

Handbook on Environmental Projects using Onshape

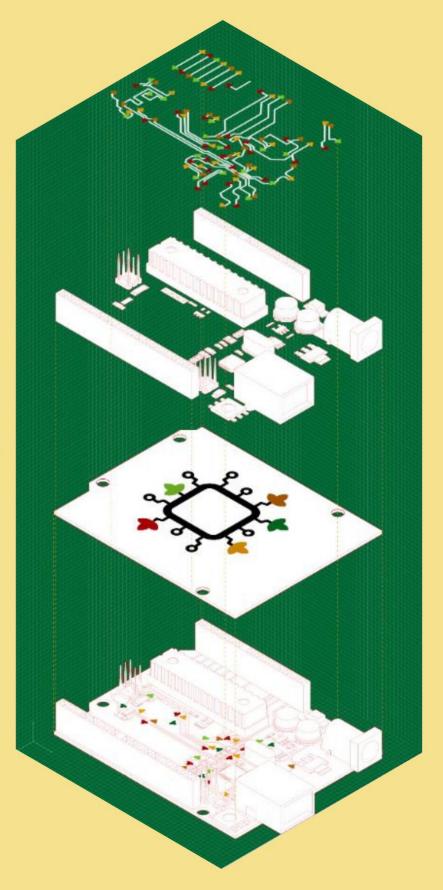


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INTRODUCTION

This handbook was created within the scope of Intellectual Output 4 of the Green STEAM Incubator project, which is related to 3D modeling. The objective was to create a document with ideas for environmental projects that make use of 3D modeling as a resource to promote sustainable ways of designing and using 3D objects.

Every partner of the consortium of this project came up with different project ideas that were gathered in a single document. Each project has its own different objectives and learning results, but one thing is common to all of them: to promote the use of 3d modeling as a way of producing less waste through the consumption of less energy.

Together, these project ideas serve as a 15-hour workshop to promote the aforementioned objectives. All the essential information for each session is given at the beginning of each project idea.



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PROJECT: 3D DRAWING OF A BIRD FEEDER

- **<u>STEM field:</u>** Technology, Engineering.
- Indicative calendar: Any time of the year.
- Activity duration: 3 hours.
- Type of activity: Workshop.
- **Educational objectives:** By the end of the course, the learners are expected to draw a bird feeder on the Onshape software.
- Learning outcomes and acquired competencies: By the end of the course, the learners: (1) will have designed a 3D object, specifically, a bird feeder; (2) will get familiarized with the Onshape features and functionalities; (3) will have enhanced their 3D design skills.
- <u>Required material and resources:</u>
 - Computer.
 - o Internet access.
 - Onshape account (or other similar).
- Description and/or step-by-step instructions

This project consists of a 3D design of a bird feeder, then we will present the step-by-step for its elaboration.

First, when designing a 3D model, it is essential to have a strategy. For instance, for designing a bird feeder (see figure 1), we can see the main cylinder. So, a good piece of advice is to start with the design of this part of the object.

Second, we can see that the cylinder is hollow or emptied out with a missing top and a couple of holes. Third, we have some additional features on the main cylinder, the so-called perches (i.e., in the figure below, those appear like circular stands for the birds; those could also be simpler cylindrical features).

Let's start with the design of the 3D object now.





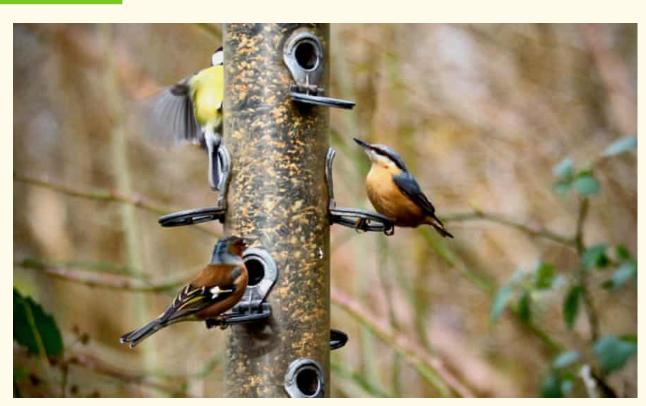


Figure 1 A bird feeder, source: <u>https://www.theguardian.com/environment/2020/feb/27/country-diary-</u> <u>the-gatherings-at-the-bird-feeder-are-anything-but-random</u>

Open Onshape.

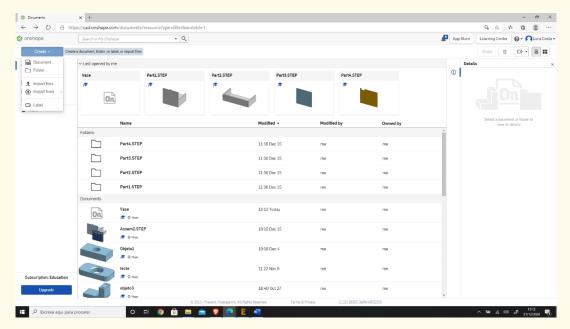
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Create a document.



Step 3

Give a name to your document such as "Bird Feeder".

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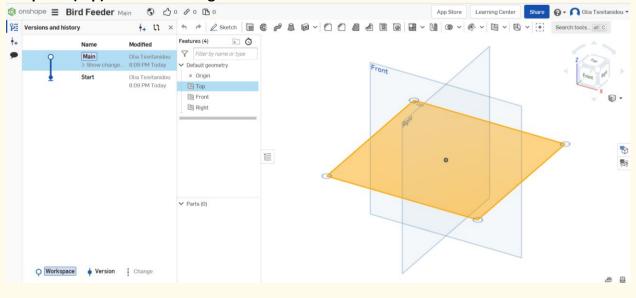


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Start with the creation of the main cylinder. This can be done in multiple ways: we can create a circle, extrude it up, draw a rectangle, revolve it, etc. We will not worry about any dimensions right now; we will take care of those later. Select

the plane (top) to start drawing.



Select sketch, then circle.

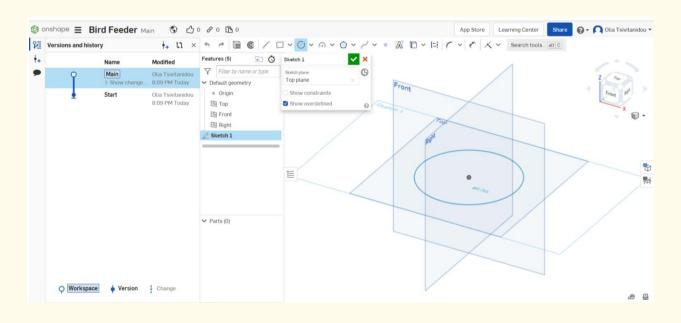
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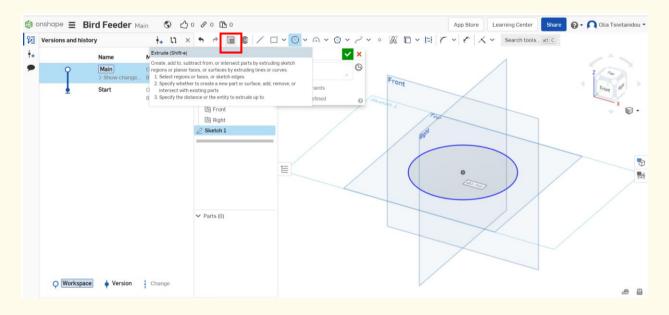


Draw a circle (do not worry about the dimensions at this stage).



Step 7

Select extrude.

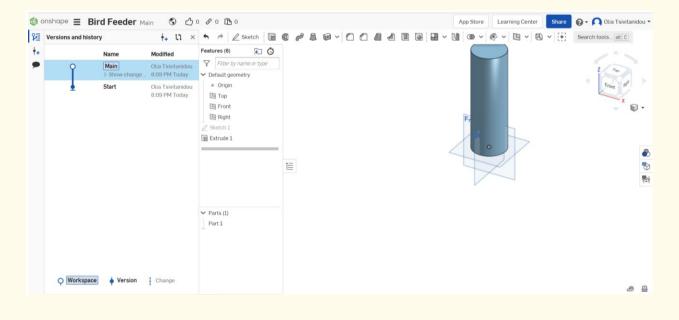


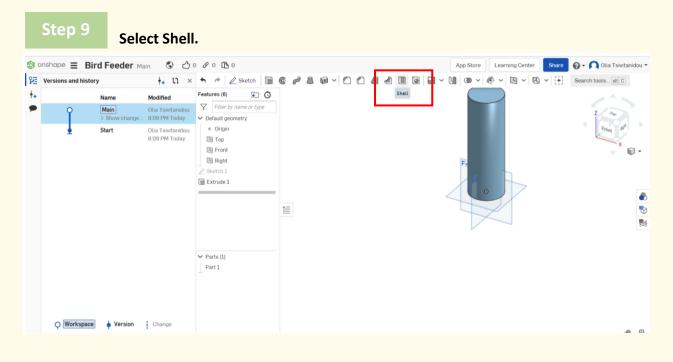


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Extrude that circle.

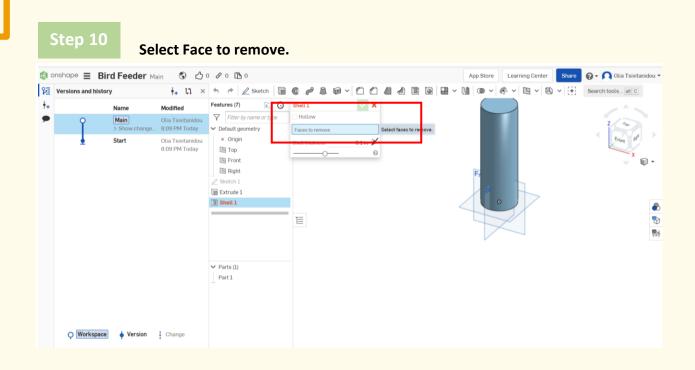




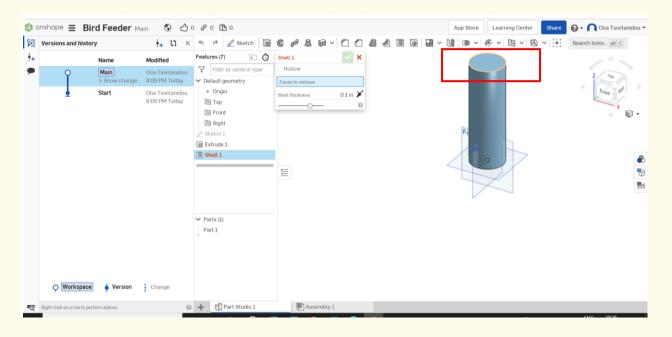


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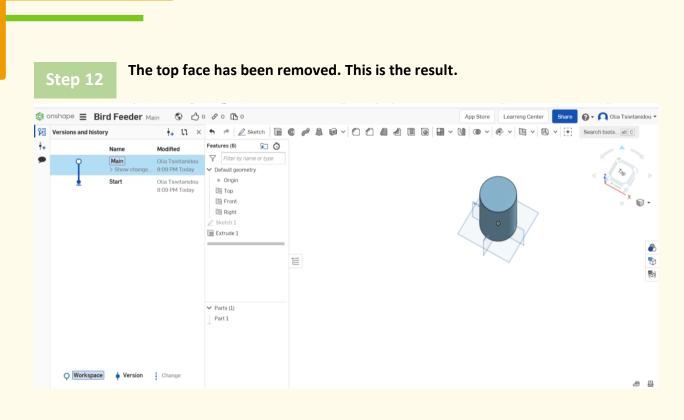
Click on the top cylinder.



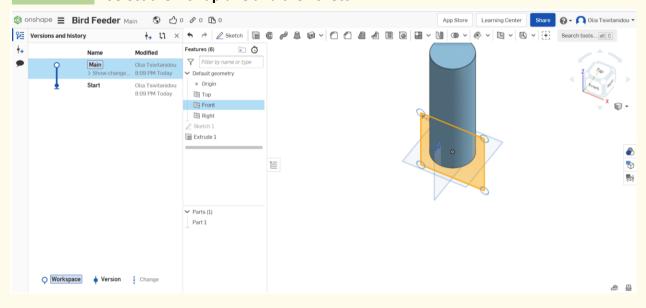


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Next, we are going to put the large holes on the cylinder surface. Select the front plane and then sketch.





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To create the first hole, we put our mouse down to the origin, so that's tethered or linked together, and it will be somewhere in the middle. Click the circle and draw a circle.

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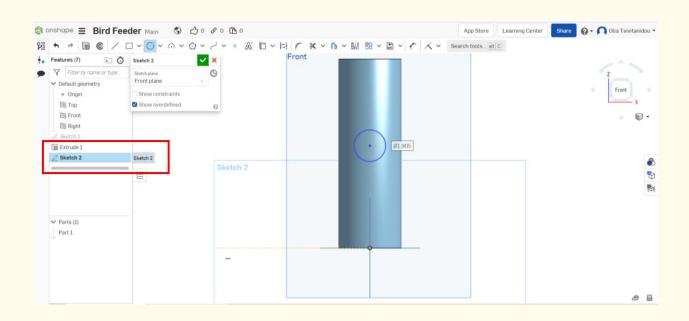


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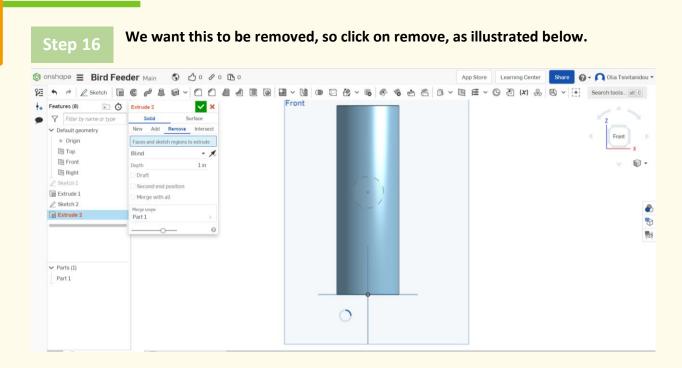
Go on the right side of your screen, where it says sketch 2 (see red box), and pick it from there.



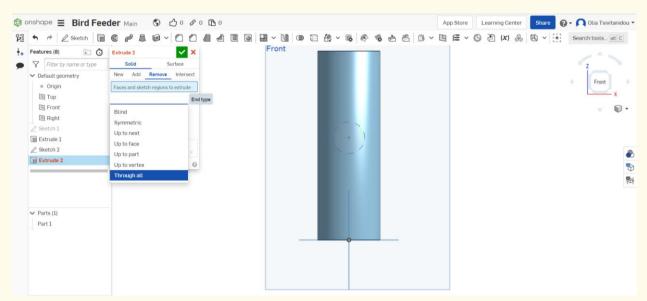


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Select "through all."

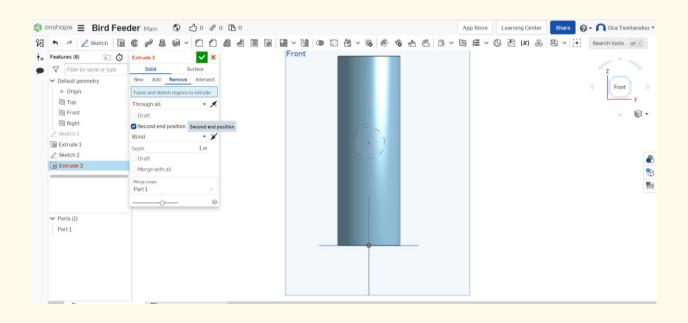




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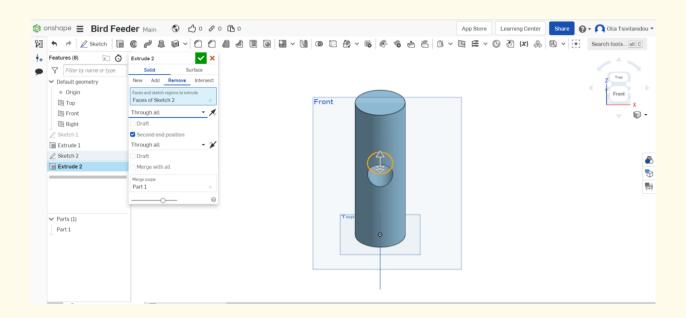


Select "second end position" to have the same exact hole on the opposite side of the inner cylinder surface.



Step 19

Select "second end position" and make that also a "through all."

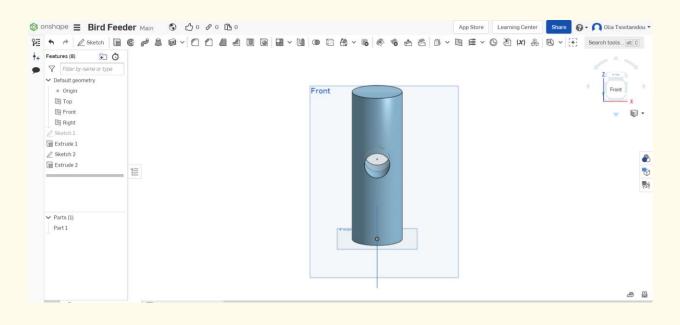




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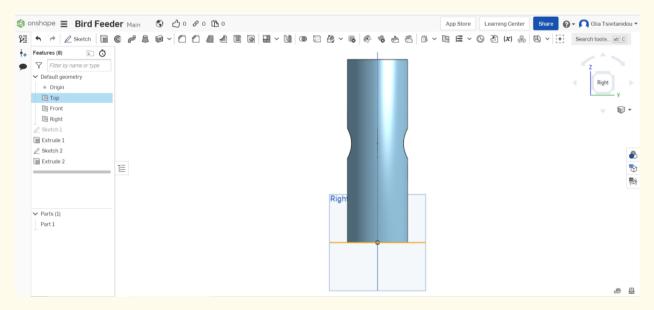


So, now we are one hole that goes all the way through the front face.



Step 21

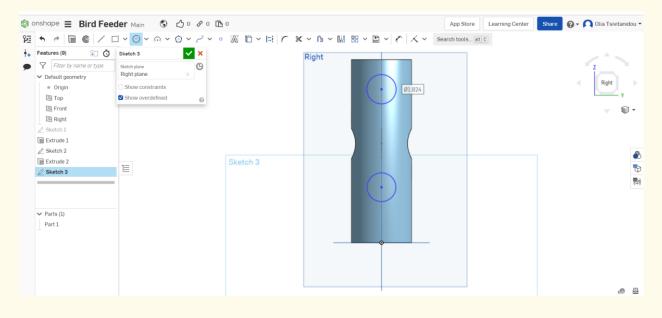
Let's go-ahead to create two more holes. We are going to do a new sketch on the right side face. Click on the right side plane, then sketch and circle.





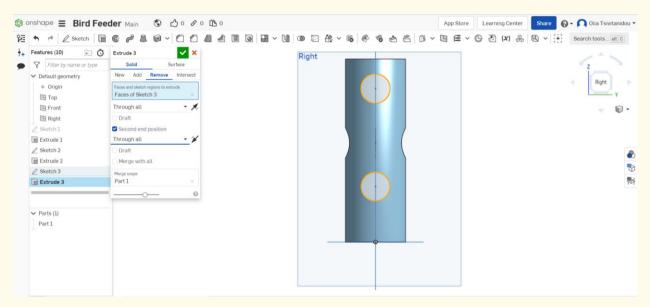


Let's go ahead and draw two new circle sketches, in the same way as done previously.



Step 23

As before, we will extrude by selecting first by clicking on sketch 3, then extrude, then remove, through all, second end position, and again, through all.

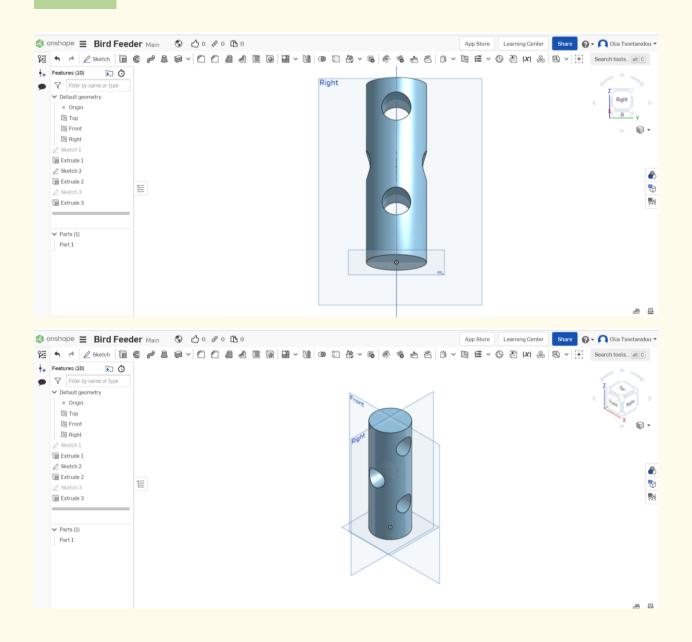




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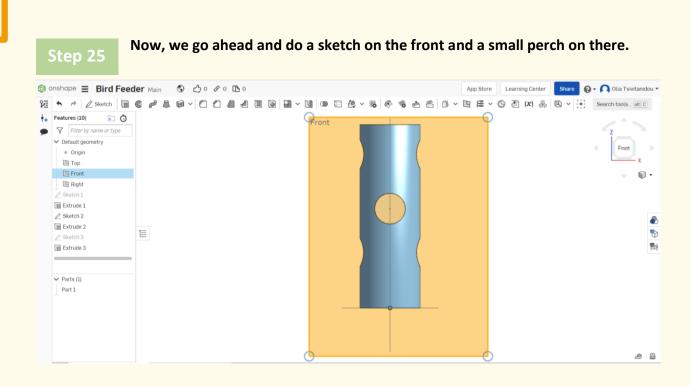
And now we have two more holes on the cylinder's surface.



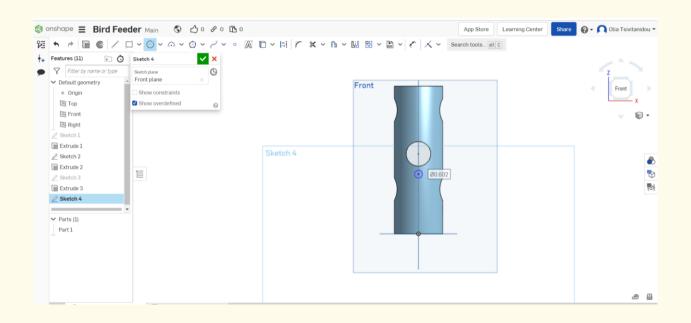


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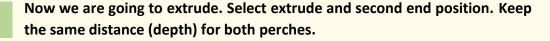
Select the front plane, click sketch and draw a small circle.

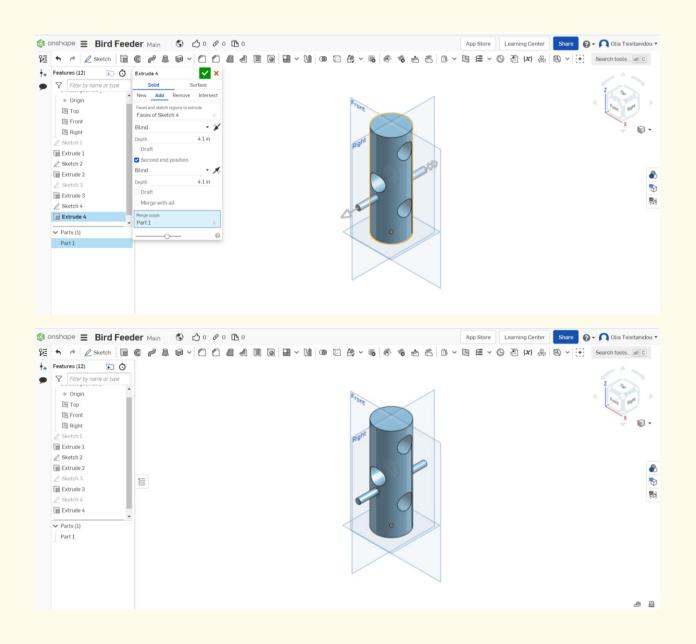




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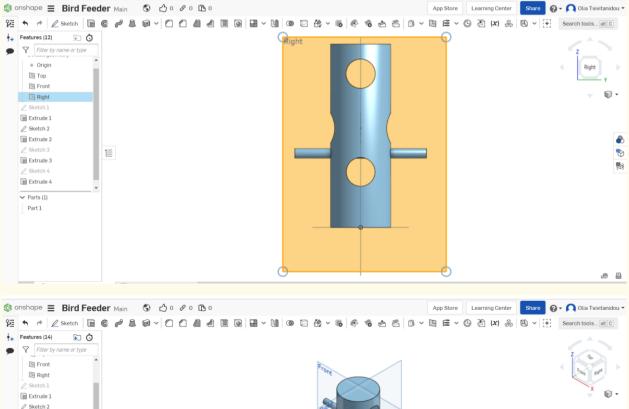


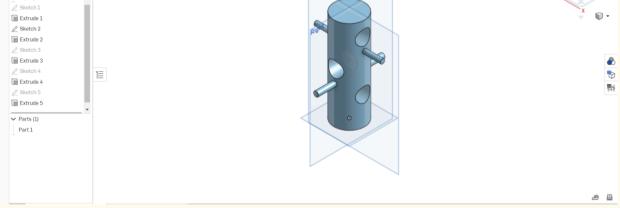


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Now, we are going to repeat steps 26 and 27 for creating two more perches. Click on the right plane and proceed in the same way.



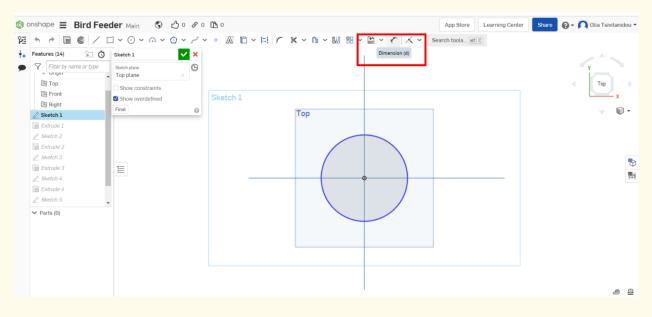




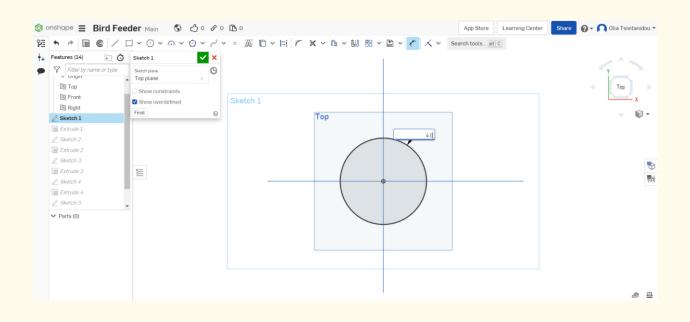
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As a final step, we are going to put on the dimensions that we want. Let's start with the diameter size and set it put to 4 inches. Click on sketch 1, and then dimensions (red box below).



Change the size of the diameter and set it to 4 inches.



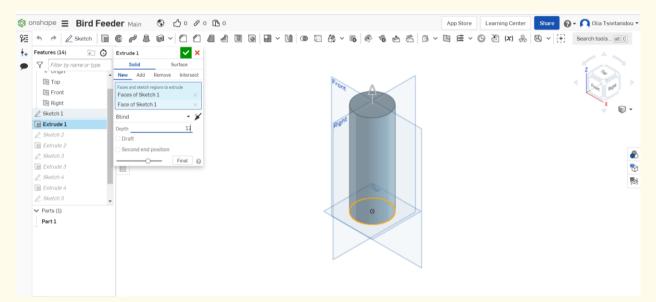


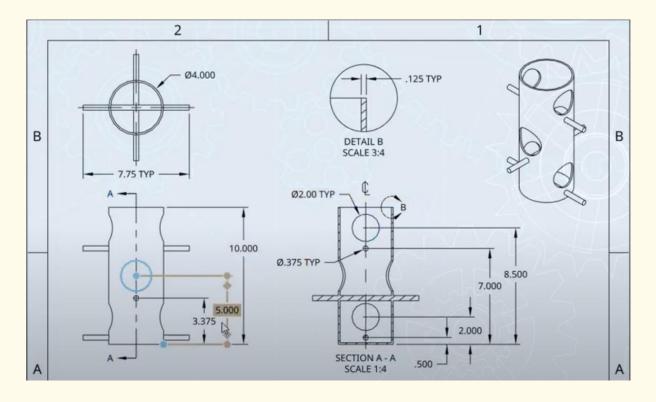
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Set realistic sizes. Repeat step 30 for resizing all sketches to give the real dimensions (see figure below). In the same rationale, we can correct extrusion. For instance, select extrusion 1 from the features list. We set the

depth into 12 inches.

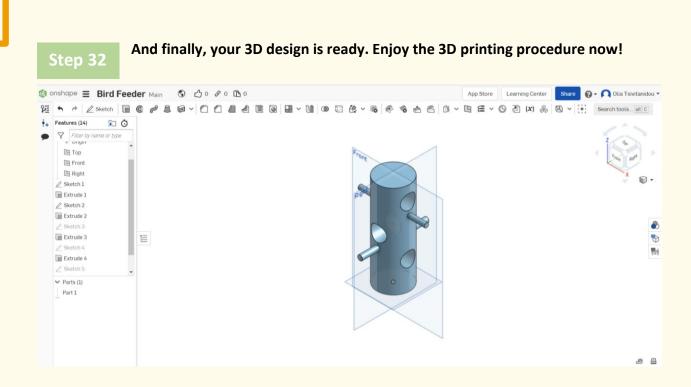






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PROJECT: 3D DRAWING OF A SHOVEL

- **<u>STEM field:</u>** Science, technology, and electronics.
- Indicative calendar: Any time of the year.
- Activity duration: 3 hours
- <u>Type of activity:</u> Drawing of a shovel.
- **Educational objectives:** By the end of the course, the learners are expected to draw a shovel on the Onshape software.
- Learning outcomes and acquired competencies:
 - How to do a shovel on Onshape.
- <u>Required material and resources:</u>
 - Computer;
 - Internet access;
 - Onshape account (or other similar).

Description and/or step-by-step instructions

This project consists of the 3D design of a shovel, then we will present the step-by-step process of its elaboration.

PRELIMINARY STEPS:

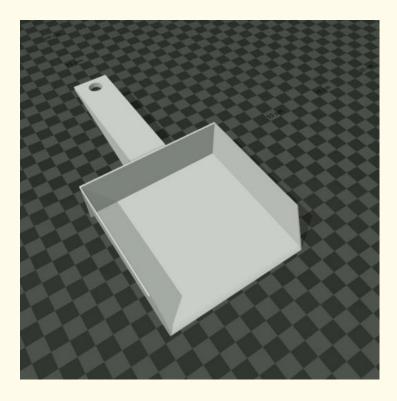
It is important to have an idea of the final object and make sure measures are relevant; you need to do some small research.





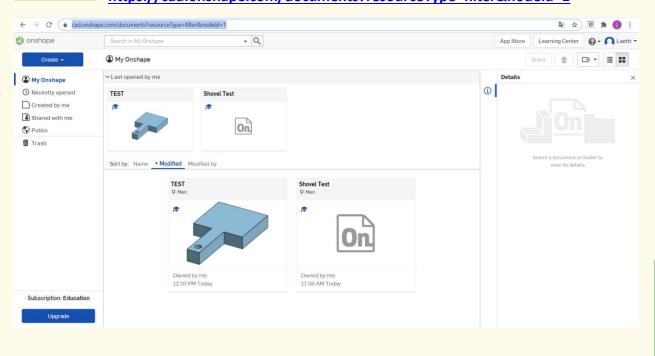
We have found an example of what our shovel will look like.

We took this example for inspiration: <u>https://cults3d.com/fr/mod%C3%A8le-3d/divers/pala-manual</u> and selected/defined our own measurement.



Step 1

Open Onshape. https://cad.onshape.com/documents?resourceType=filter&nodeId=1

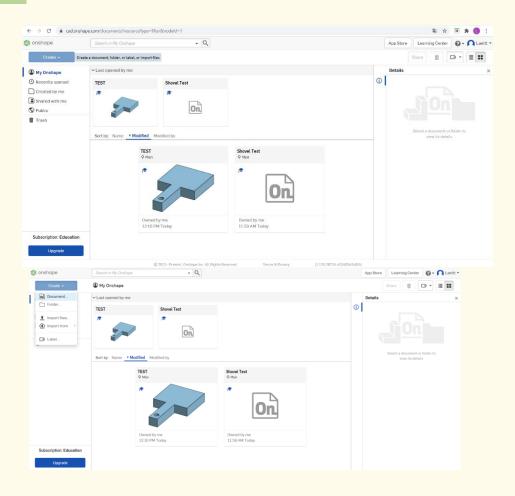




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Create a document.



Step 3

Give a name to your document such as Shovel.

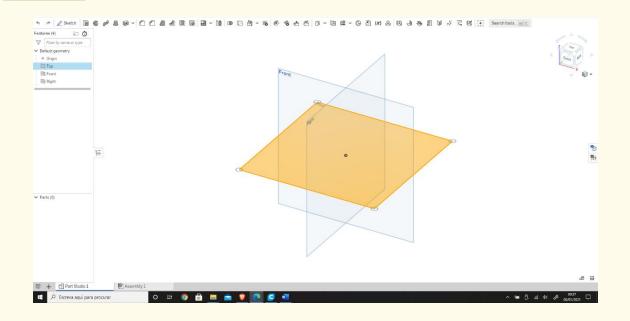
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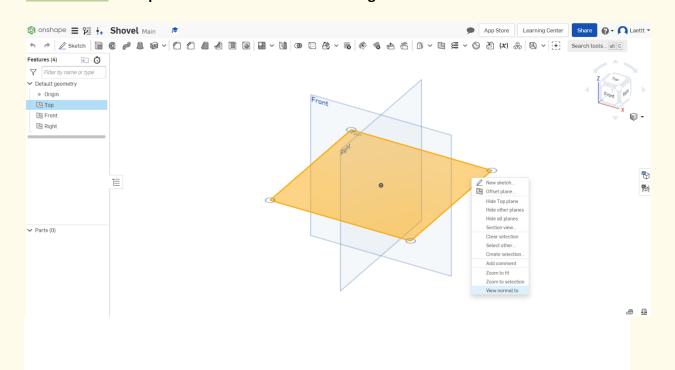


Select the plane (top) to start drawing.



Step 5

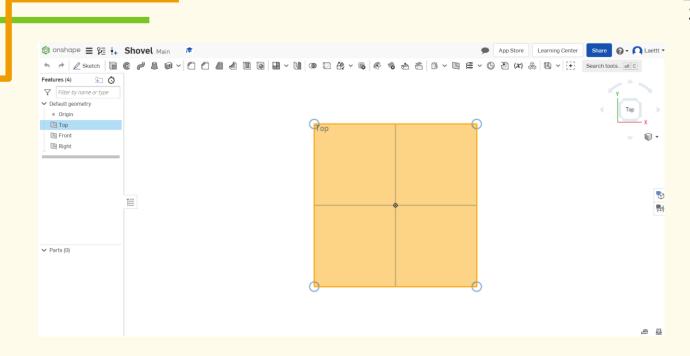
Right-click and select normal view. The plan should look like the 2nd image.



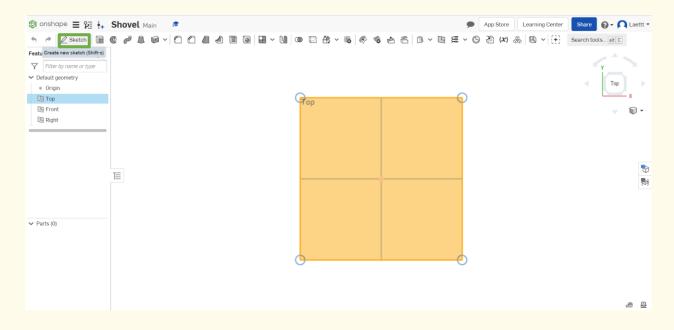


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Click on Sketch.

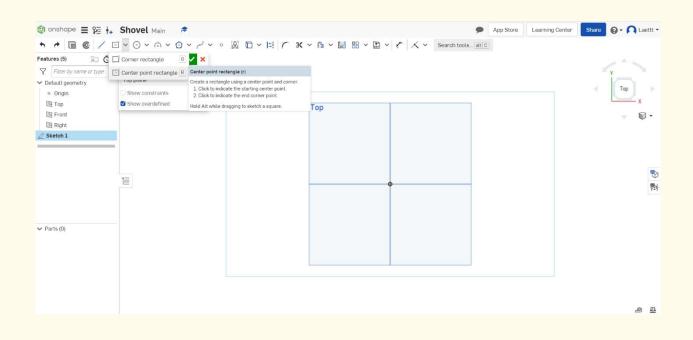




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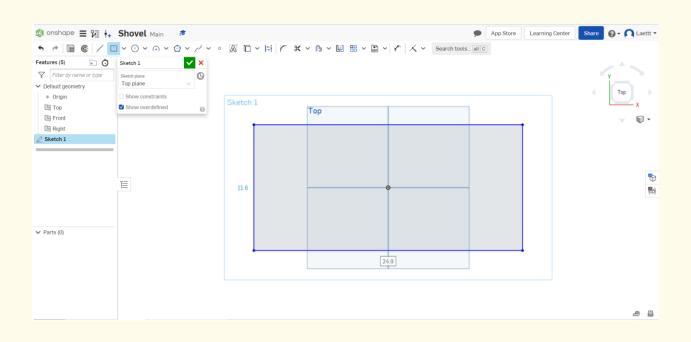


Select the center point of the rectangle to draw.



Step 8

Draw the rectangle.

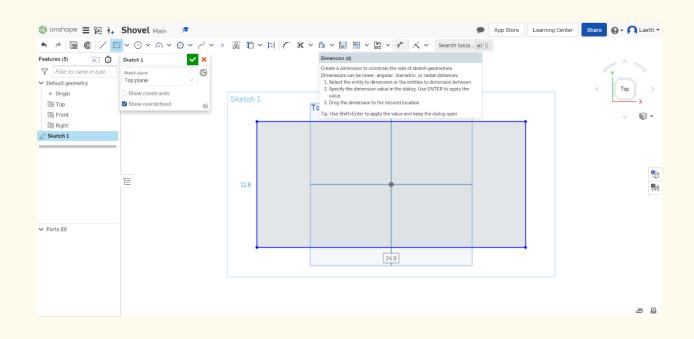




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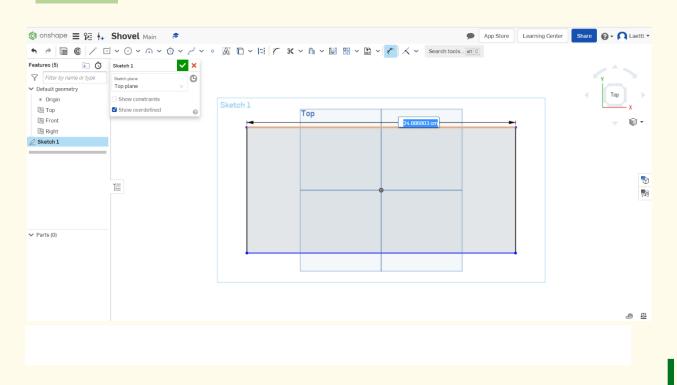


n g Select Dimension to give the measures.



Step 10

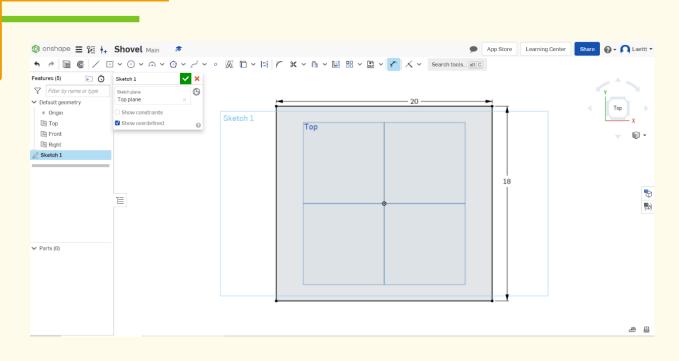
Select the line (first left then top) and give the measure 2nd figure.



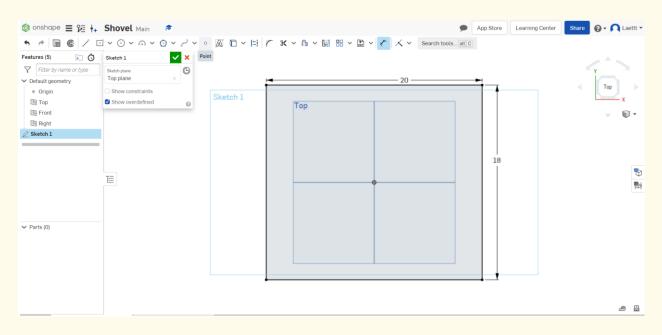


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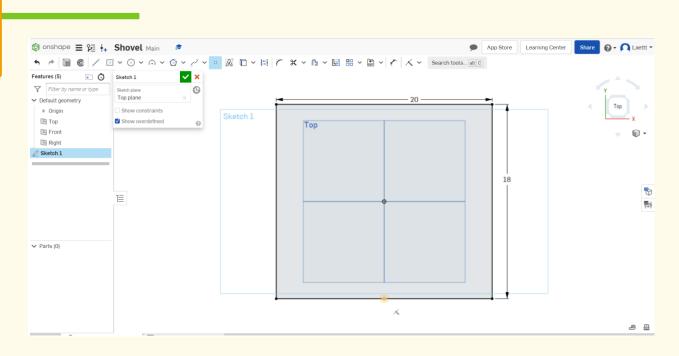
Step 11 Select the function point and place a point on the lower part of the rectangle.



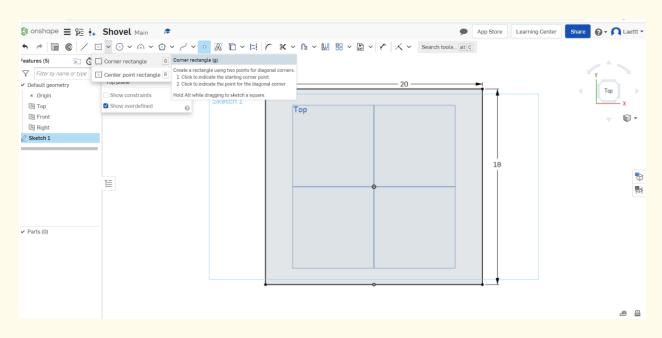


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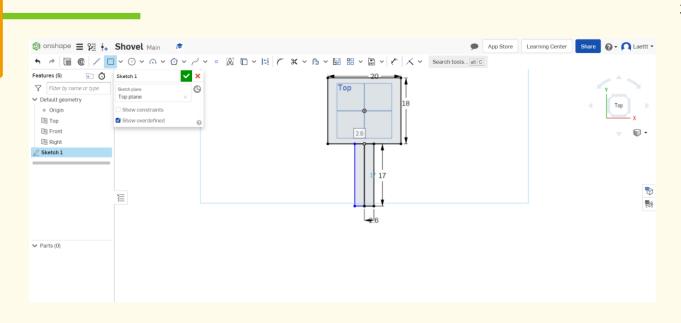
Draw 2 similar rectangles as shown.



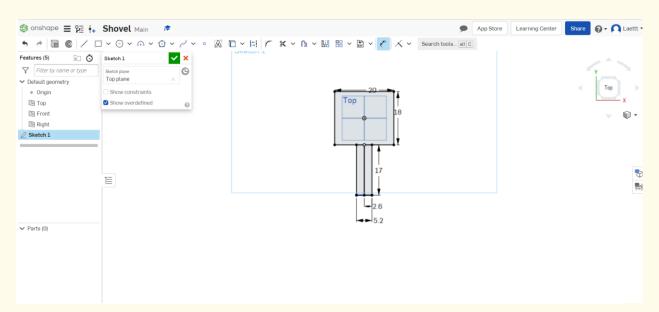


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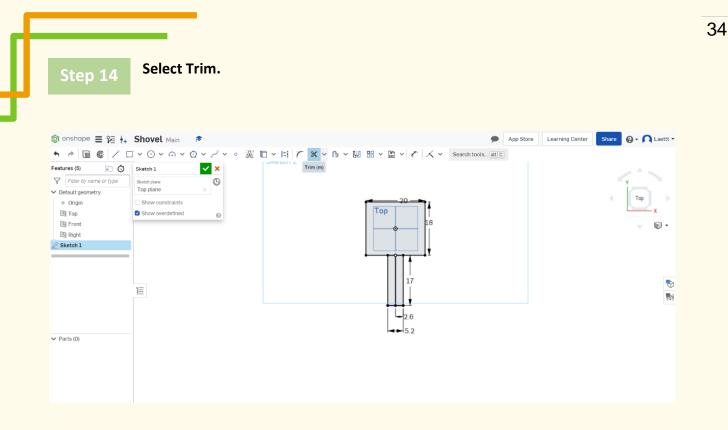
Select Dimension to give the measures and enter the measures as shown in the image below.



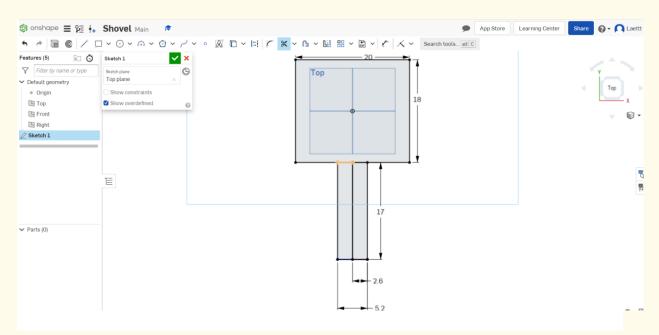


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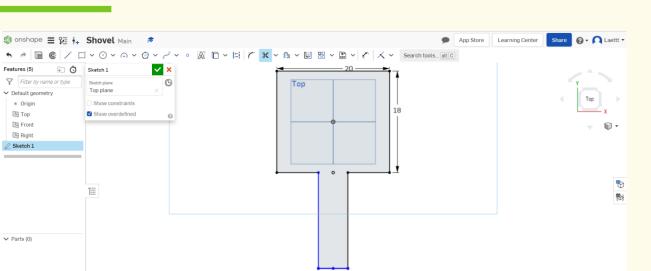
Select the orange lines (then the lines will disappear).





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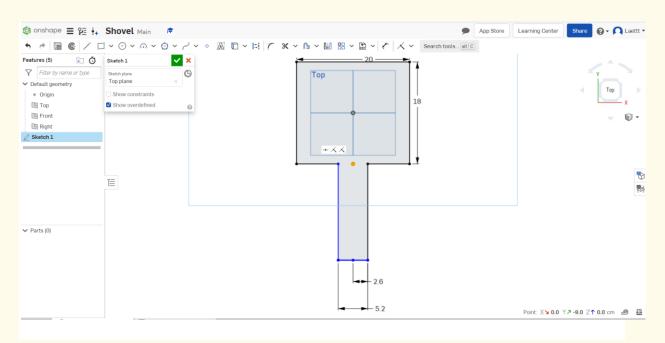




2.6

Step 16

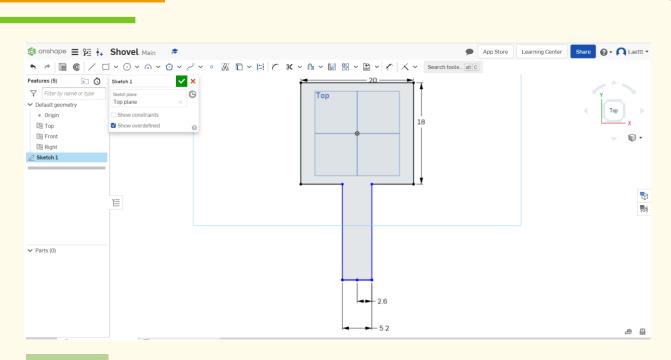
Remove the dot.



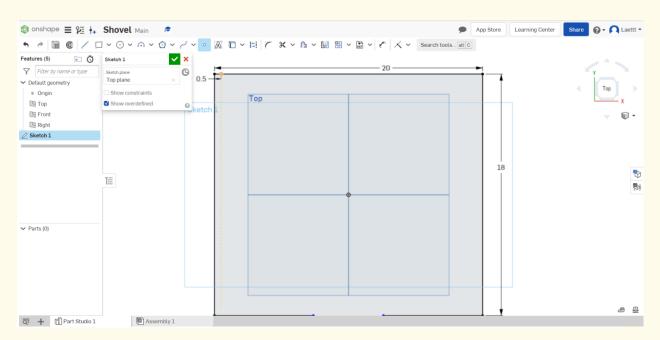


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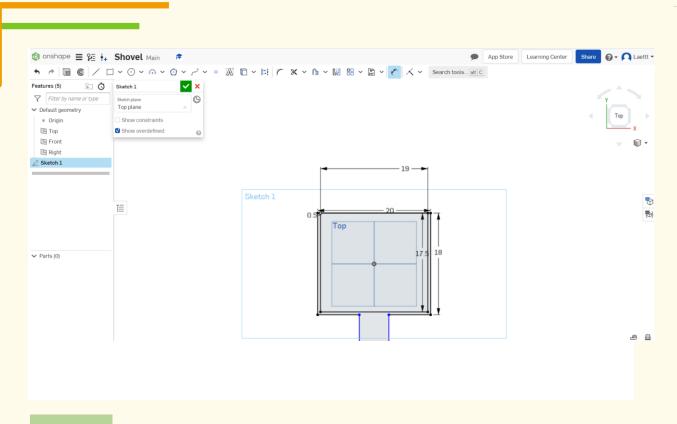
Place a dot at 0.5cm from the edge of the first rectangle and draw another rectangle inside (the rectangle is 19cm and 18.5cm). This will be the base of the shovel.



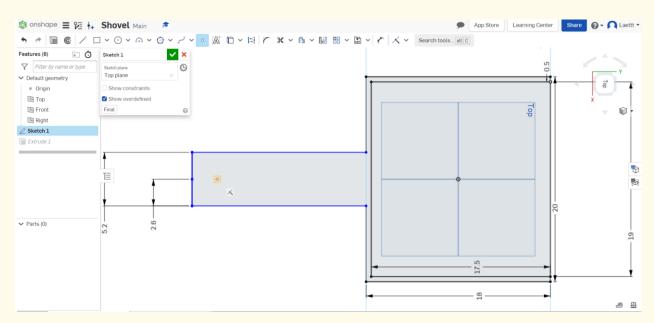


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To add a hole in the handle, start with a dot in the handle.

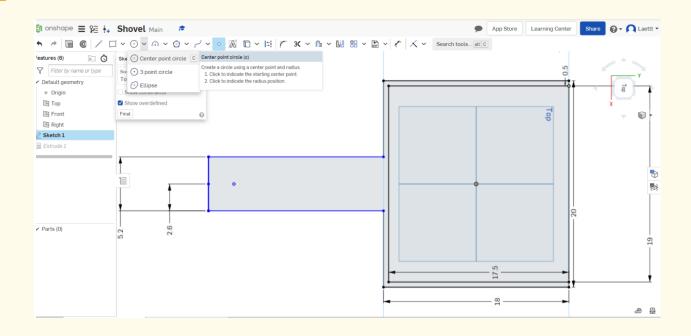


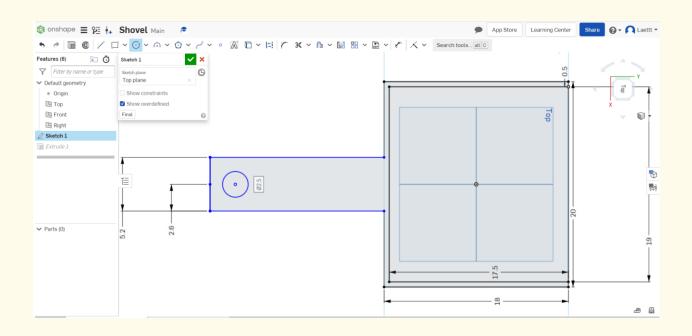


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Draw a center dot circle.



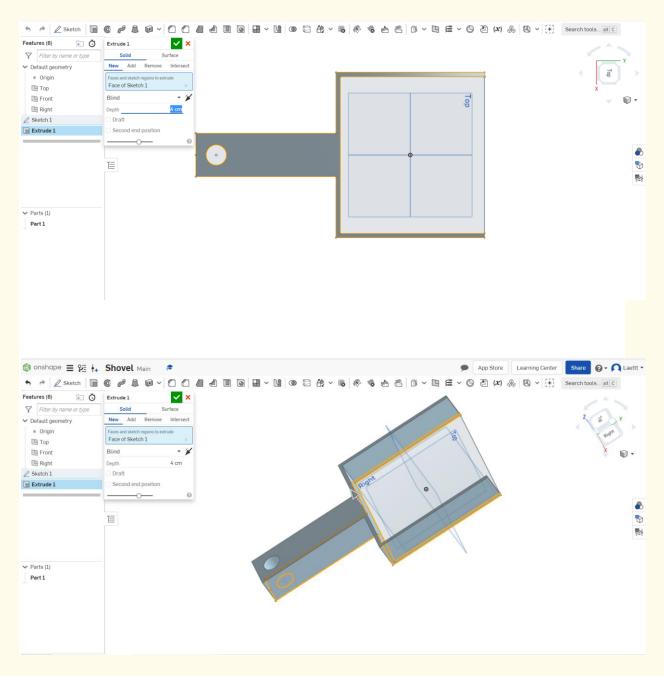




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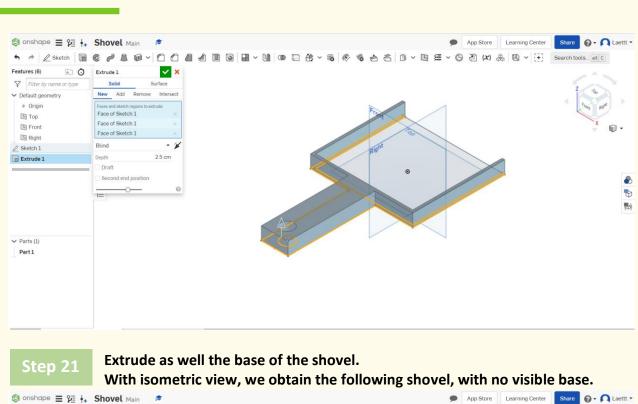
Click on Extrude and select the lines you want to extrude (the orange lines). Change the measure to 4cm.

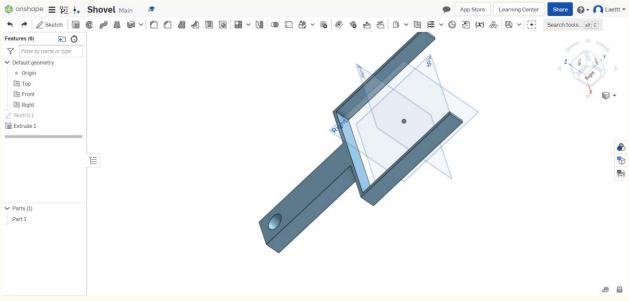




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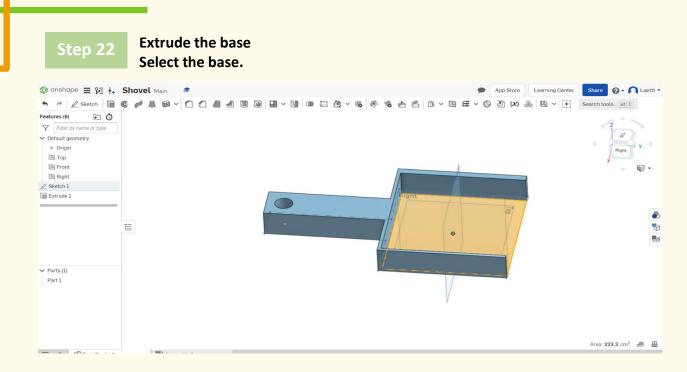


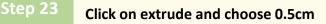


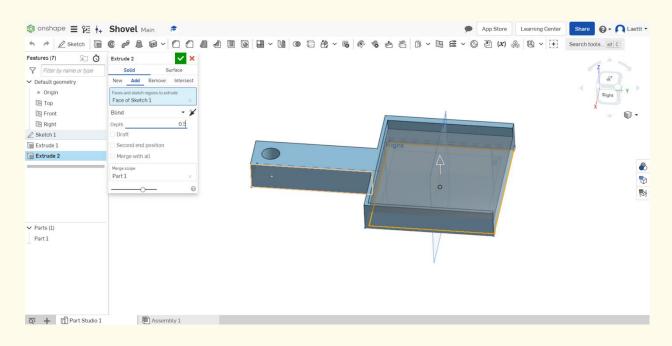


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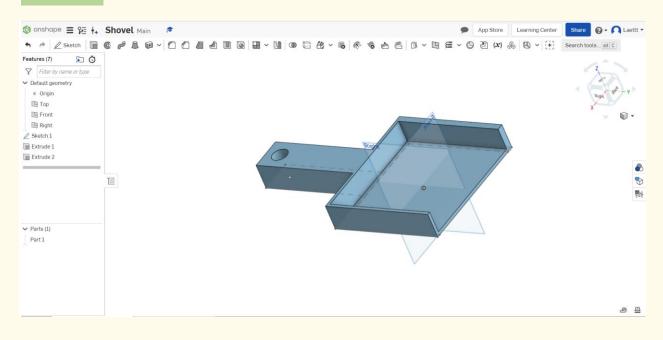




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ep 24 This should be your result.





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PROJECT: 3D DRAWING OF A WATERING CAN

- **<u>STEM field:</u>** Science, technology, and electronics.
- Indicative calendar: Any time of the year.
- Activity duration: 3 hours.
- <u>Type of activity:</u> Drawing a watering can.
- **Educational objectives:** By the end of the course, the learners are expected to draw a watering can on the Onshape software.
- Learning outcomes and acquired competencies:
 - How to do a watering can on Onshape.
- <u>Required material and resources:</u>
 - Computer;
 - Internet access;
 - Onshape account (or other similar).
- Description and/or step-by-step instructions:

This project consists of the 3D design of a watering can, then we will present the step by step for its elaboration:





Open Onshape (the free version).

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Step 2

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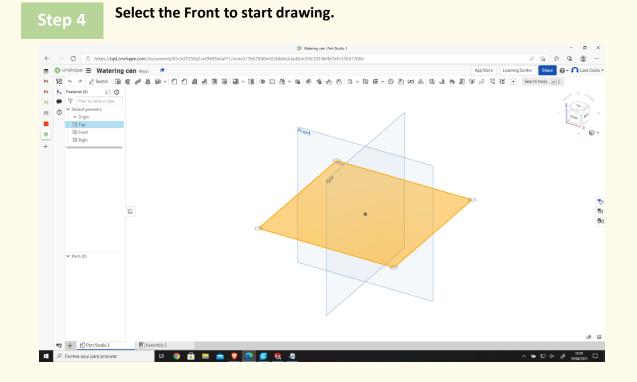
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Give a name to your document, such as Watering Can.

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A My Onshape	~ Last opened by me		Document name				Details		
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		Part2.STEP	11	36 Dec 15 2020	me	me			
		Part1.STEP	11	36 Dec 15 2020	me	me			
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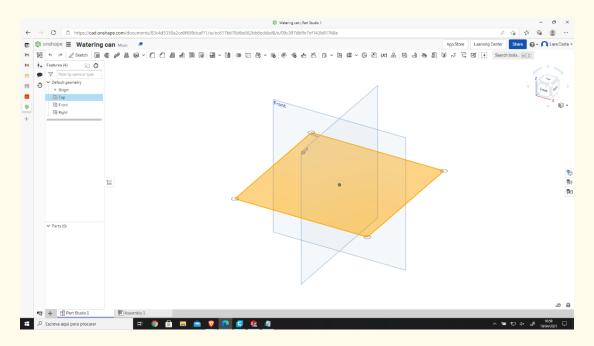


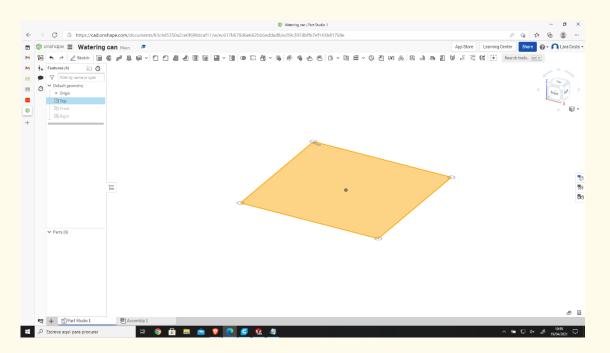


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Right-click and select "Hide other planes." The plan should look like the 2nd image.



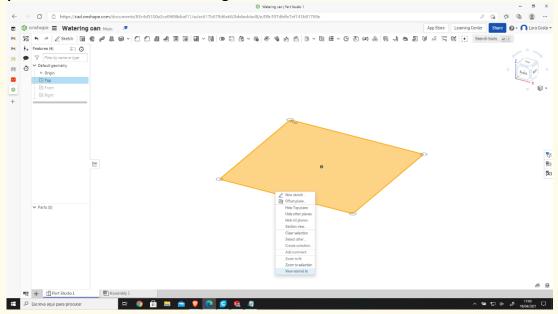


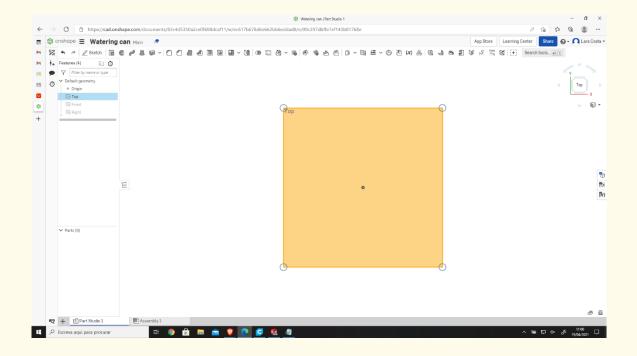


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Right-click and select "View normal To." The plan should look like the 2nd image.



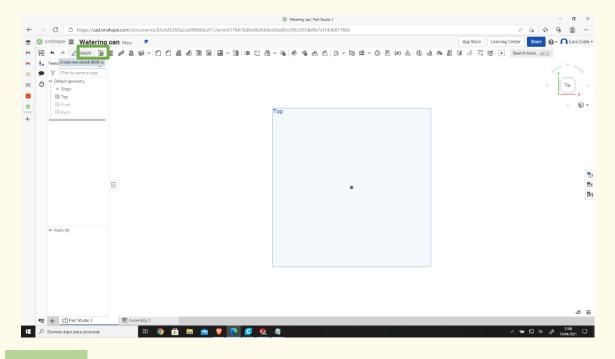




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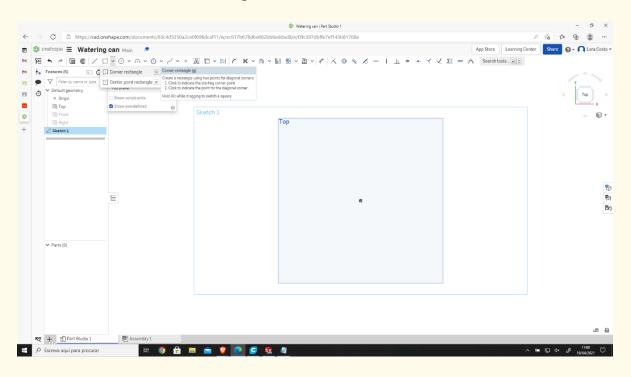


Click Sketch.



Step 7

Select corner rectangle to draw.

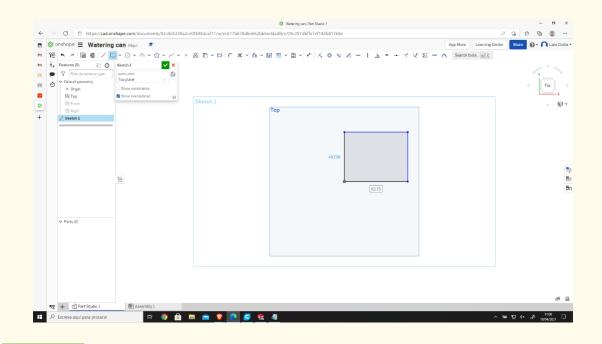




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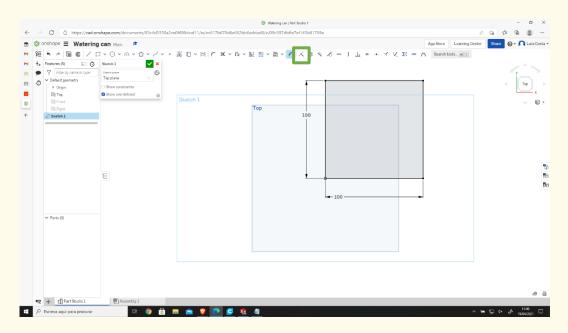


Draw the rectangle.



Step 9

Click on Dimension and give 100mm to each side.

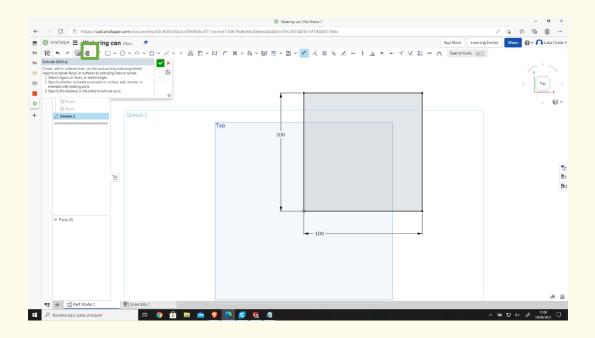




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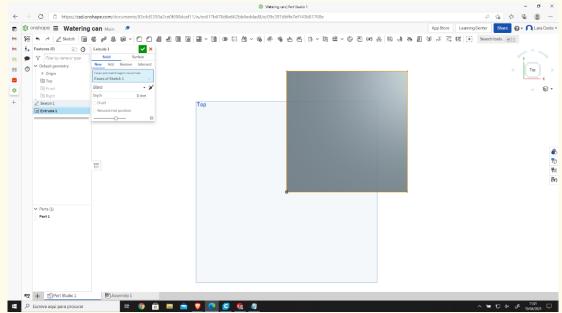


Select extrude.



Step 11

The Depth should be 5mm.

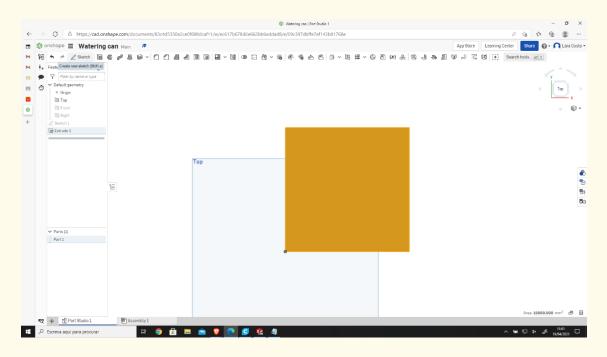




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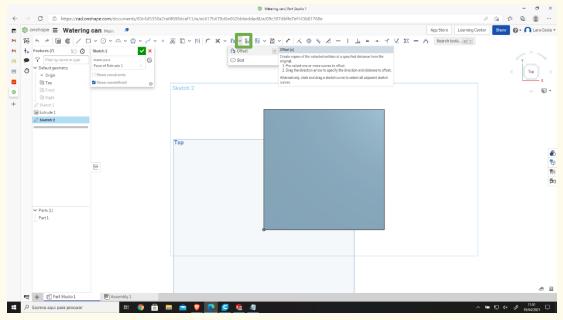


Select the plan and sketch.



Step 13

Select OFFSET.



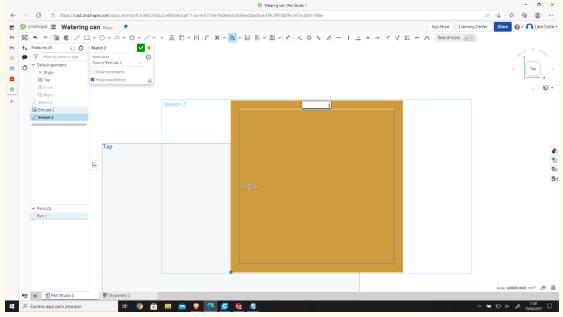


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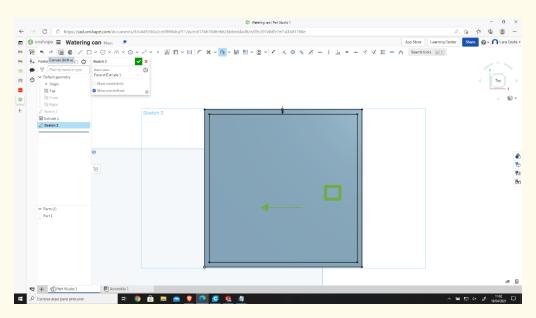


Select the outlines and give a dimension of 3mm.



Step 15

Do the same as steps 13 and 14, but this time the dimension is 0 mm.



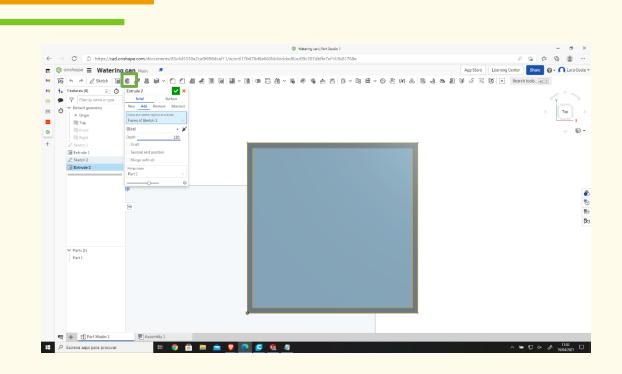
Step 16

Select Extrude and give the measure of 120mm.

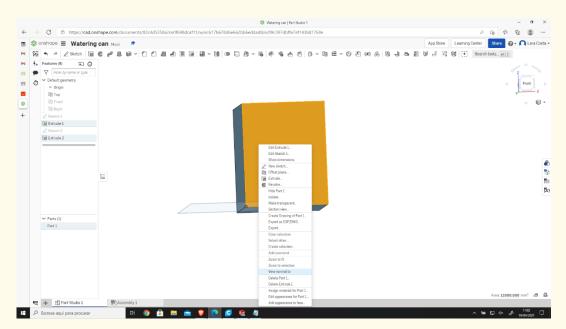


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Select the Front side, and with the right-click, choose "View normal to."

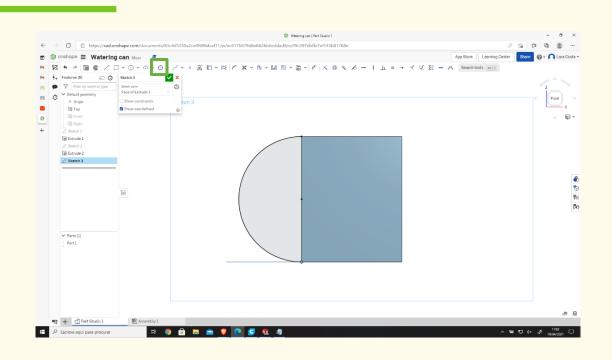


Select 3 arc points and do it like the image.

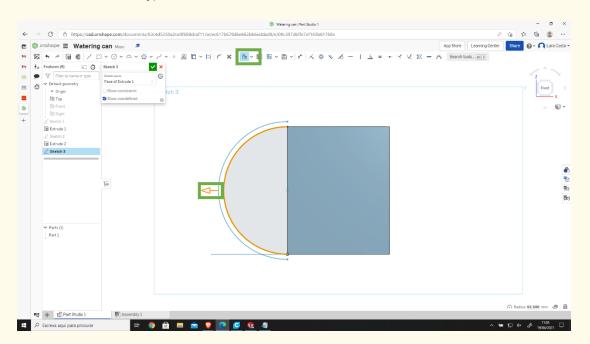


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Click on offset, then orange line, and then in the arrow to change the side (if necessary).

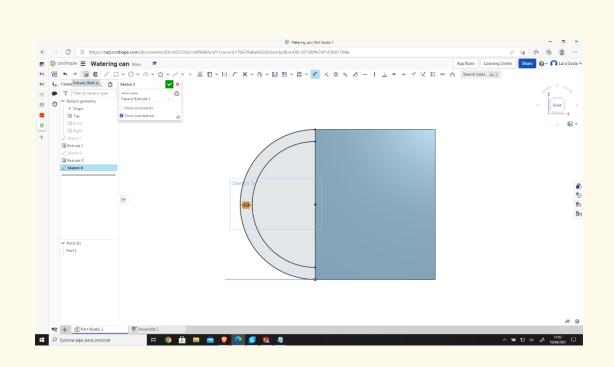


O Give 10 mm as a measure.

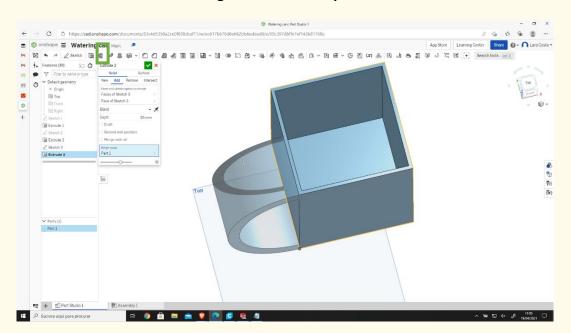


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Click on Extrude and give 55 mm as Depth.

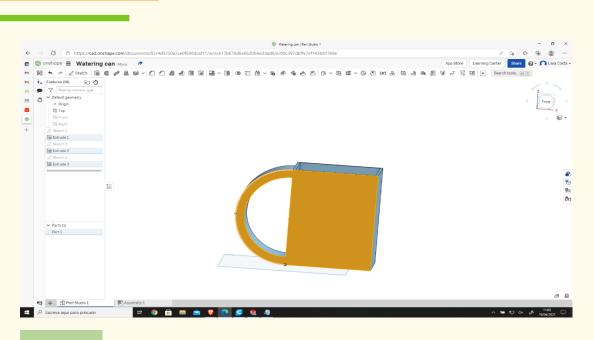


22 Select the front side again and sketch.

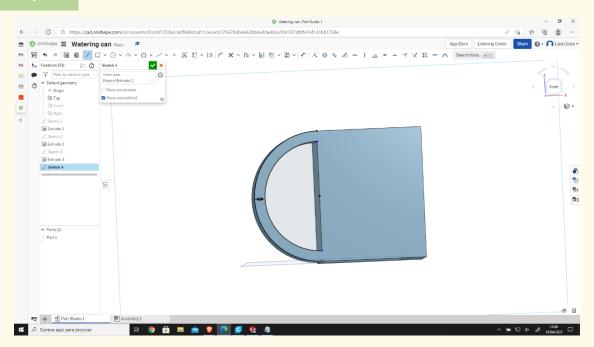


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Do the same as steps 18, 19, and 20.

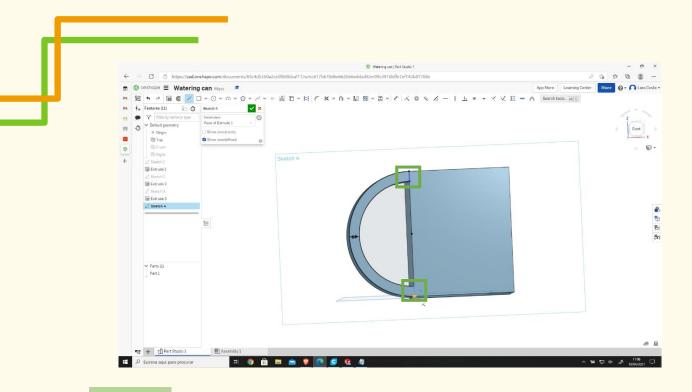


Step 24 Select line and do it as the image shows (blue lines).

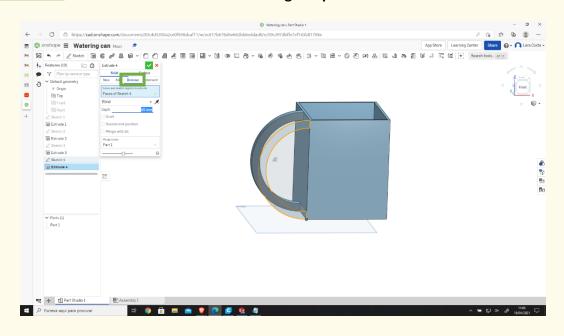


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Select Extrude – remove – change Depth to 45mm.



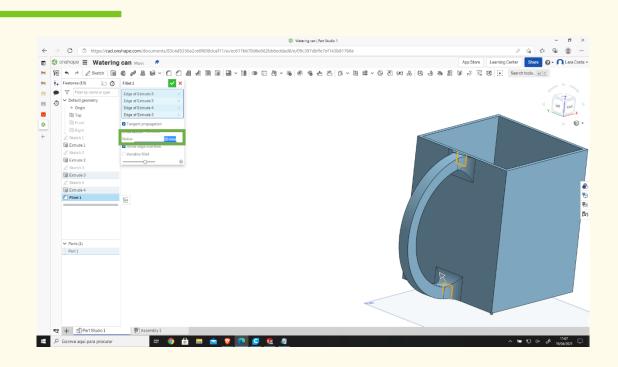
Step 26

Select Filet, then select the lines at orange and change the radius to 10mm.

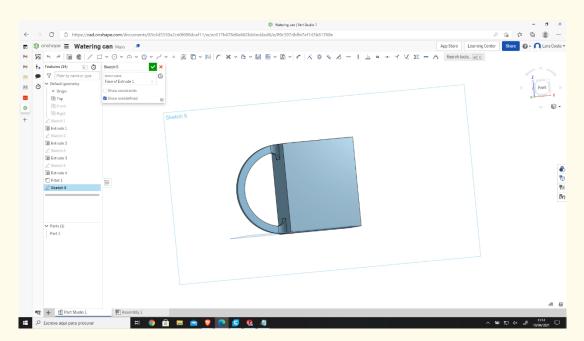


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Select the front plane (like on step 17) and click Sketch.



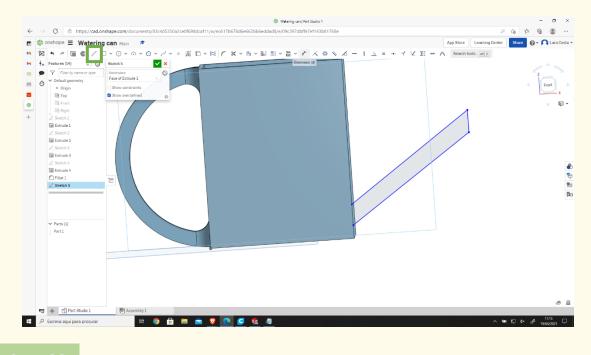


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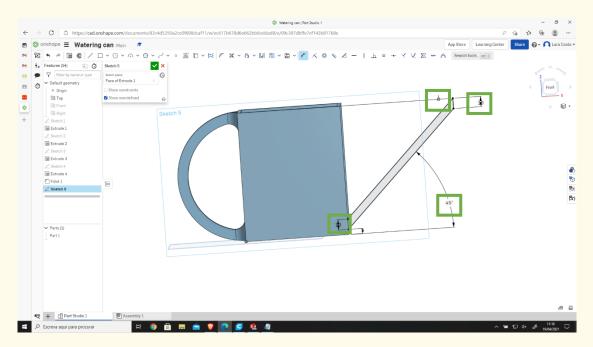


Select line and do it like the image shows (blue lines).



Step 29

Select Dimension and put the right dimensions.



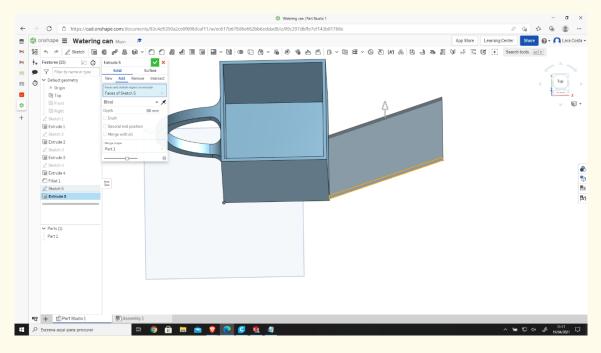


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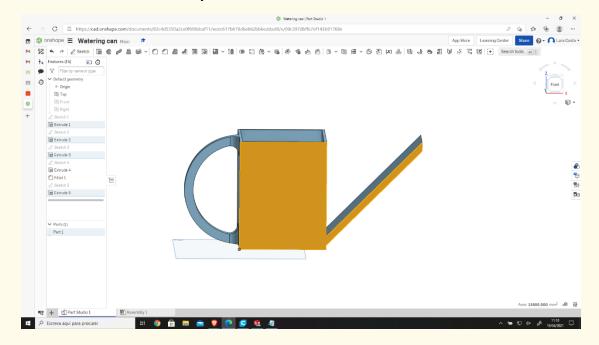


Step 31

Select Extrude and give 60 mm as Depth.



Select the front plan and then sketch.



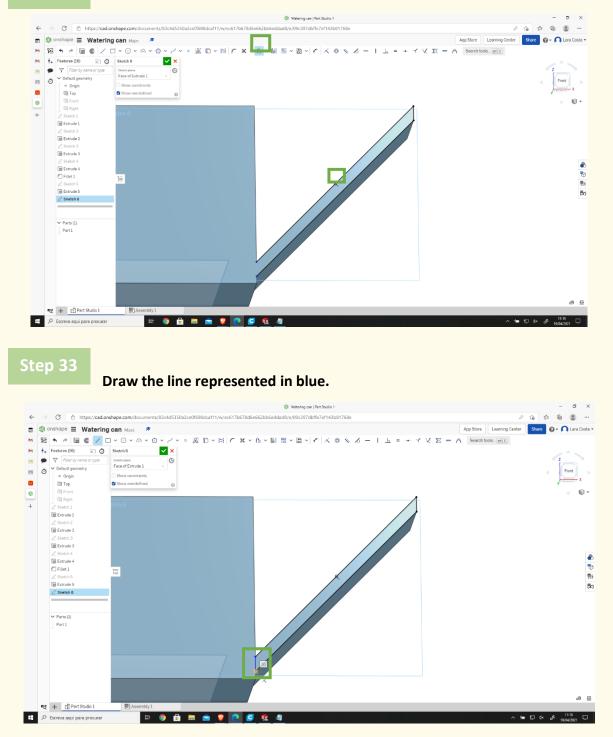


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Select offset and choose the lines in black (the measure should be 0mm).

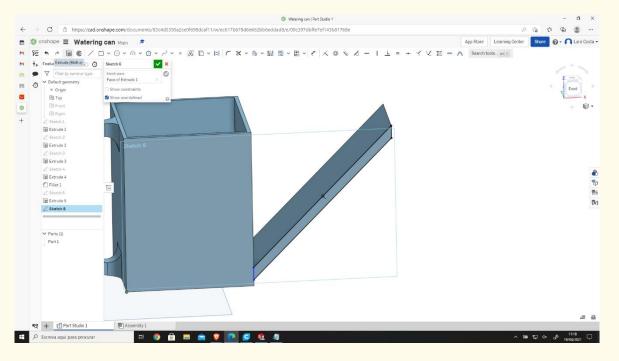




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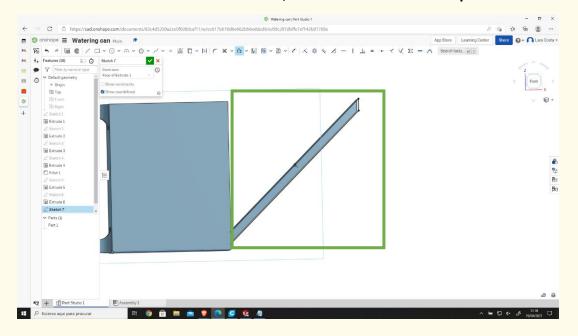


Select Extrude.



Step 35

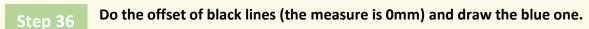
Then choose "Remove 57mm", then click on the drawn tip and then sketch.

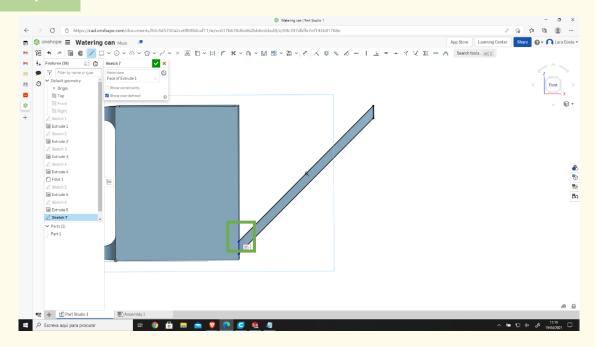




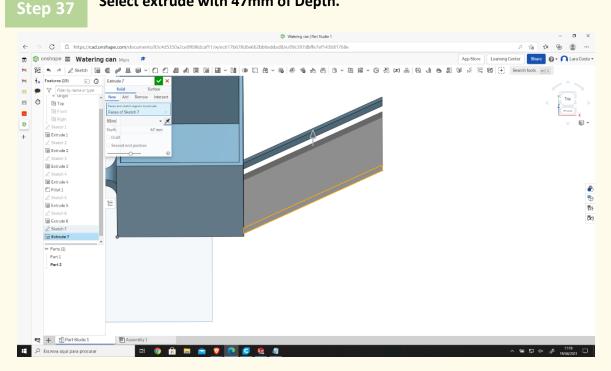
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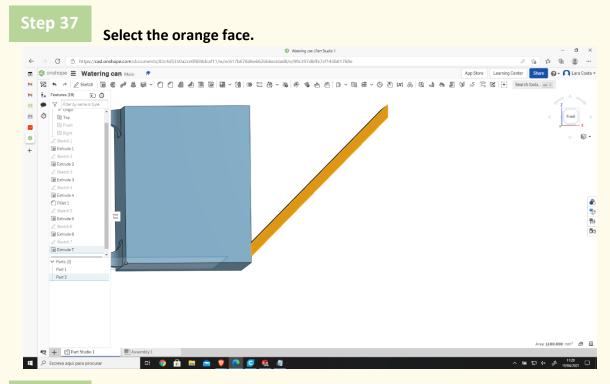
Select extrude with 47mm of Depth.





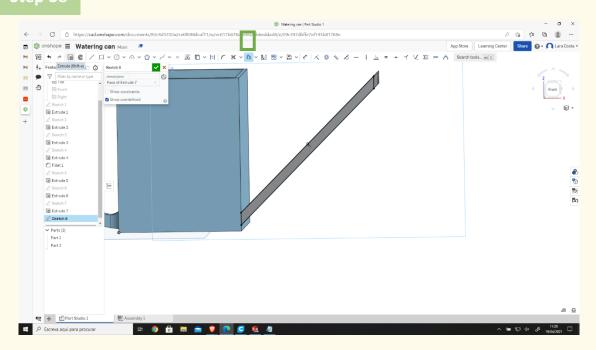
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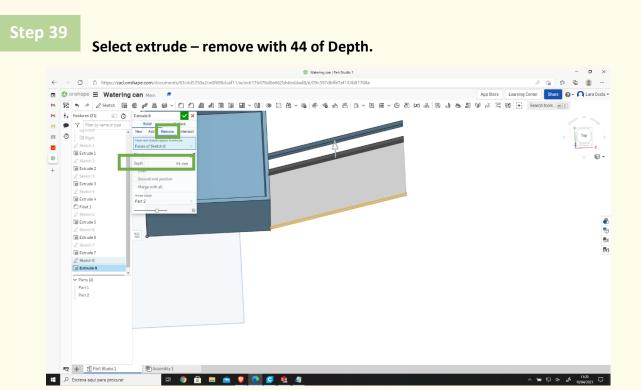
Select Sketch and do the offset (the measure is 0mm) of the black lines.

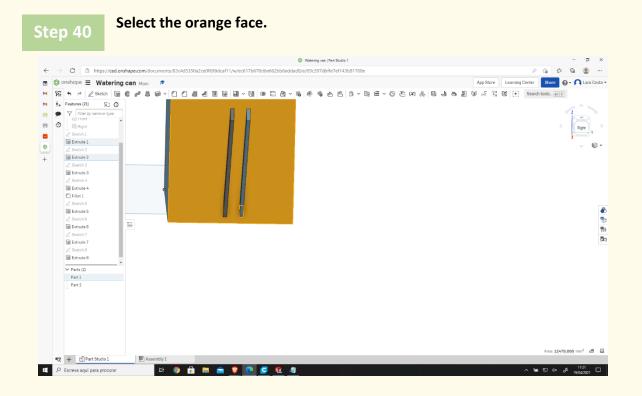




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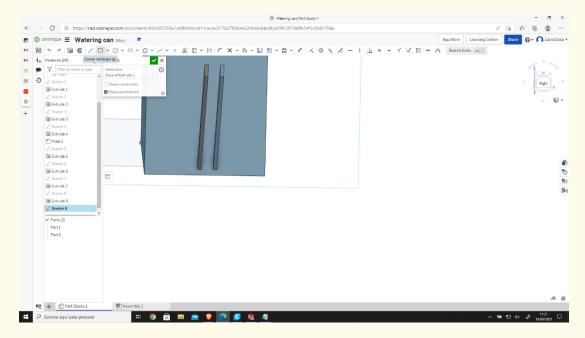


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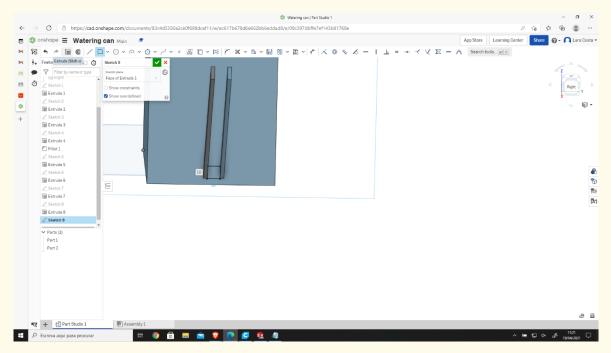


Select corner rectangle.



Step 42

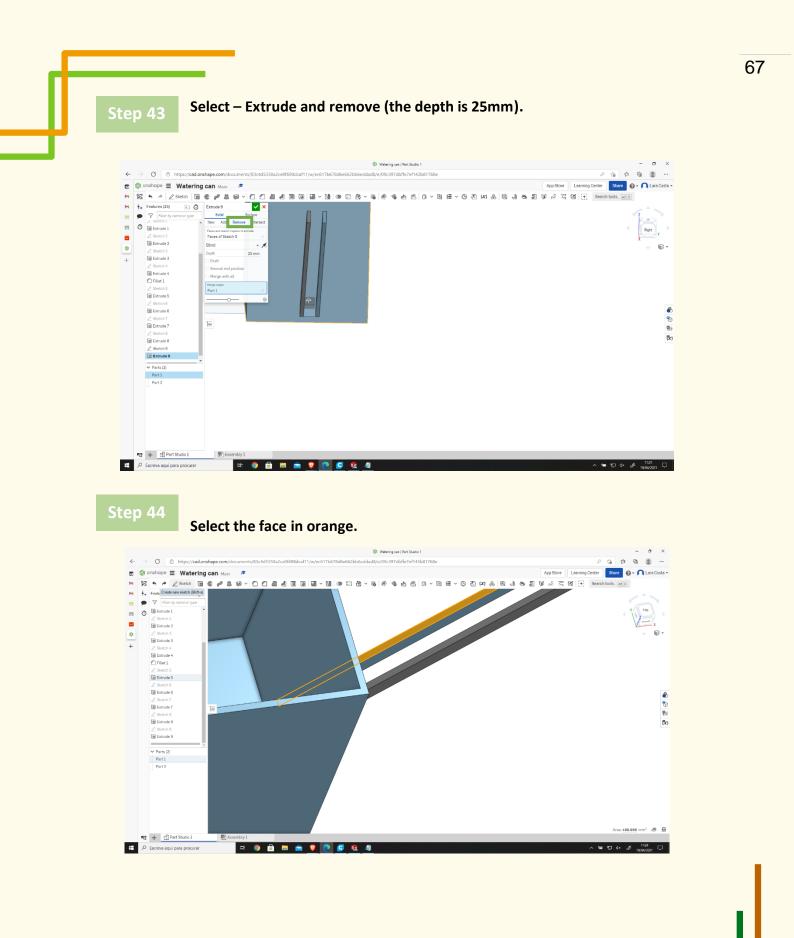
Do the rectangle as represented.





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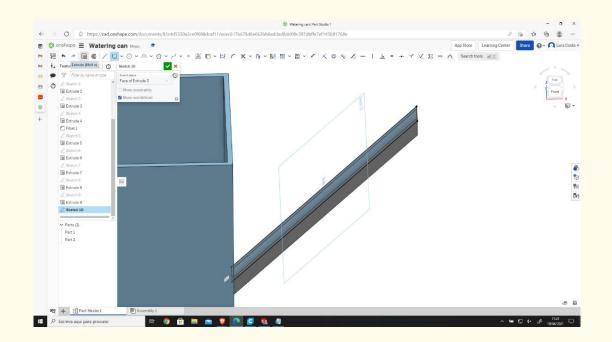


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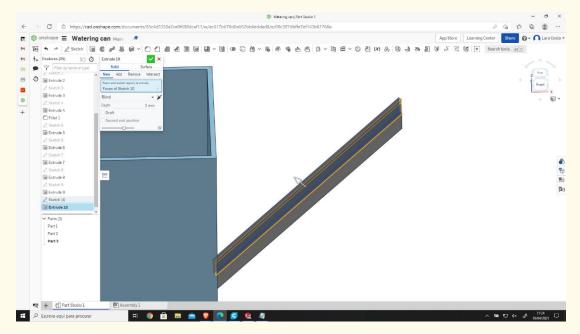


Do a rectangle as represented (on the top of the spout of the watering can).



Step 46

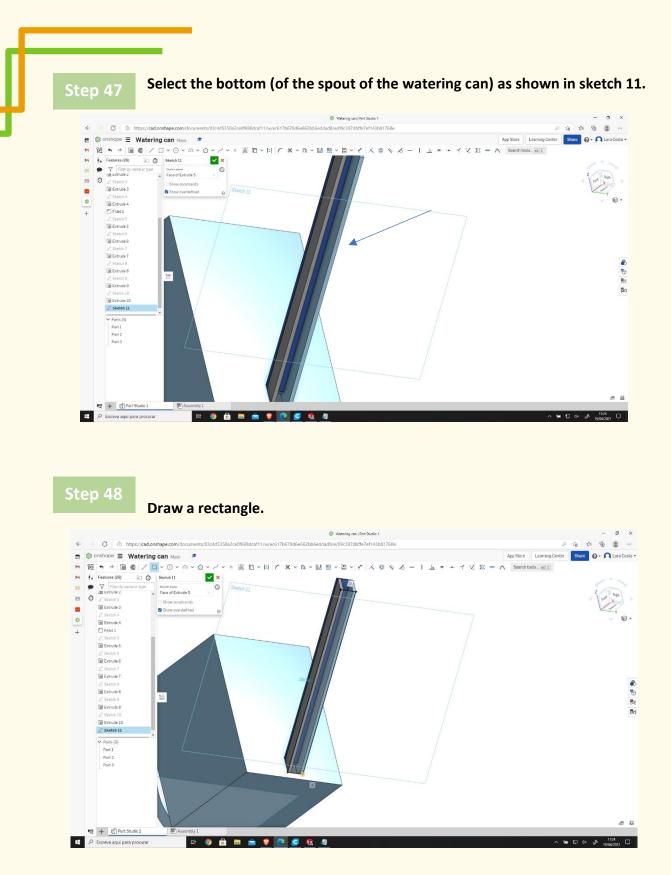
Select Extrude, then put the Depth to 2mm.





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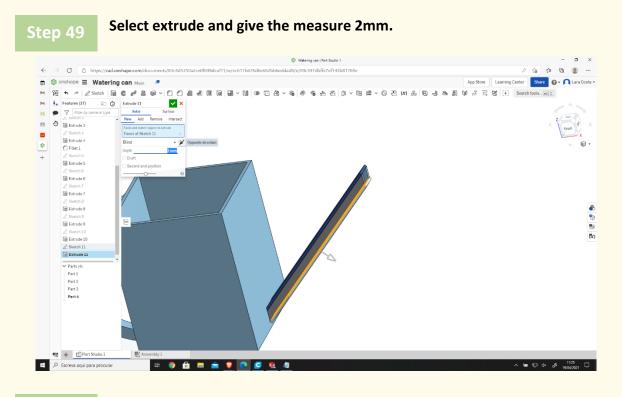




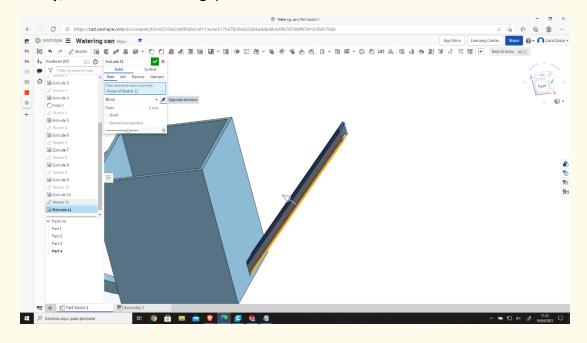


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Select opposite direction (click on the arrow to change the direction if necessary, as shown in the image).

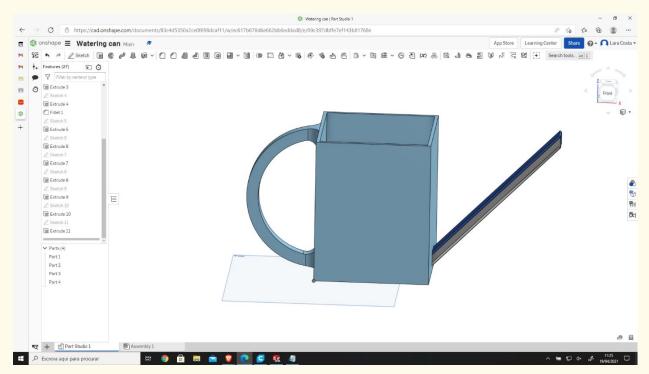


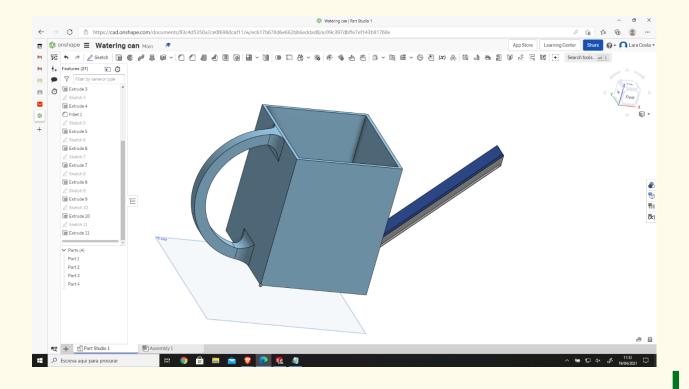


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The final result should look like this.







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PROJECT: 3D DRAWING OF A GARDEN RAKE

- **<u>STEM field</u>**: Science, technology, and electronics.
- Indicative calendar: Any time of the year.
- Activity duration: 3 hours.
- **<u>Type of activity:</u>** Drawing of a Garden Rake.
- **Educational objectives:** By the end of the course, the learners are expected to draw a garden rake on the Onshape software.
- Learning outcomes and acquired competencies:
 - How to do a garden rake on Onshape.
- <u>Required material and resources:</u>
 - Computer;
 - Internet access;
 - Onshape account (or other similar).

• Description and/or step-by-step instructions

This project consists of the 3D design of a garden rake, then we will present the step by step process of its elaboration:





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			2013 - Present, Onshape Inc. All Rights Reserved	Terms & Privacy	(1.123.26505.3aB/o4305253)		



Create a document.

Open Onshape.

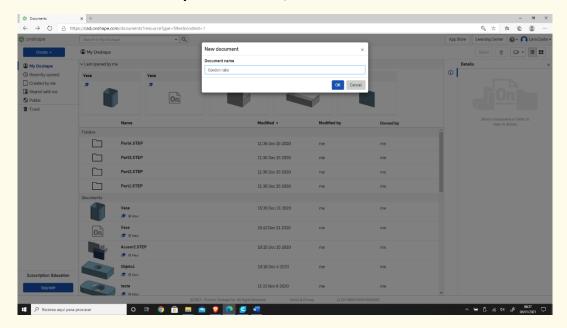
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Give a name to your document, such as Garden Rake.



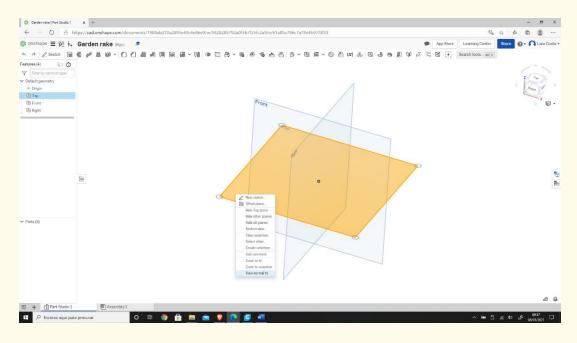
Select the plane (top) to start drawing. - o × 血 喧 🙁 … Q \$ App Store Learning Center Share @ - 🔍 Lara Costa -🔕 onshape 🗮 輝 🗛 Garden rake Main 🛛 🕏 700 From pit **.** も であ ĭ ✓ Parts (0) e 01 🛅 As 0 🛱 🌖 🔒 🛤 😭 🦁 💽 🗲 P Escreva aqui para procurar .

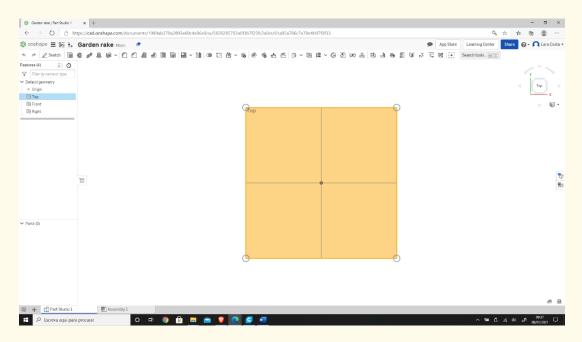


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Right-click and select normal view. The plan should look like the 2nd image.



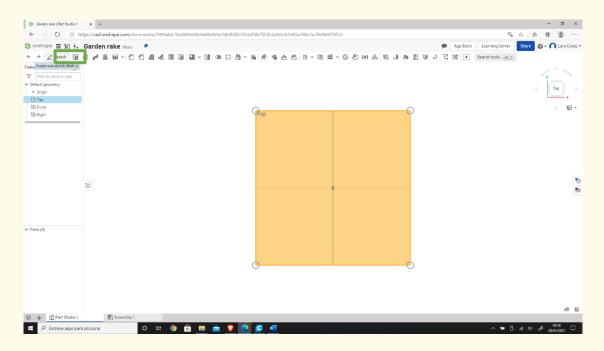




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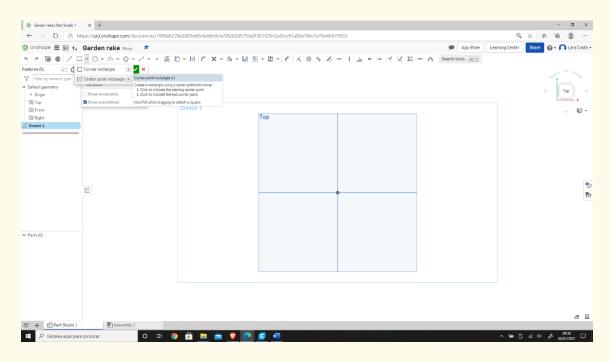


Click Sketch.



Step 7

Select the center point of the rectangle to draw.

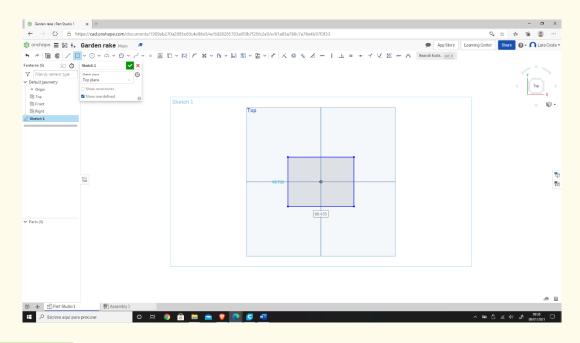




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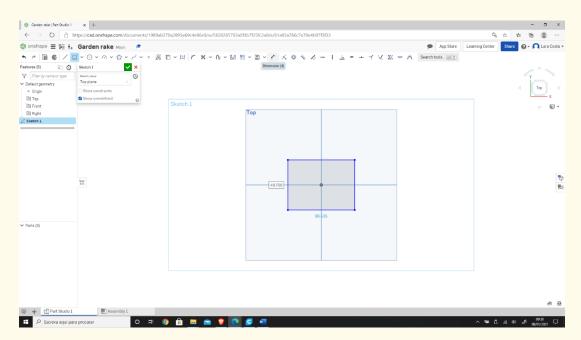


Draw the rectangle.



Step 9

Select "Dimension" to define the measures.

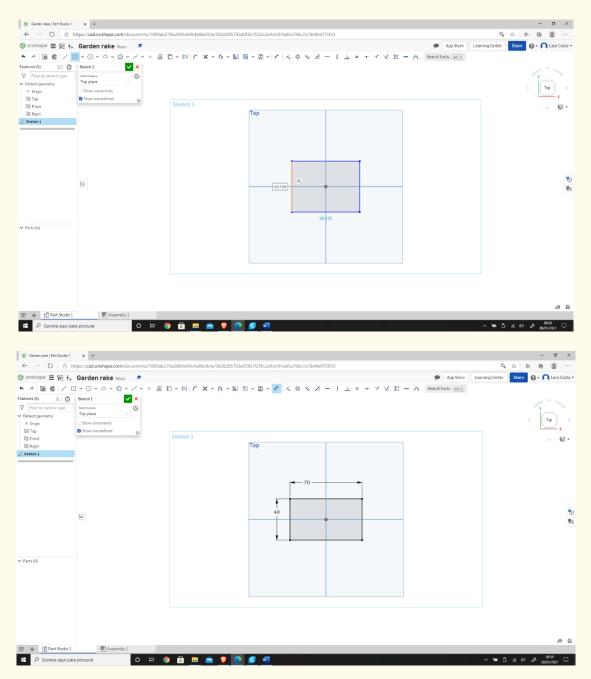




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Select the line (first left then top) and give the measure 2nd figure.





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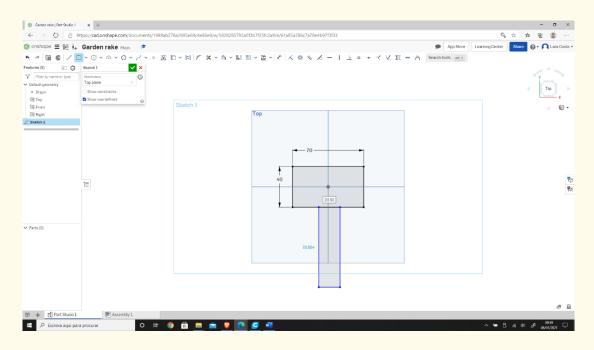


Select corner rectangle.

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Filter by name or type	Center point rectangle R 1 Click to india	le using two points for diagonal corners. ata the starting corner point			\sim
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Step 12

Draw a similar rectangle as shown (start by drawing on top of the bottom line of the first rectangle drawn).

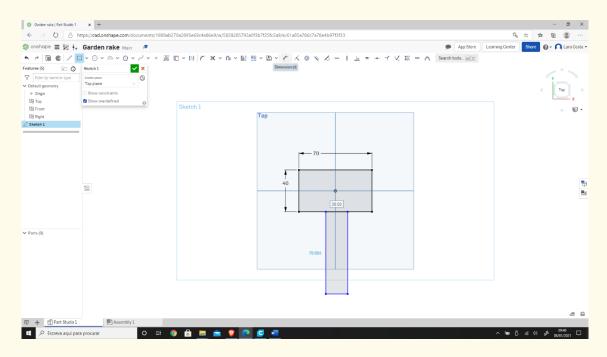




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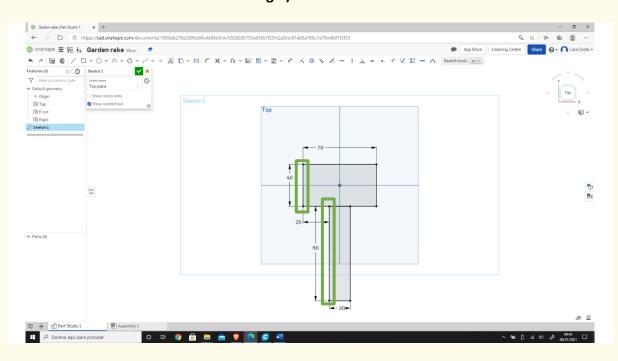


Select Dimension to give the measures.



Step 14

Enter the measures as shown in the image below: (to measure 25mm, click on the left line of the 1st rectangle and then on the left line of the 2nd rectangle).

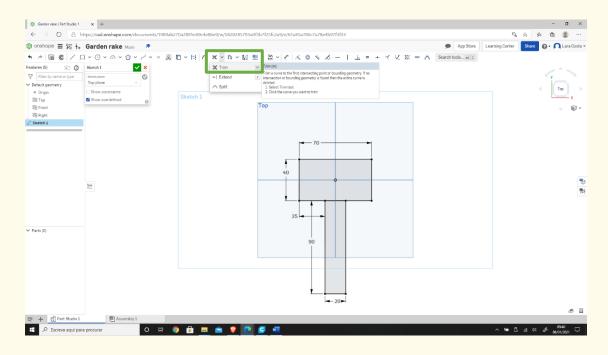




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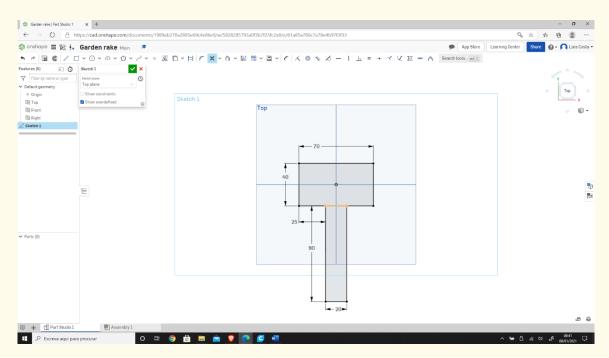


Select Trim.



Step 10

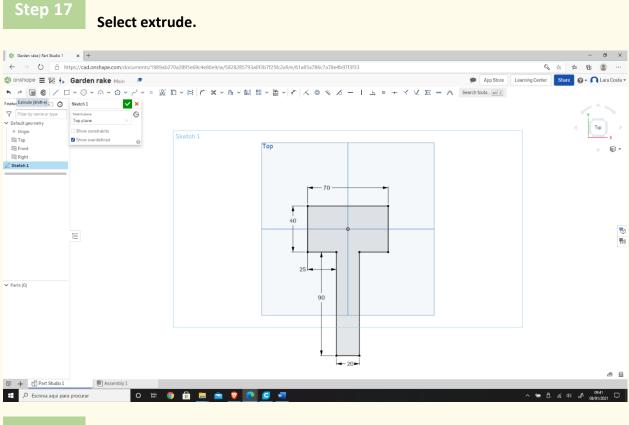
Select the line in orange (then the line will disappear).



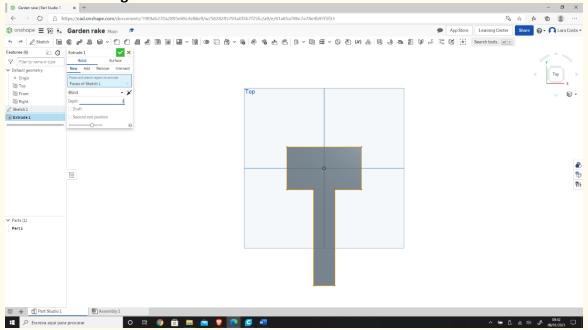


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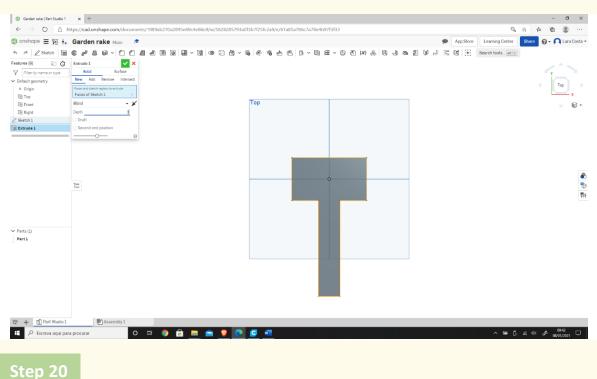




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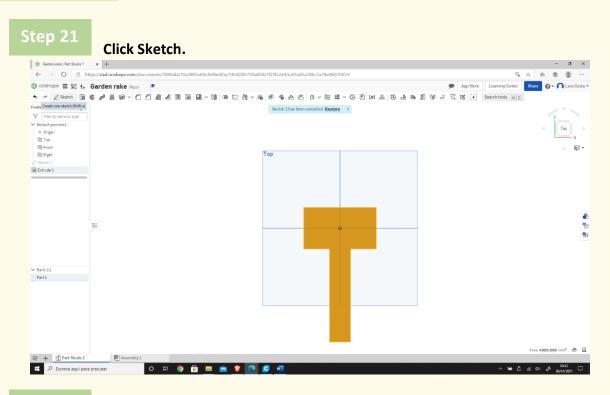
Select the front.

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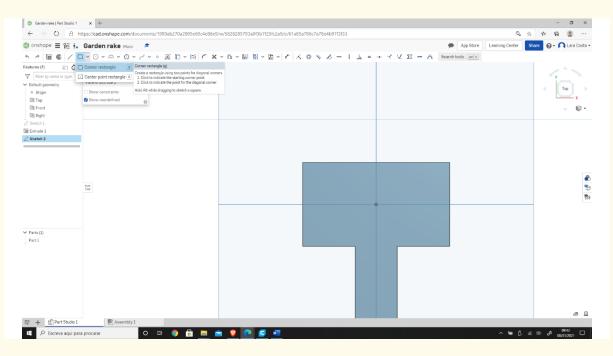


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Select Corner rectangle.

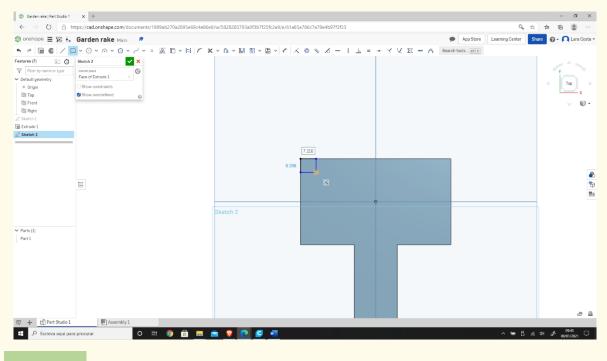




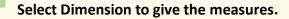
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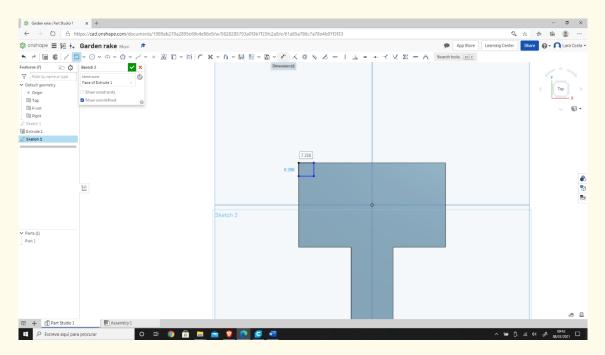


Draw a similar rectangle as shown.



Step 24



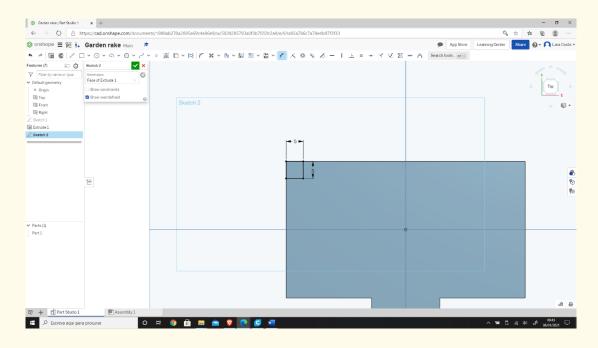




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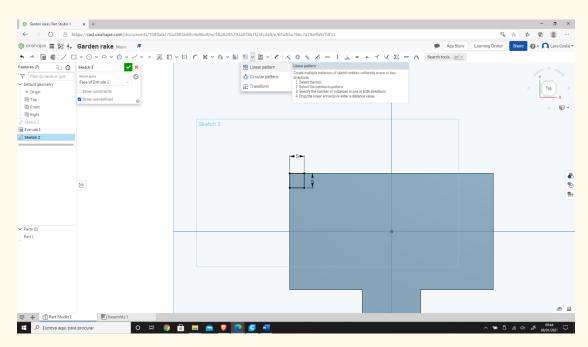


Enter the measures as shown in the image below.



Step 26

Then select a Linear pattern.

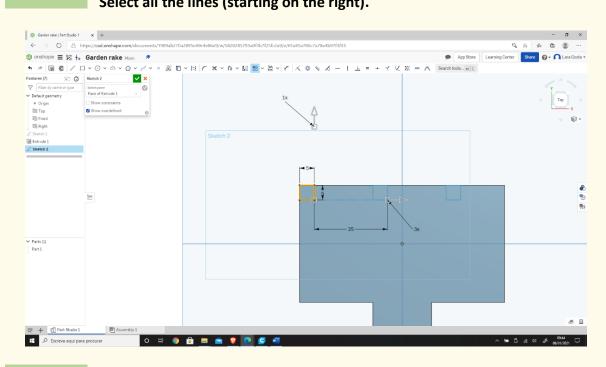




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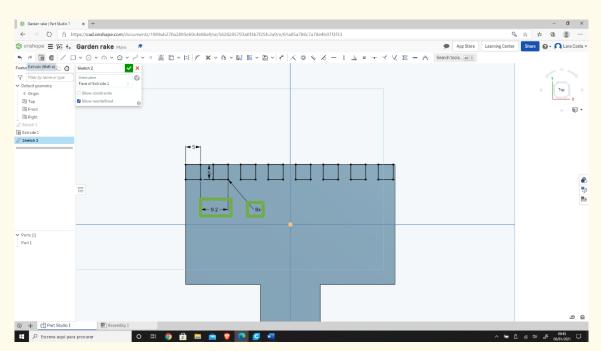
Select all the lines (starting on the right).



Step 28

Enter the parameters as shown in the image below (inside the green

rectangles).

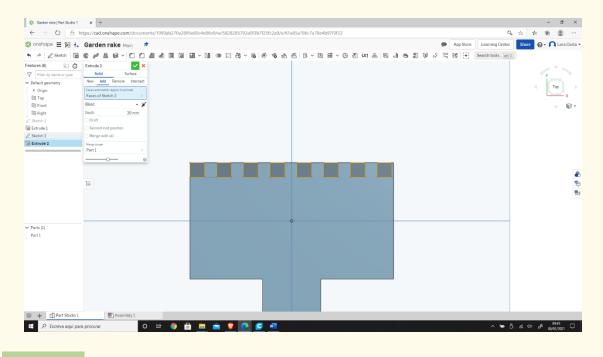




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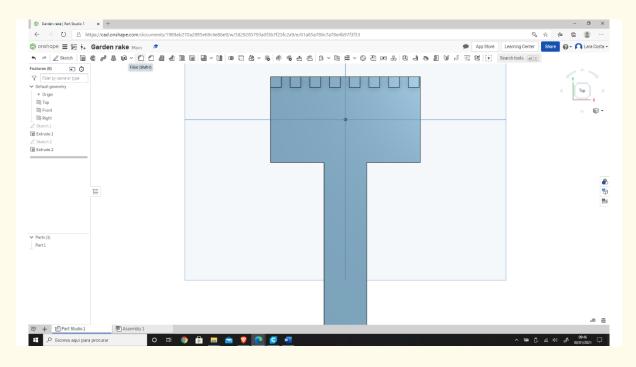


Select Extrude, and the measure is 20mm.



Step 30

Select Fillet.

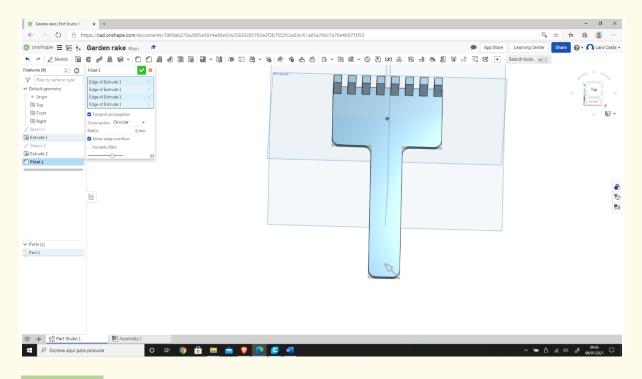




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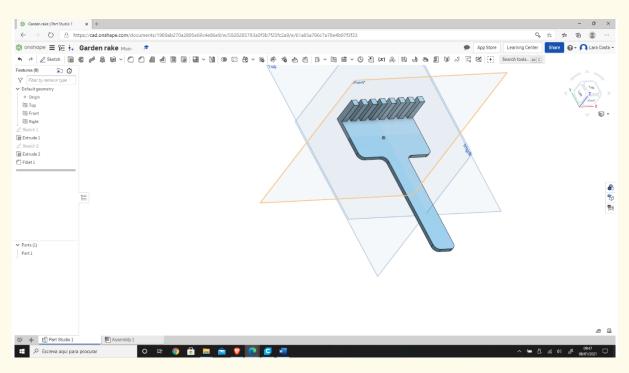


Select all the corners (edges) that we want to fillet, as the image shows.



Step 32

This should be your result.





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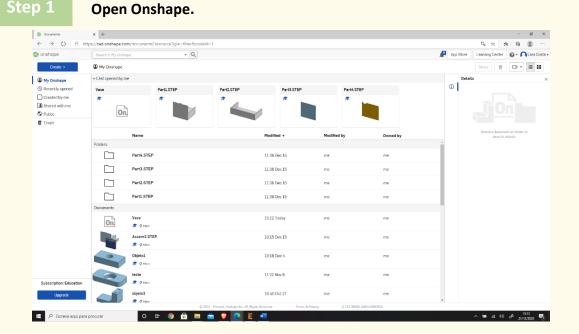
PROJECT: 3D DRAWING OF A VASE

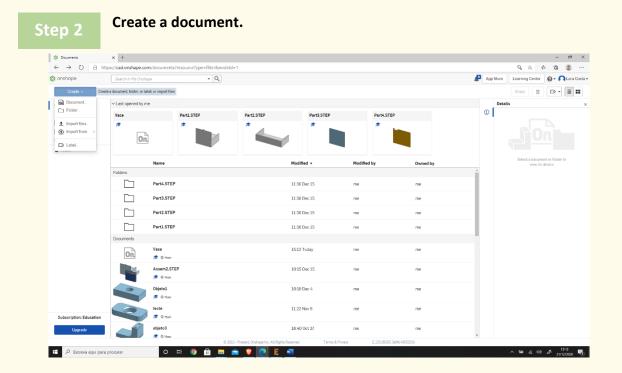
- **<u>STEM field:</u>** Science, technology, and electronics.
- Indicative calendar: Any time of the year.
- Activity duration: 3 hours.
- <u>Type of activity:</u> Drawing of a vase.
- Educational objectives
- Learning outcomes and acquired competencies:
 - How to do a vase on Onshape.
- <u>Required material and resources:</u>
 - Computer.
 - Internet access.
 - Onshape account (or other similar).
- Description and/or step-by-step instructions:

This project consists of a 3D design of a vase, then we will present the step by step for its elaboration:











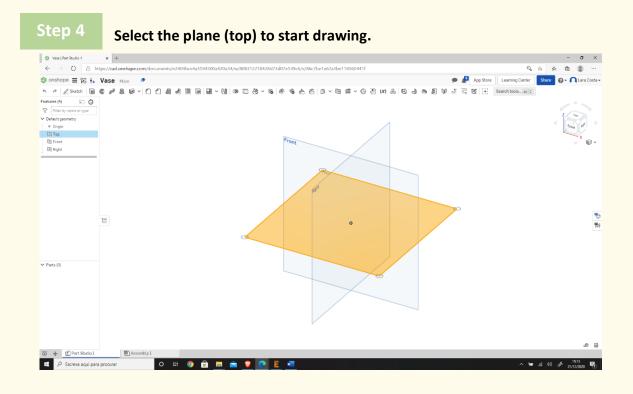
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Give a name to your document such as "Vase."

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	Documents					
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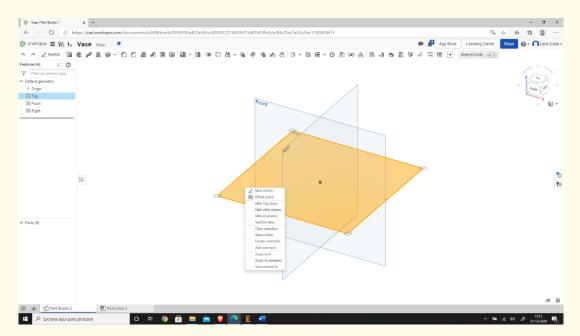


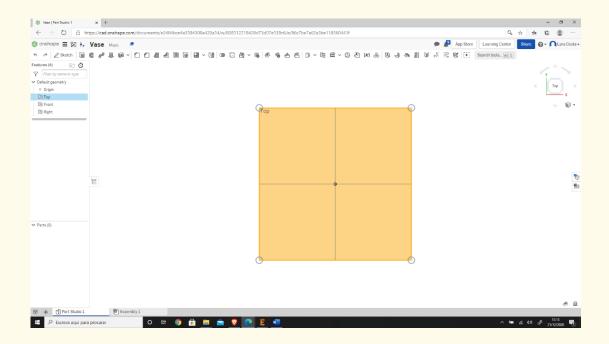


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Right-click and select "normal view to". The plan should look like the 2nd image.

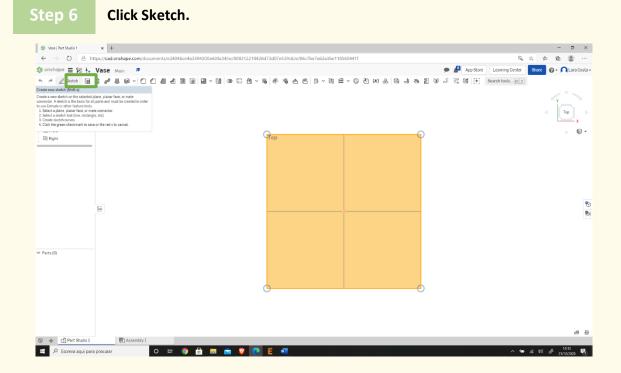






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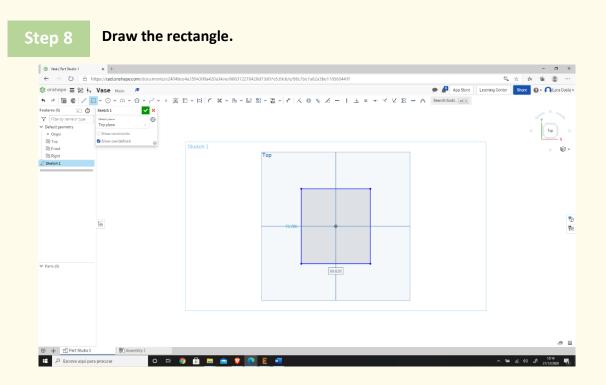


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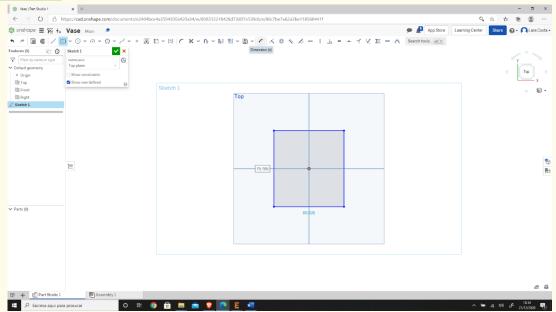


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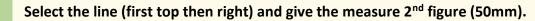


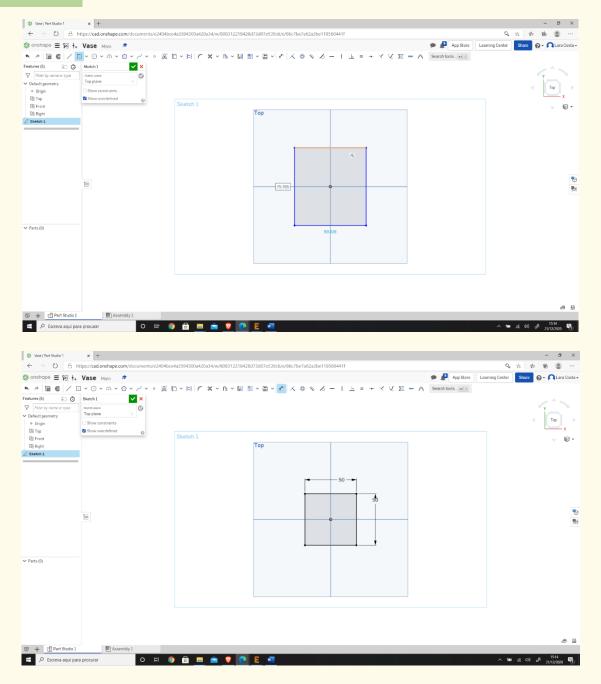
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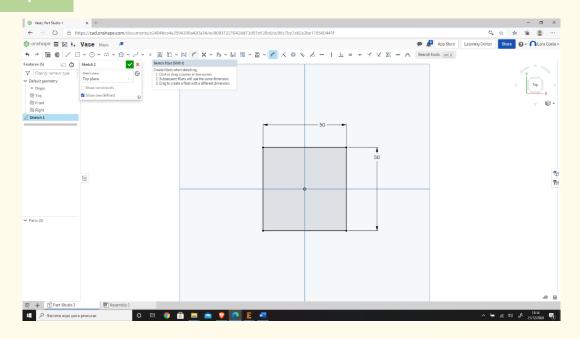


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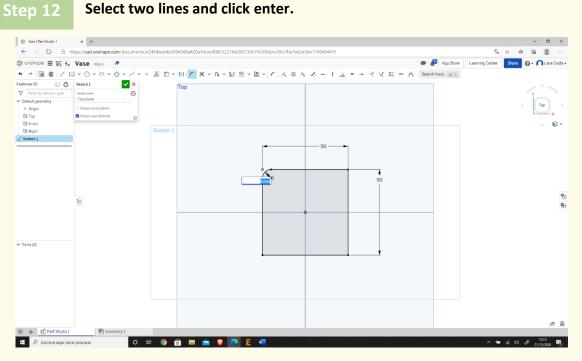


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Select sketch fillet. Step 11



Select two lines and click enter.

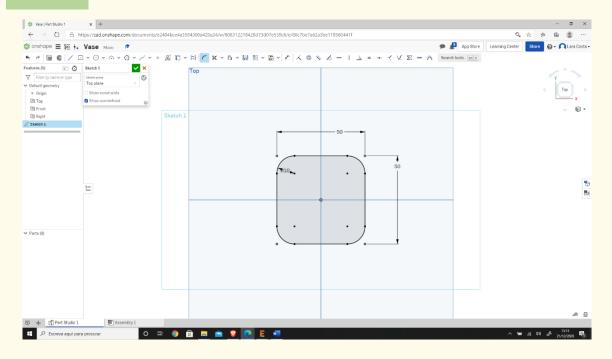




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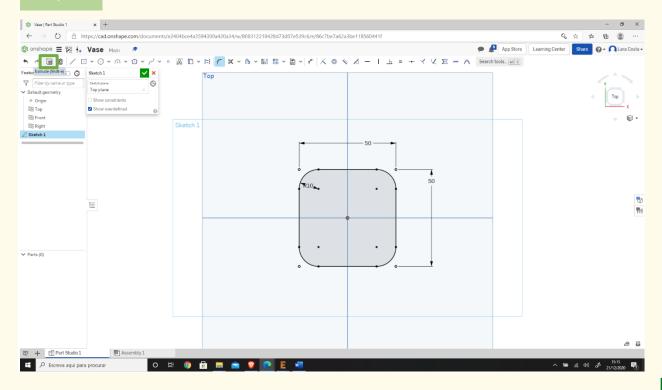


tep 13 Enter the measure 10mm and repeat the process to every corner.



Step 14

Select Extrude.



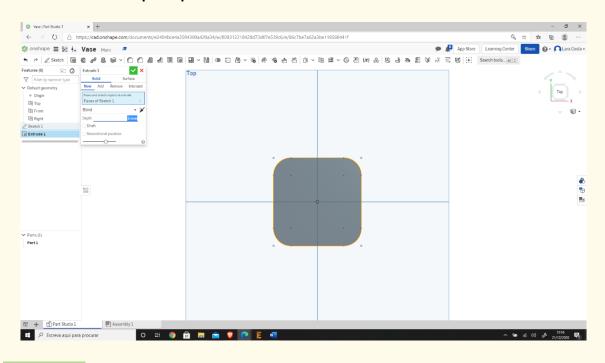


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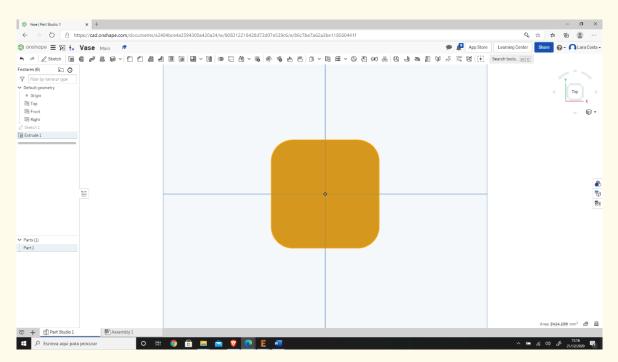


Put Depth equal 3mm and click 🗹 .



Step 16

Select the front face.



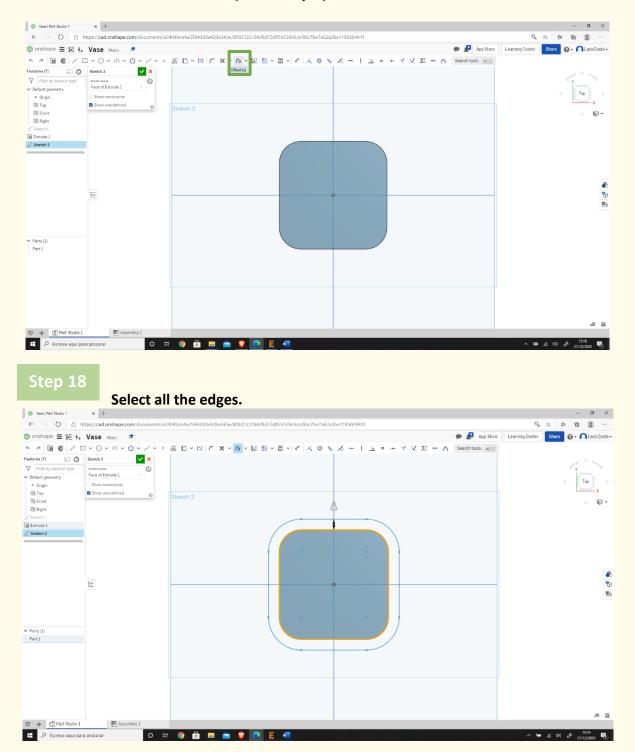


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First select Sketch (like in step 6) and then offset.

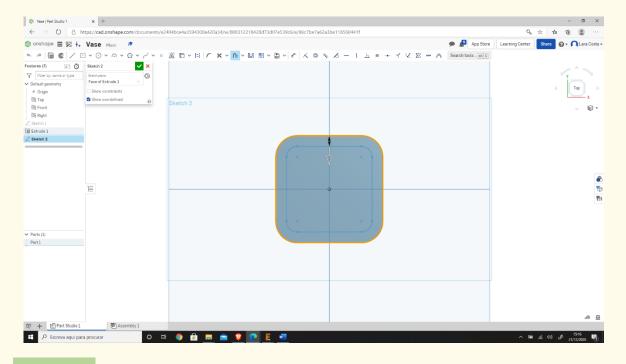




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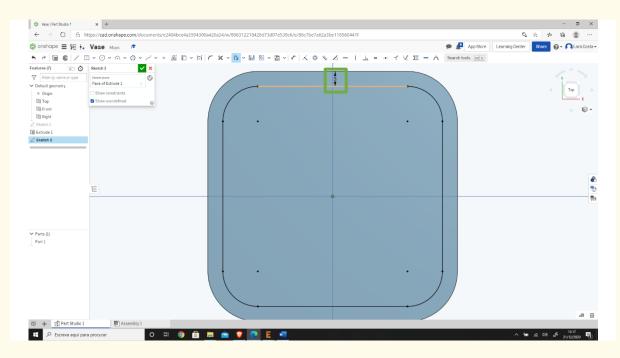


Click on the arrow if necessary to have this result.



Step 20

Change the measure to 3mm.

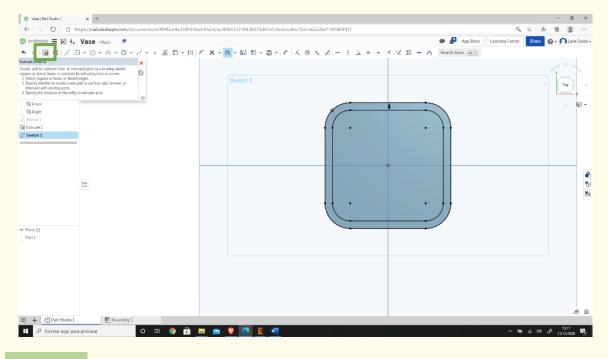




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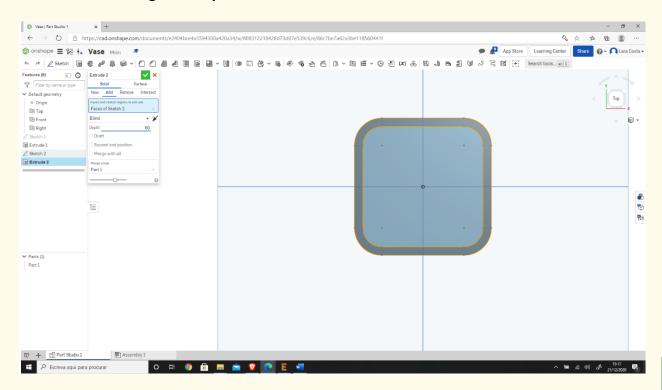


Do it again (like it is done on steps 18,19 and 20) but now the measure is 0, and click extrude.



Step 22

Change the Depth to 60mm and click **2**.

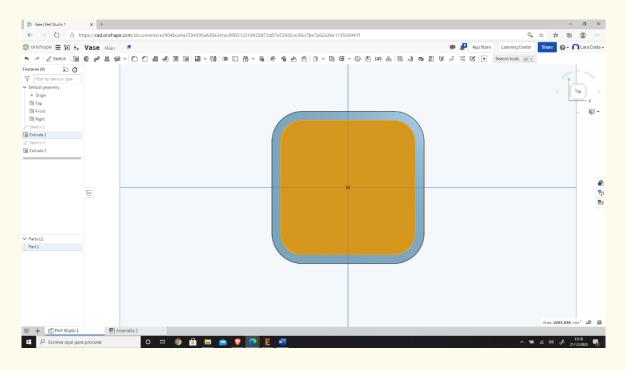




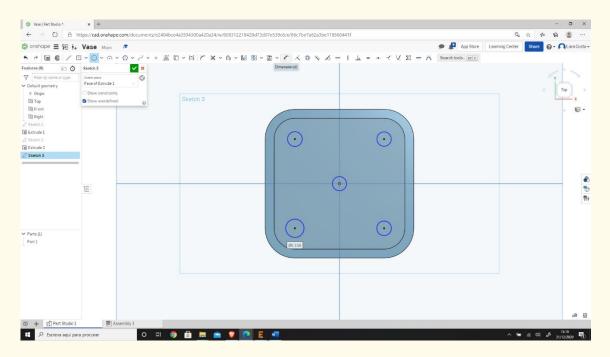
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Select the base and click sketch (like on step 6).



Draw a circle in the center and one in each corner (the circles do not need the same exact measures from the origin).

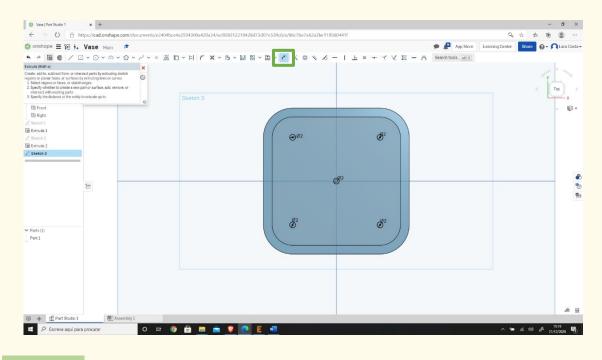




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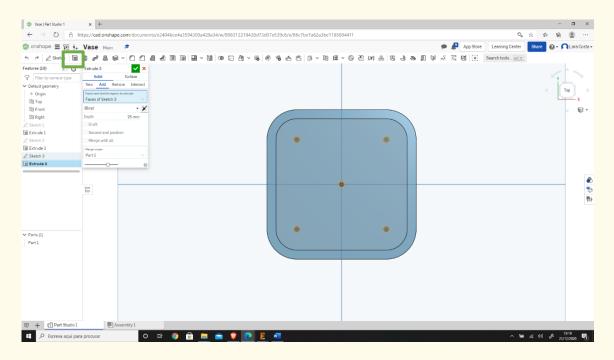


Insert the measure (2mm) by clicking in dimension and then in each circle.



Step 26

Then select extrude.



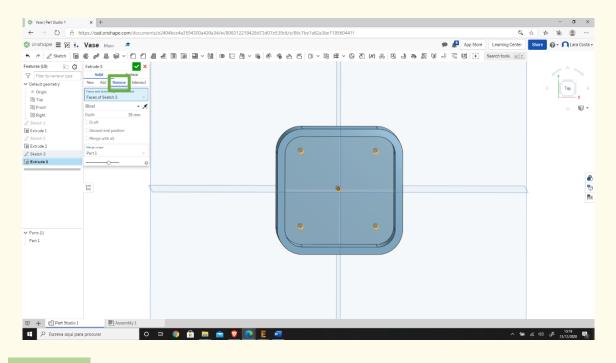


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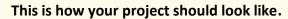


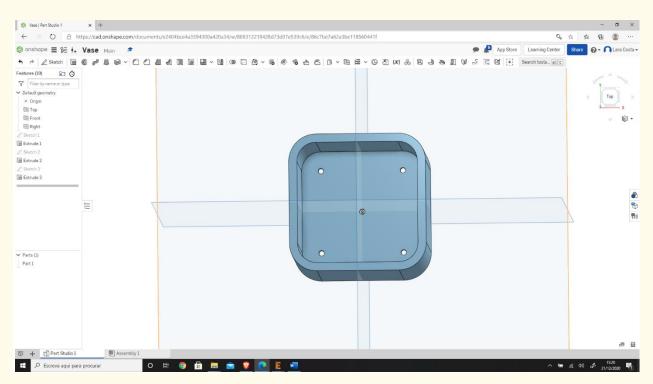


Click Remove 🔼



Step 28

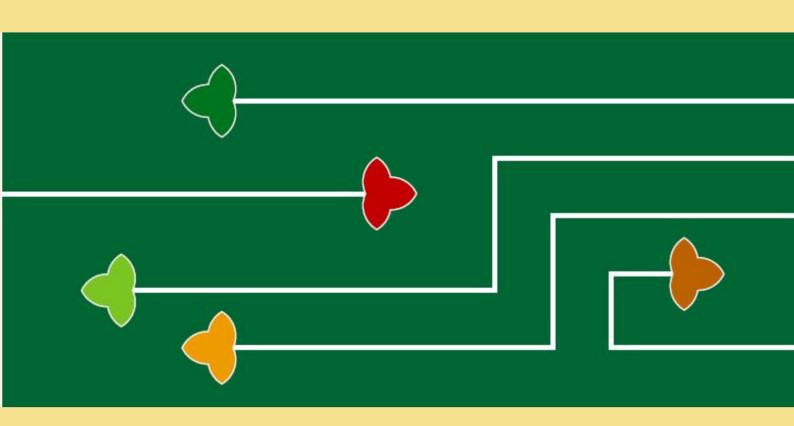






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