click *Ok*  to close command.



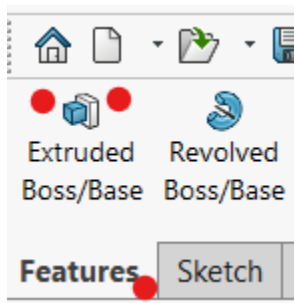
- Now there should be a hole directly through your part.

## Step 9 : Extrusion #2

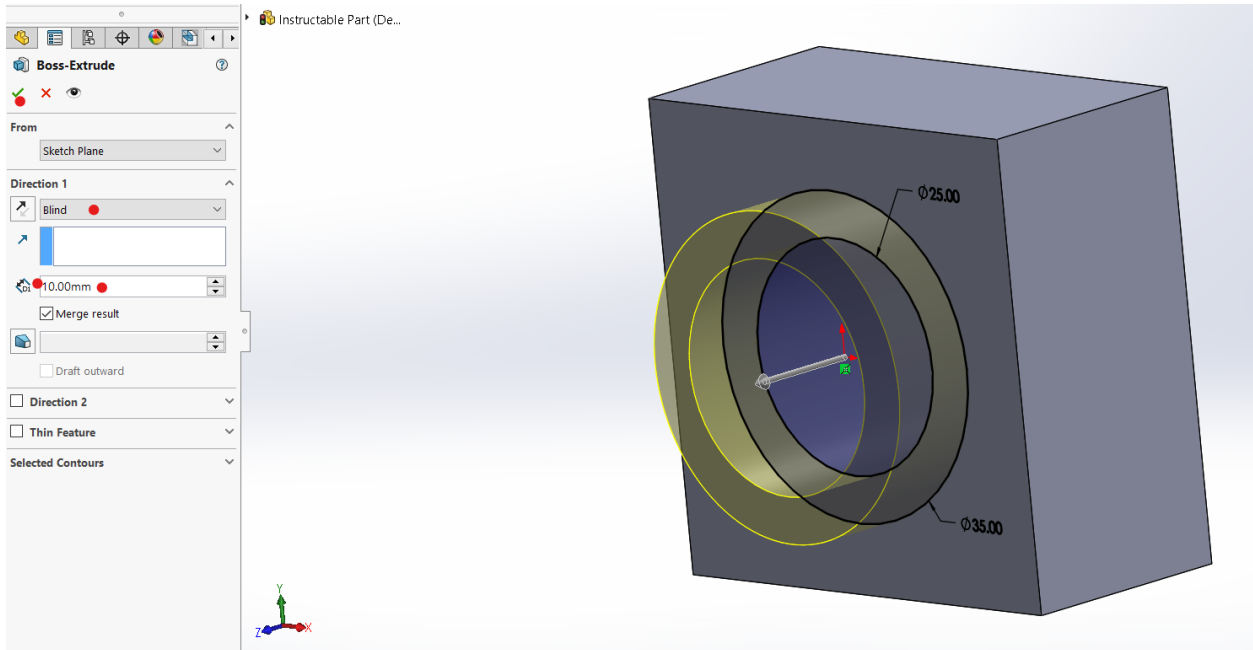
- First, open a sketch on the front face of the cube. (Refer to first part of Step 7 if confused)
- Next, sketch two circles at the origin  point, one circle has a diameter of “25”, the next has a diameter of “35”.




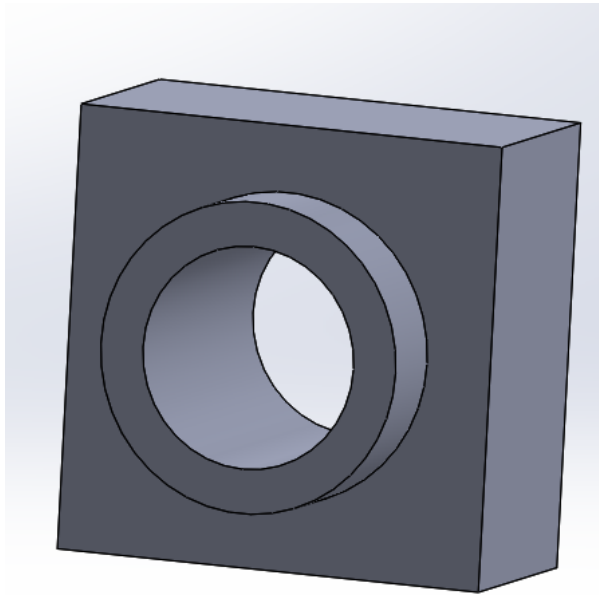
- Next, under the features tab, select the *Extrude Boss/Base* command.



- Lastly, under the Direction 1 pulldown, select *Blind*. Then type in “10” in the depth box.



- Select *Ok*  to finish the extrusion.

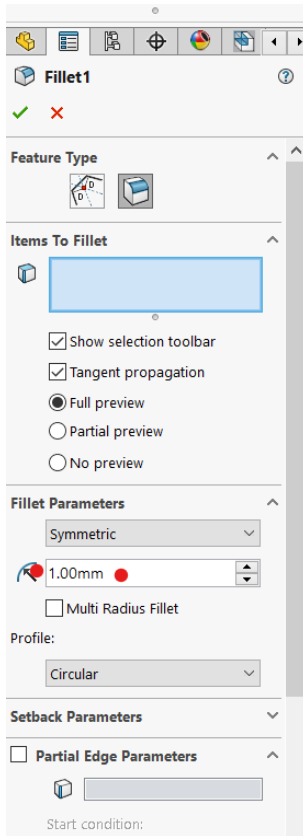


- This is what you should be left with.

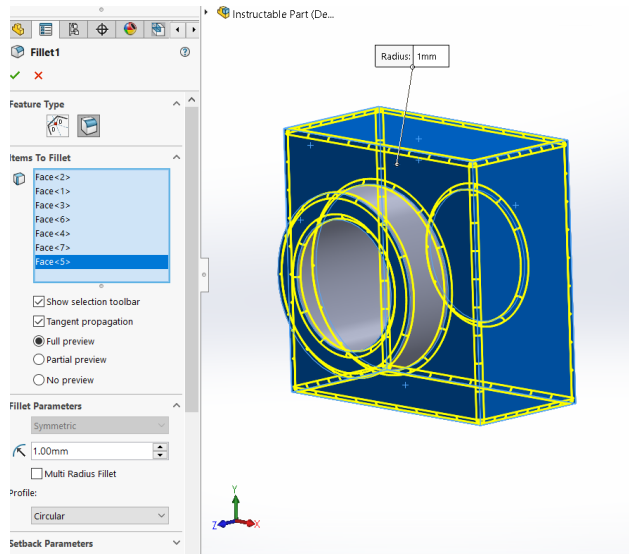
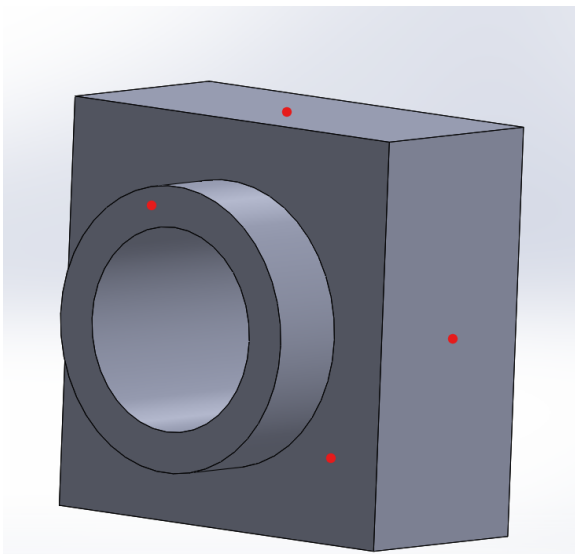
## Step 10 : Fillet the part


- First, under the features tab, select the *Fillet* command.

- With the *Fillet* command open, set the fillet parameters to “1.00” mm.



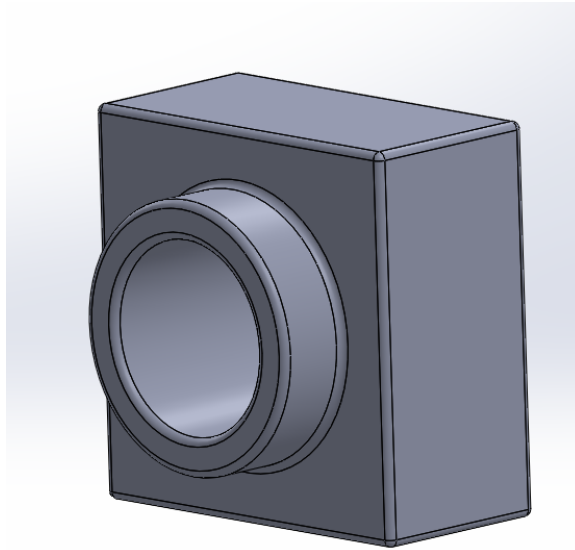
- Next we are going to select every face of the object. Red dots represent the faces you should select. DO NOT forget the 3 faces you cannot see in the pictures. Bottom, backside, and left side.



- Click ok  to fillet the product

Your part is now complete, but we are not done.

Your part should look like this :



## Step 11 : Check Mass Properties

- Under the Evaluate tab, select the *Mass Properties* command.
- A pop up window should appear, your *Mass* should equal “58.40” grams. If not, please go back and check your steps.

The screenshot shows the SolidWorks interface with the **Mass Properties** dialog box open. The **Evaluate** tab is selected in the software's ribbon. The dialog box displays the following information:

- Mass Properties** window for **Instructure Part.SLDPRT**.
- Override Mass Properties...** and **Recalculate** buttons.
- Include hidden bodies/components
- Create Center of Mass feature
- Show weld bead mass
- Report coordinate values relative to: -- default --
- Mass properties of instructure part: Configuration: Default, Coordinate system: -- default --
- Density = 0.00 grams per cubic millimeter
- Mass = 54.80 grams**
- Volume = 54800.87 cubic millimeters
- Surface area = 12490.05 square millimeters
- Center of mass: ( millimeters )
  - X = 0.00
  - Y = 0.00
  - Z = -11.00
- Principal axes of inertia and principal moments of inertia: ( grams \* square millir
- Taken at the center of mass.
 

lx = ( 1.00, 0.00, 0.00)	Px = 16979.09
ly = ( 0.00, 1.00, 0.00)	Py = 16979.09
lz = ( 0.00, 0.00, 1.00)	Pz = 26072.38
- Moments of inertia: ( grams \* square millimeters )
 

Taken at the center of mass and aligned with the output coordinate system. (Usin		
Lxx = 16979.09	Lyy = 0.00	Lzz = 0.00
Lxy = 0.00	Lyy = 16979.09	Lyz = 0.00
Lxz = 0.00	Lzy = 0.00	Lzz = 26072.38
- Moments of inertia: ( grams \* square millimeters )
 

Taken at the output coordinate system. (Using positive tensor notation.)		
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