

Green
STEAM
Incubator

Handbook on Environmental Projects using Onshape

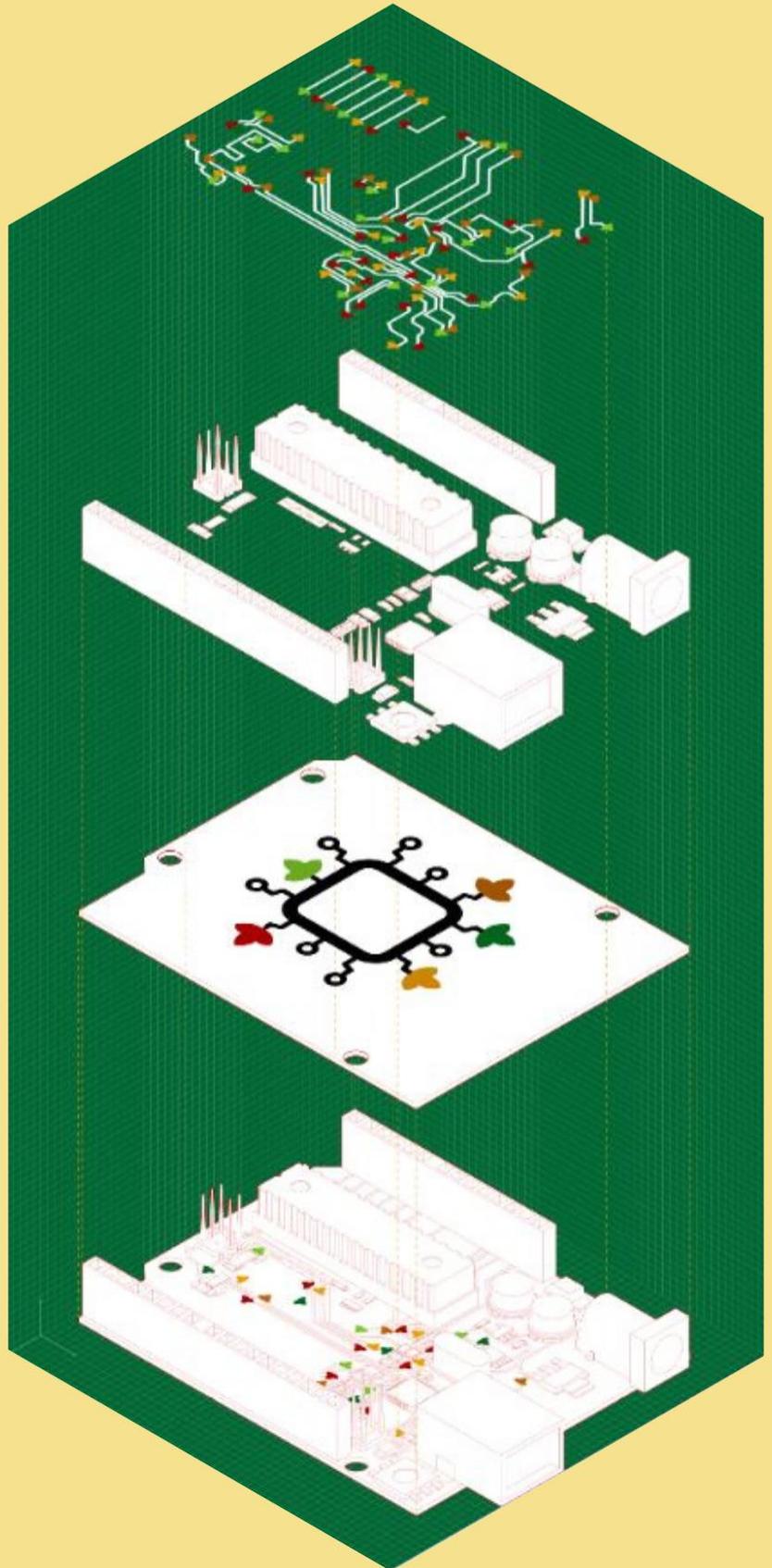


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INTRODUCTION

This handbook was created within the scope of Intellectual Output 4 of the Green STEAM Incubator project, which is related to 3D modelling. The objective was to create a document with ideas for environmental projects that make use of 3D modelling 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 modelling 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 to each session is given at the beginning of each project idea.



PROJECT: 3D DRAWING OF A BIRD FEEDER

- STEM field: 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 know how to draw a bird feeder on the Onshape software.
- Learning outcomes and acquired competences: 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.
- Required material and resources:
 - Computer.
 - 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 that the main element is the cylinder. So, a good practice is to start with the design of this part of the object.

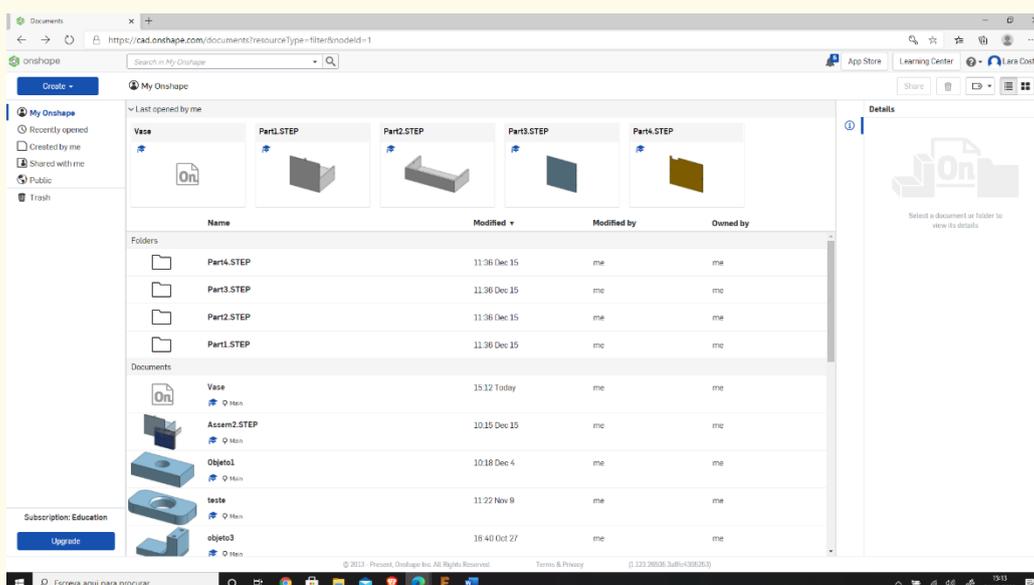
After that, we can see that the cylinder is hollow or emptied out with a missing top and a couple of holes. In addition, we have some more features on the main cylinder, the so-called perches (i.e., in the figure below, those appear like circular stands for the birds; those could also be simpler cylindrical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

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



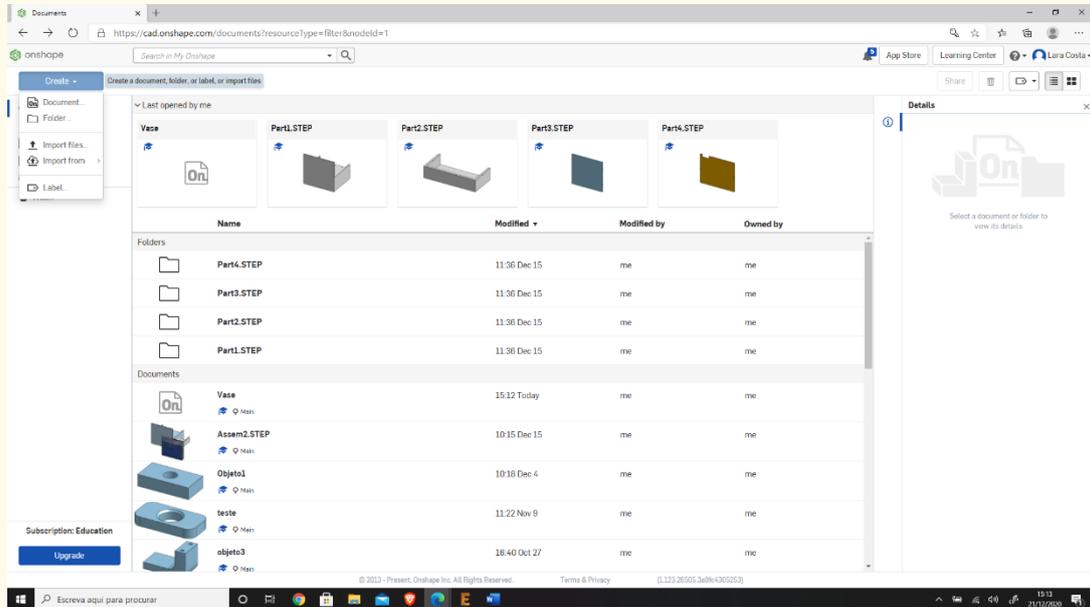
Figure 1 A bird feeder, source: <https://www.theguardian.com/environment/2020/feb/27/country-diary-the-gatherings-at-the-bird-feeder-are-anything-but-random>

Step 1 Open Onshape.



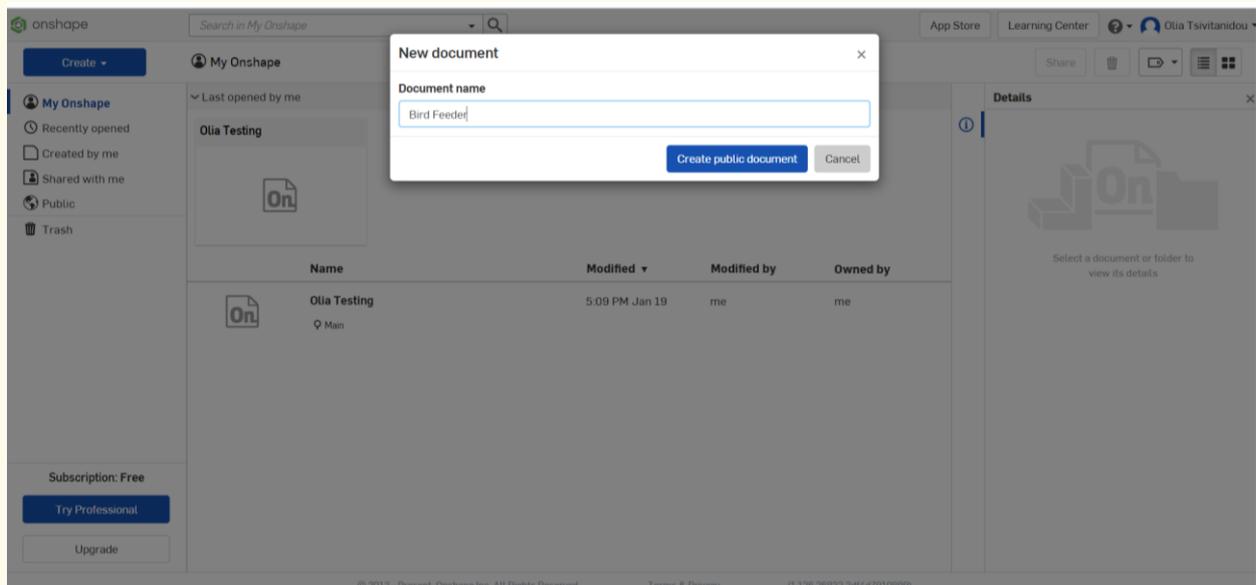
Step 2

Create a document.



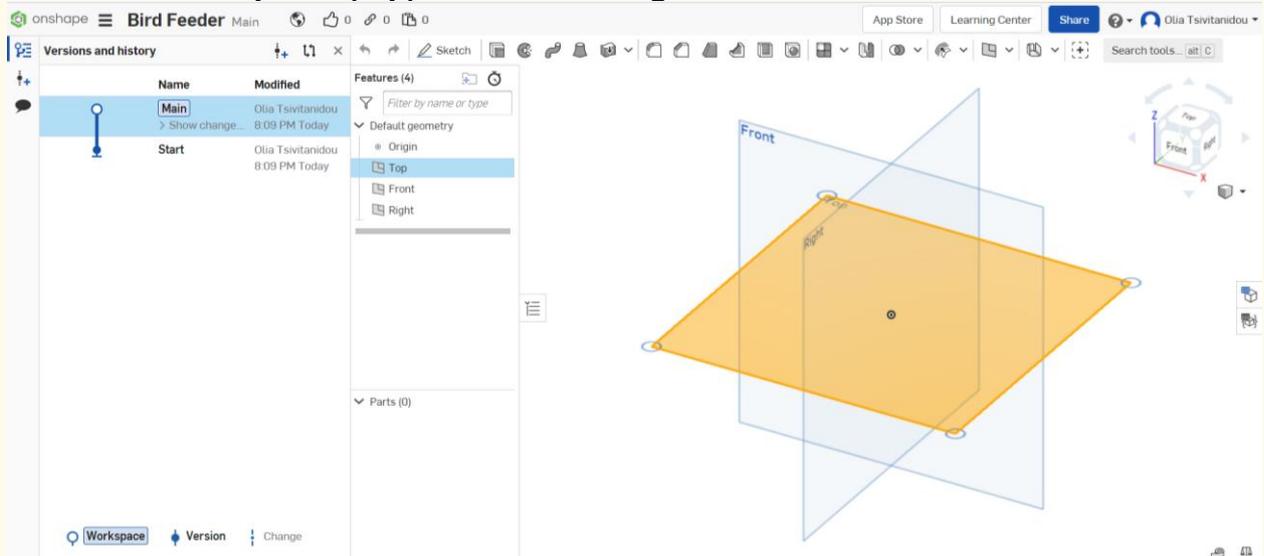
Step 3

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



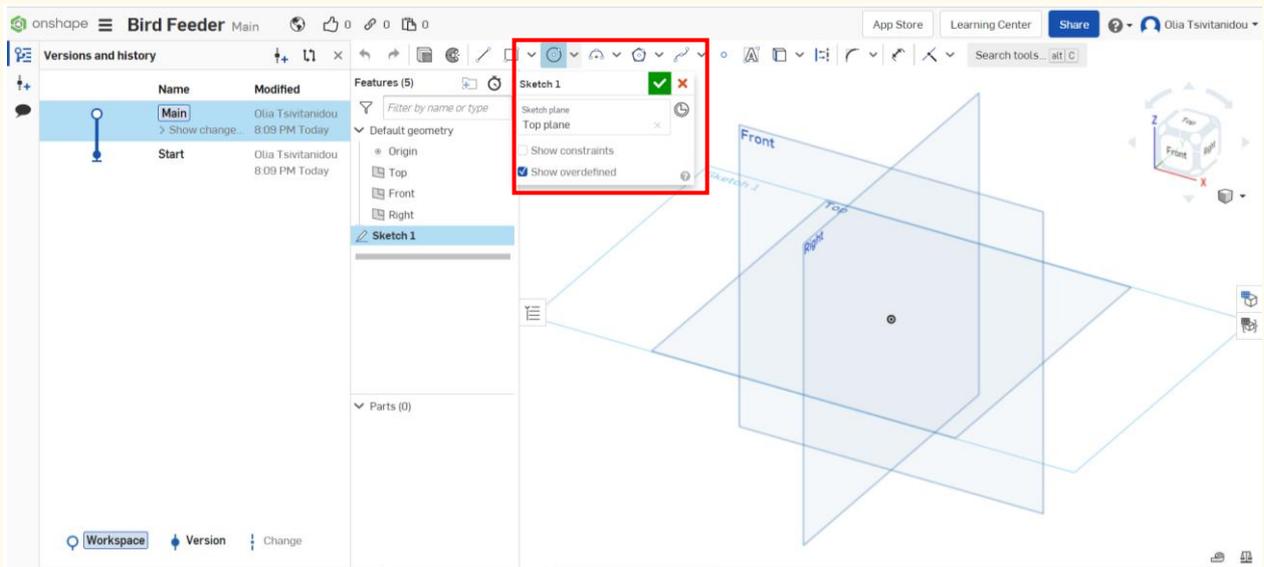
Step 4

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.



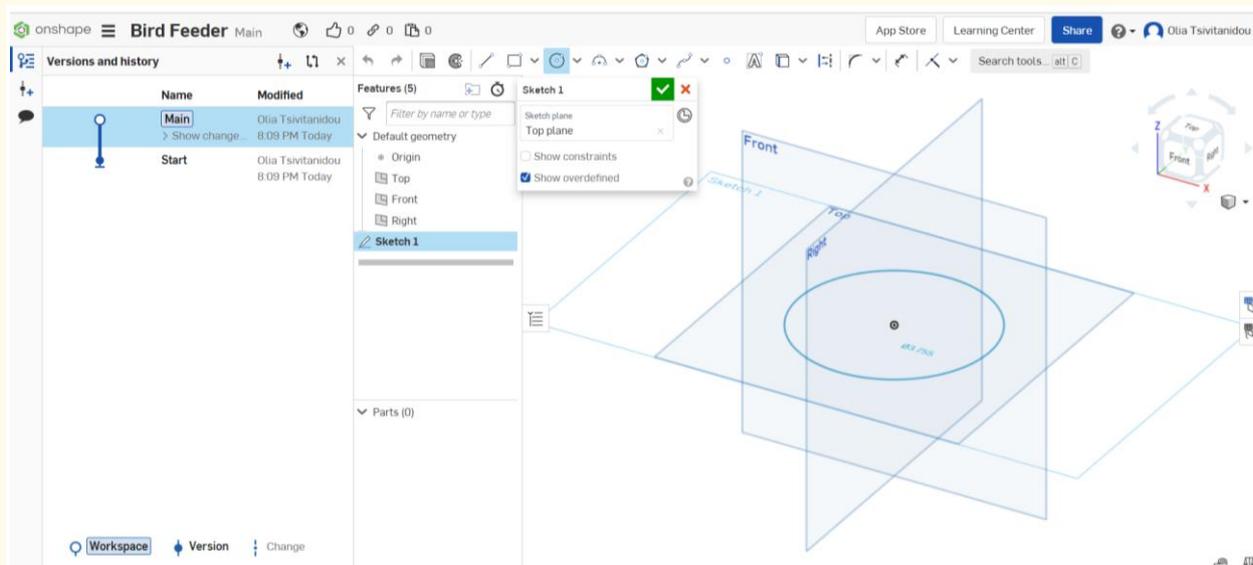
Step 5

Select "Sketch," then "Circle."



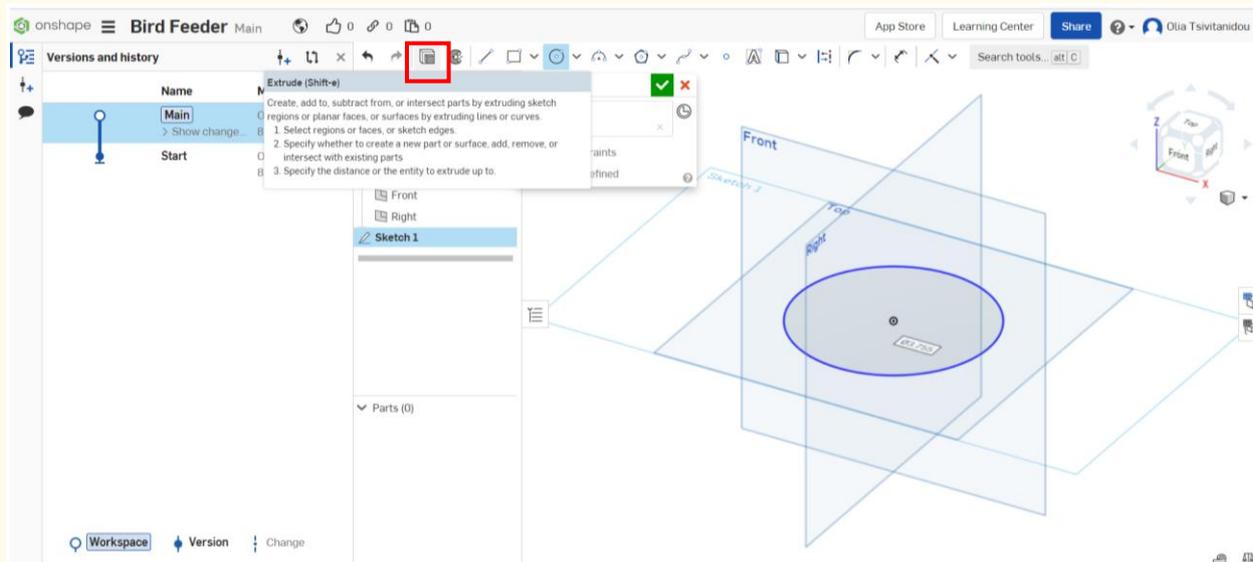
Step 6

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

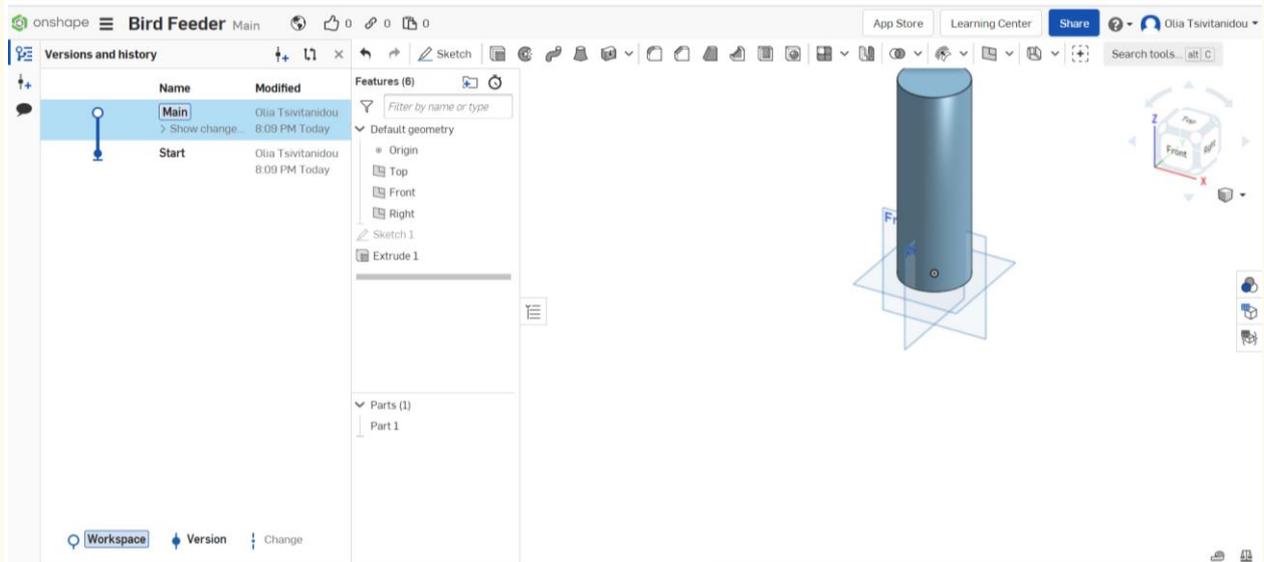


Step 7

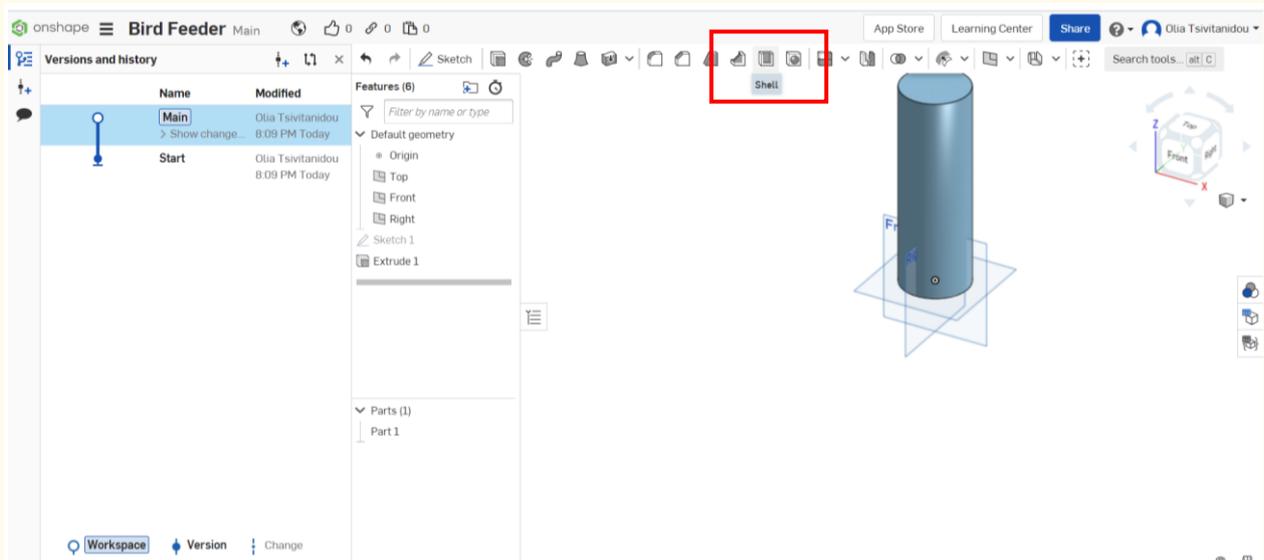
Select "Extrude."



Step 8 Extrude that circle.

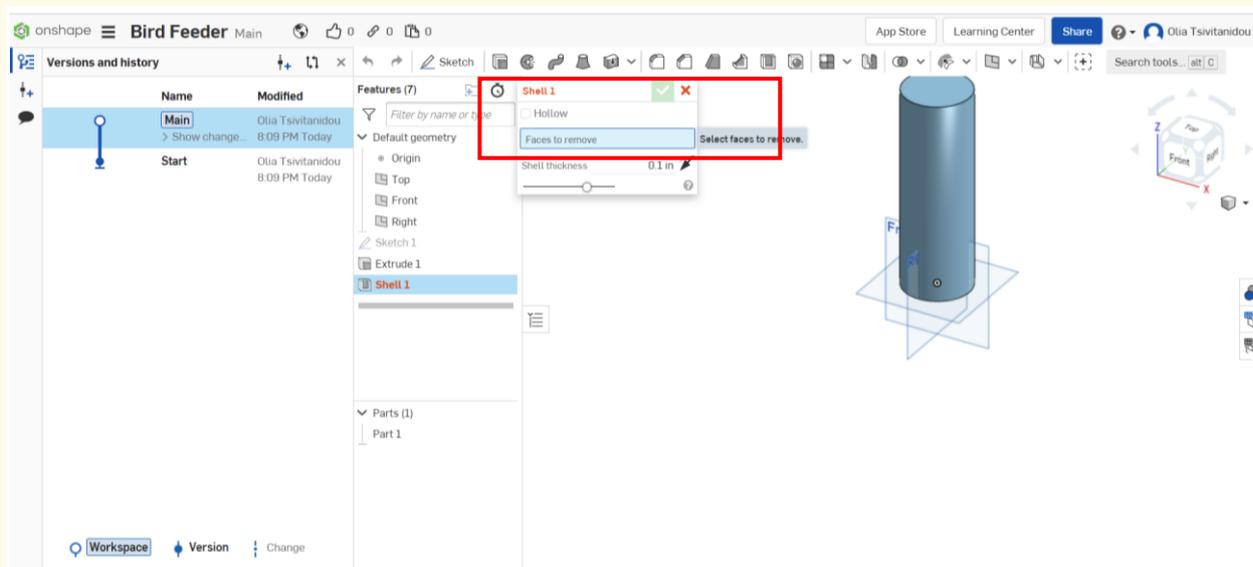


Step 9 Select "Shell."



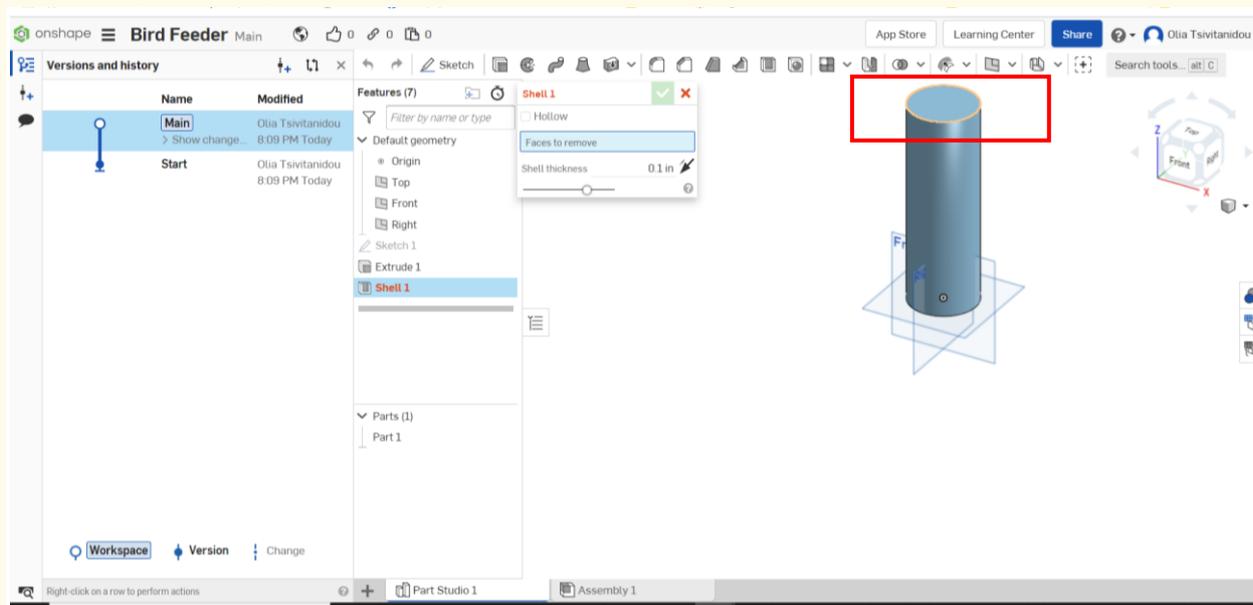
Step 10

Select "Face" to remove.



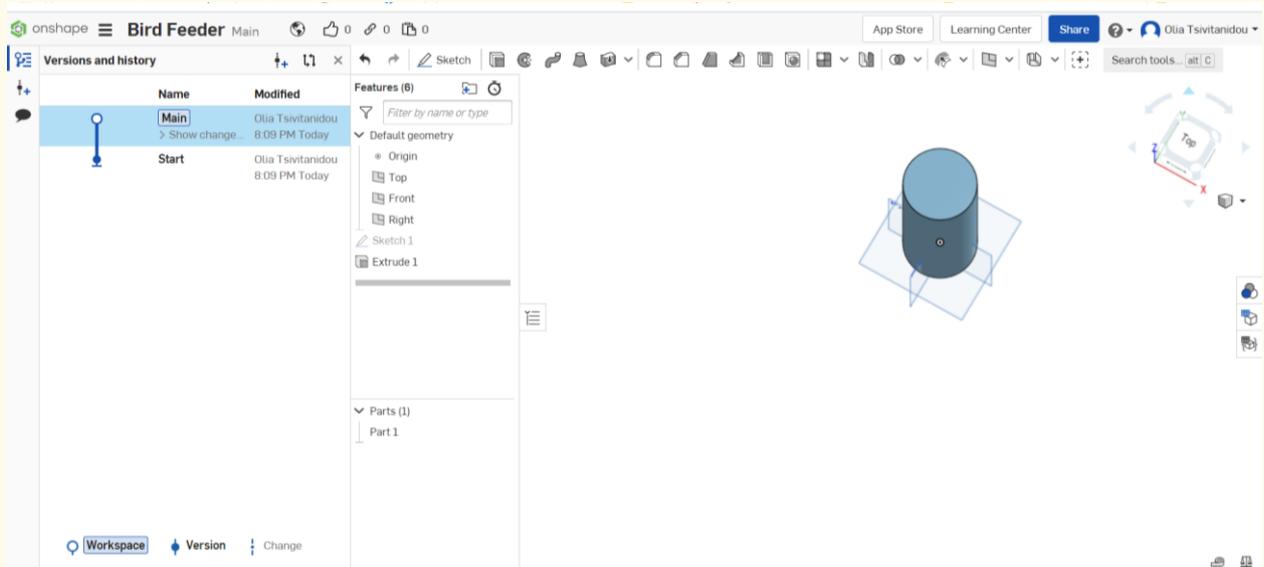
Step 11

Click on the top cylinder.



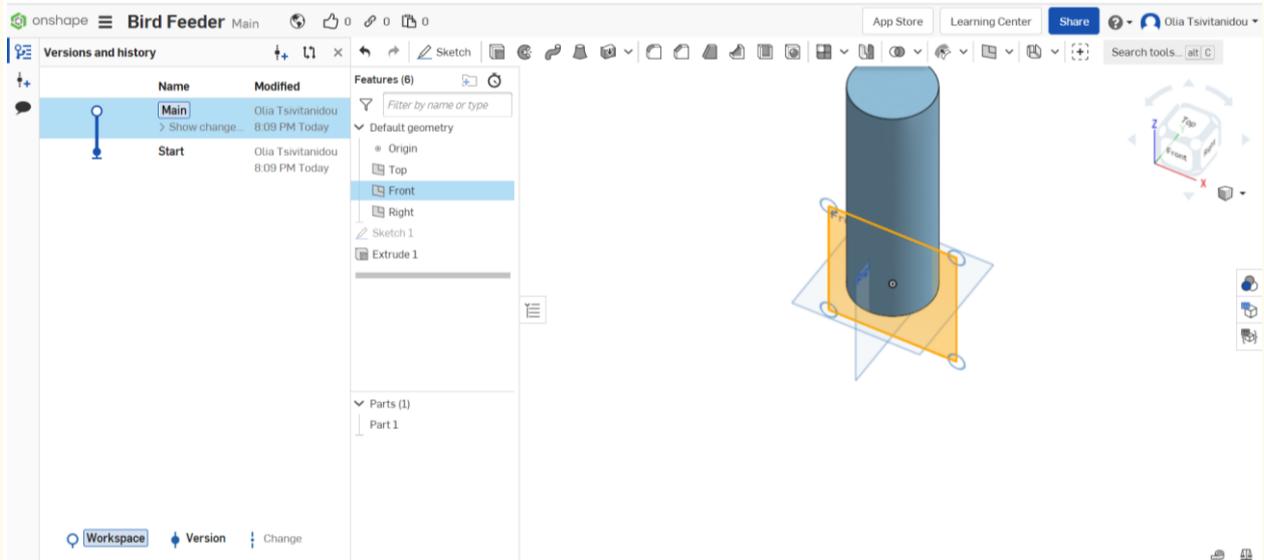
Step 12

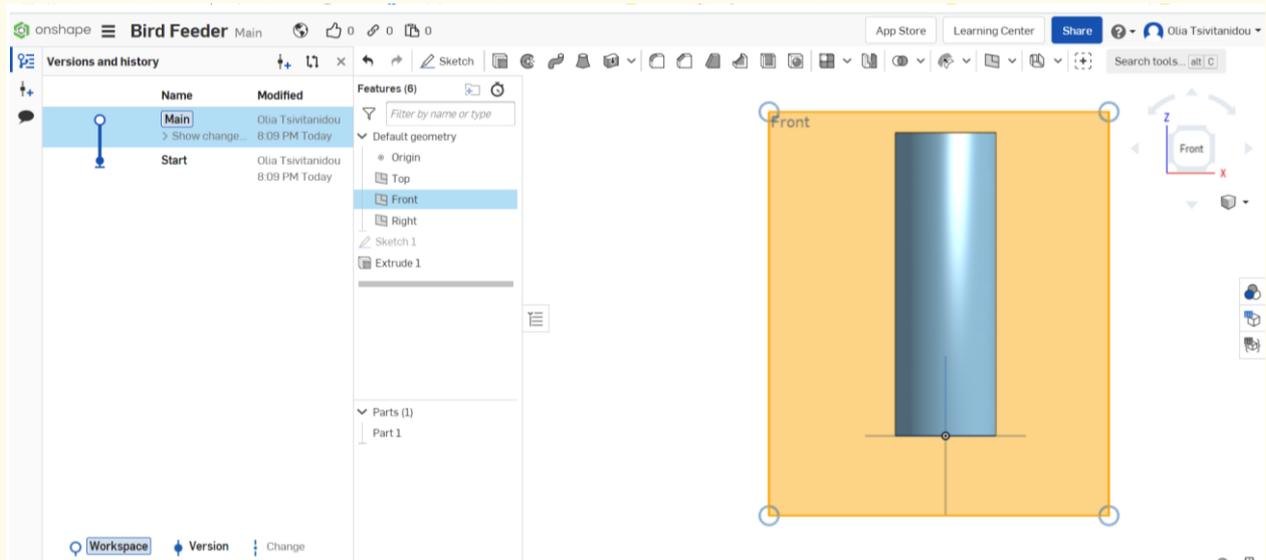
The top face has been removed. This is the result.



Step 13

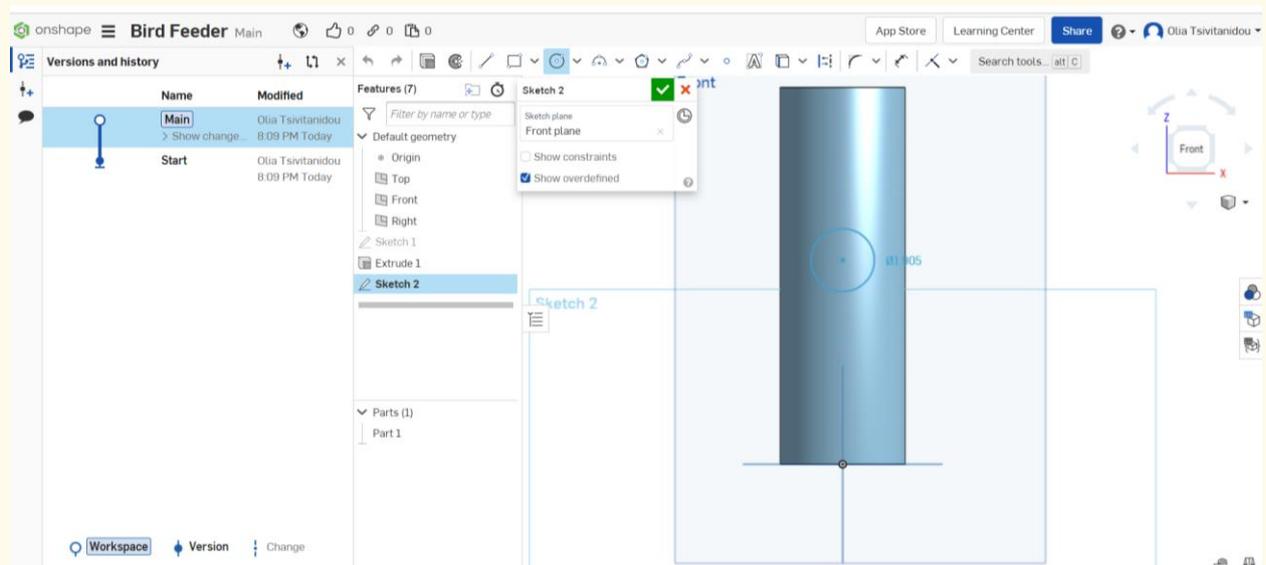
Draw the large holes on the cylinder surface.
Select the "Front Plane" and then "sketch."

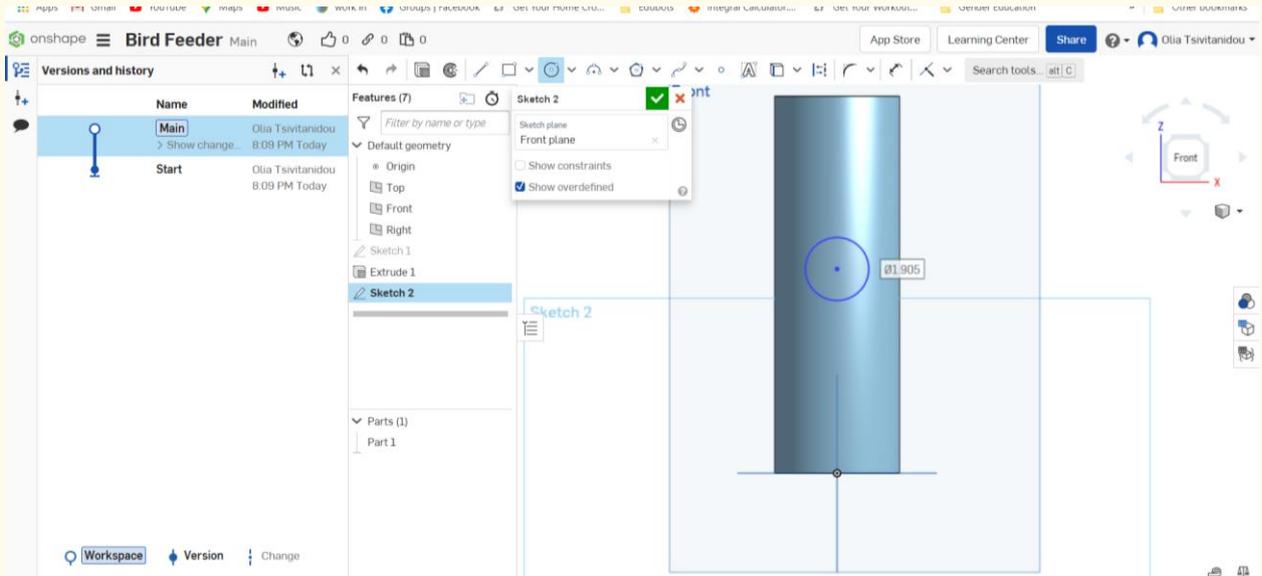




Step 14

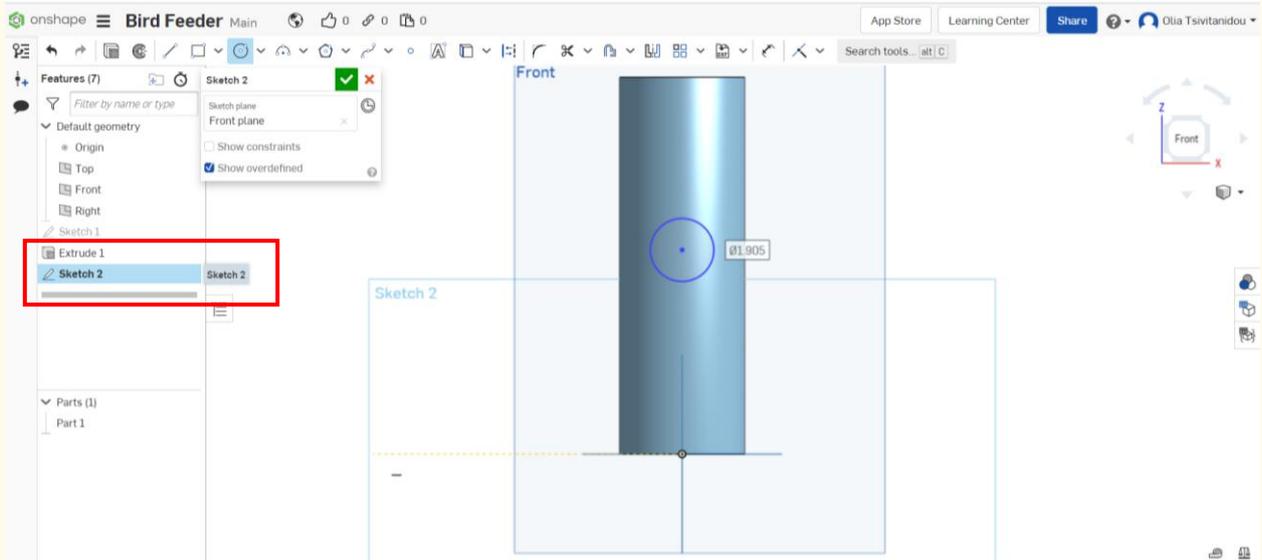
For creating the first hole, move the mouse down to the origin so that it's tethered or linked together, and it is going to be somewhere in the middle. Click "circle" and draw a circle.





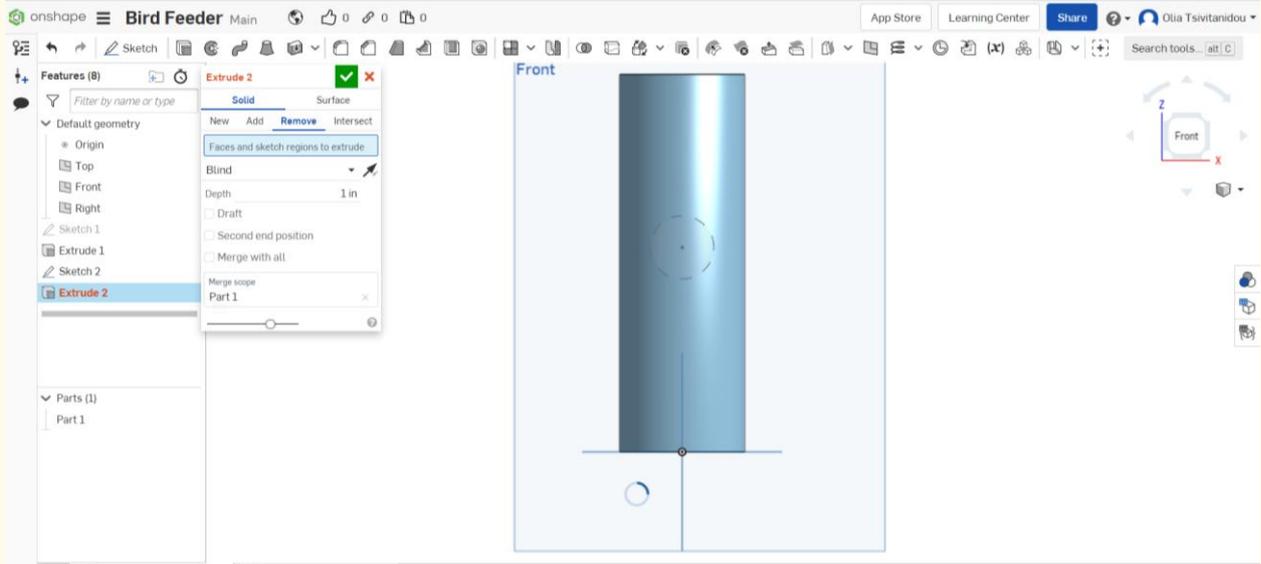
Step 15

Select "Sketch 2", and pick it from there.



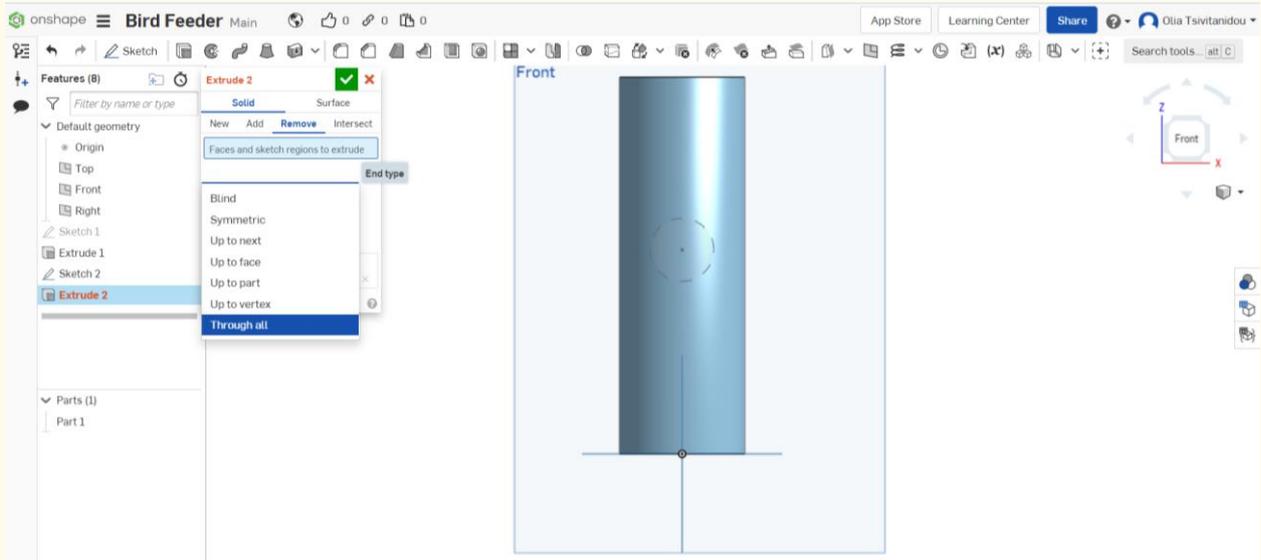
Step 16

As illustrated below, we want this to be removed, so click on "Remove."



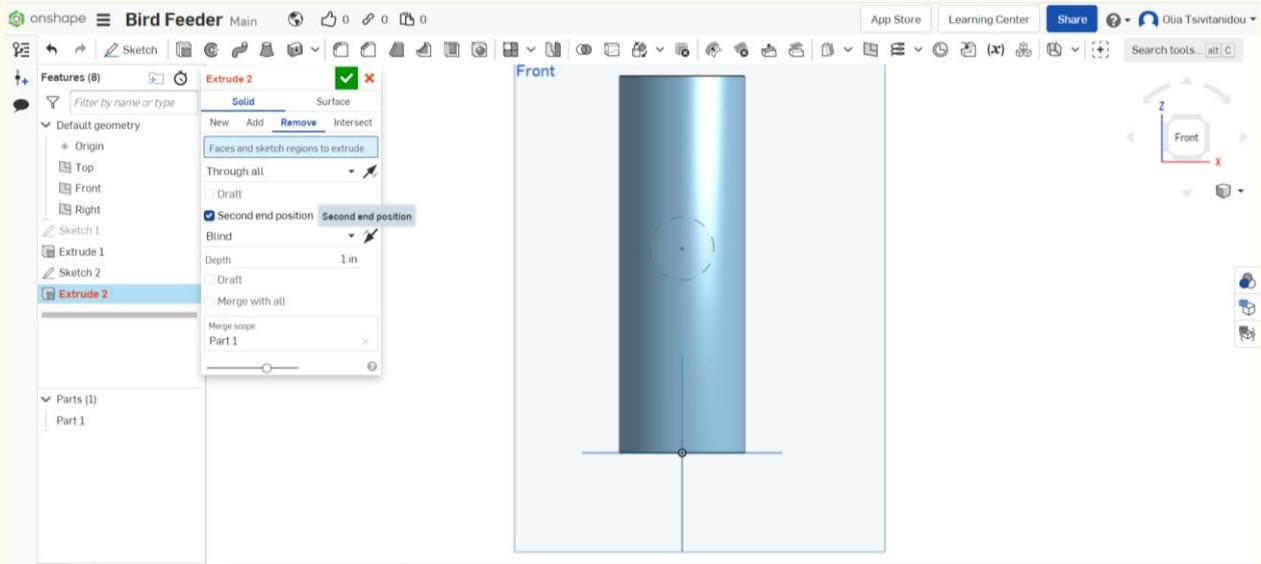
Step 17

Select "through all."



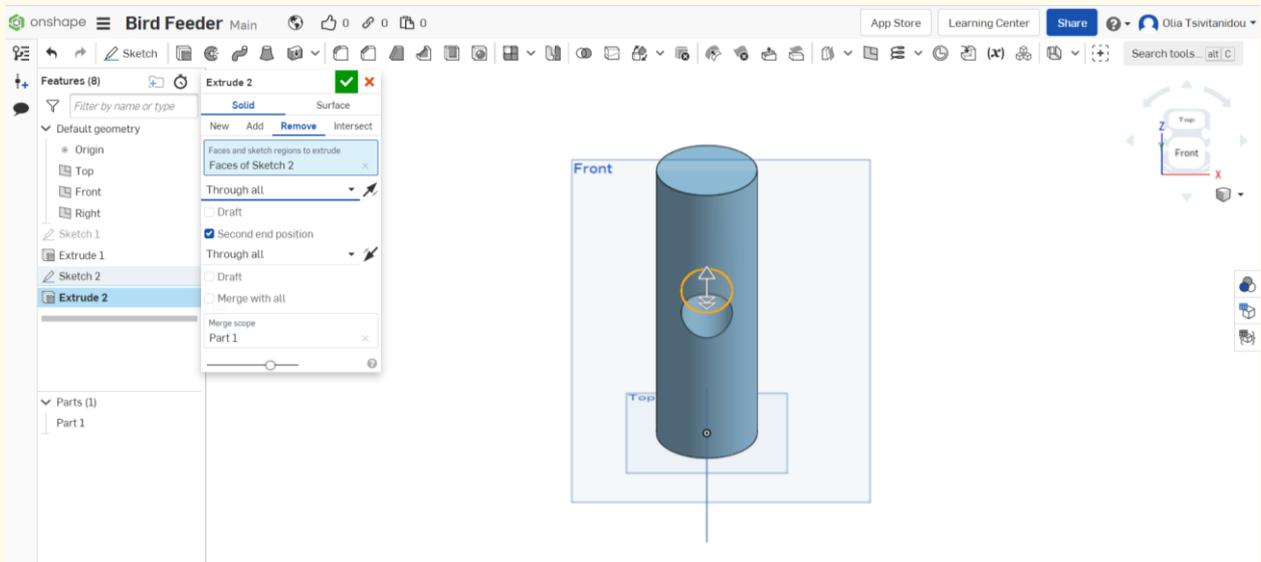
Step 18

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



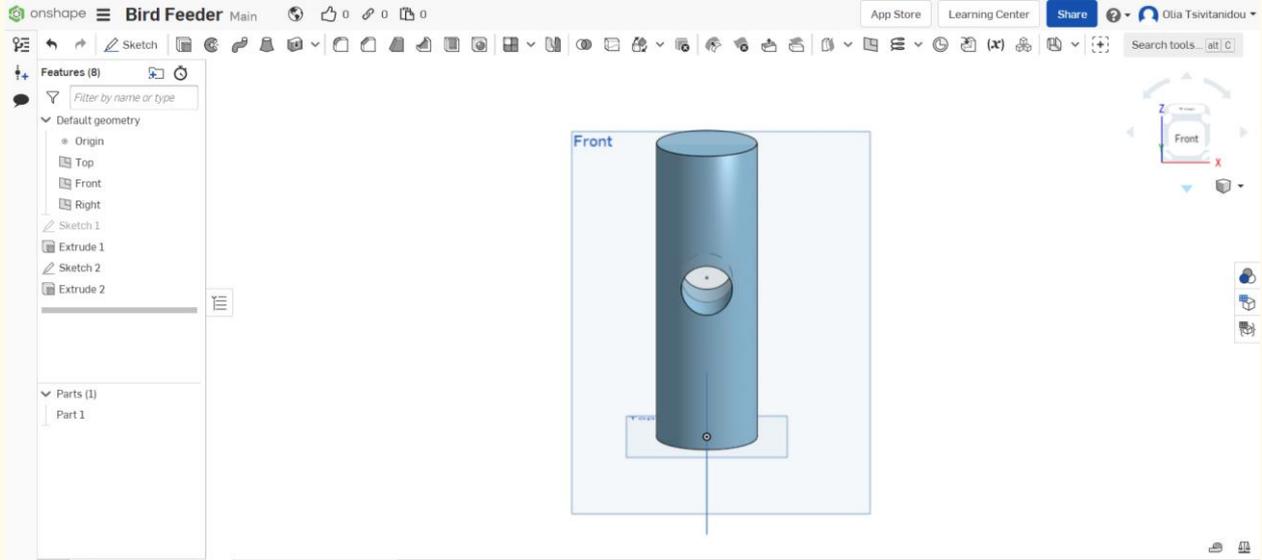
Step 19

Select "Second end position" and make that a "Through all."



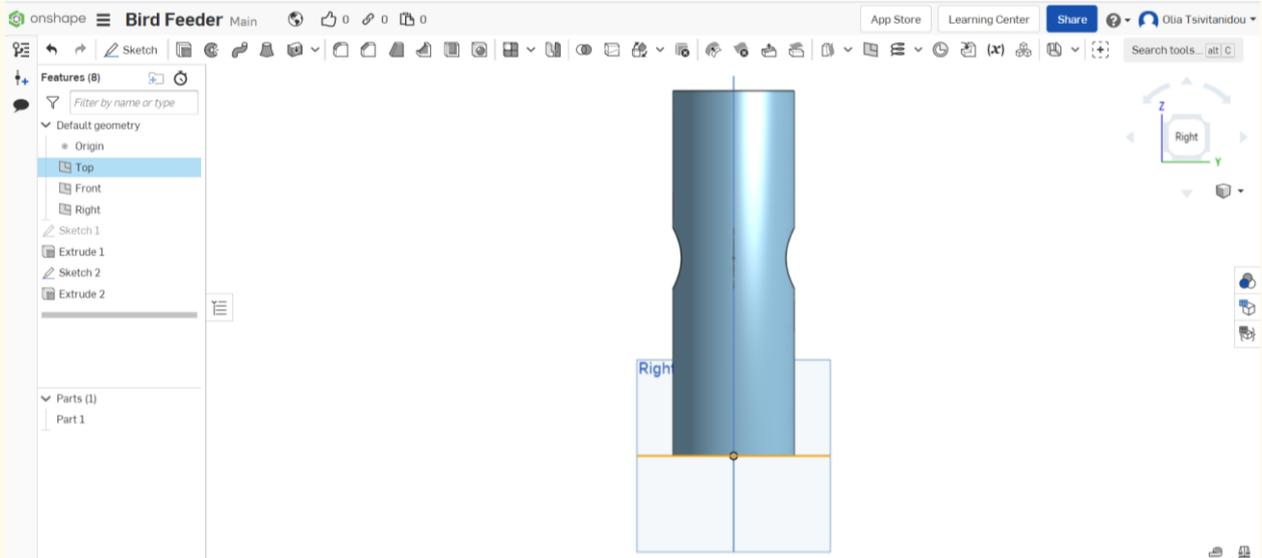
Step 20

Your design should look like this:



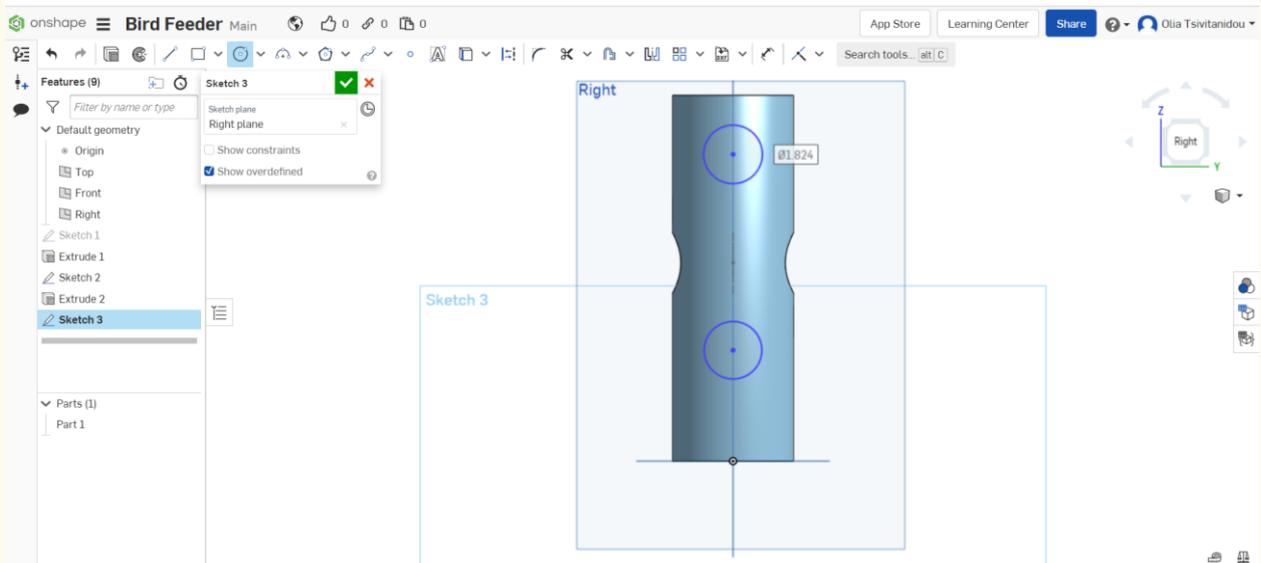
Step 21

Create two more holes. Create a new Sketch on the right side face. Click on the right side plane, then "Sketch" and "Circle."



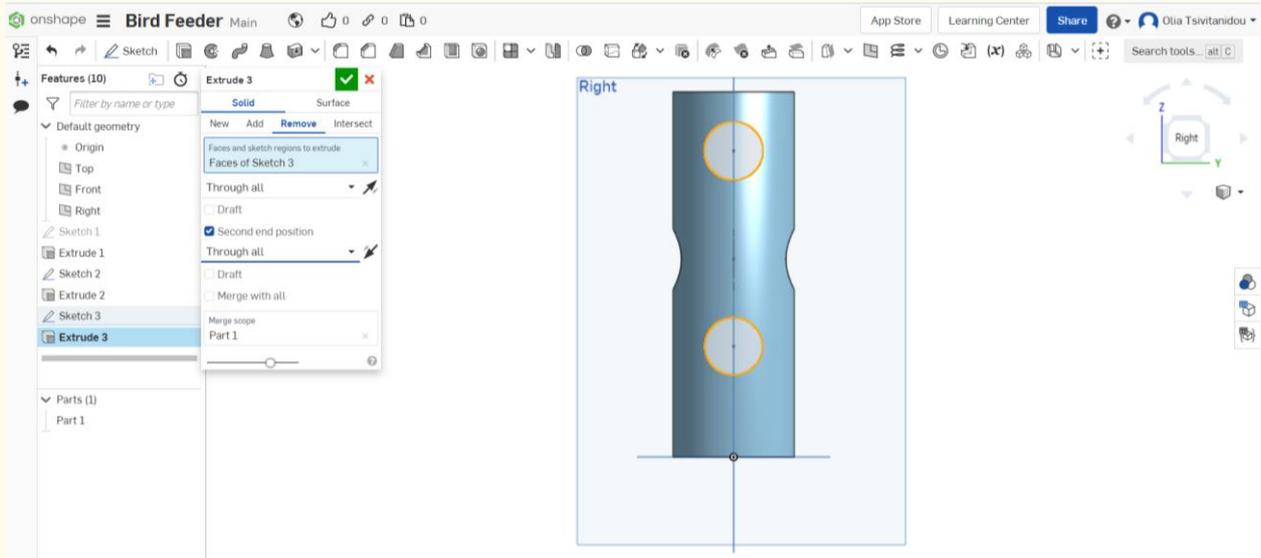
Step 22

Draw two new circle sketches in the same way as done previously.



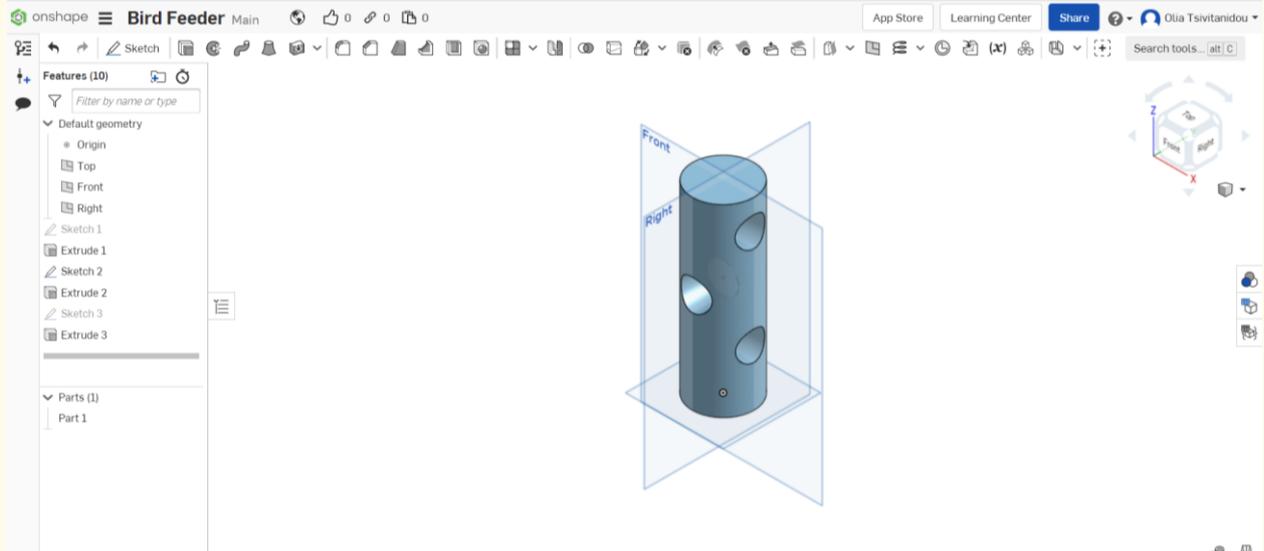
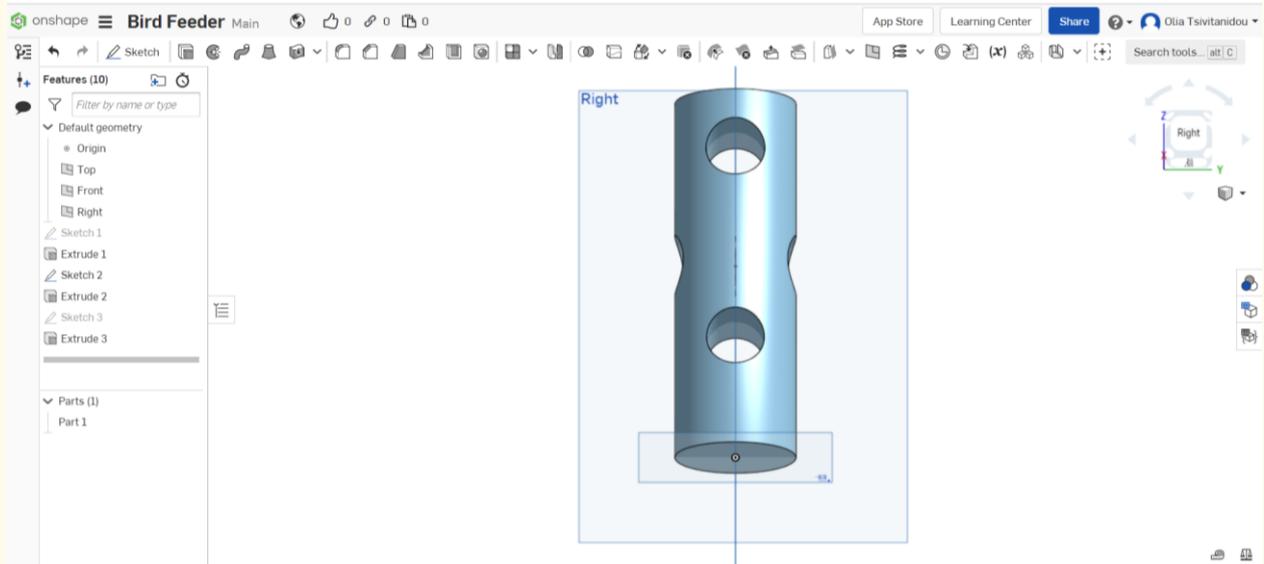
Step 23

Extrude by clicking on "Sketch 3", then "Extrude," "Remove," "Through all," "Second end position," and again, "Through all."



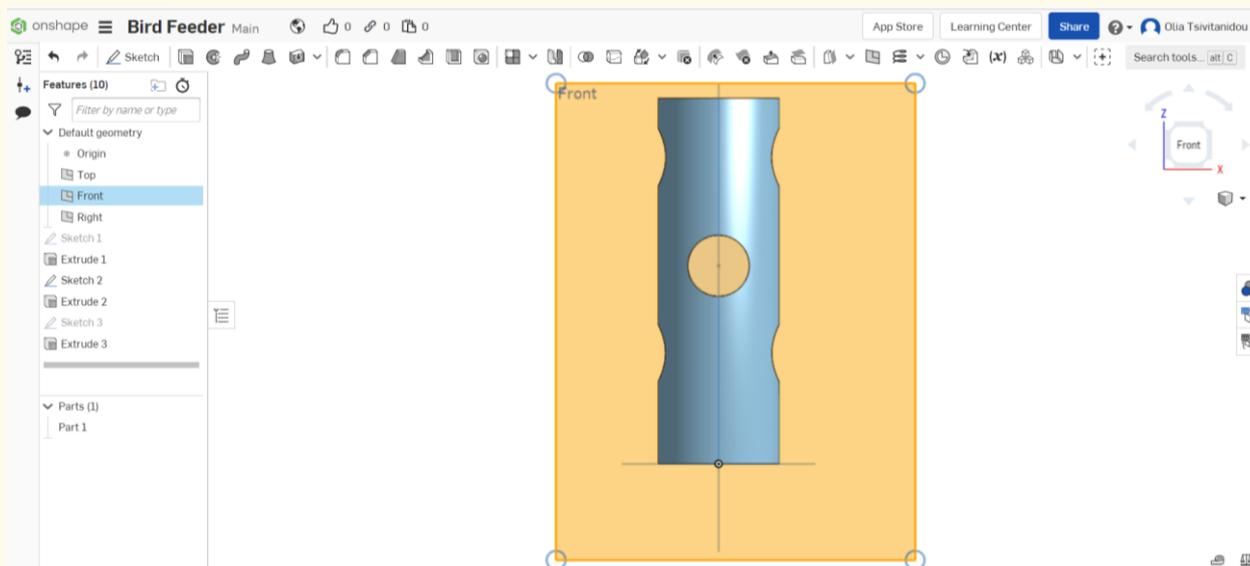
Step 24

Your cylinder should look like this:



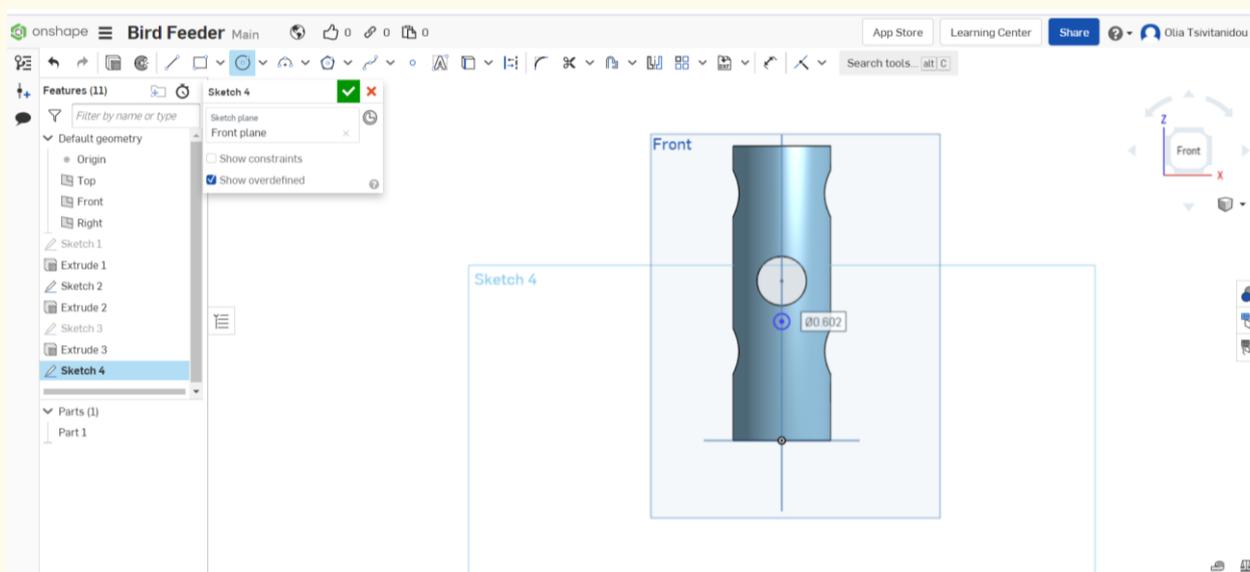
Step 25

Create a sketch on the "Front" and a small perch there.



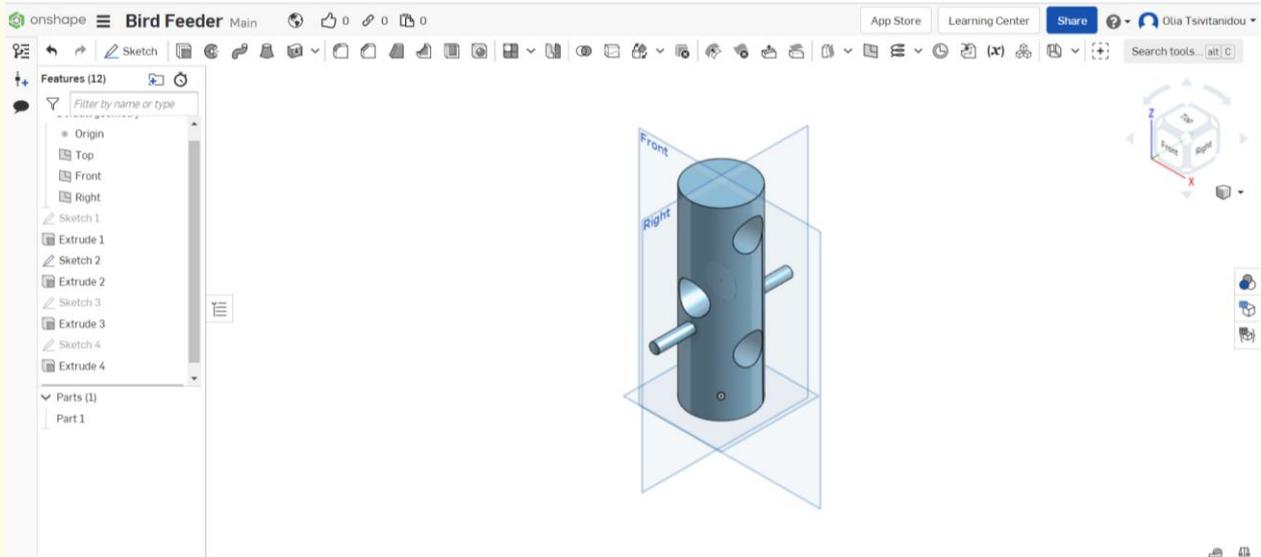
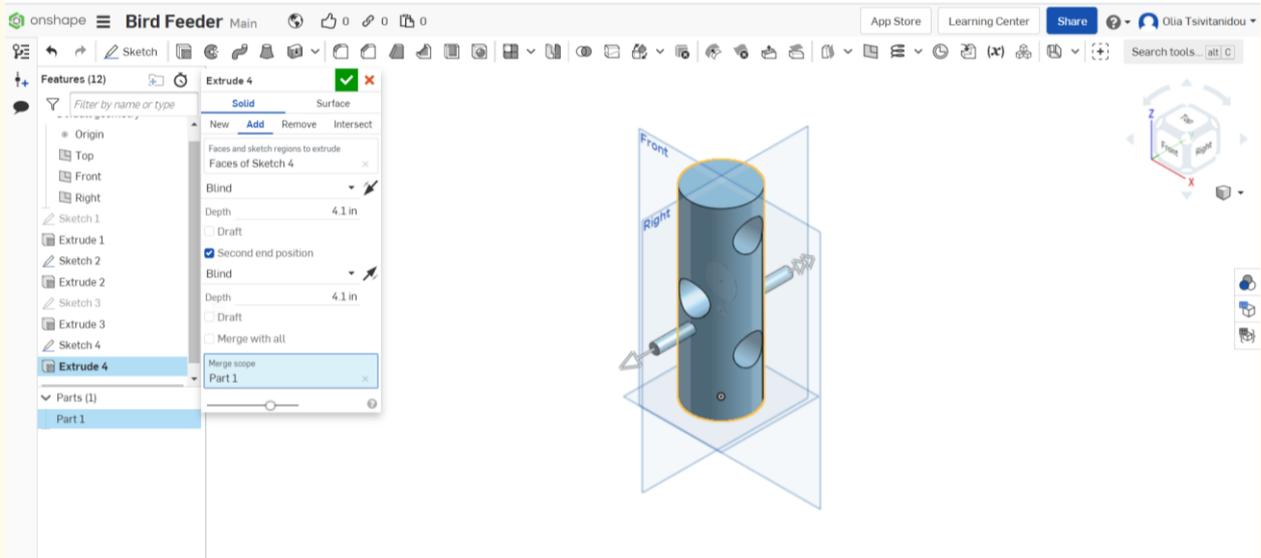
Step 26

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



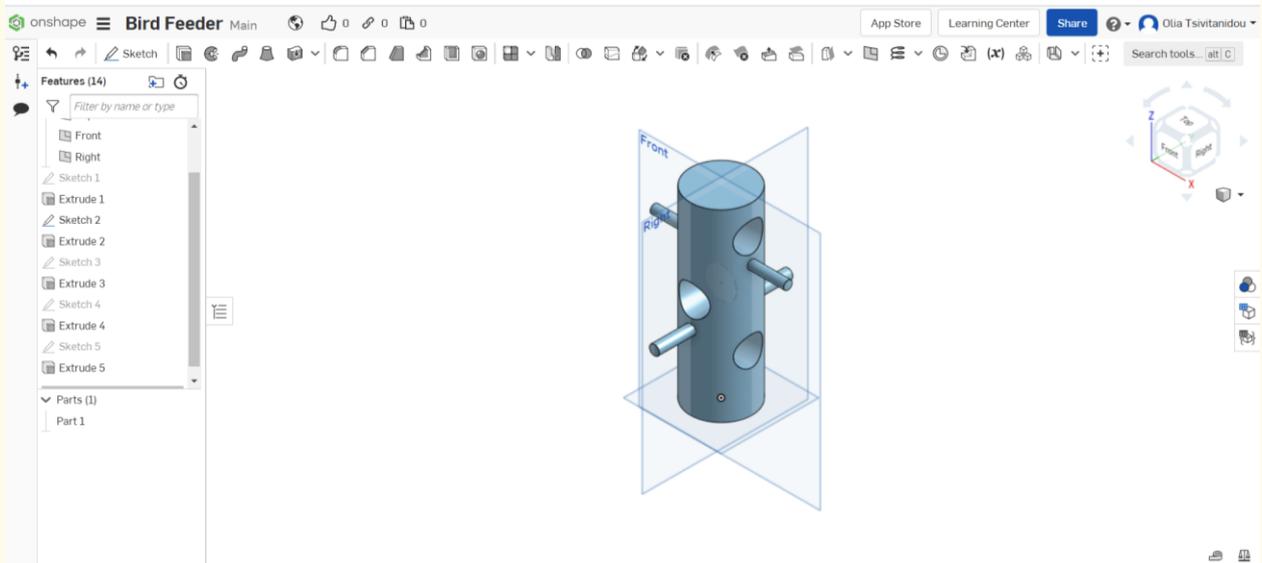
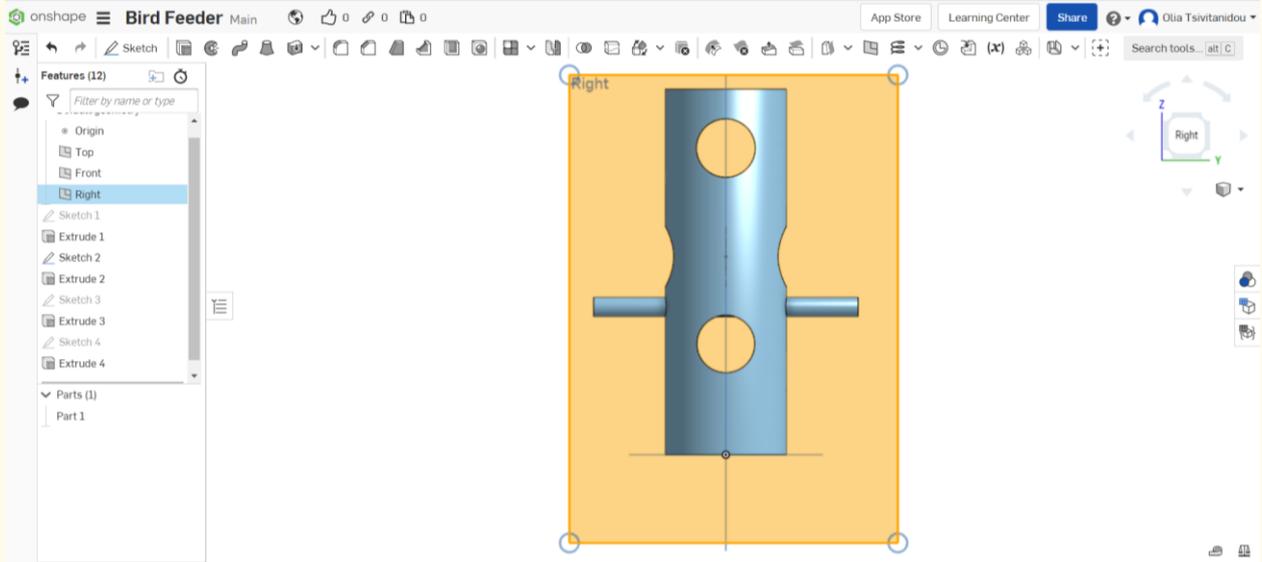
Step 27

Select "Extrude" and second end position. Keep the same distance (Depth) for both perches.



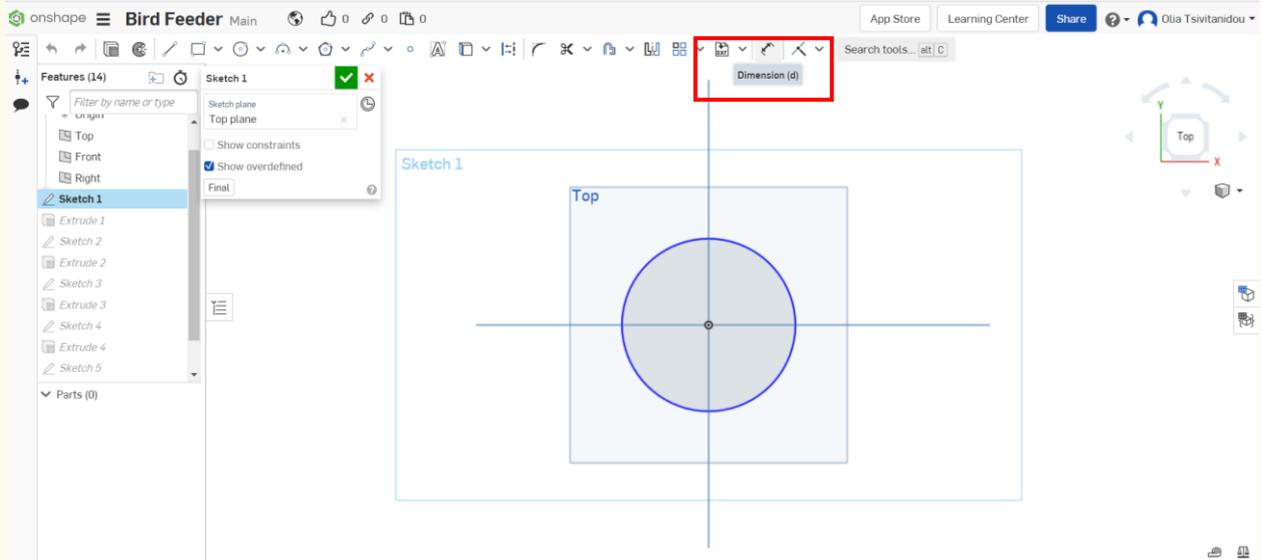
Step 28

Repeat steps 26 and 27 for creating two more perches. Click on the right plane and proceed in the same way.



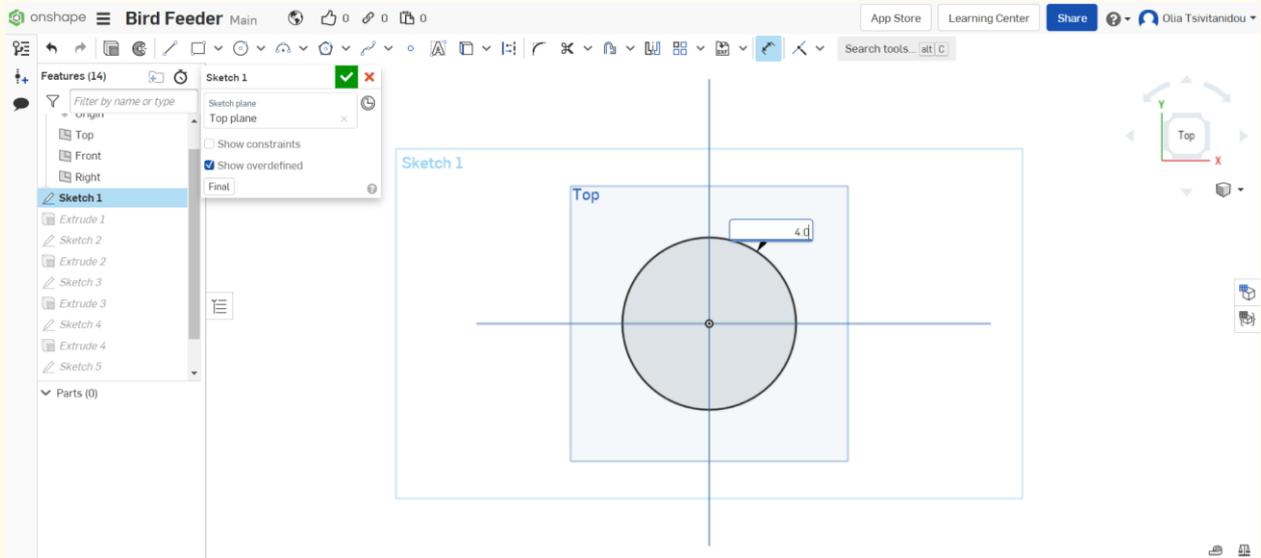
Step 29

As a final step, insert the dimensions. Start with the diameter size and set it to 4 inches. Click on "sketch 1" and then "Dimensions."



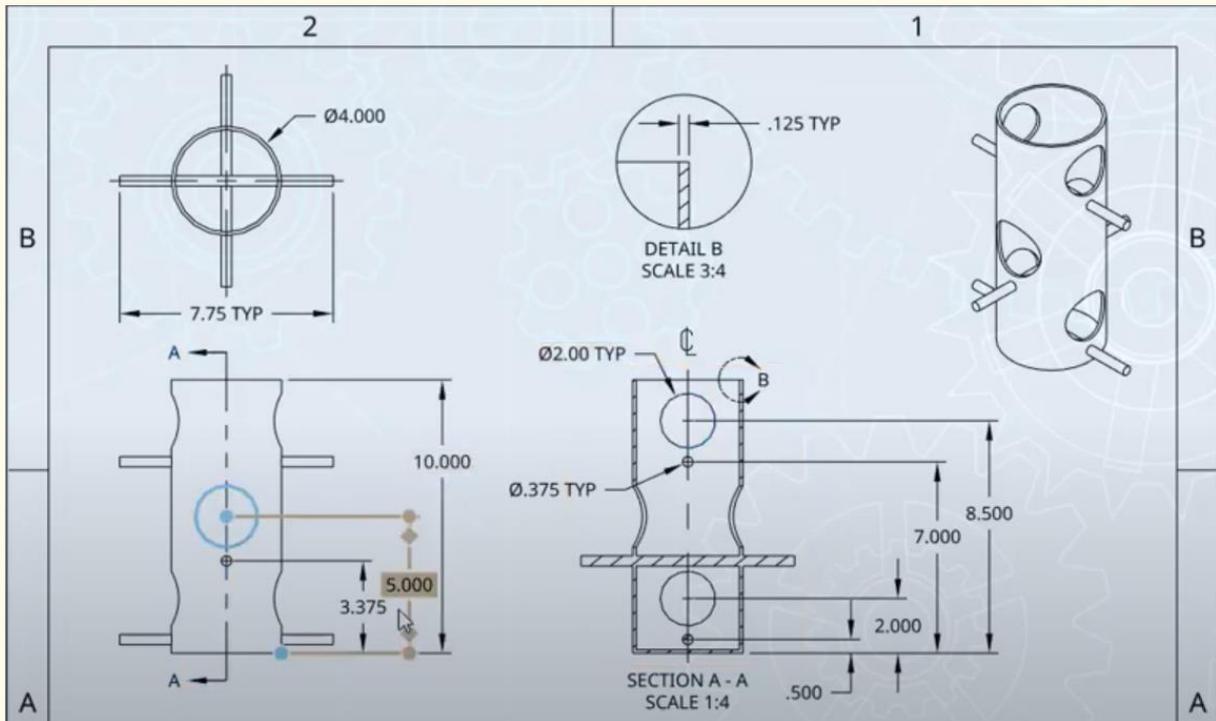
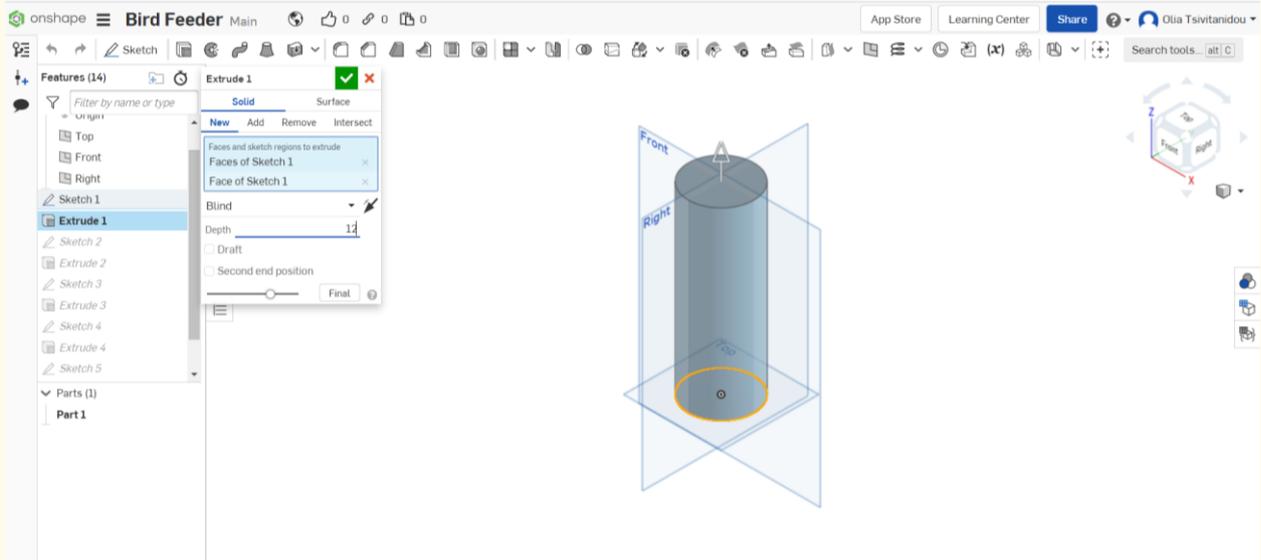
Step 30

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



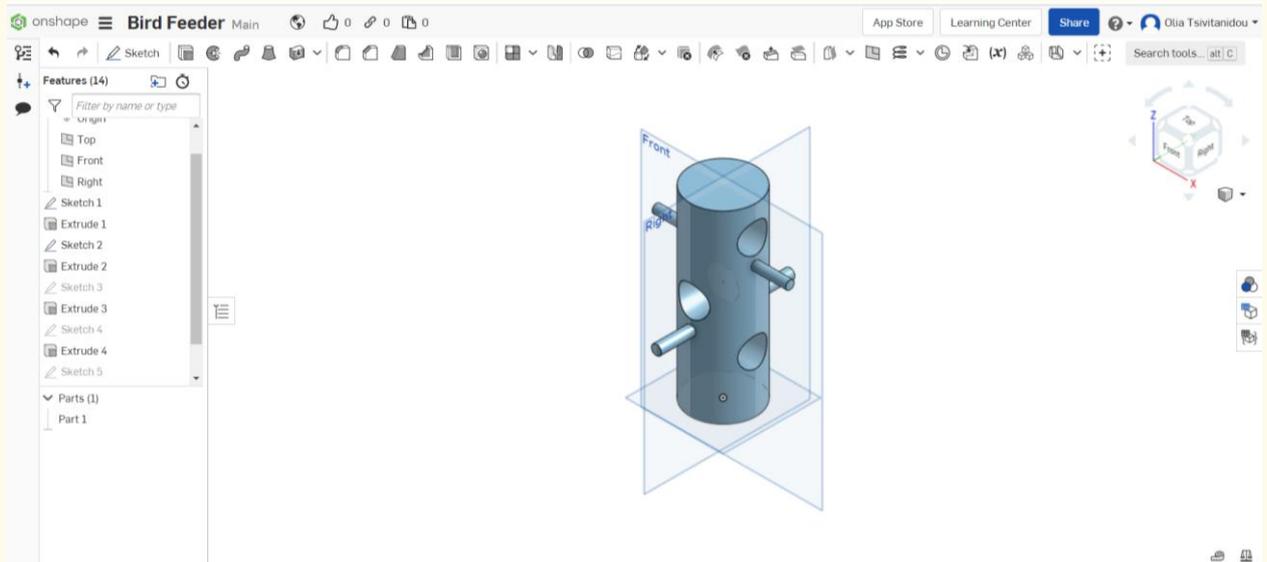
Step 31

Set realistic sizes. Repeat step 30 for resizing all sketches to give the realistic dimensions you wish (see figure below). In the same rationale, we can correct extrusion. For instance, select extrusion 1 from the features list and set the "Depth" to 12 inches.



Step 32

Your 3D design is ready. Enjoy the 3D printing procedure now!



PROJECT: 3D DRAWING OF A SHOVEL

- STEM field: Science, technology, and electronics.
- Indicative calendar: Any time of the year.
- Activity duration: 3 hours
- Type of activity: 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 competences:
 - How to do a shovel on Onshape.
- Required material and resources:
 - 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:

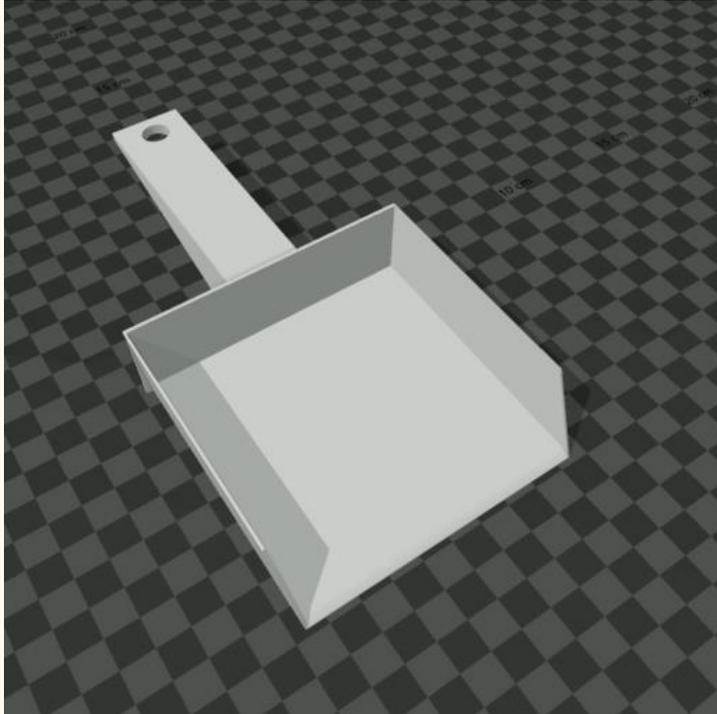
PRELIMINARY STEPS:

It is important to have an idea of what will be the final object and make sure measures are relevant you need to do some small researches.



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

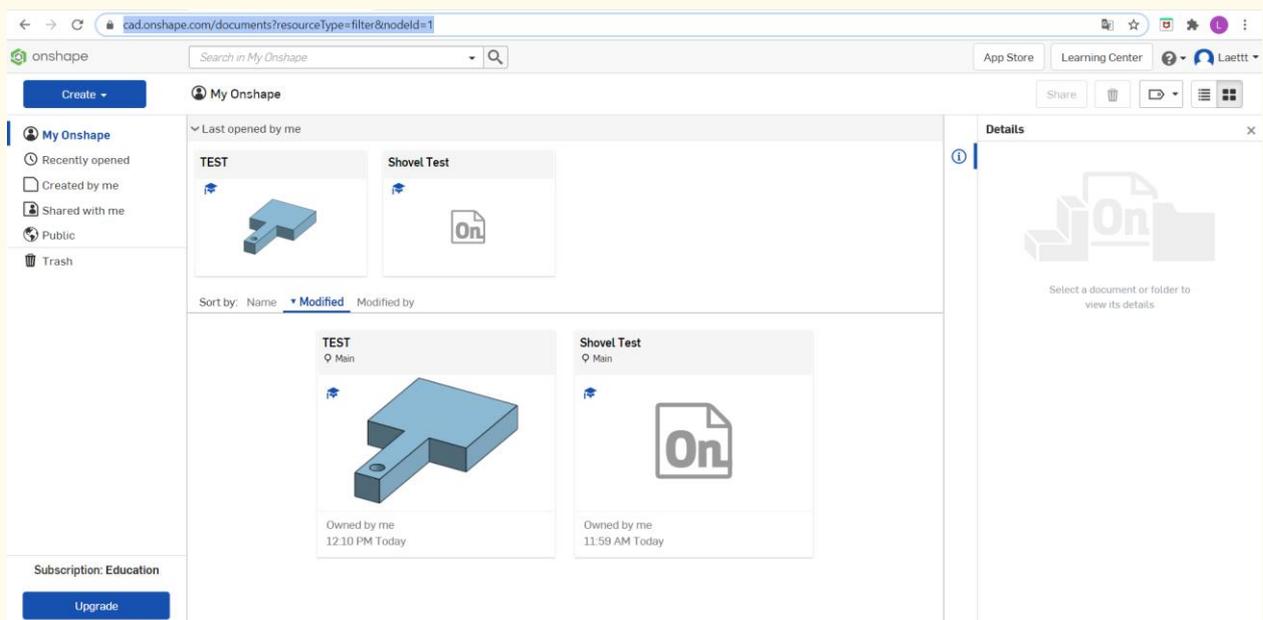
For inspiration, we took this example: <https://cults3d.com/fr/mod%C3%A8le-3d/divers/pala-manual> and selected/defined our own measurement.



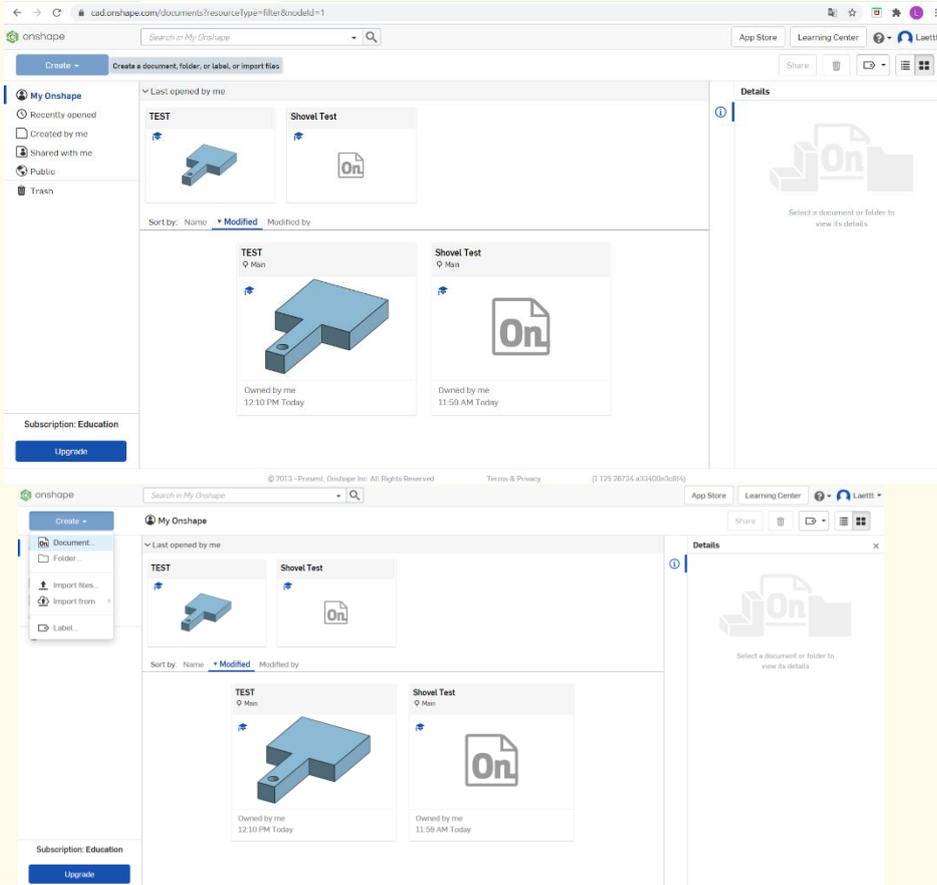
Step 1

Open Onshape.

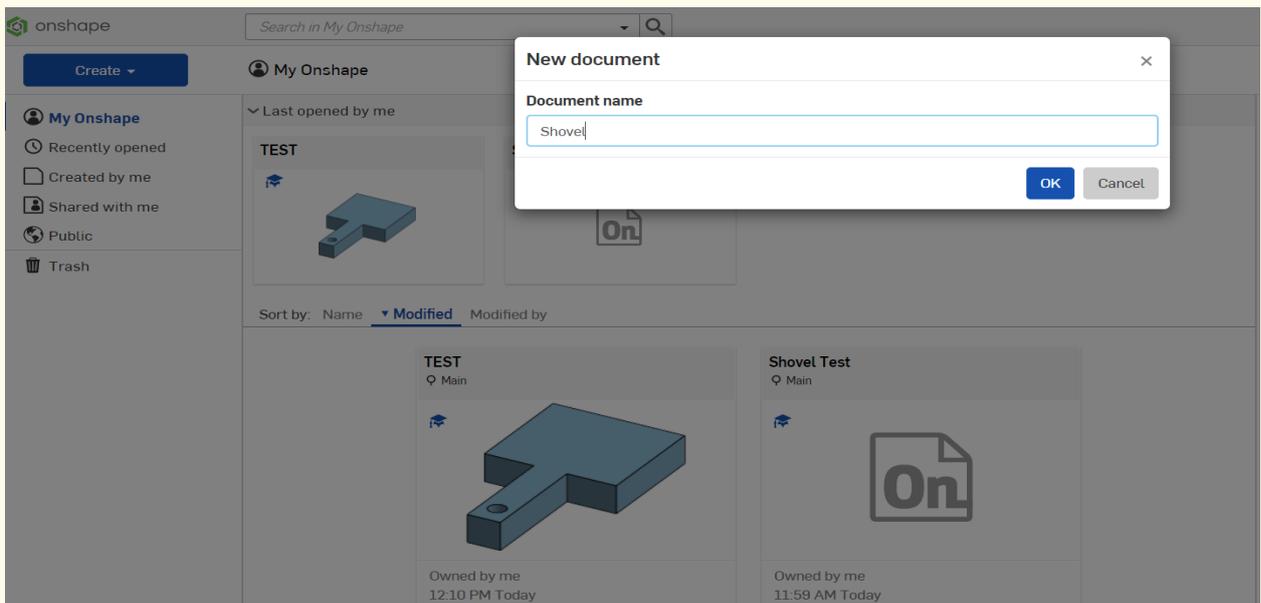
<https://cad.onshape.com/documents?resourceType=filter&nodel=1>



Step 2 Create a document.

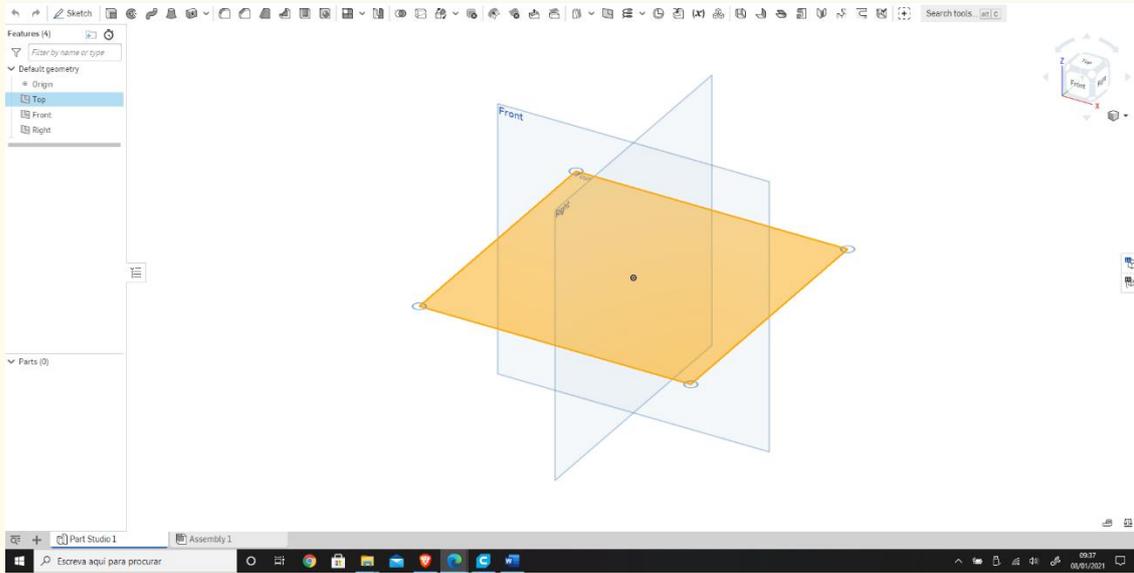


Step 3 Give a name to your document, such as Shovel.



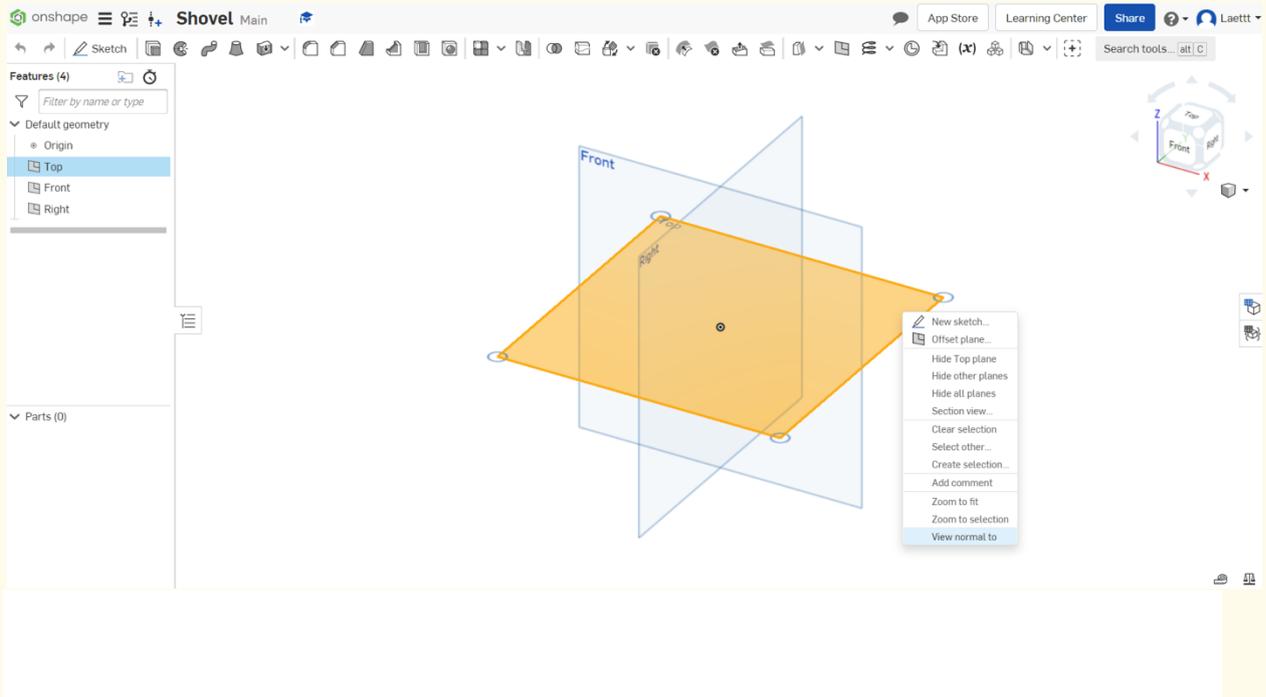
Step 4

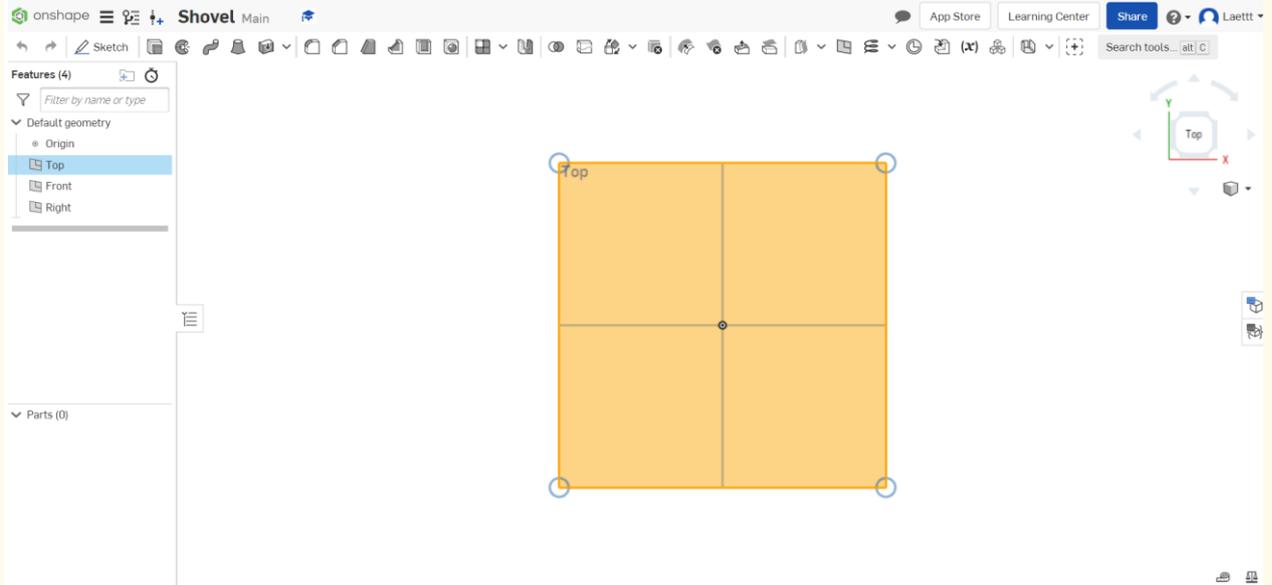
Select the plane (top) to start drawing.



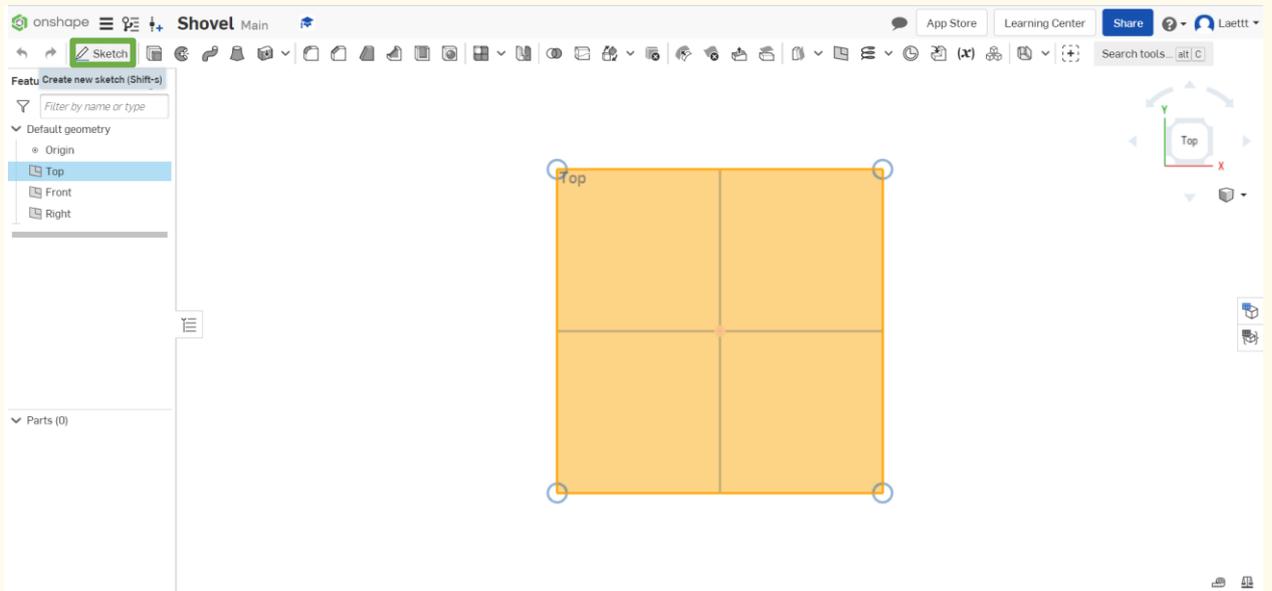
Step 5

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



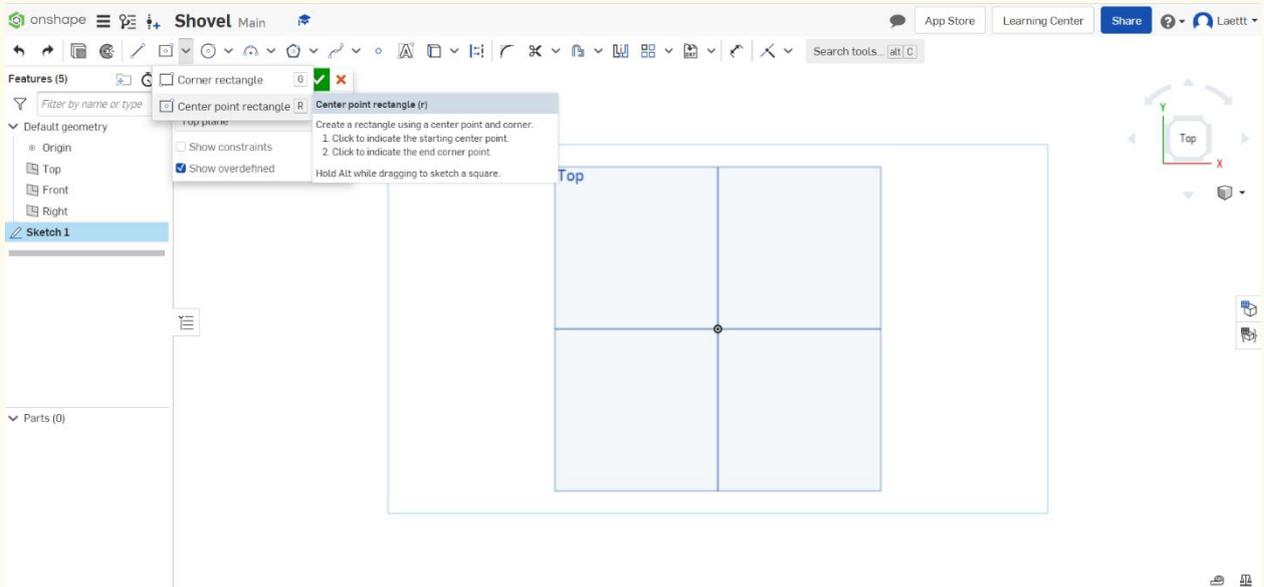


Step 6 Click on "Sketch."



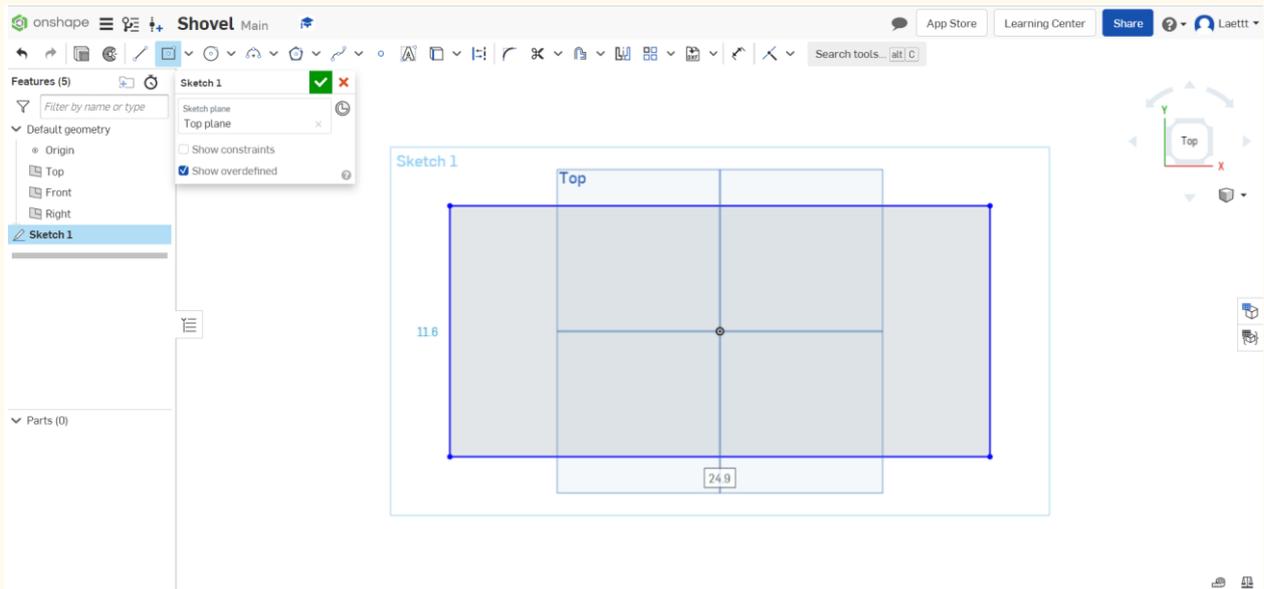
Step 7

Select the center point of the rectangle to draw.



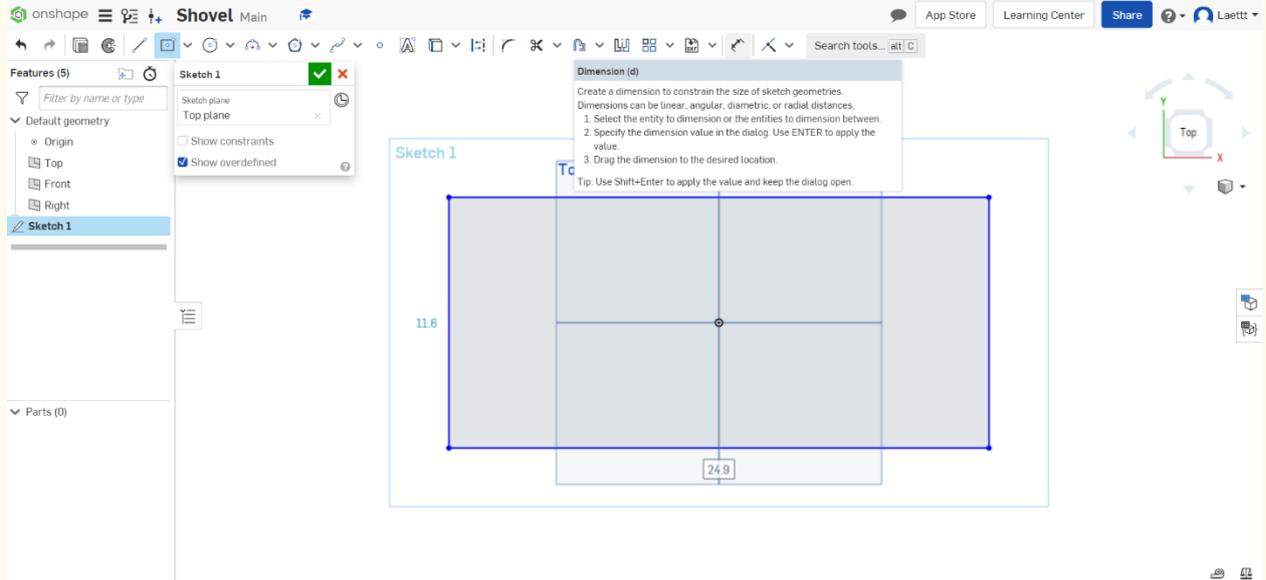
Step 8

Draw the rectangle.



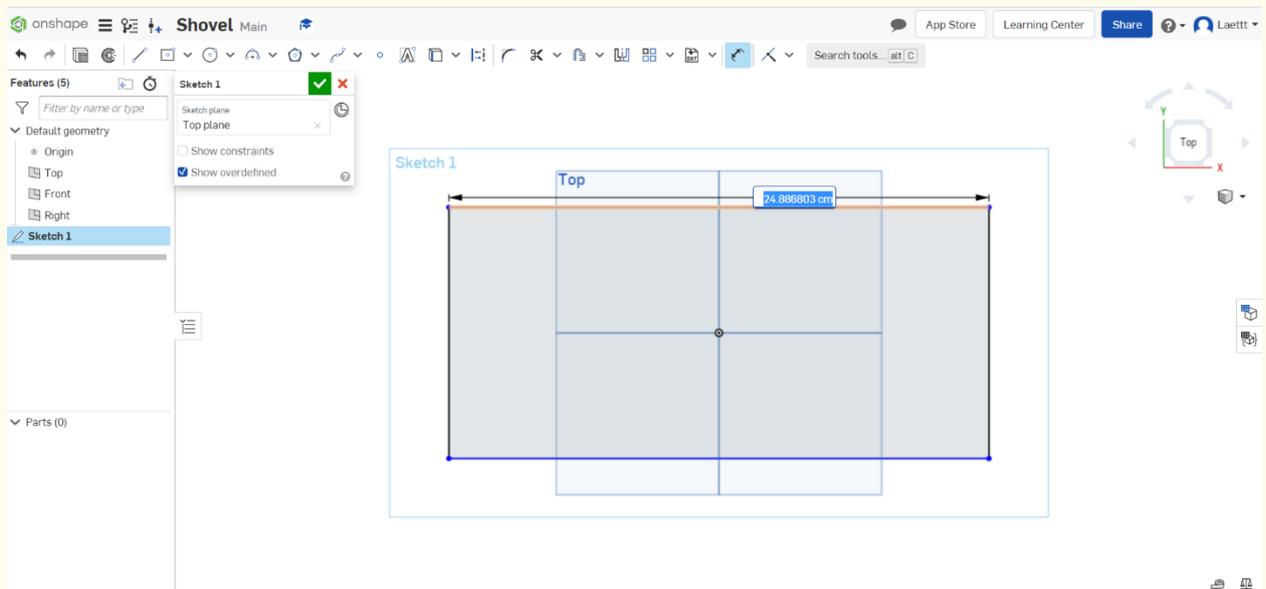
Step 9

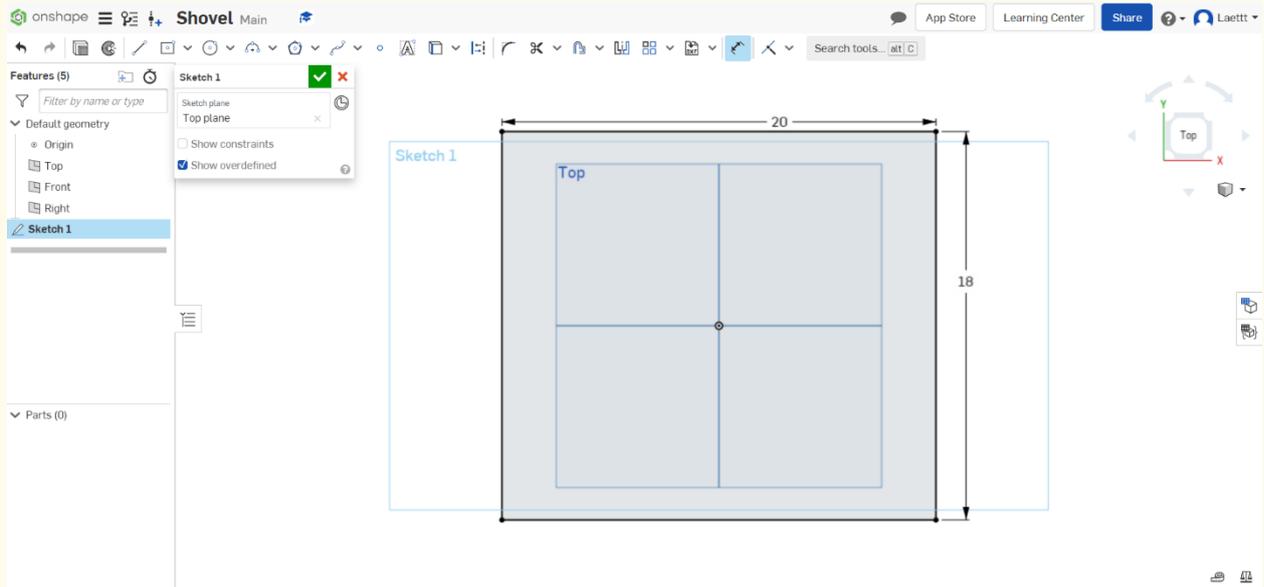
Select "Dimension" to insert the measures.



Step 10

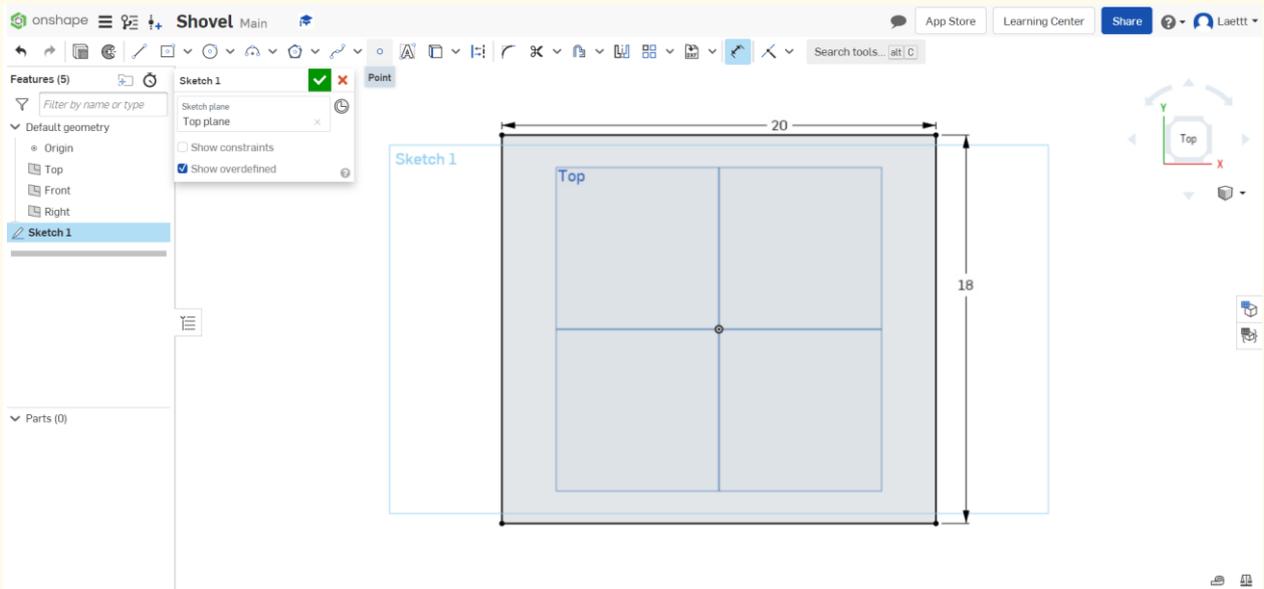
Select the line (first left then top) and give the dimensions as shown in the 2nd figure.

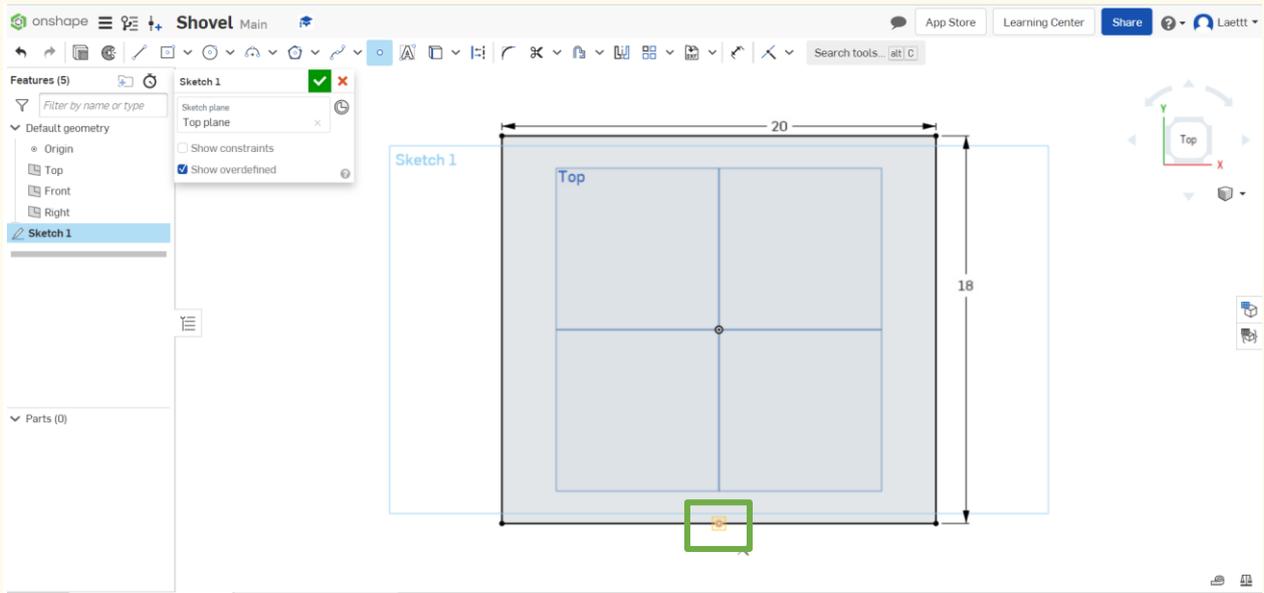




Step 11

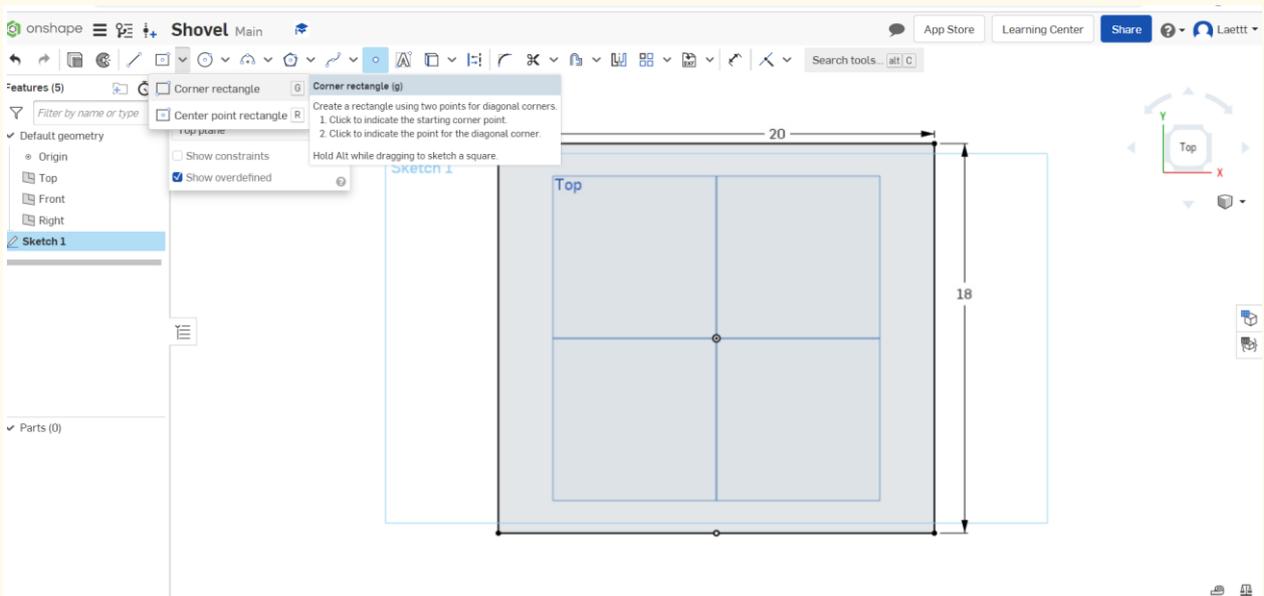
Select the function point and place a point on the rectangle's lower part, as shown in the 2nd figure.

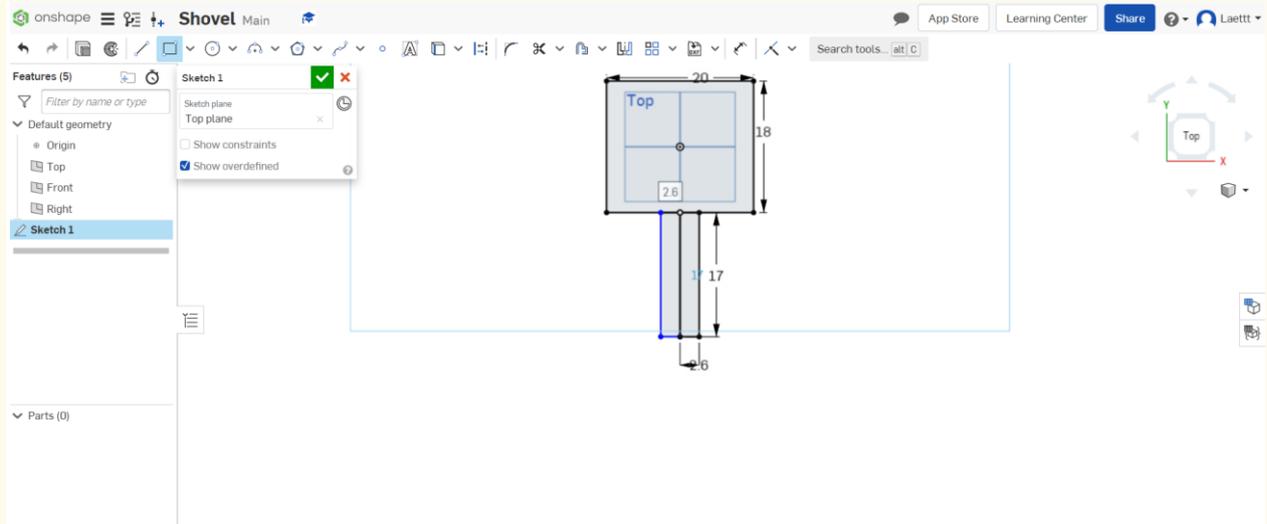




Step 12

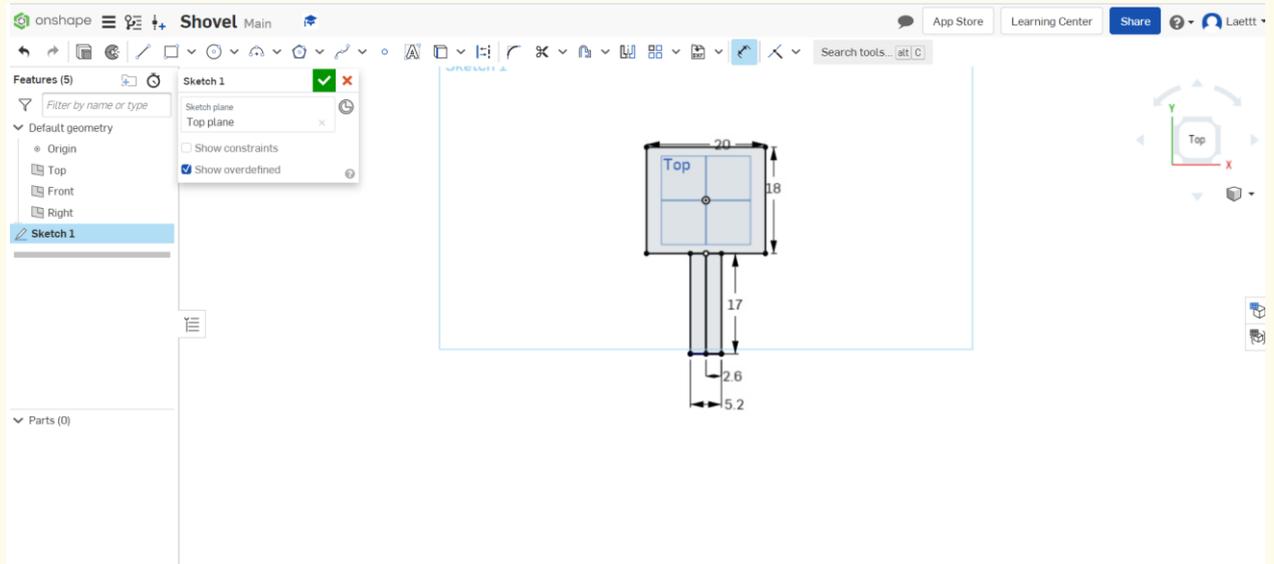
Draw 2 similar rectangles as shown (starting at the previously placed point).





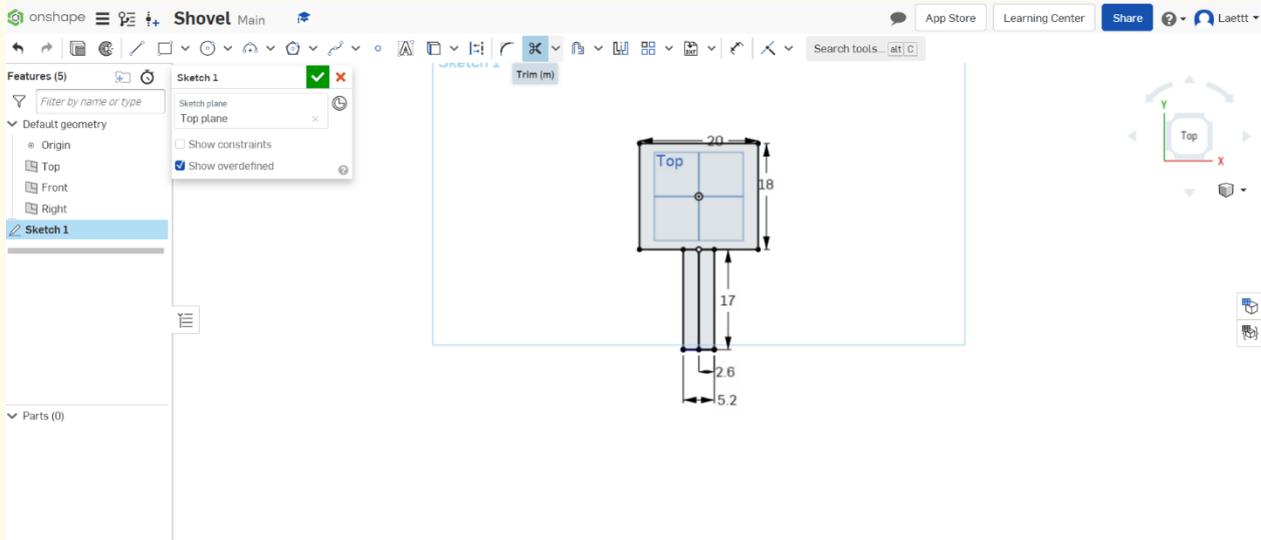
Step 13

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



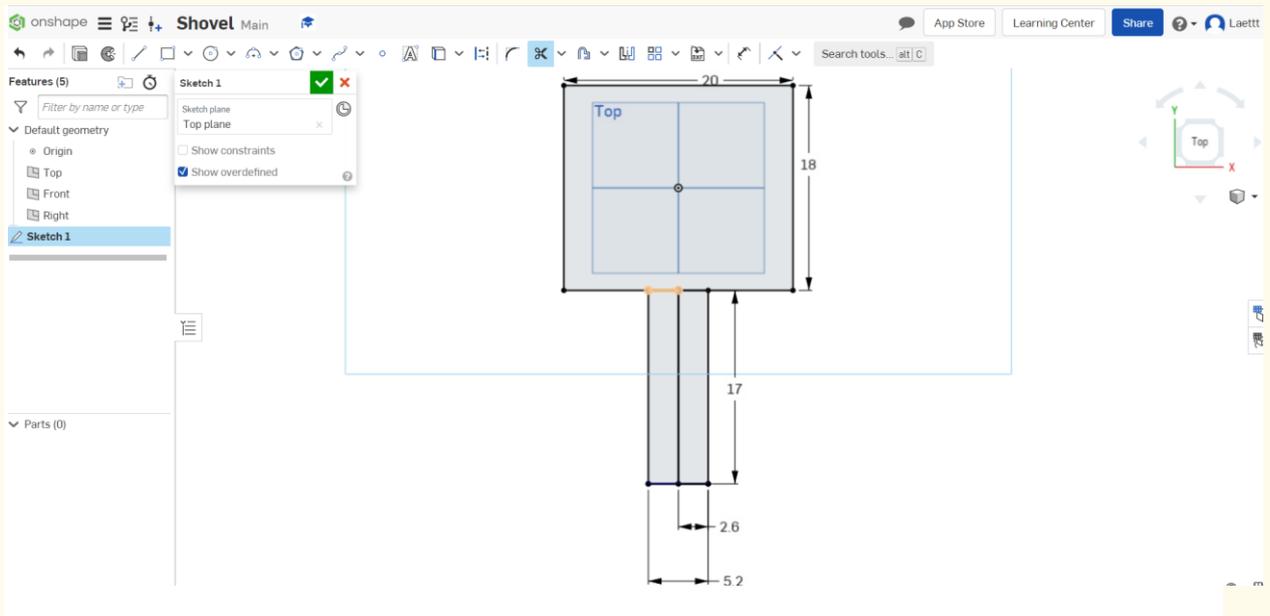
Step 14

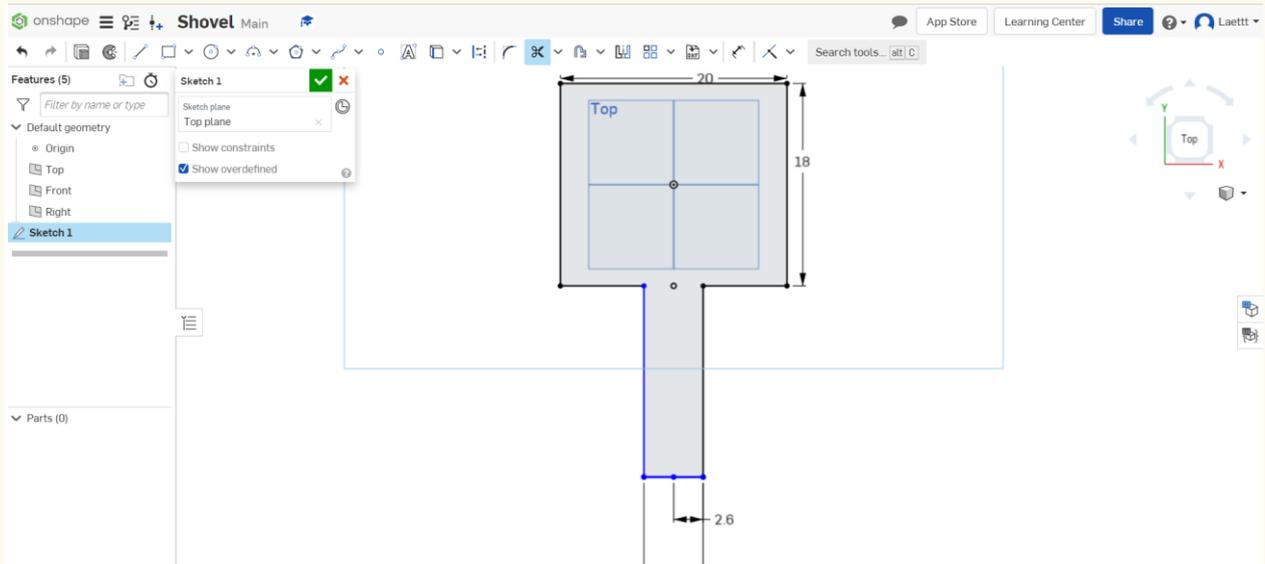
Select "Trim."



Step 15

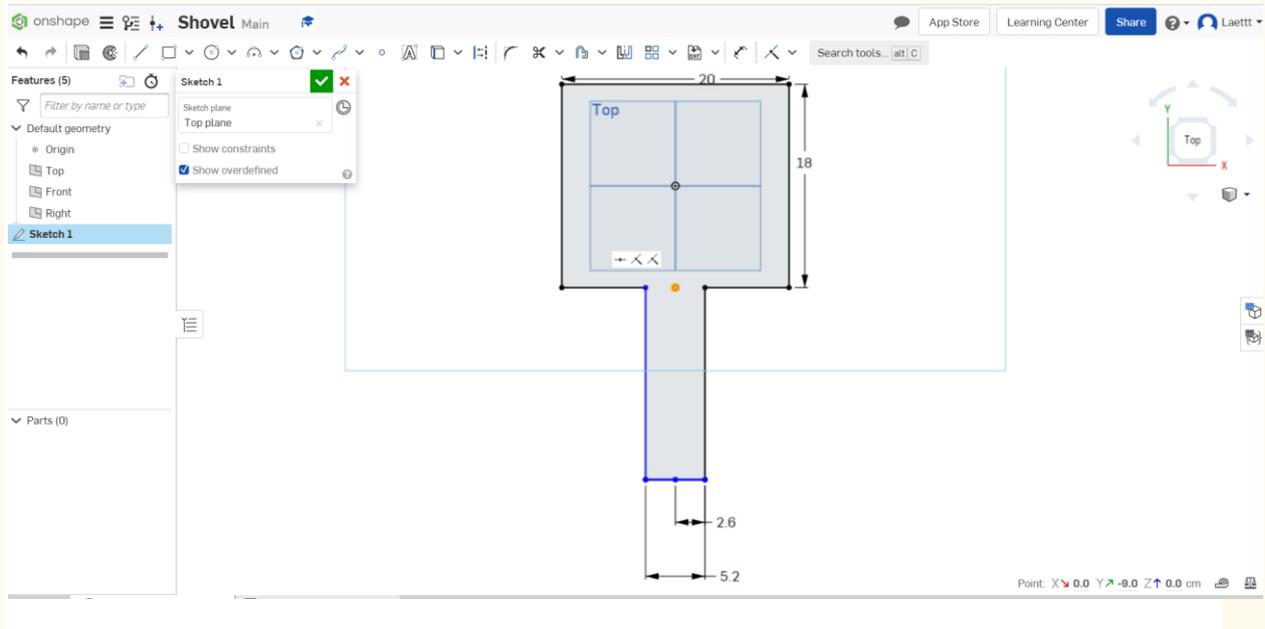
Select the orange lines (then the lines will disappear).

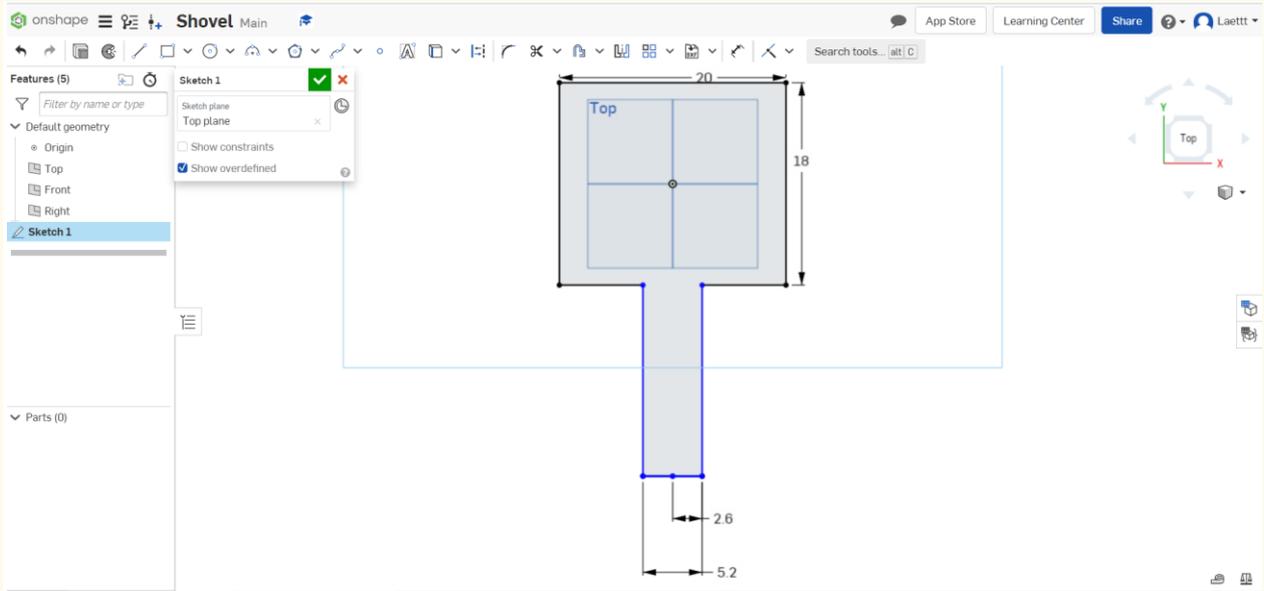




Step 16

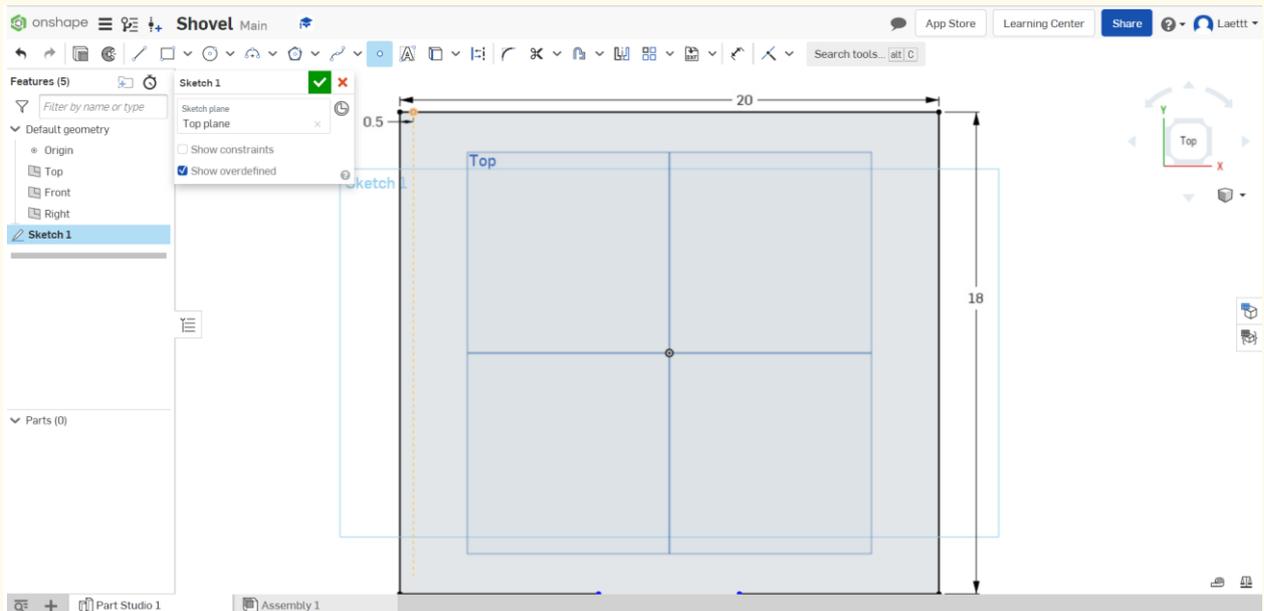
Remove the dot.

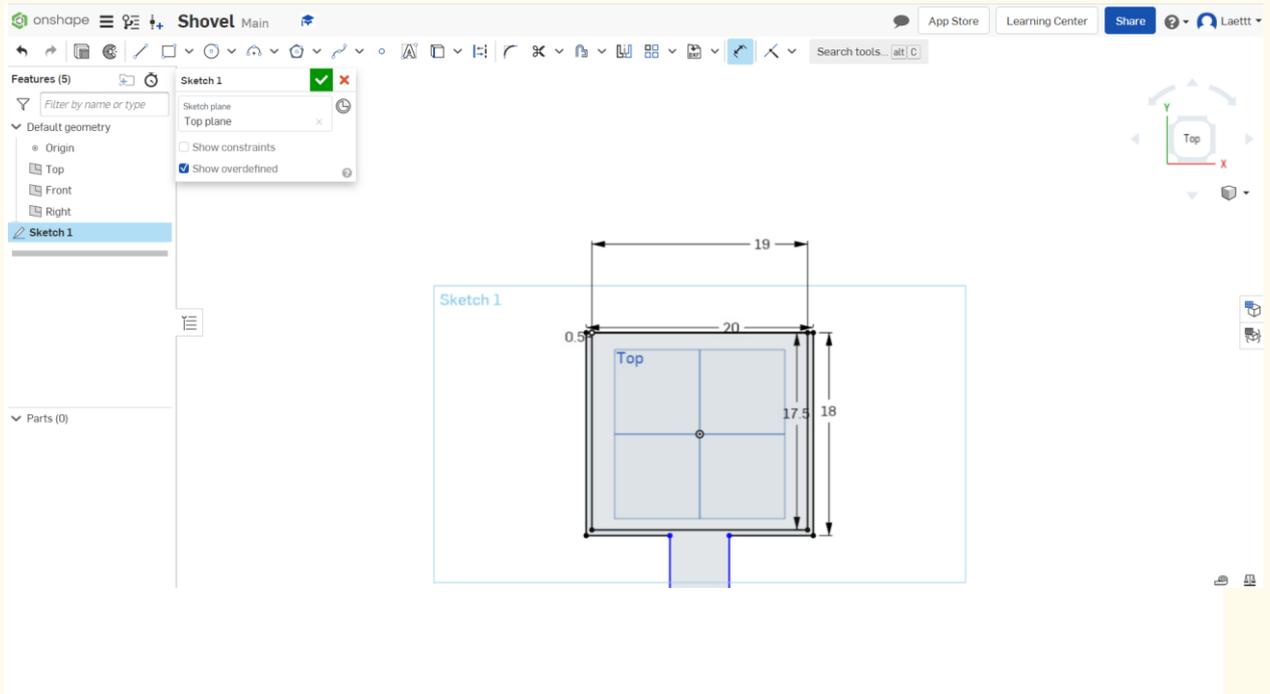




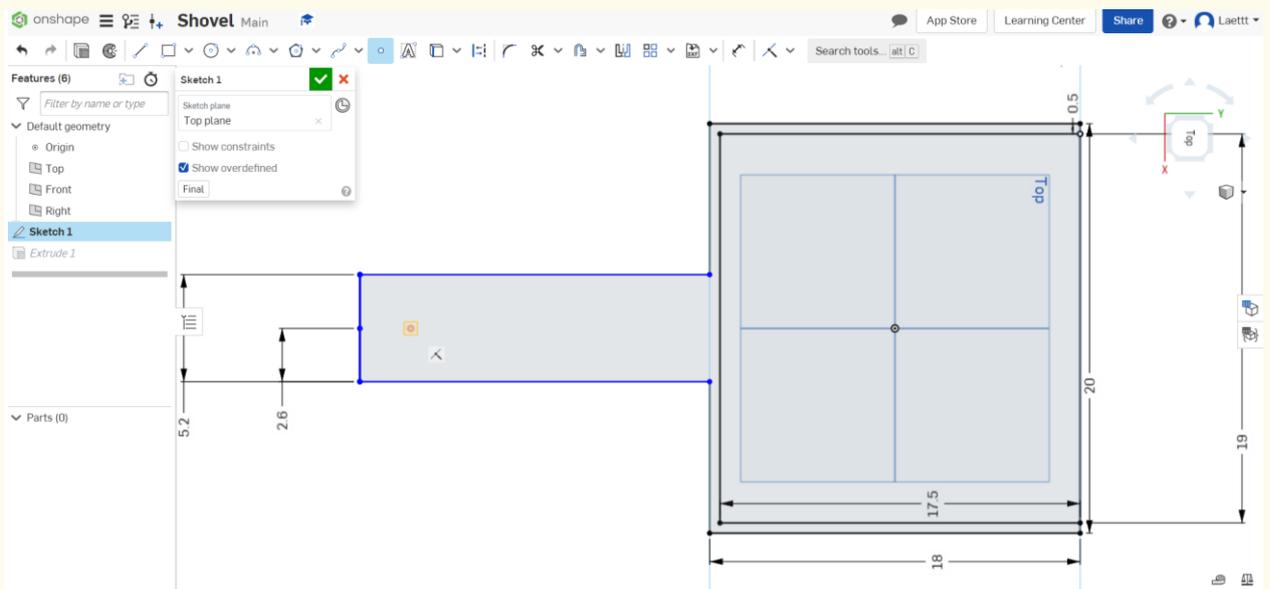
Step 17

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.



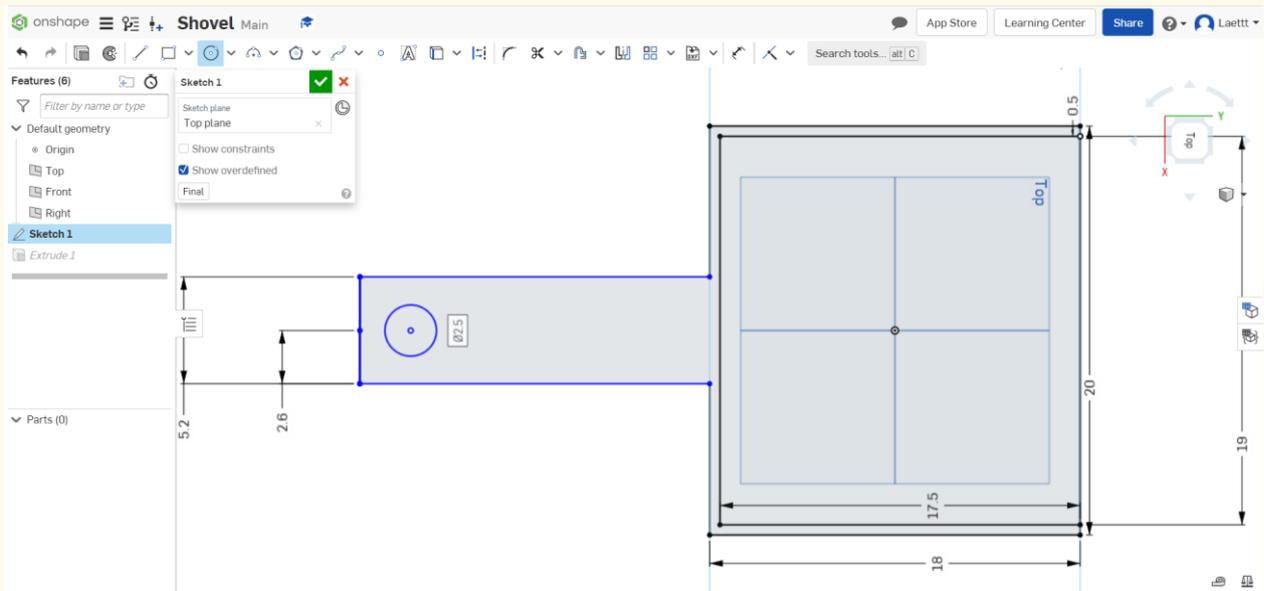
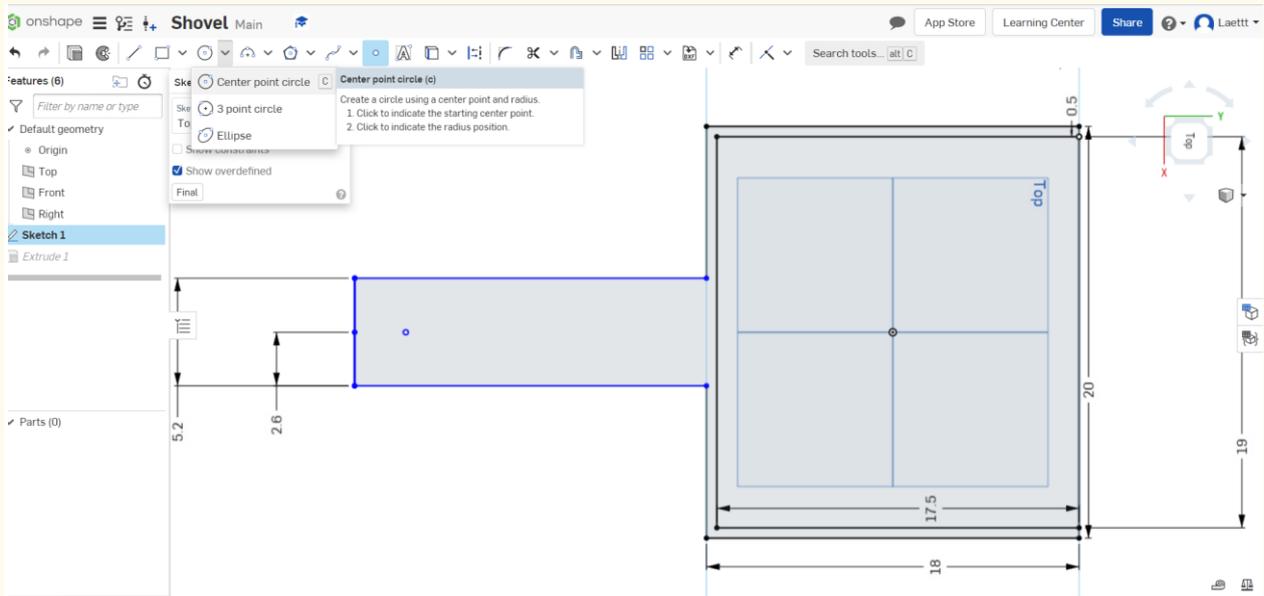


Step 18 To add a hole in the handle, start with a dot, as shown below.



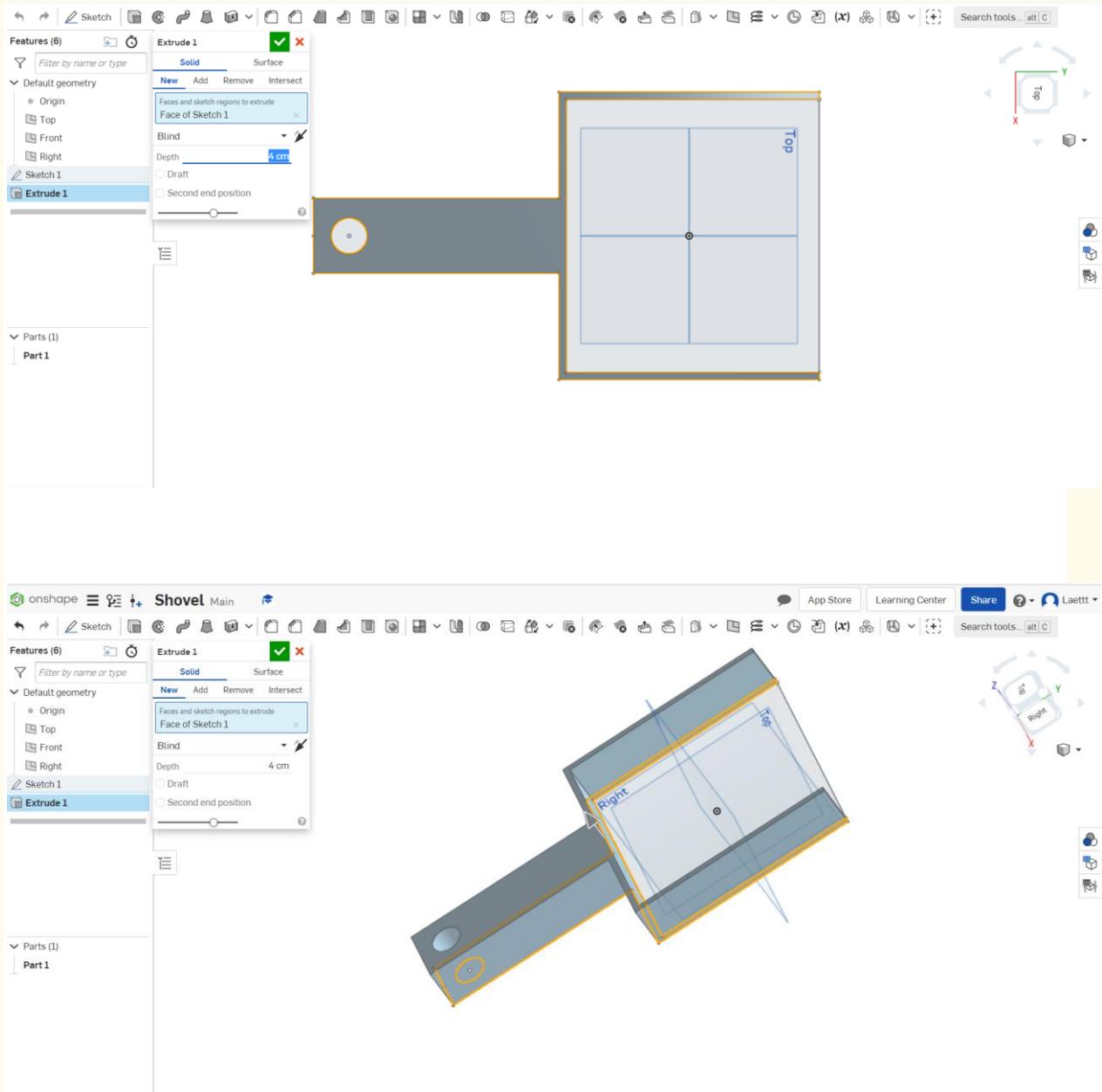
Step 19

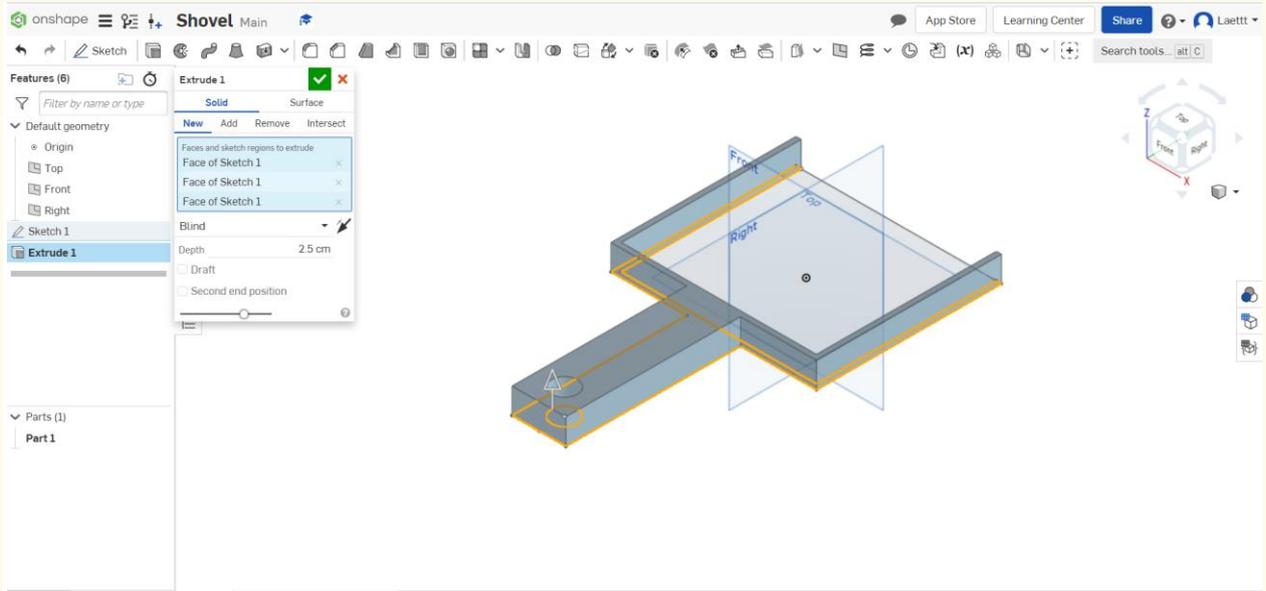
Draw a center dot circle.



Step 20

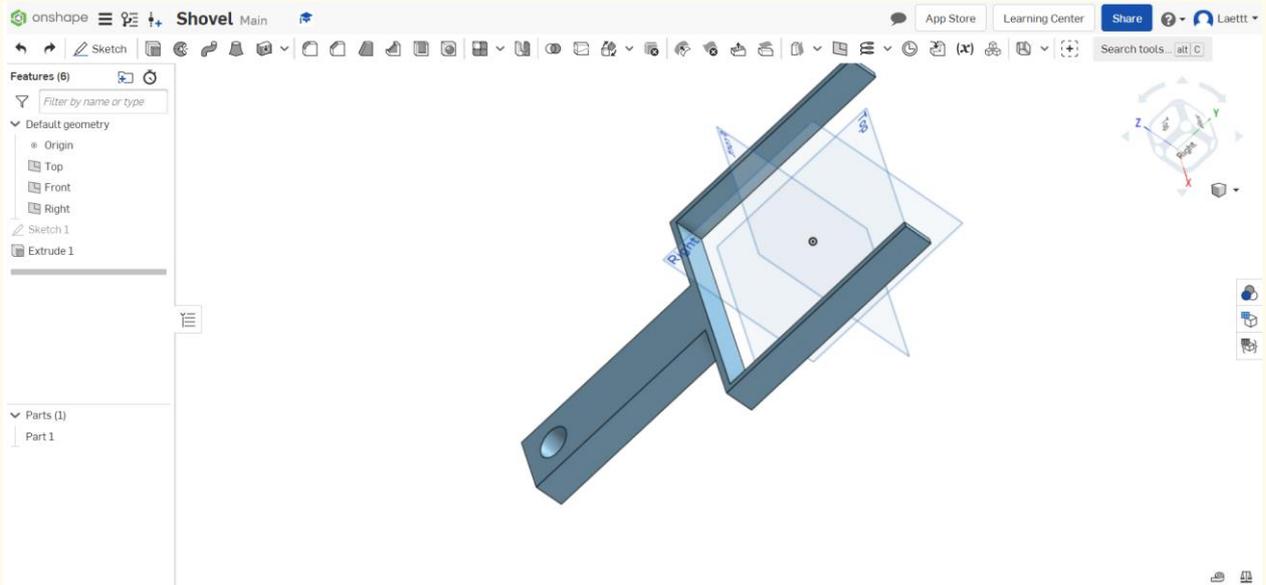
Click on "Extrude" and select the lines you want to extrude (the orange lines).
Change the dimensions to 4cm.





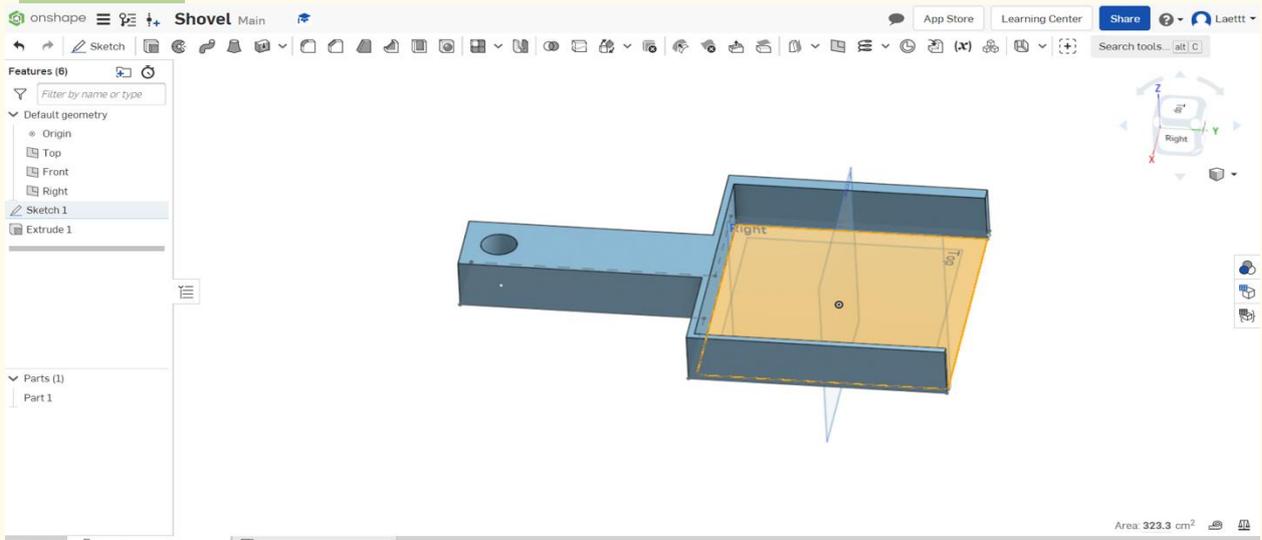
Step 21

Extrude the base of the Shovel. Select the isometric view for obtaining the following view of the Shovel.



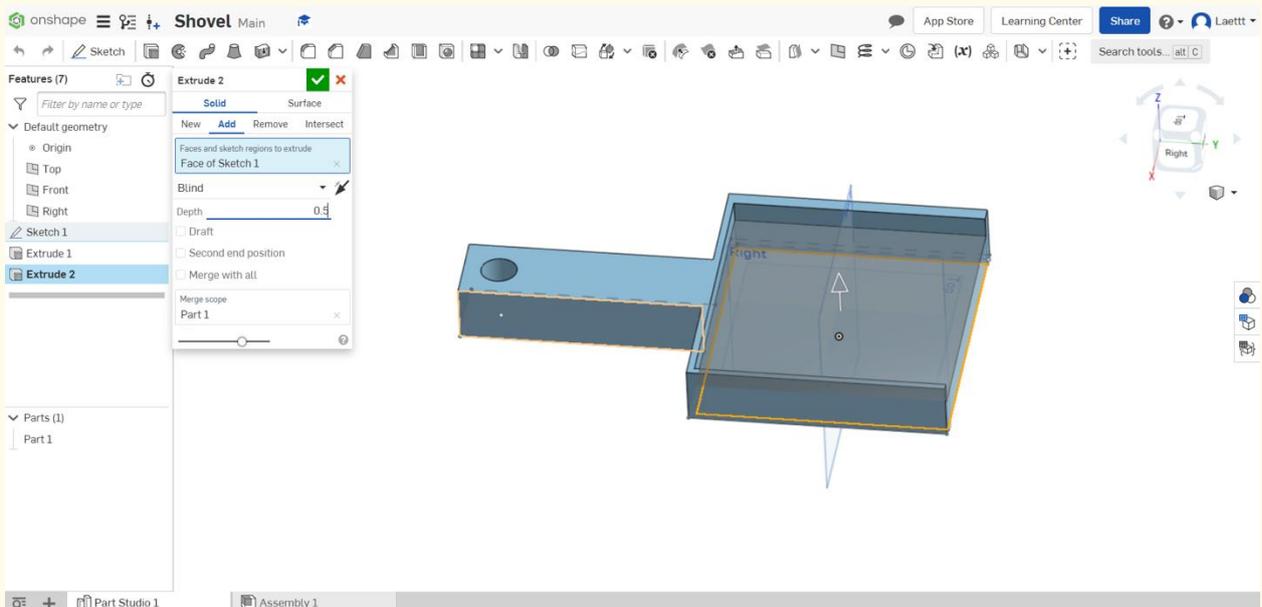
Step 22

Extrude the base and select it.

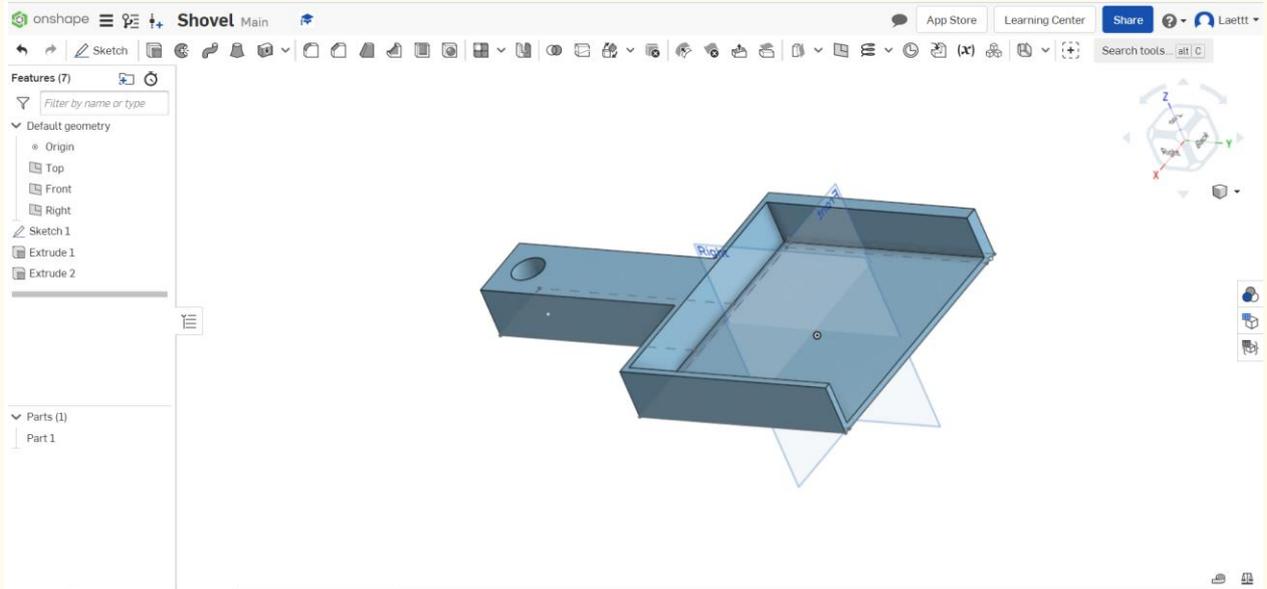


Step 23

Click on "Extrude" and choose 0.5cm. Be careful in the extrude direction (pay attention to the arrow).



Step 24 This should be your result.



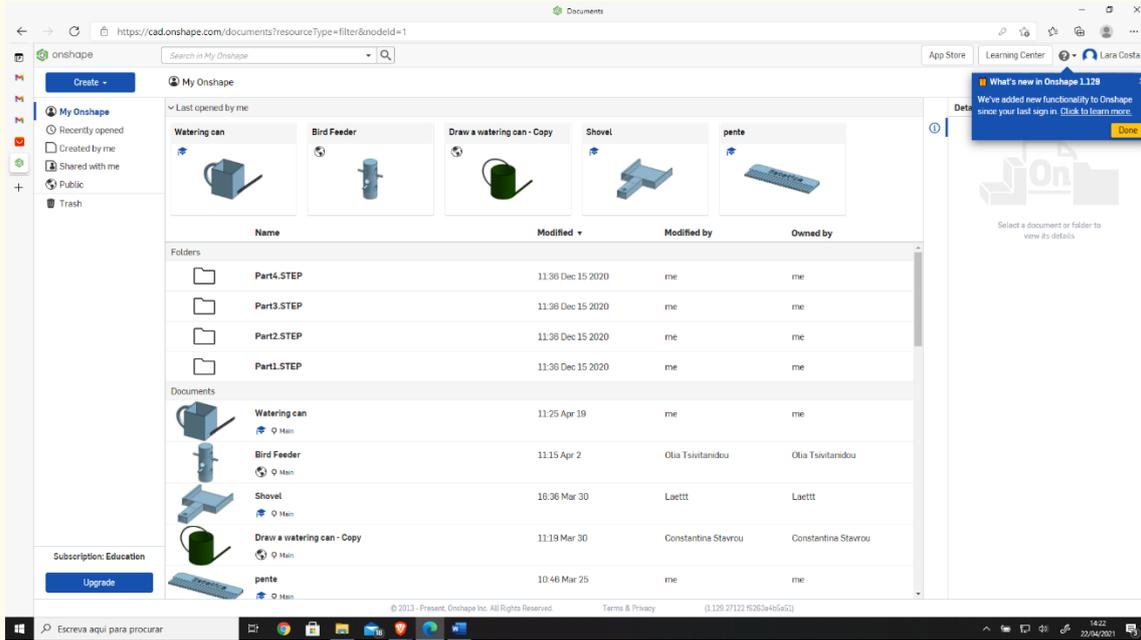
PROJECT: 3D DRAWING OF A WATERING CAN

- STEM field: Science, technology, and electronics.
- Indicative calendar: Any time of the year.
- Activity duration: 3 hours.
- Type of activity: 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 competences:
 - How to do a watering can on Onshape.
- Required material and resources:
 - 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:

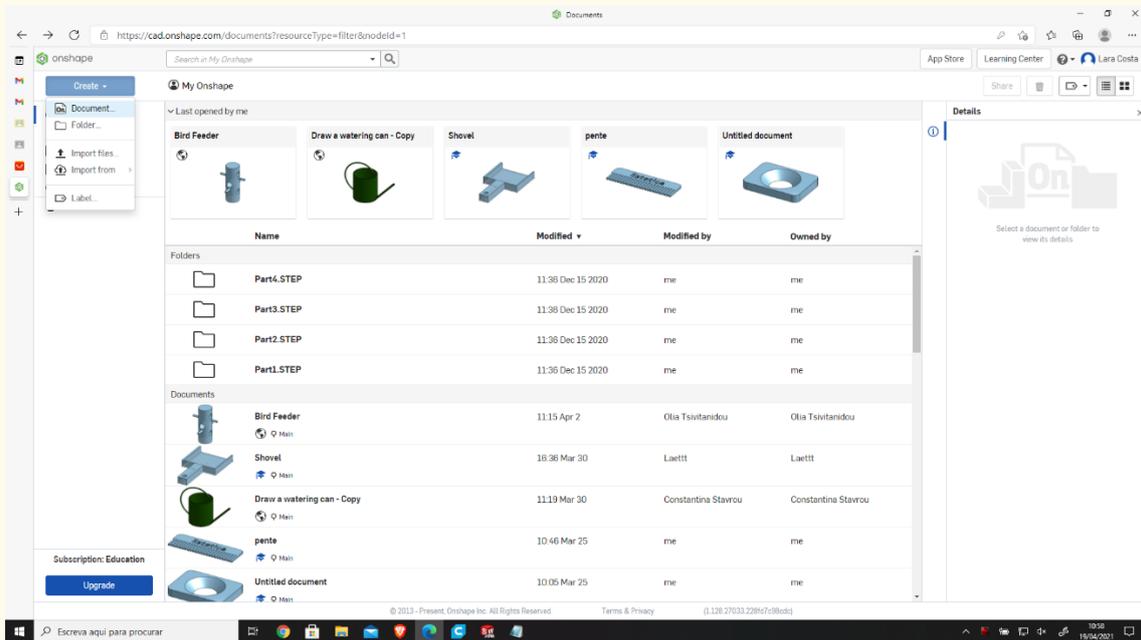
Step 1

Open Onshape (the free version).



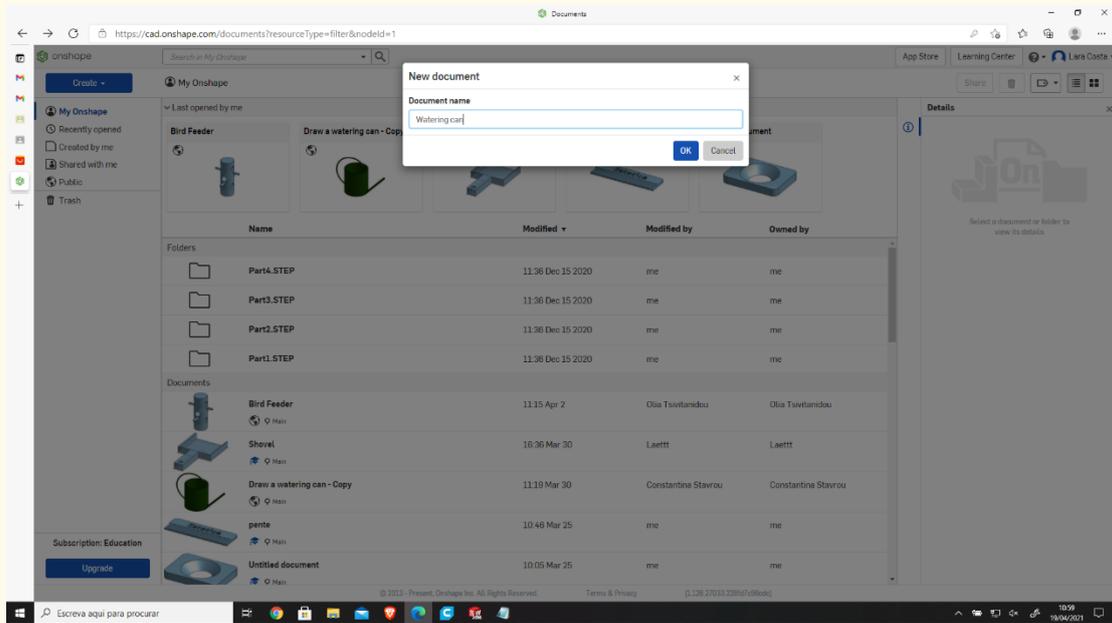
Step 2

Create a document.



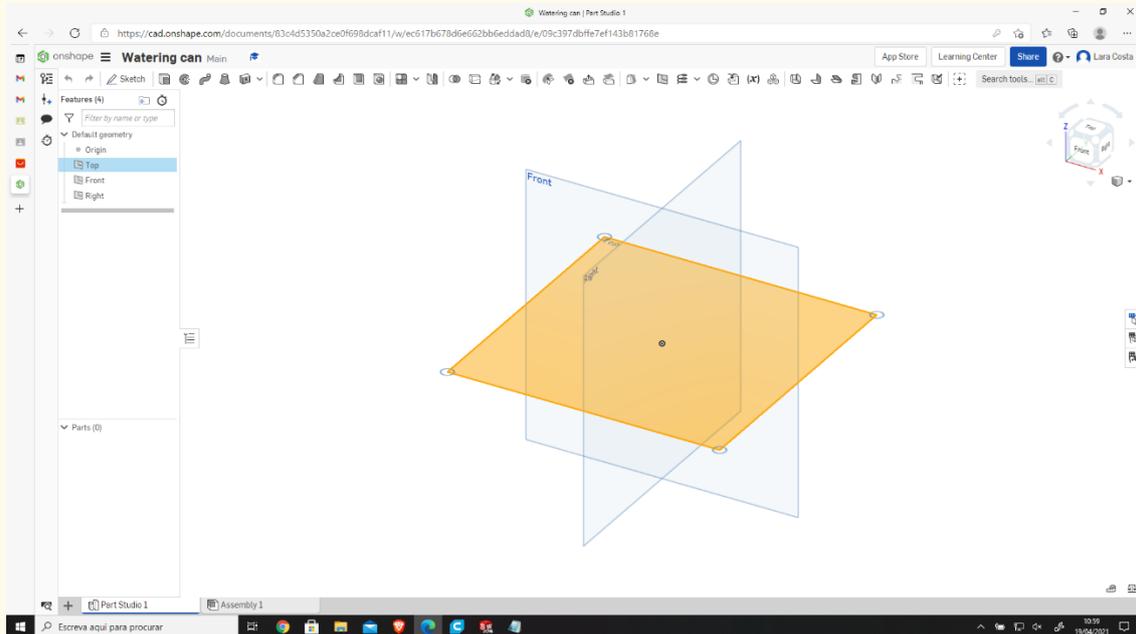
Step 3

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



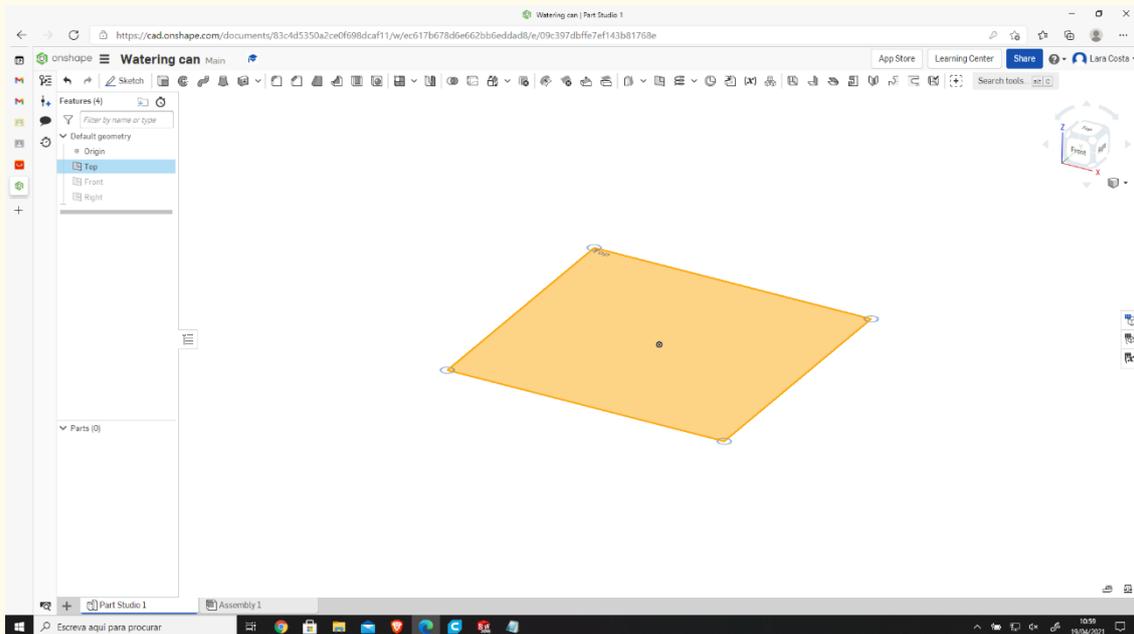
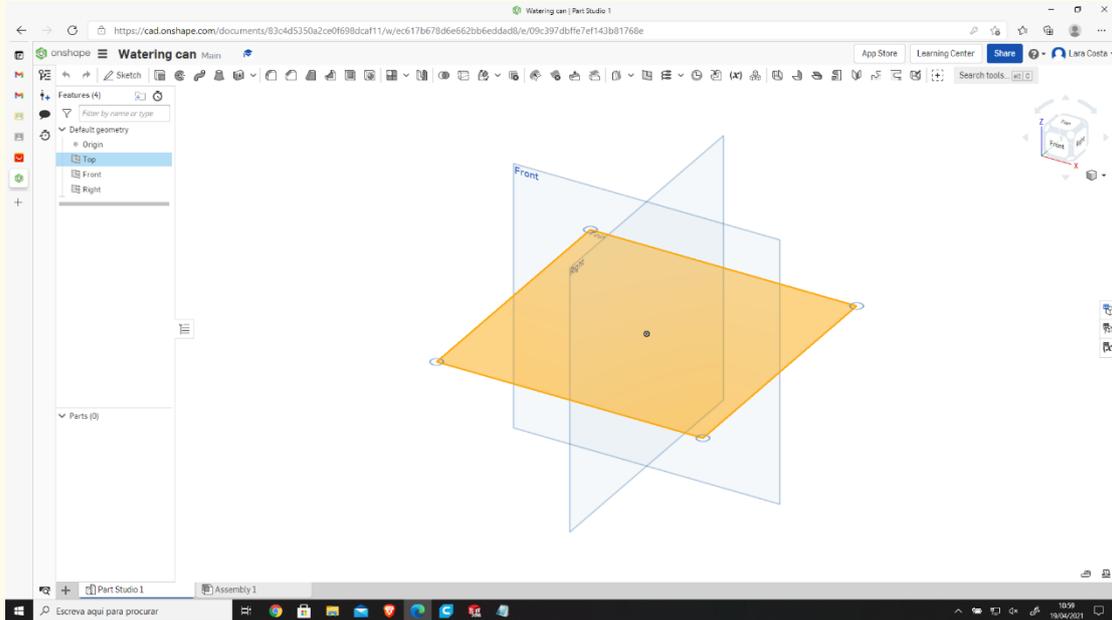
Step 4

Select the "Front" to start drawing.

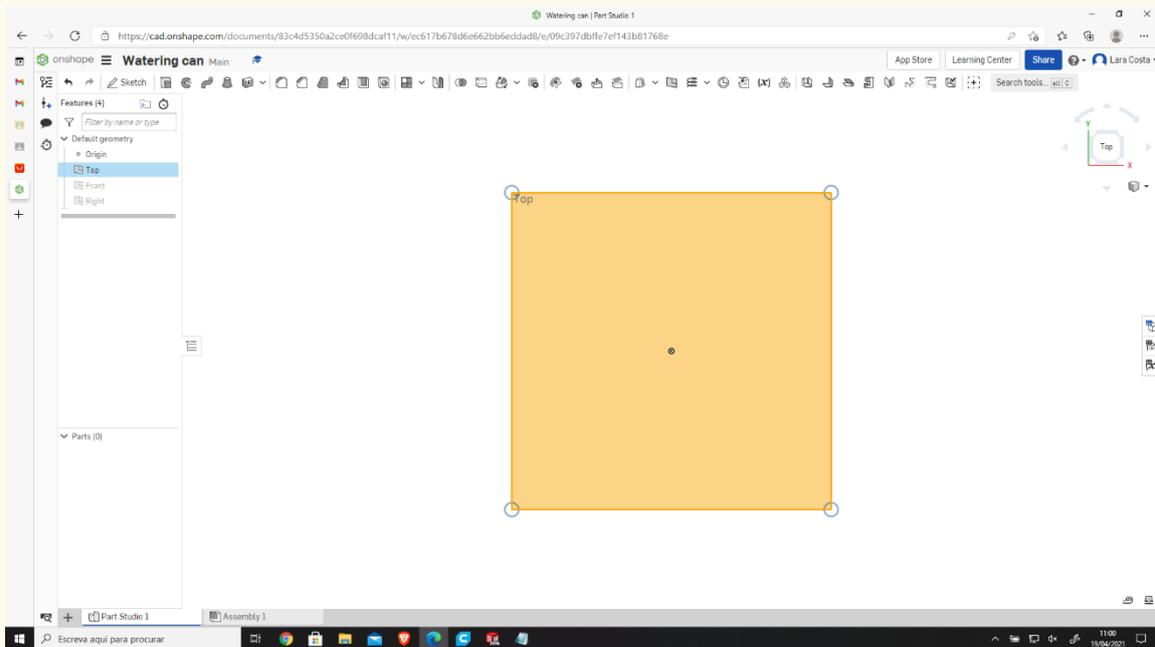
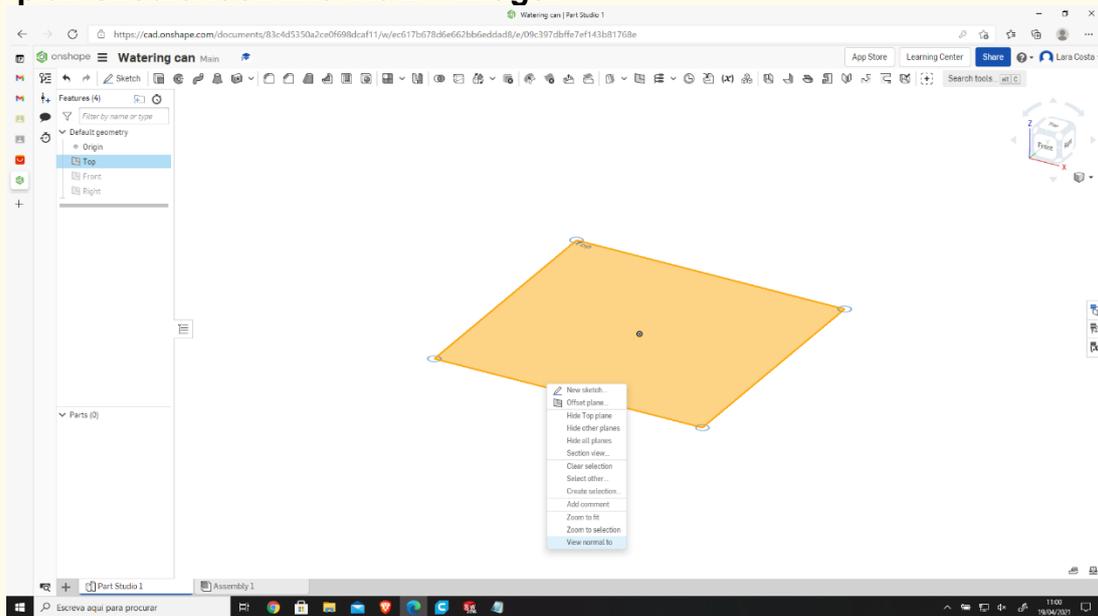


Step 5

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

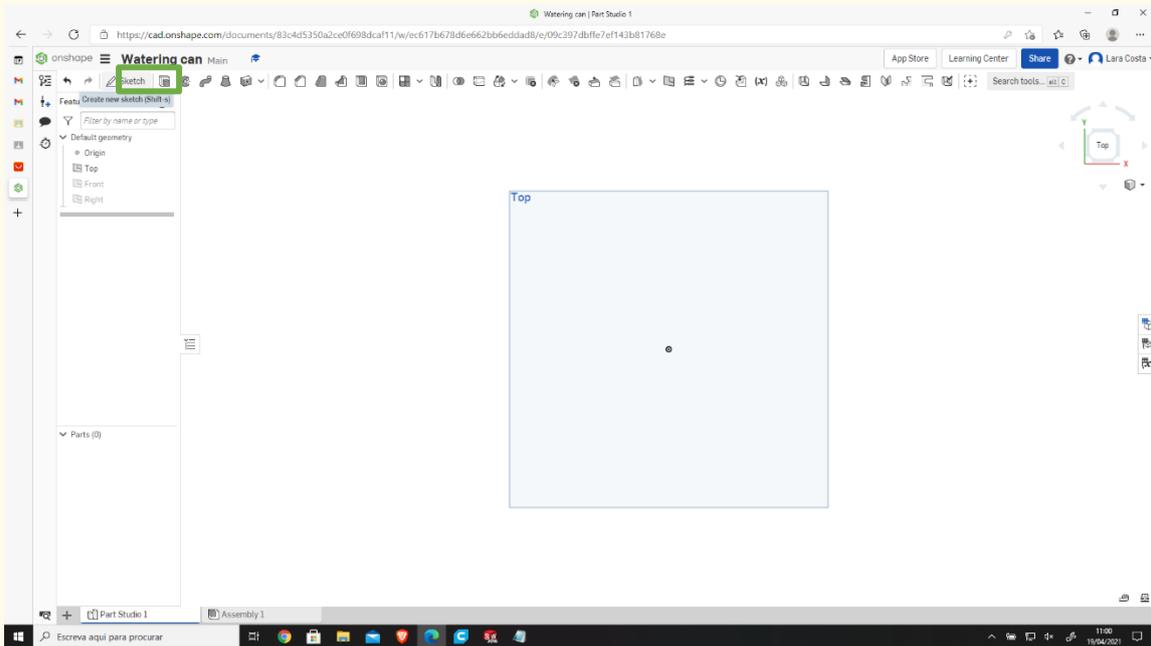


Right-click and select "View normal To."
The plan should look like the 2nd image.



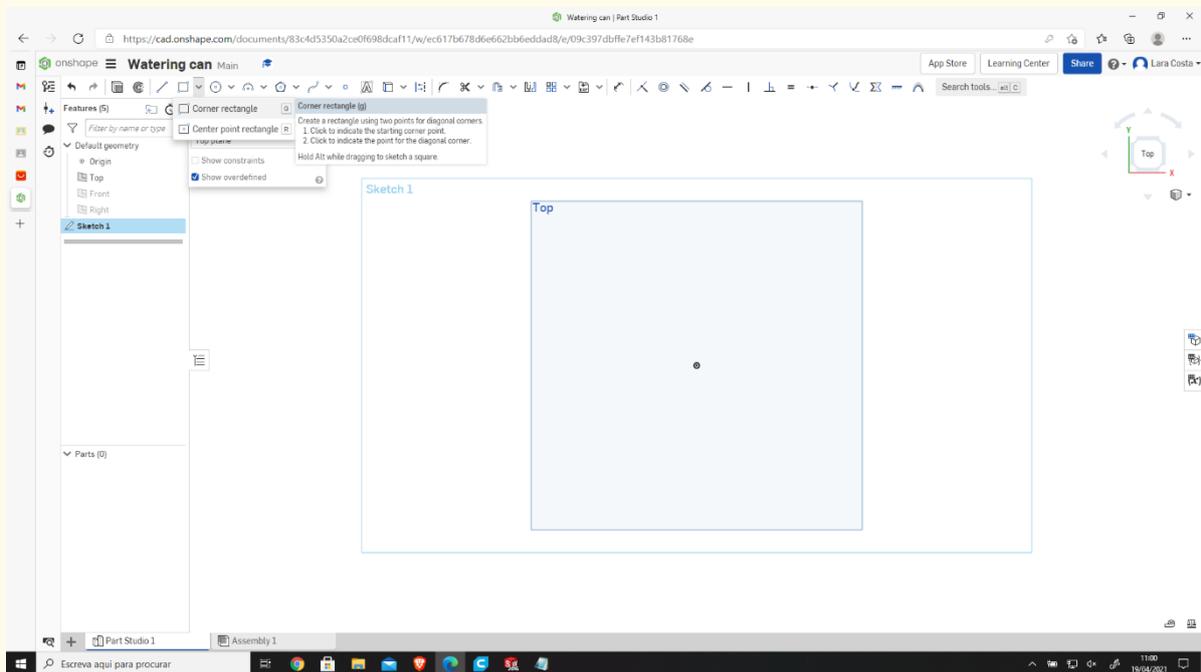
Step 6

Click "Sketch."



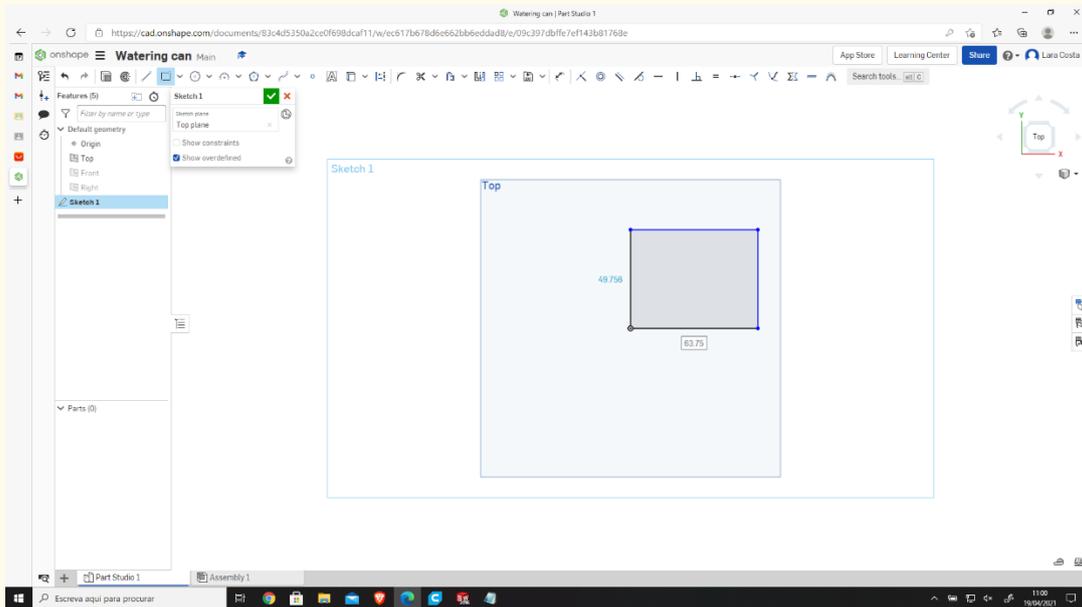
Step 7

Select corner rectangle to draw.



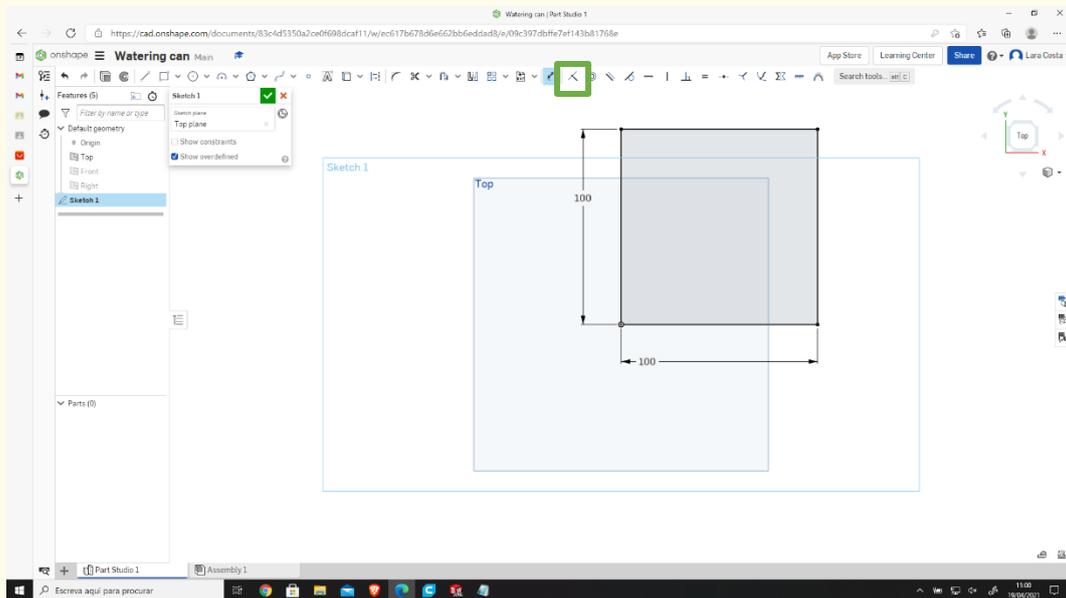
Step 8

Draw the rectangle.



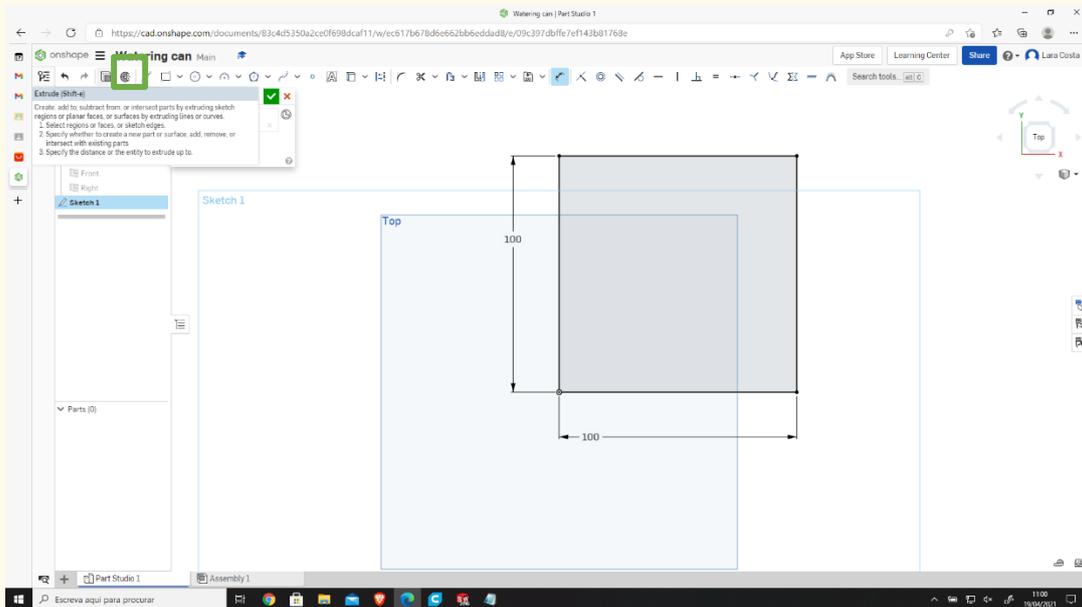
Step 9

Click on "Dimension" and give 100mm to each side.



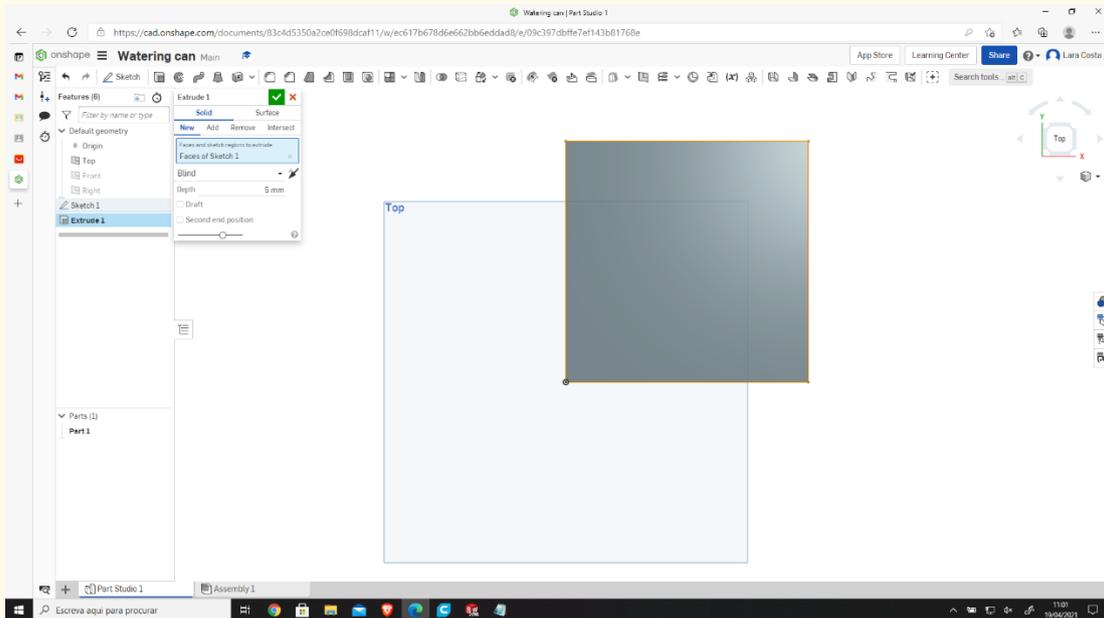
Step 10

Select "Extrude."

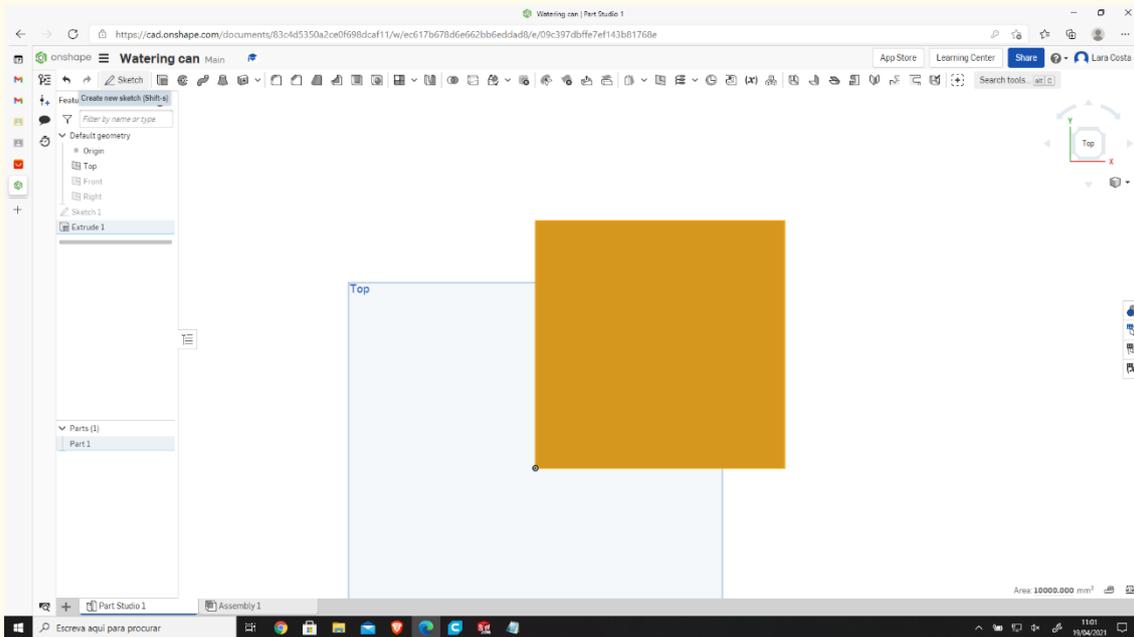


Step 11

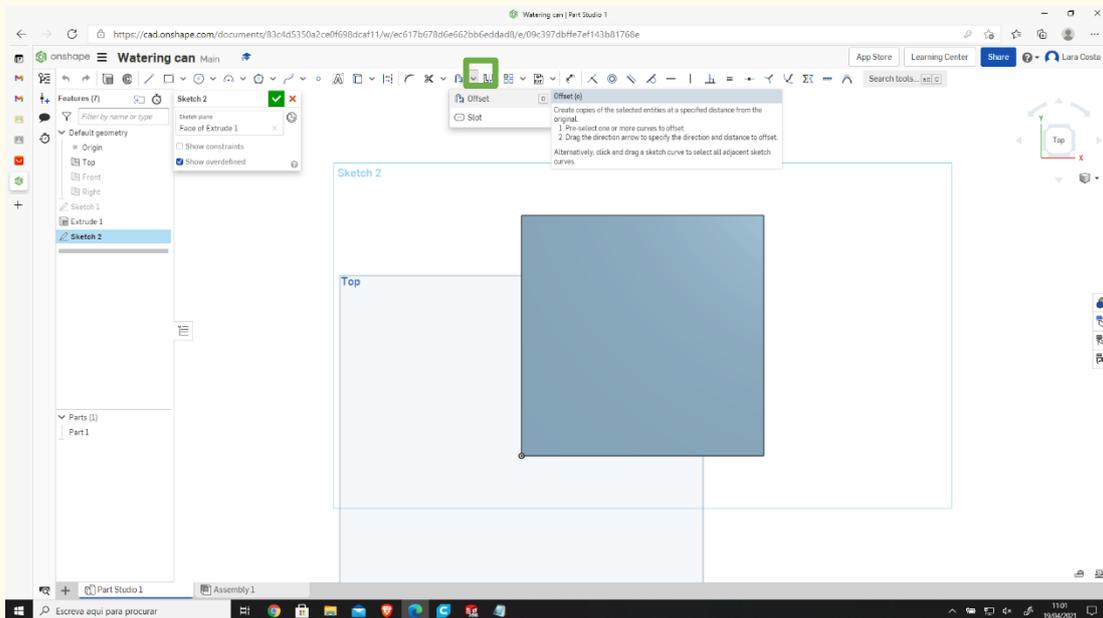
The "Depth" should be 5mm.



Step 12 Select the "Plan" and "Sketch".

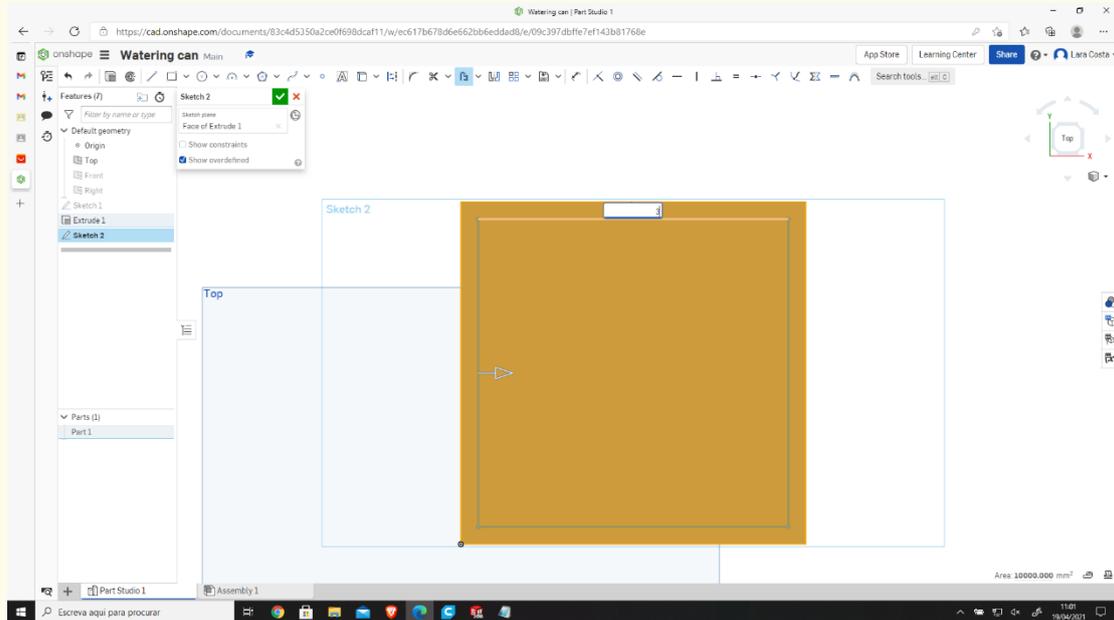


Step 13 Select "Offset."



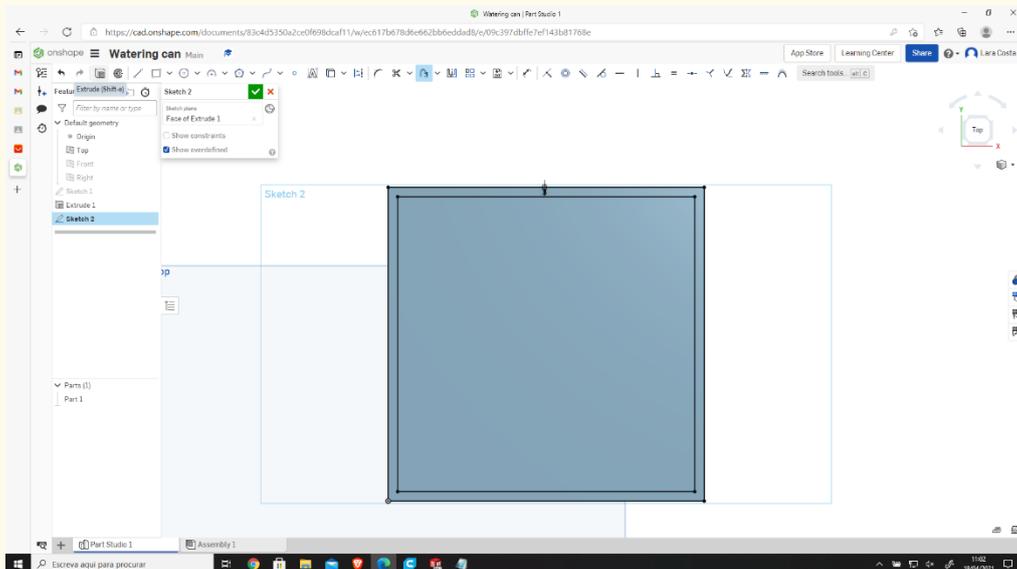
Step 14

Select the outlines and give a dimension of 3mm.



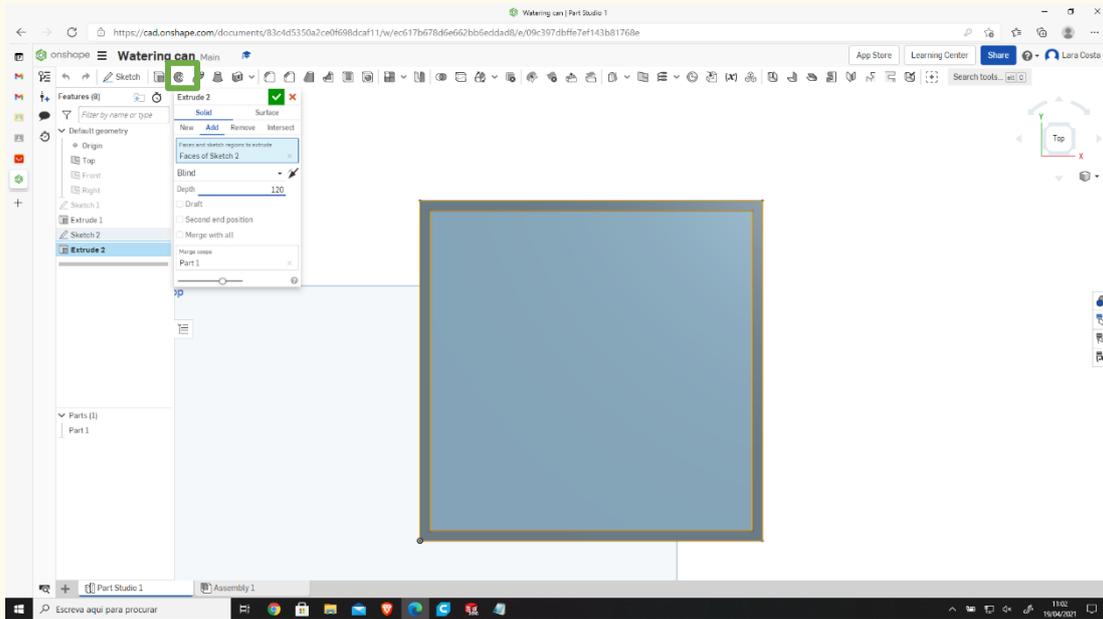
Step 15

Do the same as steps 13 and 14, with a dimension of 0 mm (use the original lines).



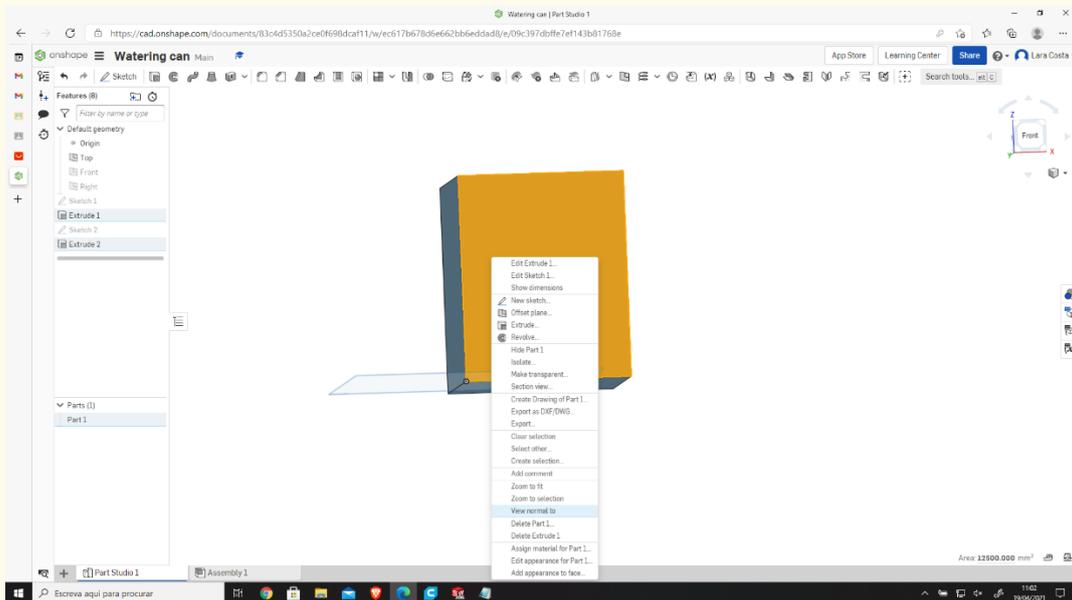
Step 16

Select "Extrude" and give a dimension of 120mm.



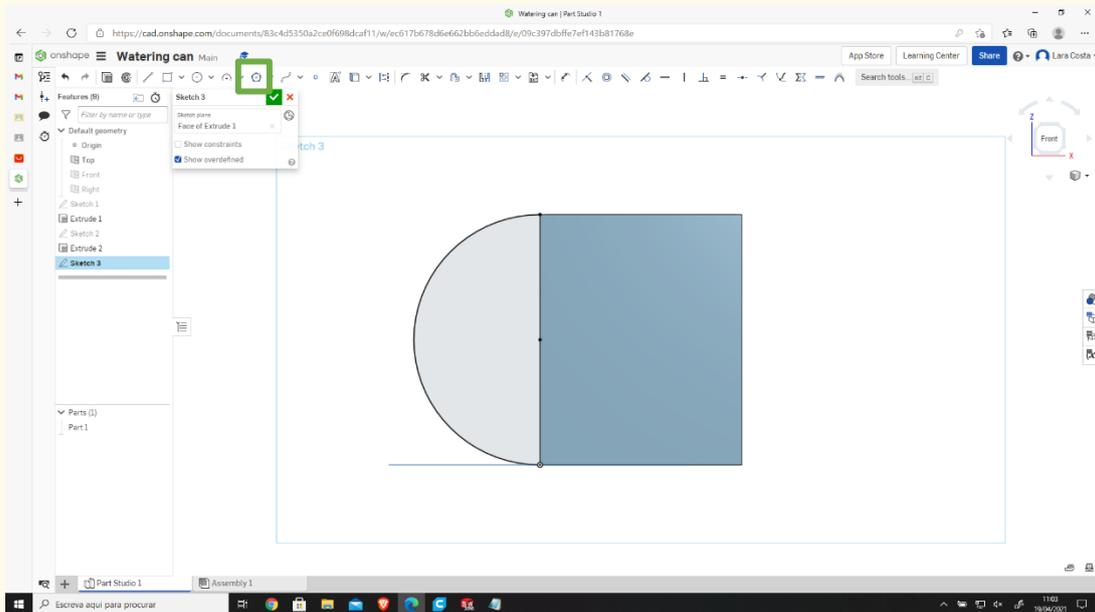
Step 17

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



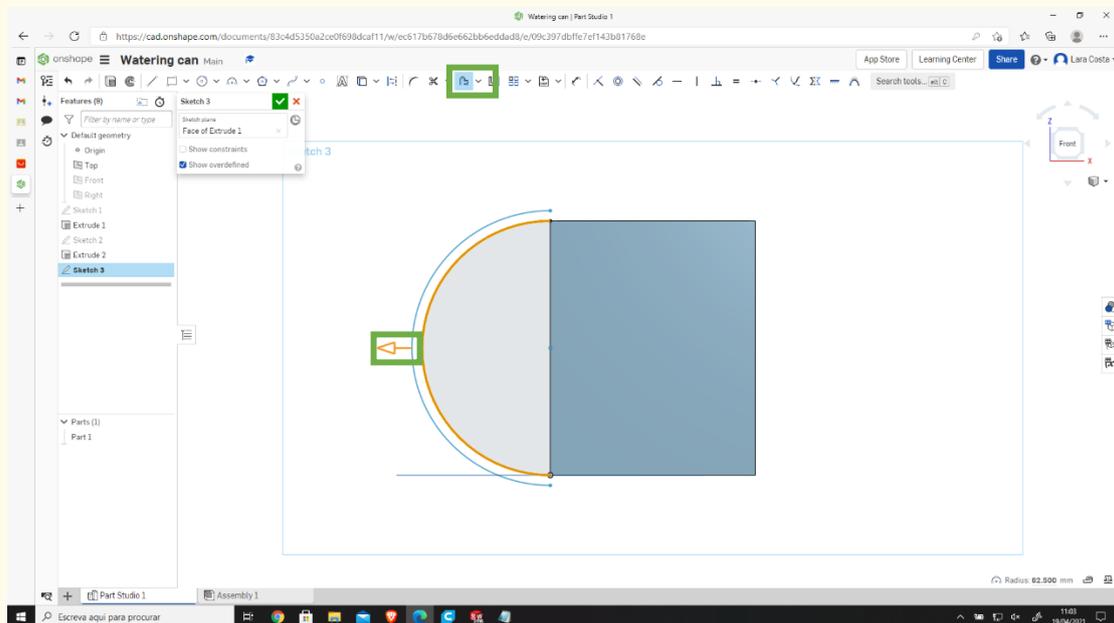
Step 18

Select 3 "arc points" and draw a semi-circle as shown below.

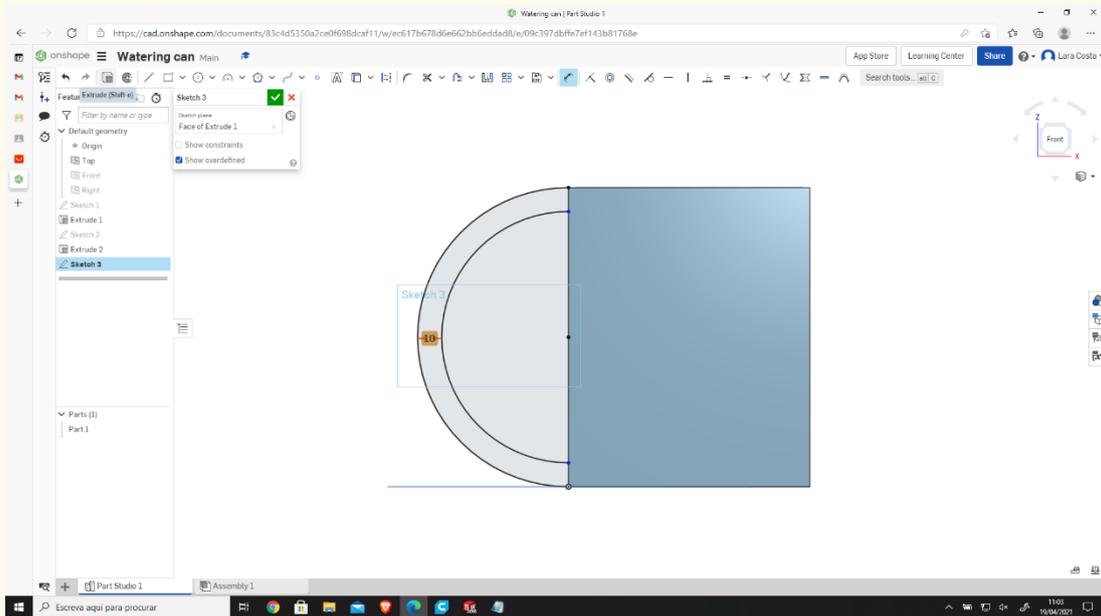


Step 19

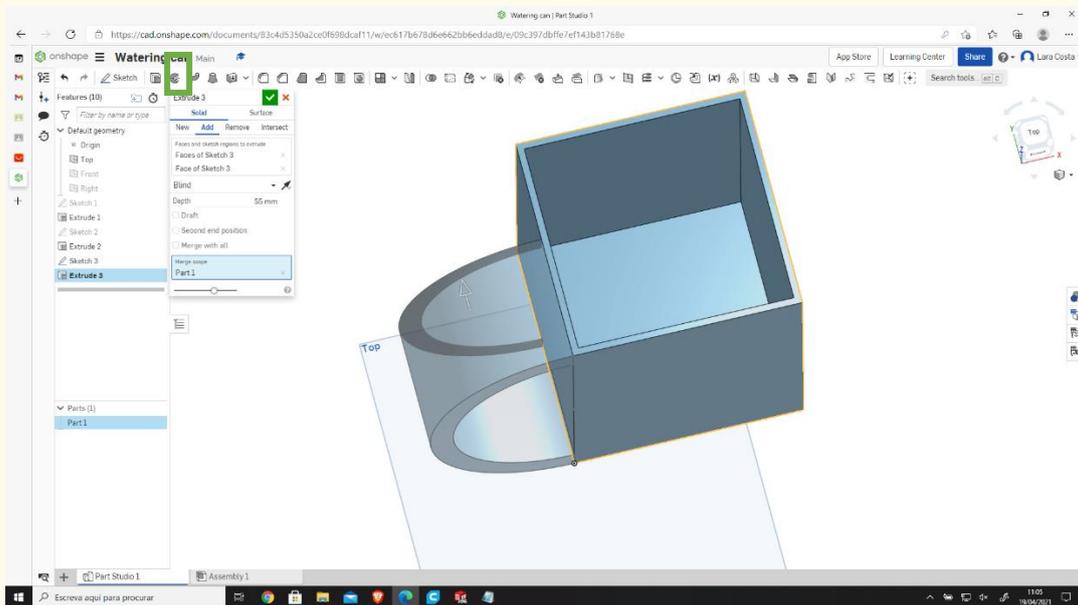
Click on "Offset," then choose the orange line and then in the arrow to change the side (if necessary).



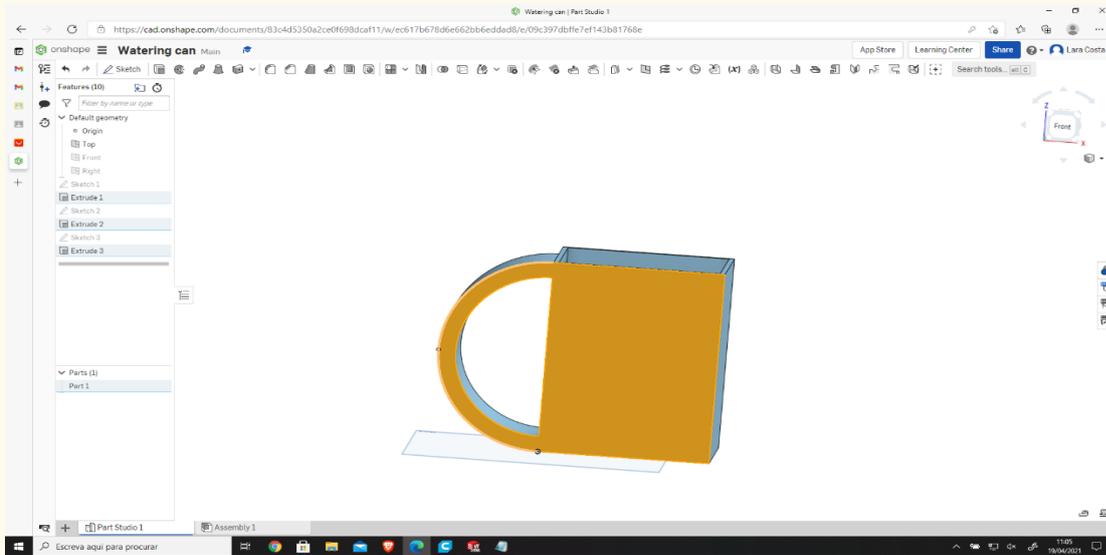
Step 20 Type 10 mm as a dimension.



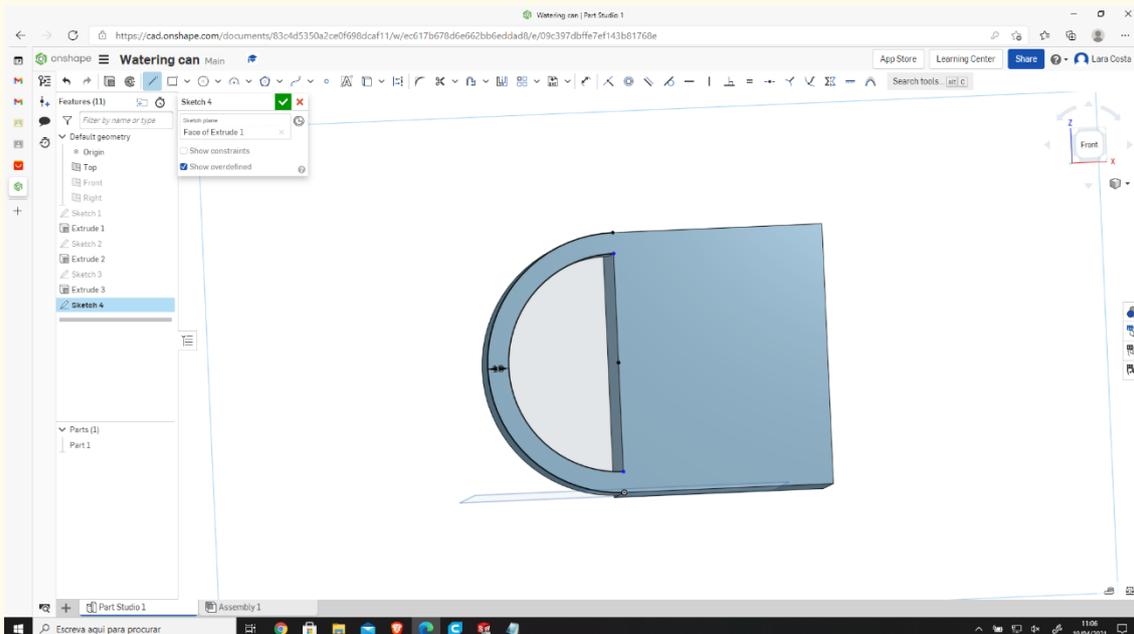
Step 21 Click on Extrude and insert 55 mm as Depth.



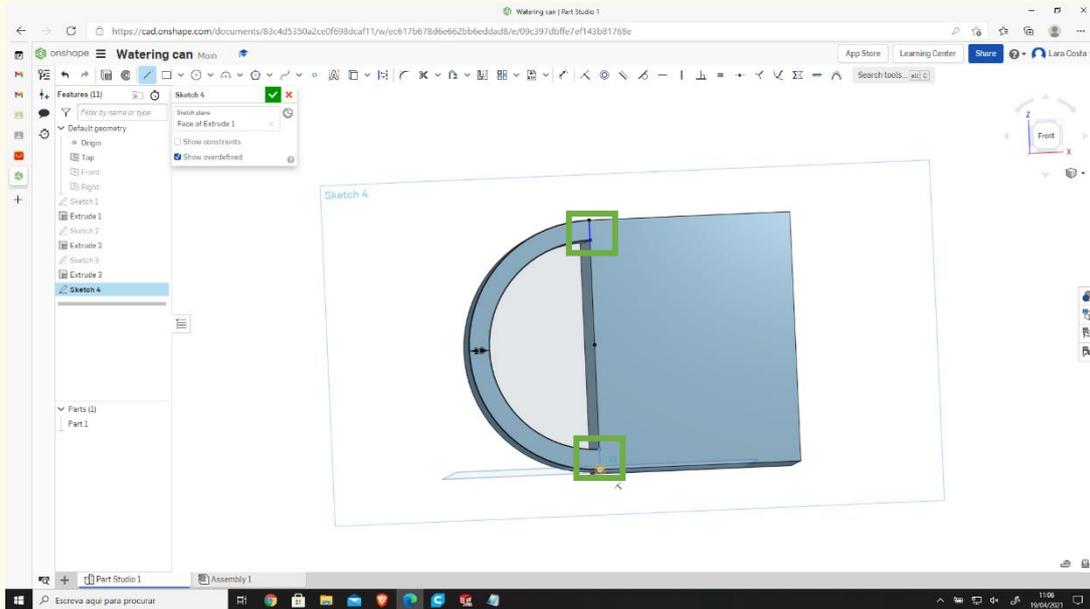
Step 22 Select the "Front" side again and sketch.



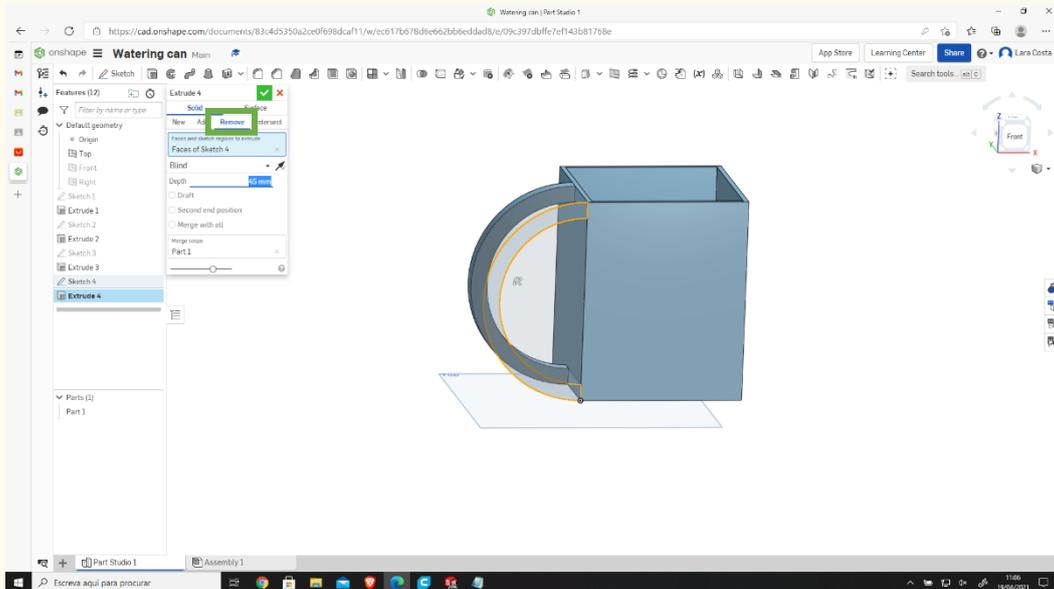
Step 23 Do the same as steps 18, 19, and 20.



Step 24 Select "Line" and draw 2 lines as the image shows (blue lines).

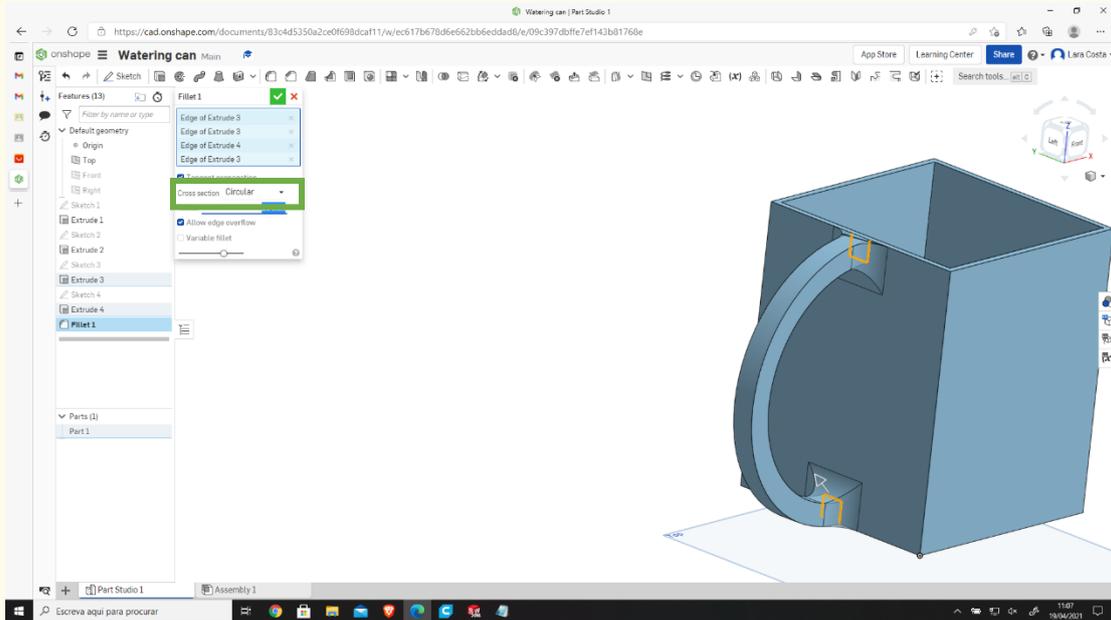


Step 25 Select "Extrude" – "Remove" and change Depth to 45mm.



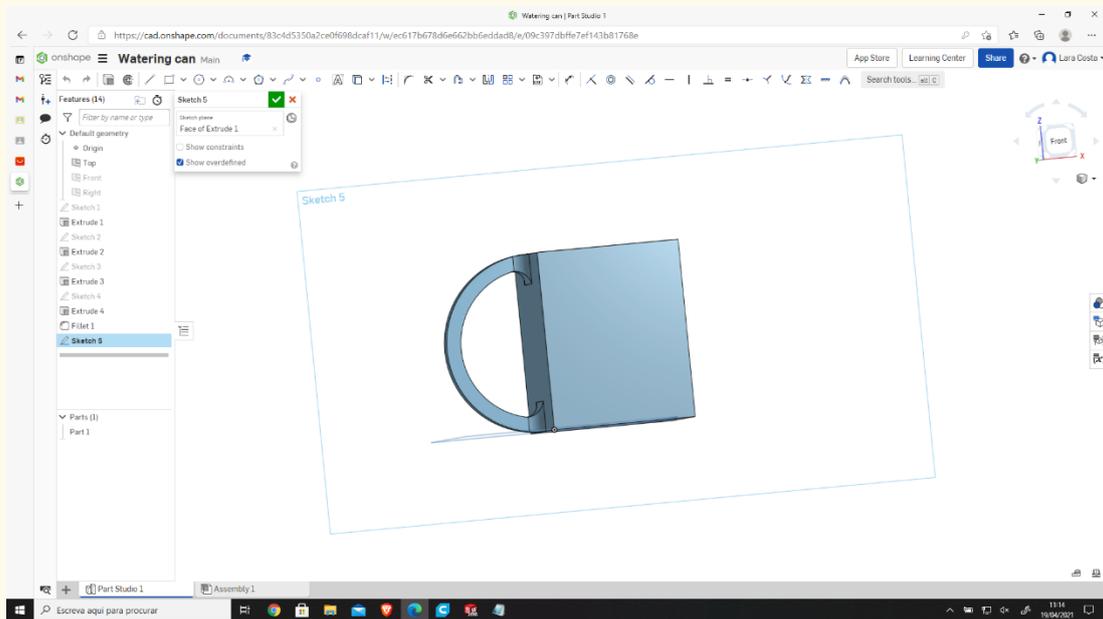
Step 26

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



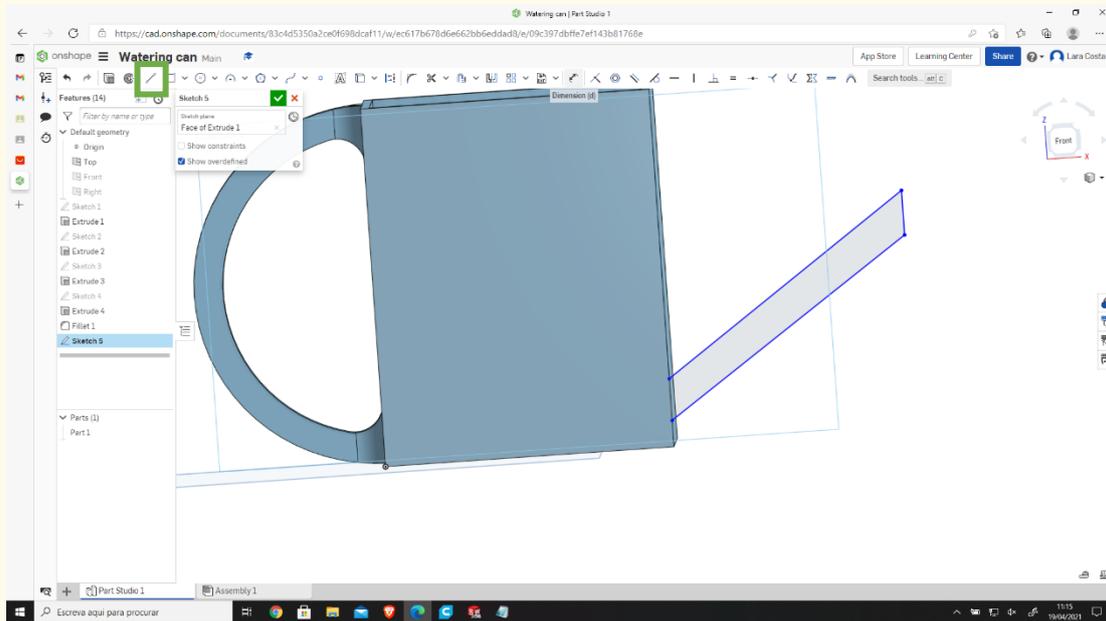
Step 27

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



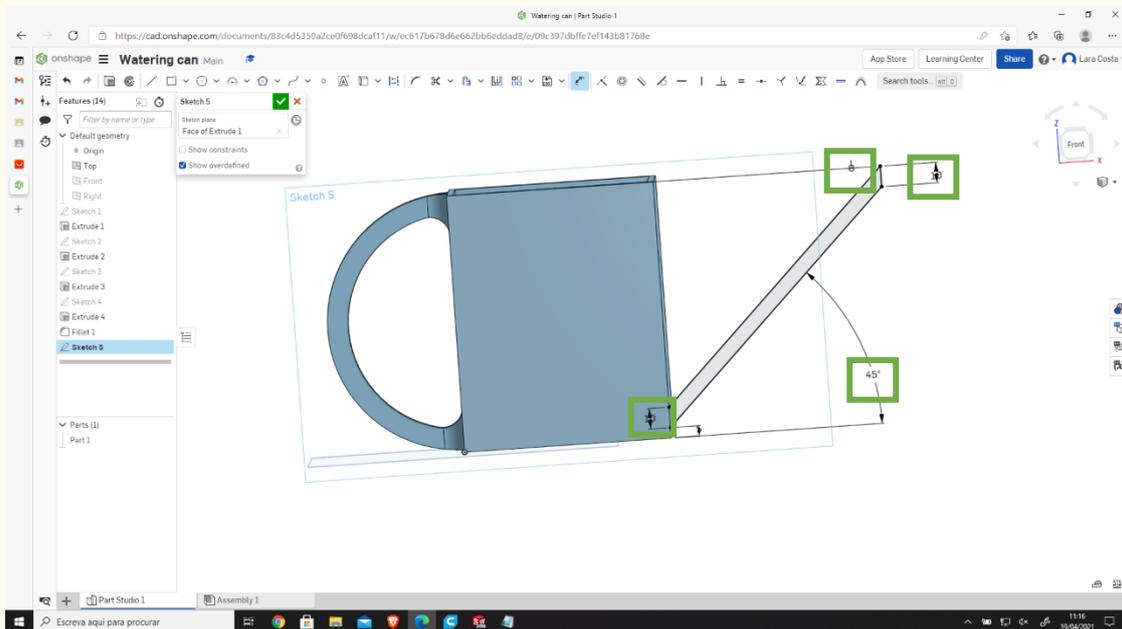
Step 28

Select "Line" and draw the lines as the image shows (blue lines).



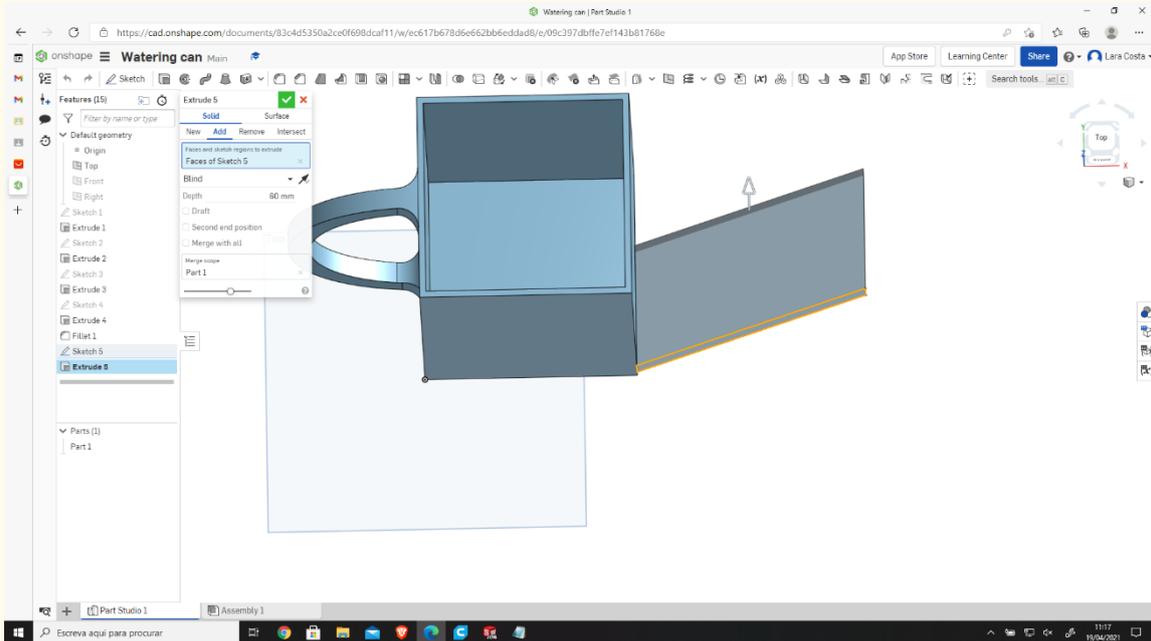
Step 29

Select "Dimension" and insert the right dimensions.



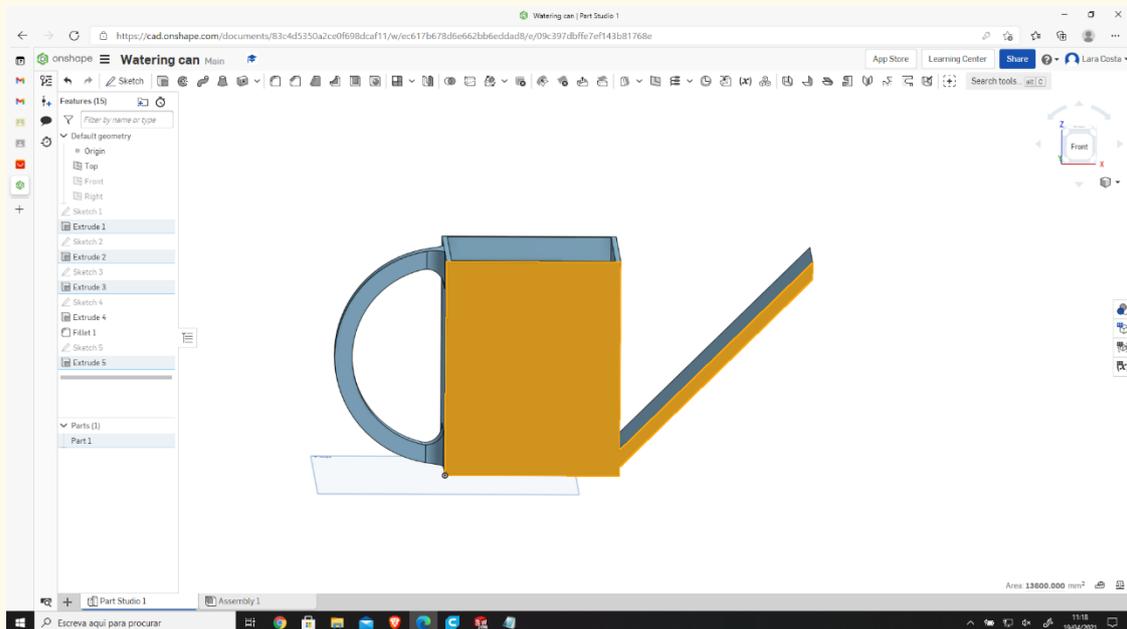
Step 30

Select "Extrude" and type 60 mm as "Depth."



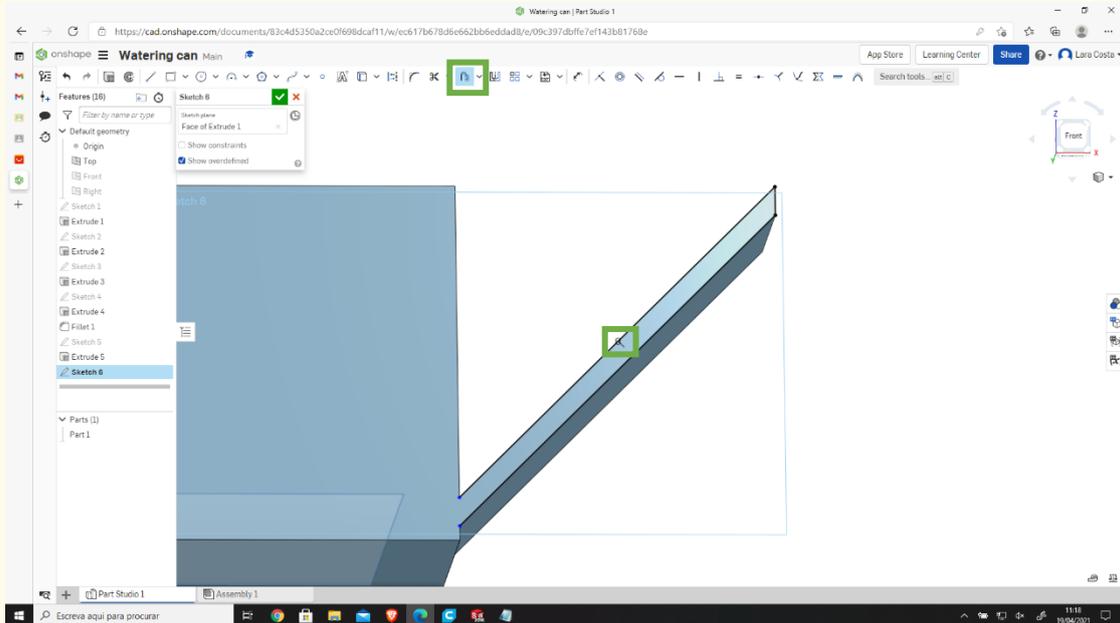
Step 31

Select the "Front" plan and then "Sketch."



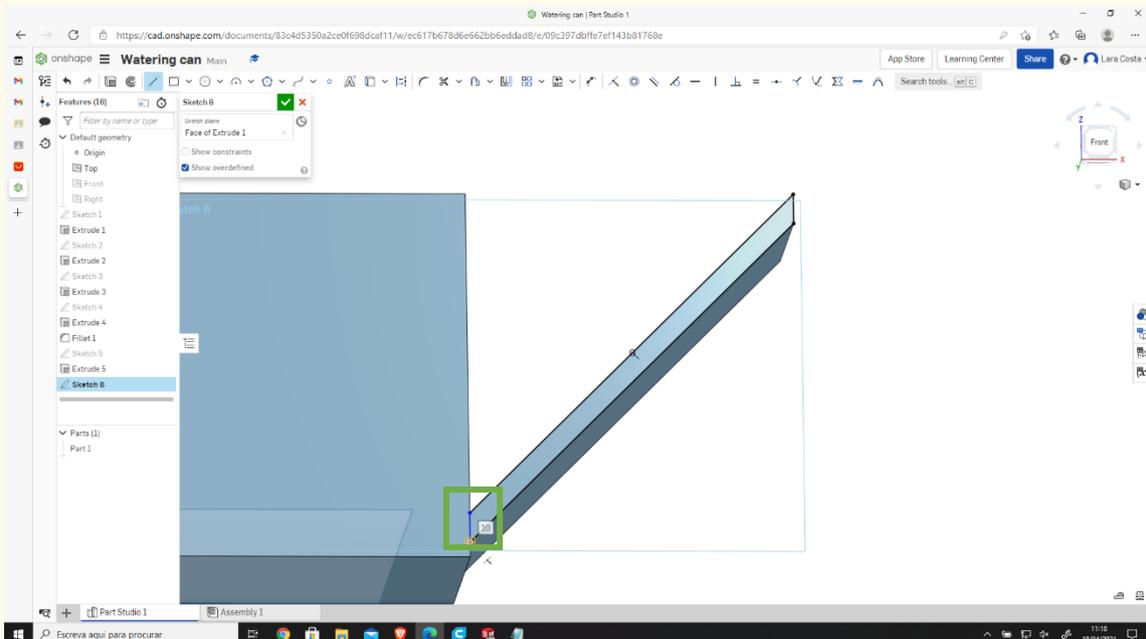
Step 32

Select "Offset" and choose the lines in black (the dimensions should be 0mm).

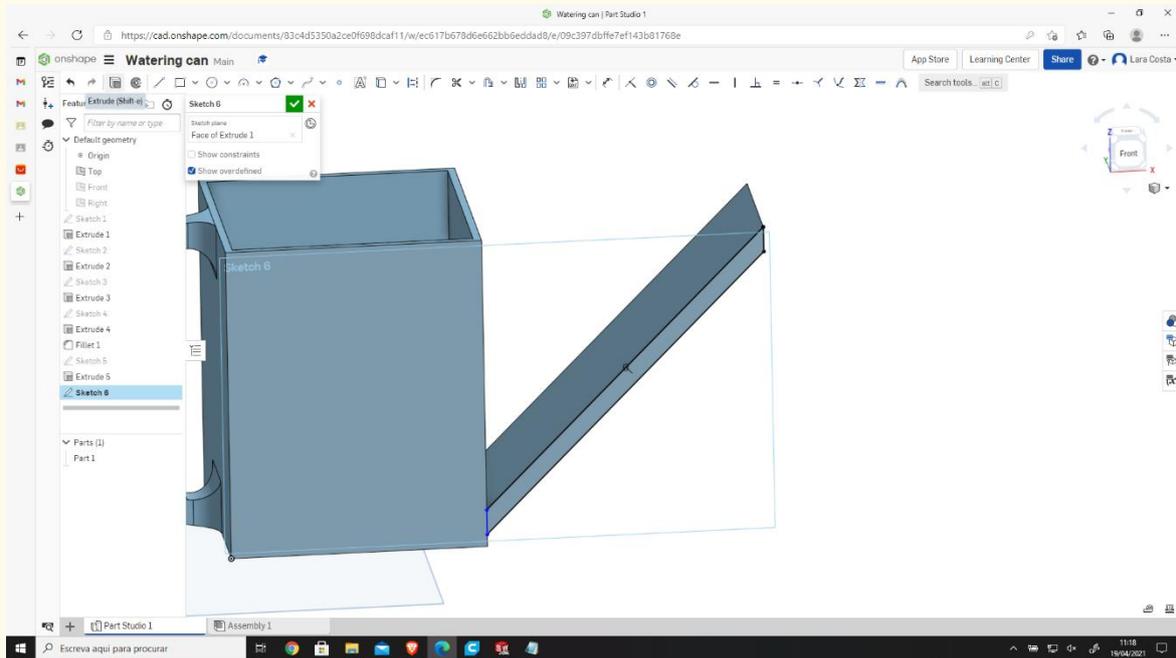


Step 33

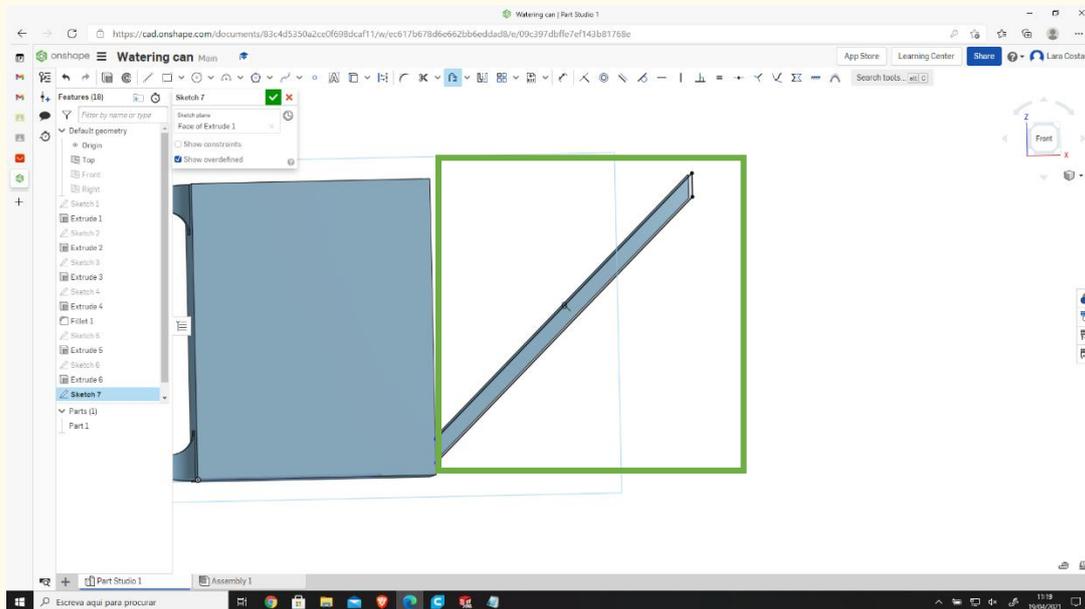
Draw the line that can be shown in blue.



Step 34 Select "Extrude."

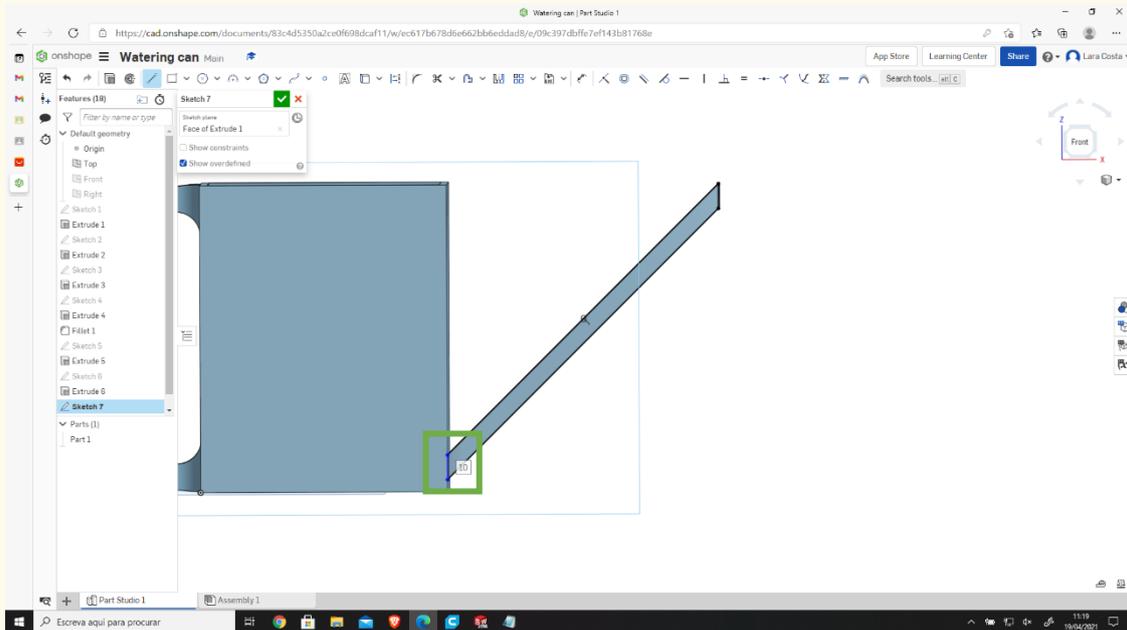


Step 35 Choose "Remove 57mm", then click on the drawn tip, and then "Sketch."



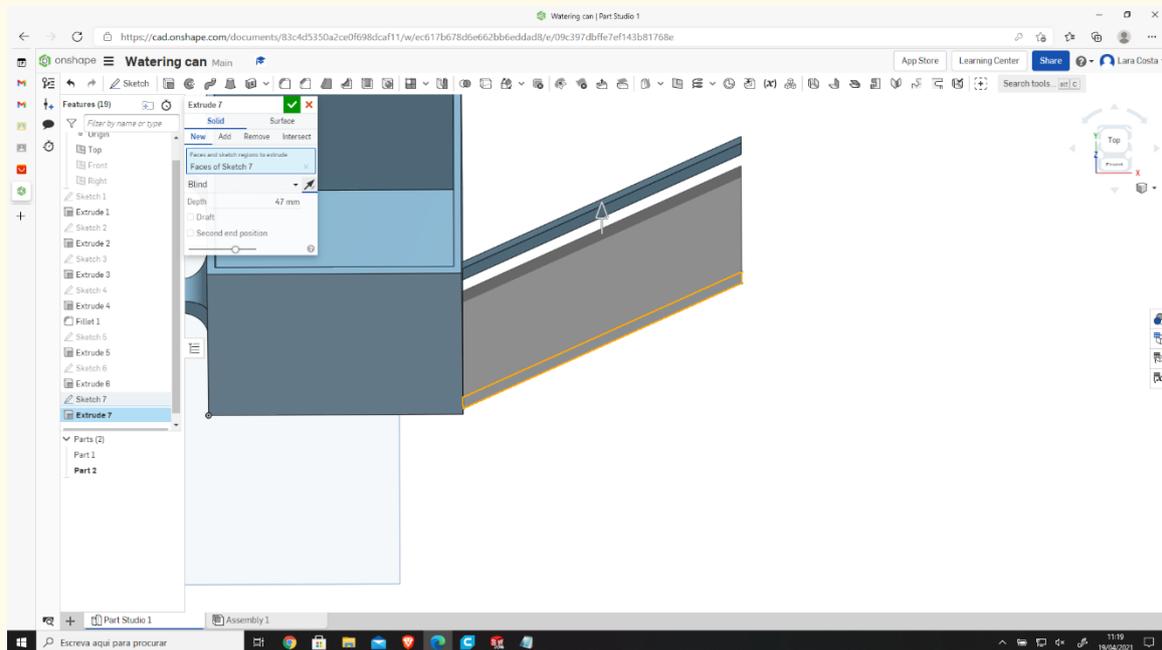
Step 36

Make an offset of the black lines (the dimension is 0mm) and draw the blue one.

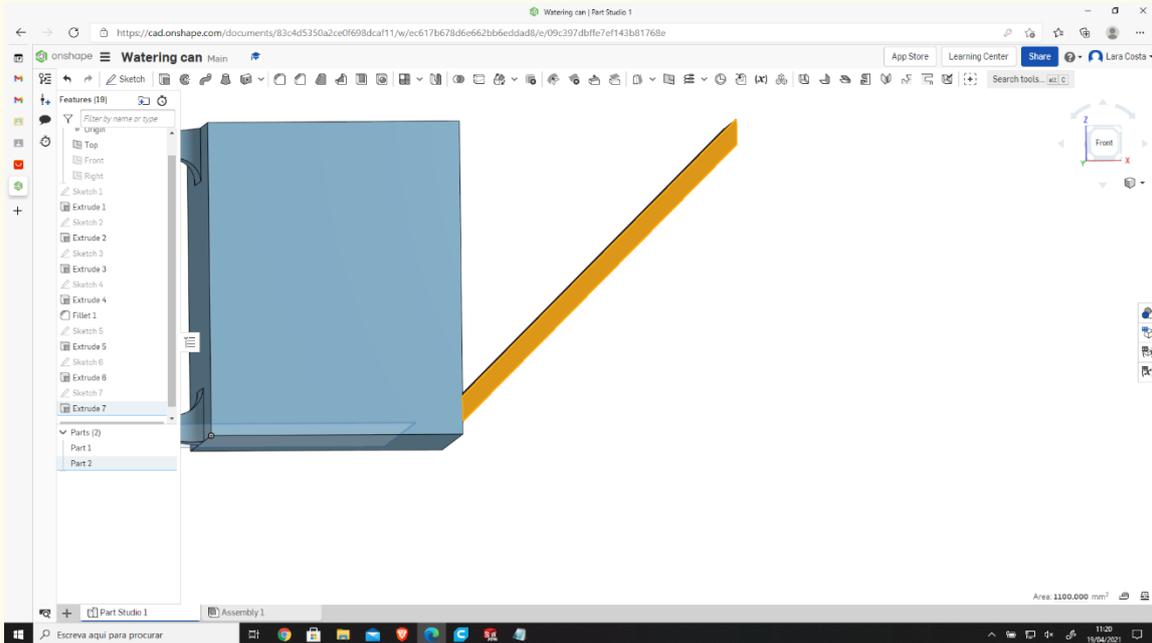


Step 37

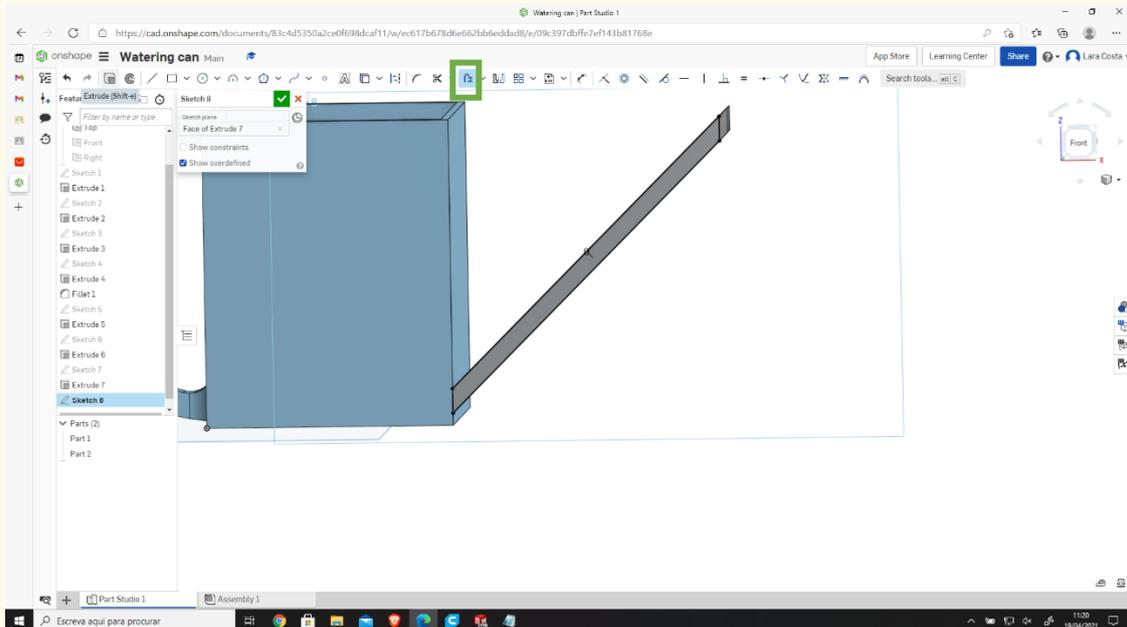
Select "Extrude" with 47mm of Depth.



Step 37 Select the orange side.

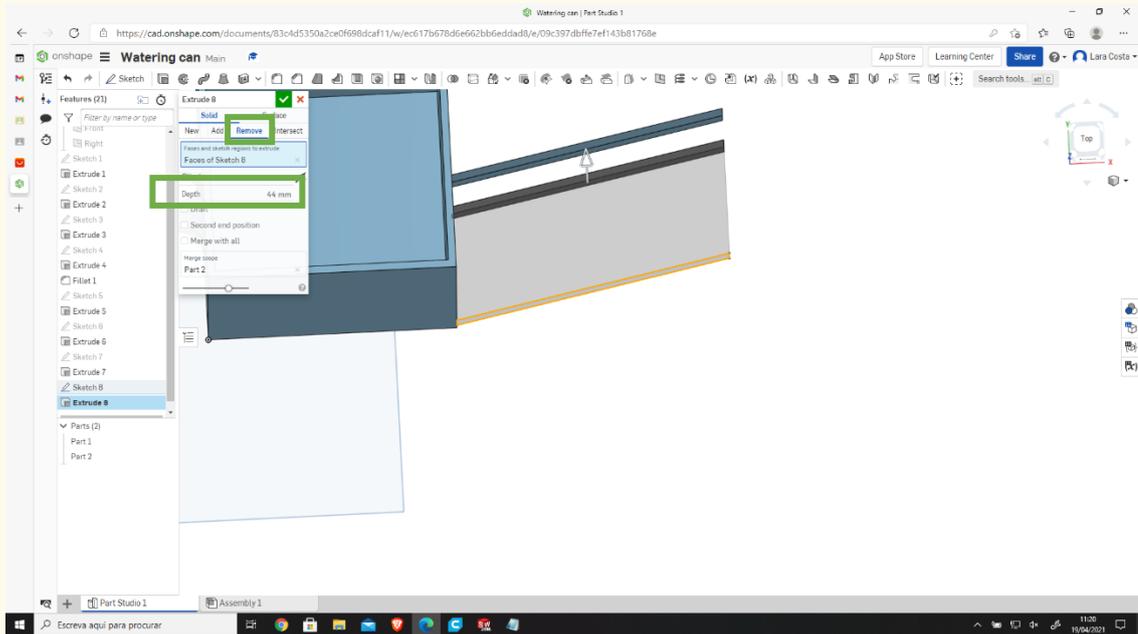


Step 38 Select "Sketch" and draw an offset (the dimension is 0mm) of the black lines.



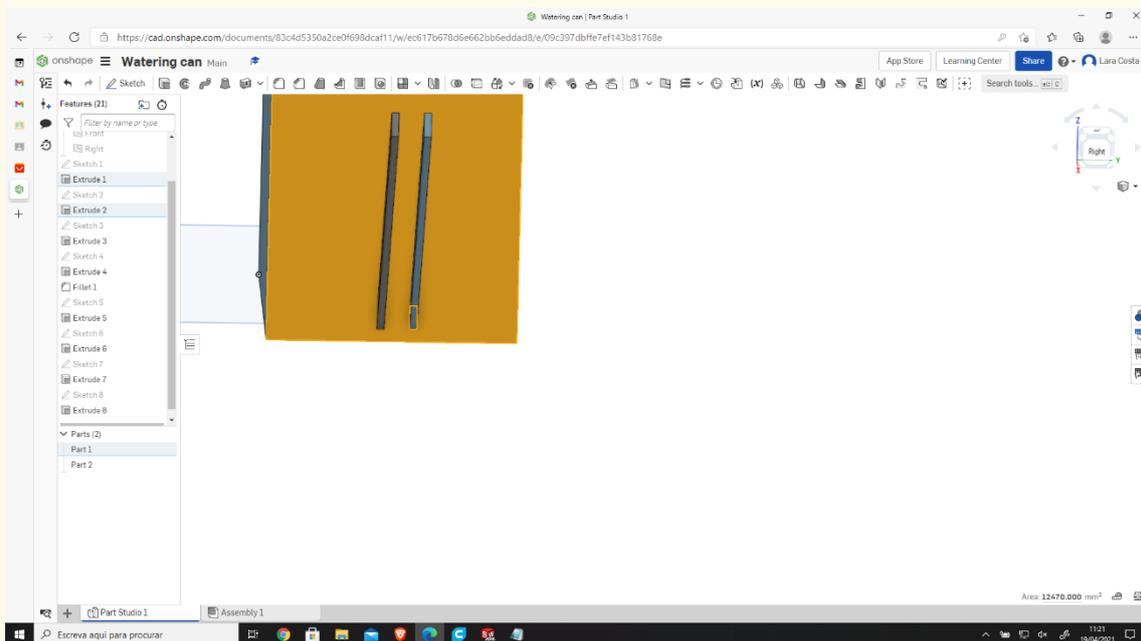
Step 39

Select "Extrude" – "Remove" with a Depth of 44mm.

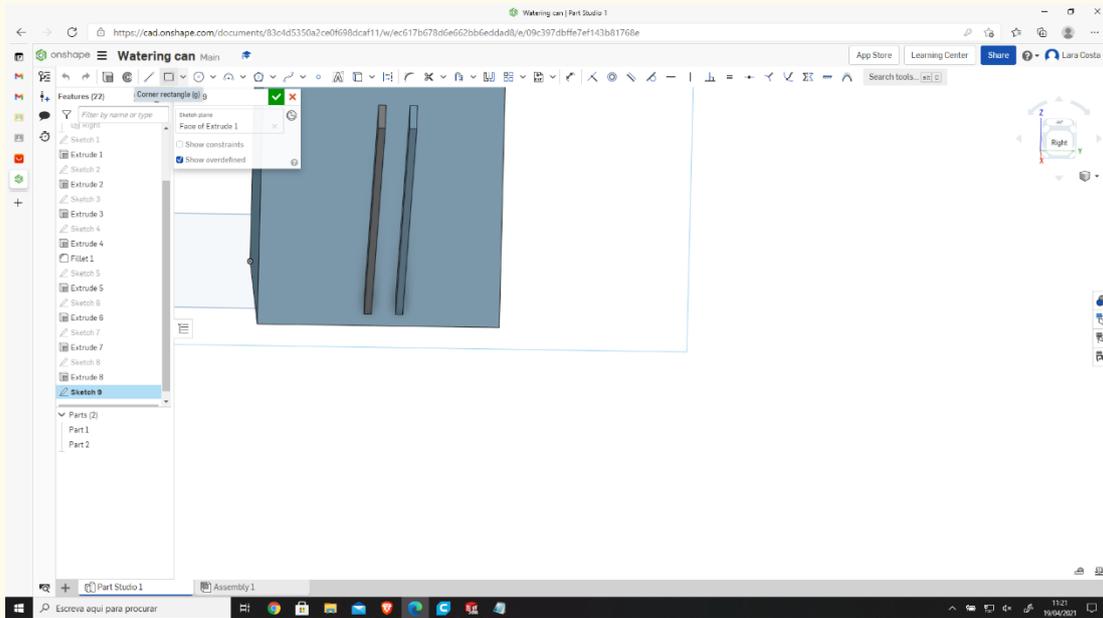


Step 40

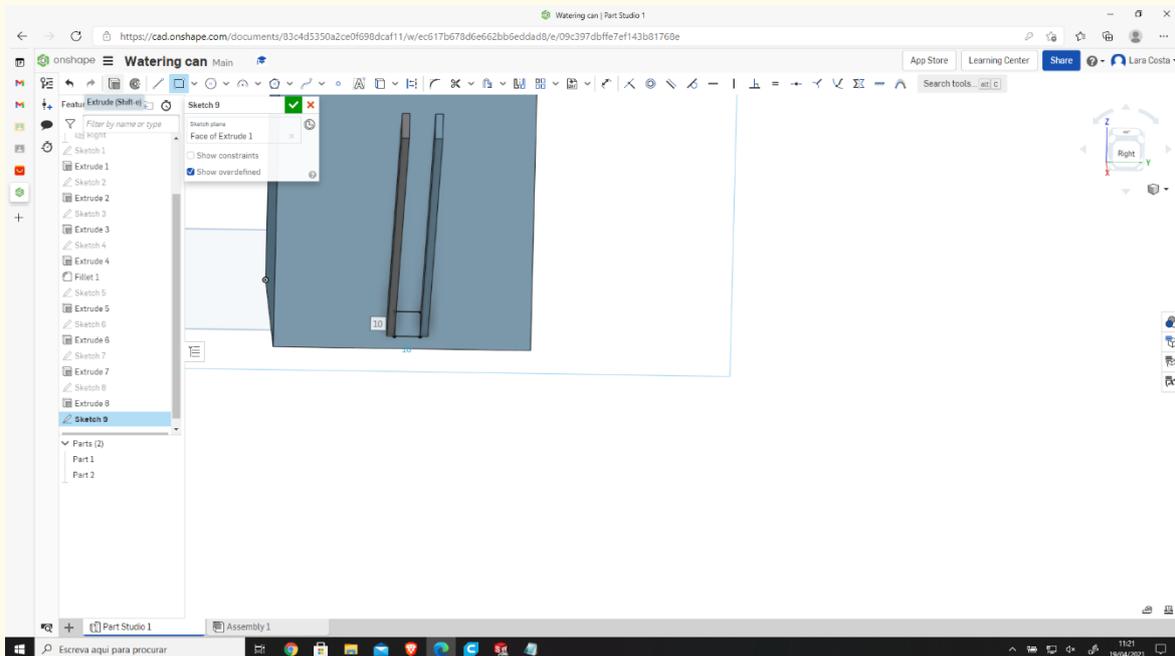
Select the orange side.



Step 41 Select "Corner rectangle."

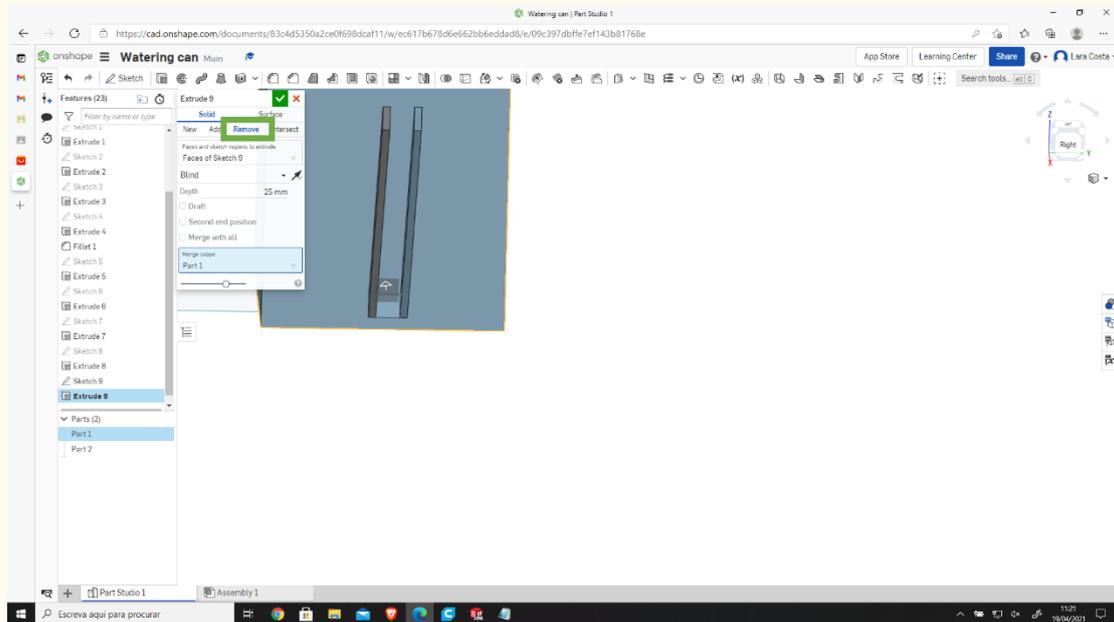


Step 42 Draw the rectangle as represented.



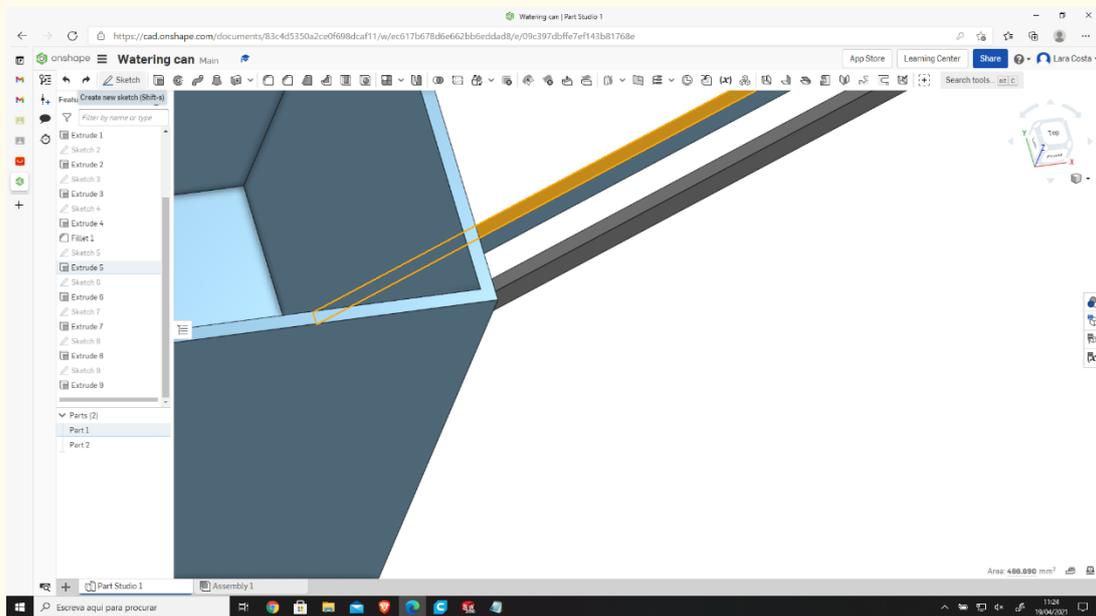
Step 43

Select "Extrude" and "Remove" (the Depth is 25mm).



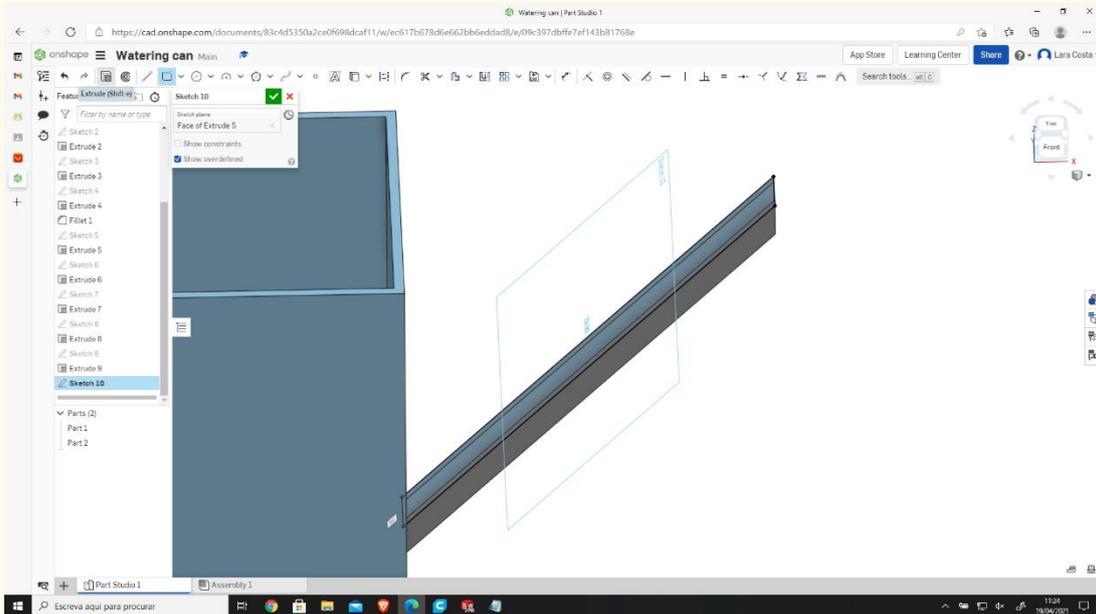
Step 44

Select the side in orange.



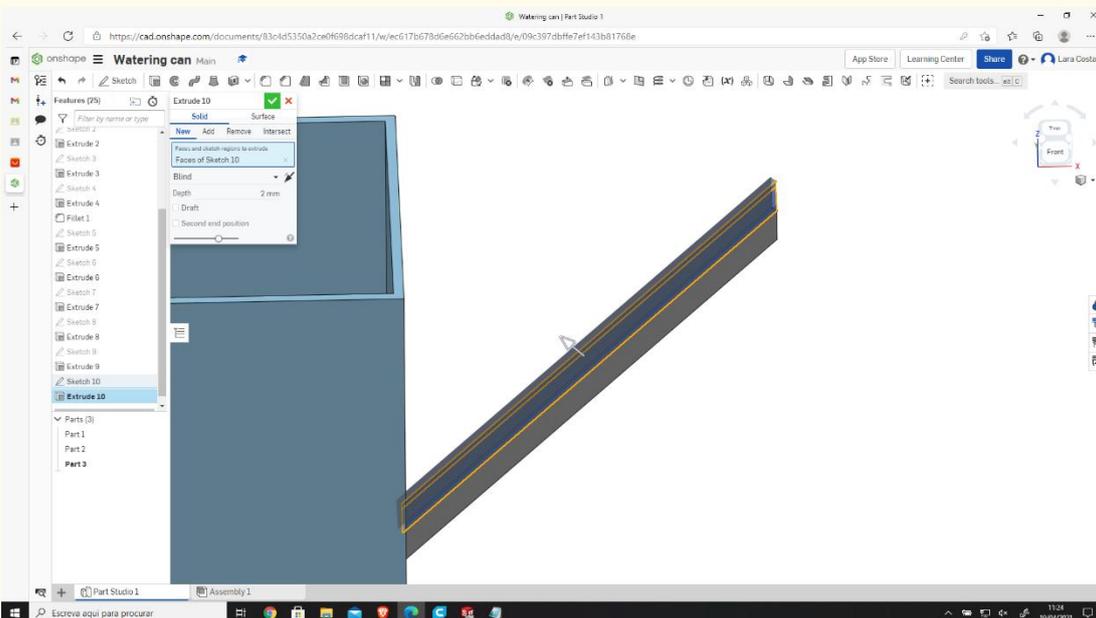
Step 45

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



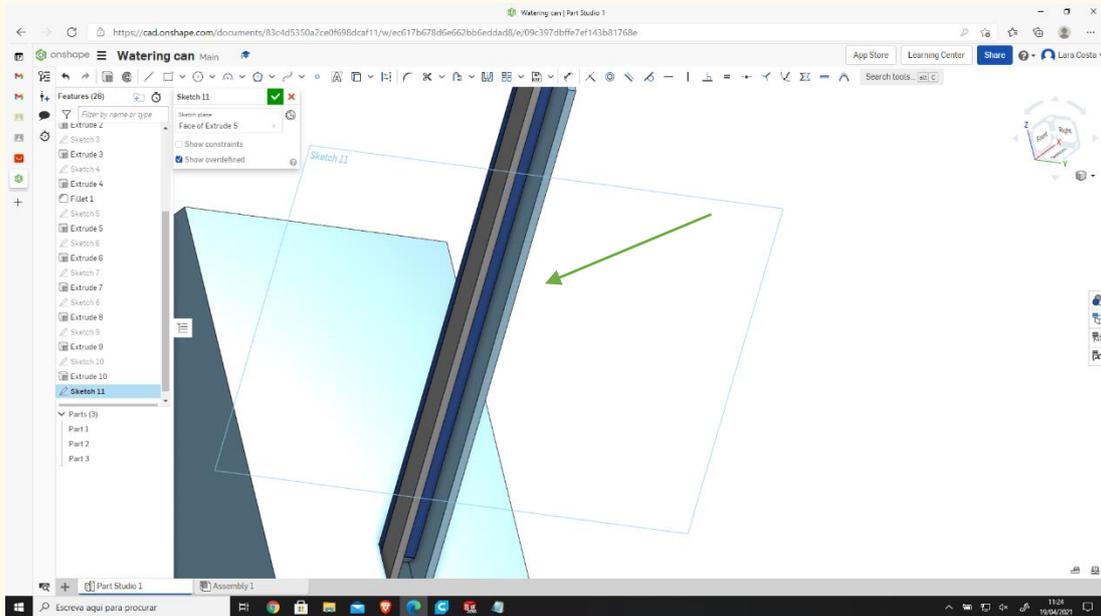
Step 46

Select "Extrude," then insert a Depth of 2mm.



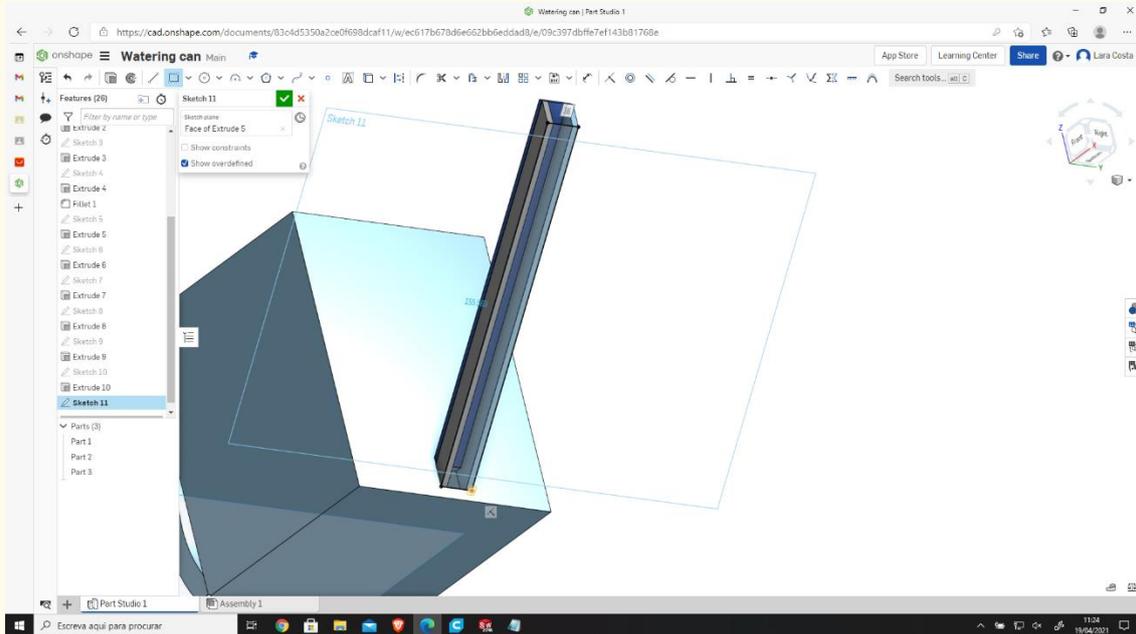
Step 47

Select the bottom (of the spout of the watering can) as shown in "Sketch 11."



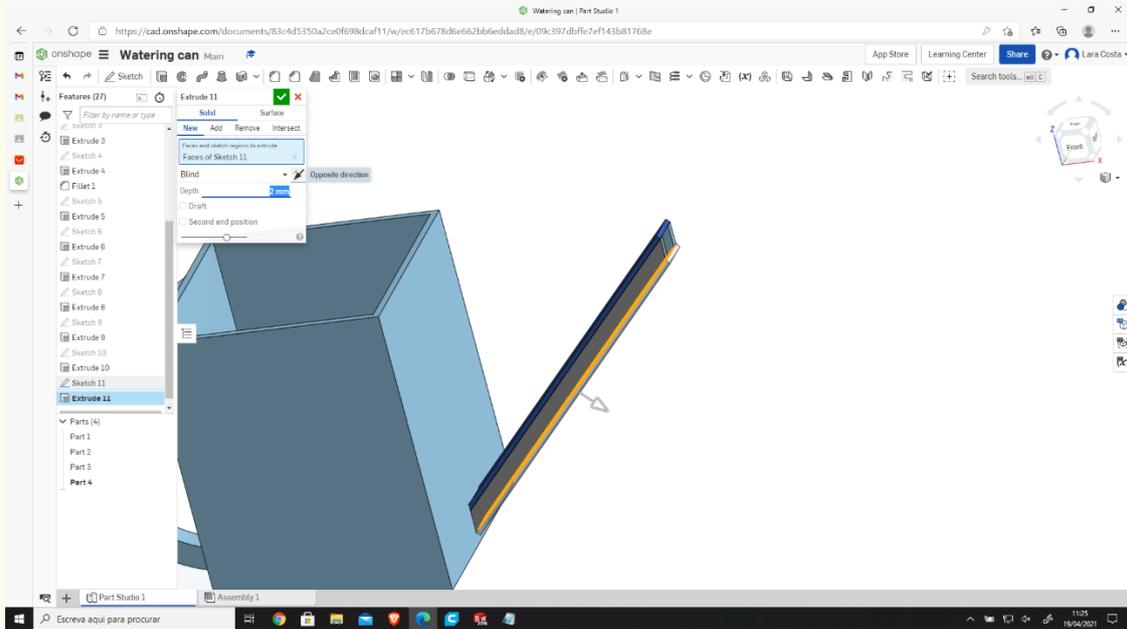
Step 48

Draw a rectangle.



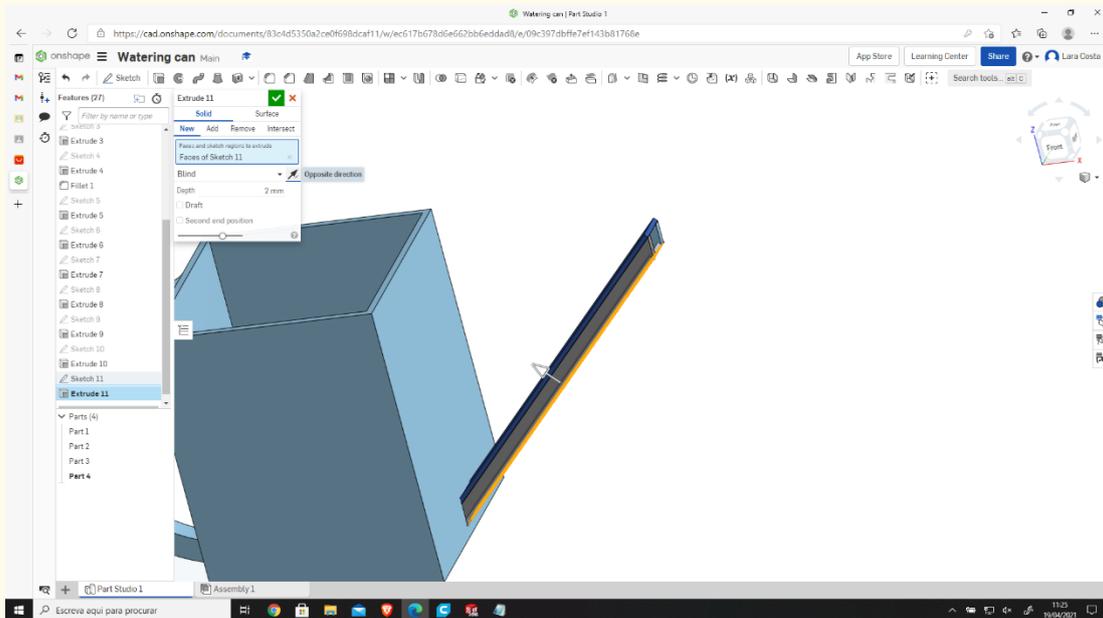
Step 49

Select "Extrude," and insert the dimension of 2mm.

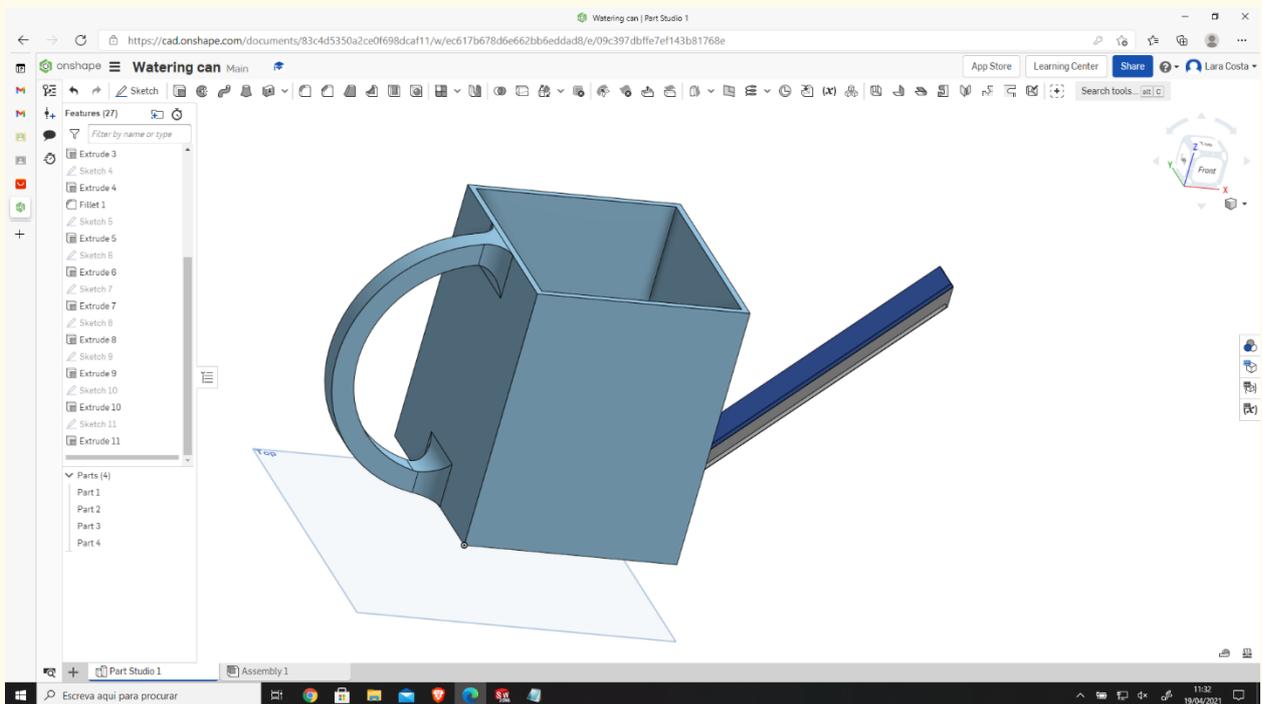
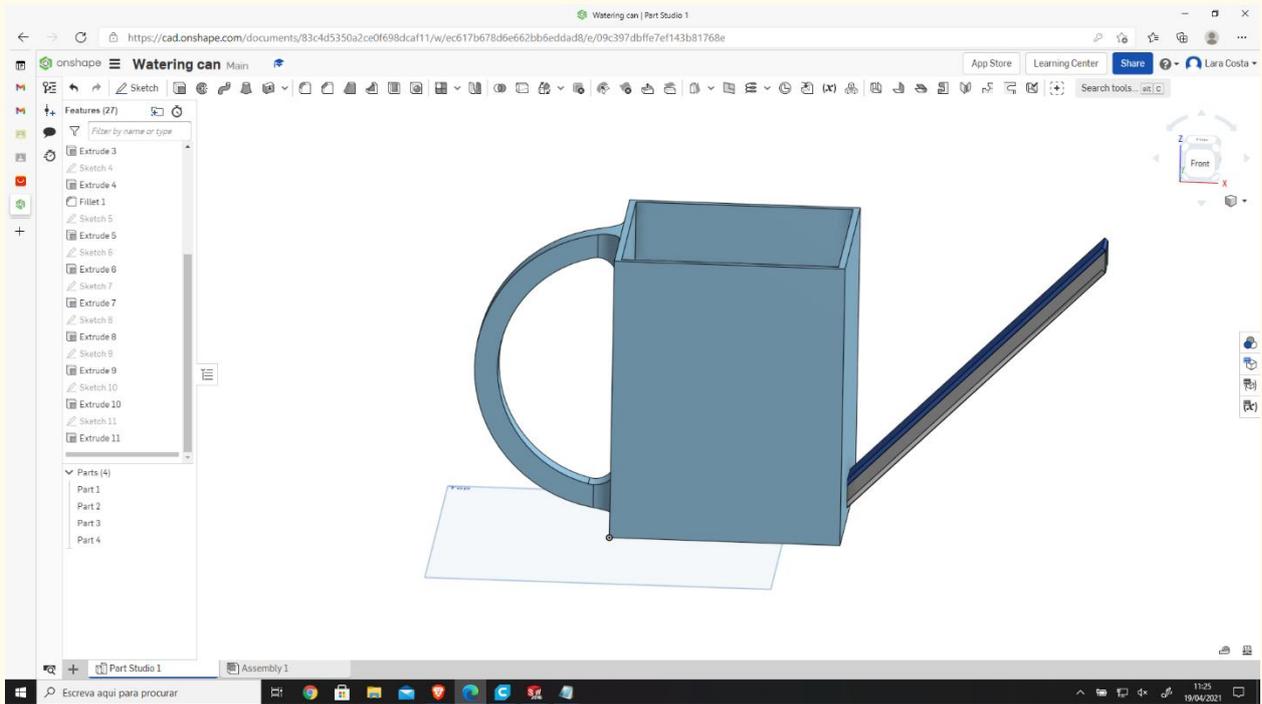


Step 50

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



The final result should look like this.



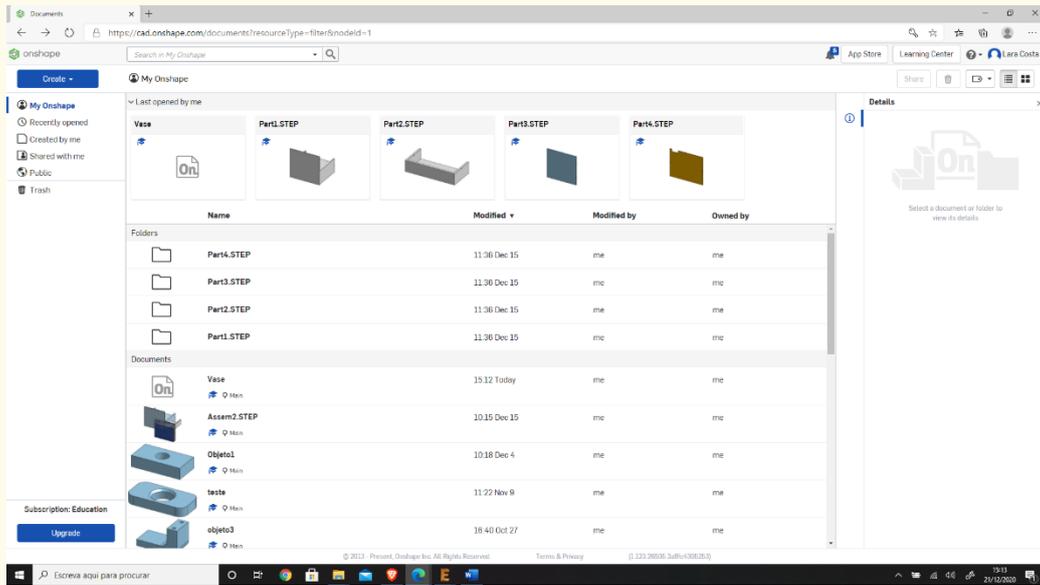
PROJECT: 3D DRAWING OF A GARDEN RAKE

- STEM field: Science, technology, and electronics.
- Indicative calendar: Any time of the year.
- Activity duration: 3 hours.
- Type of activity: 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 competences:
 - How to do a garden rake on Onshape.
- Required material and resources:
 - 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:

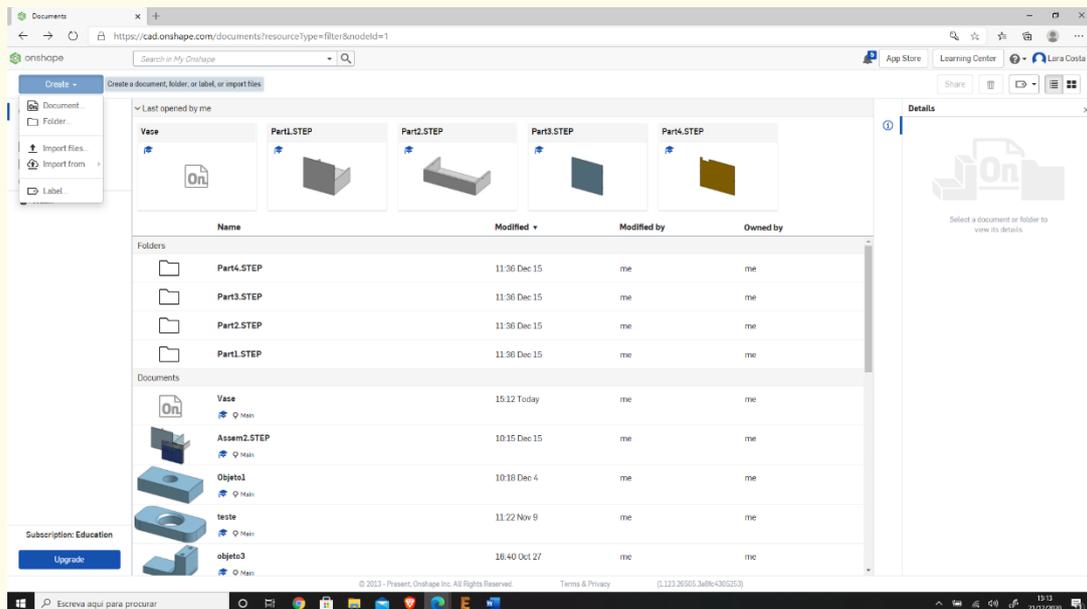
Step 1

Open the Onshape webpage.



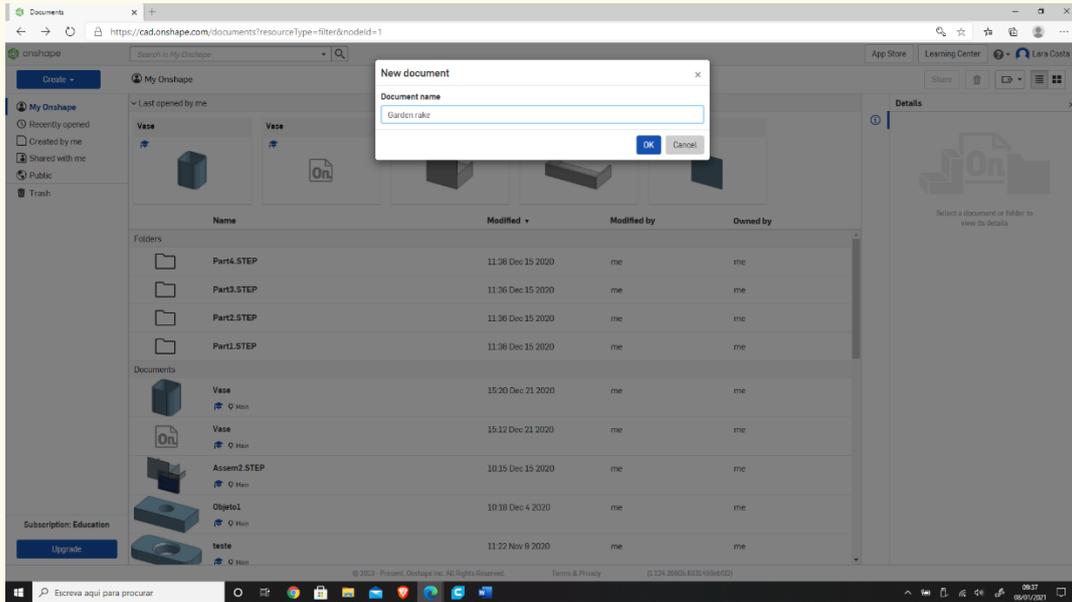
Step 2

Create a document.



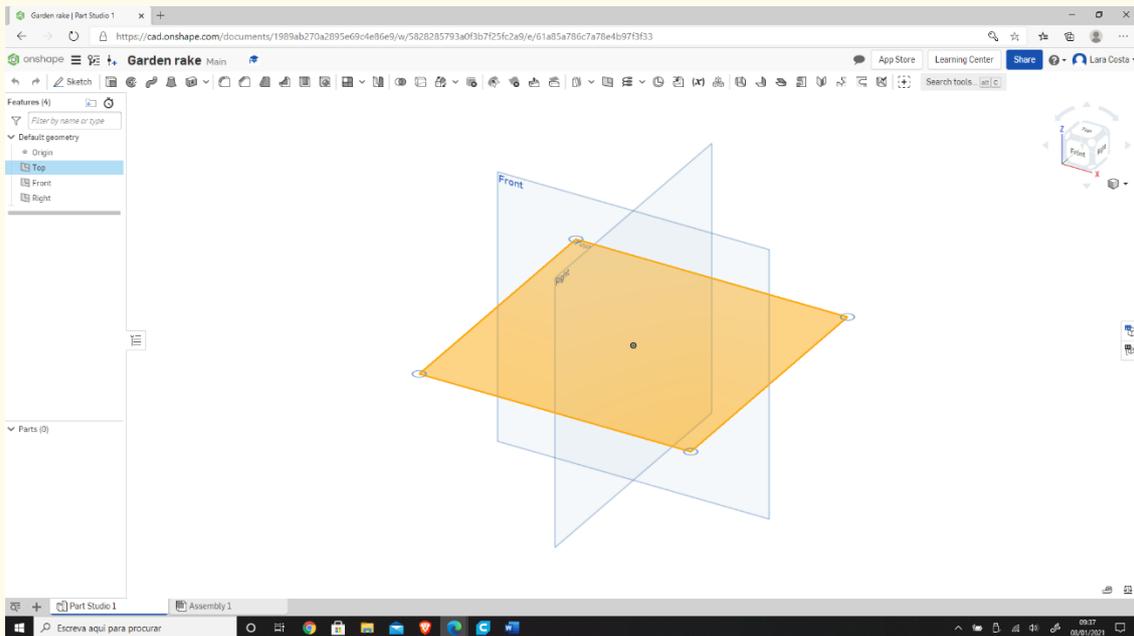
Step 3

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



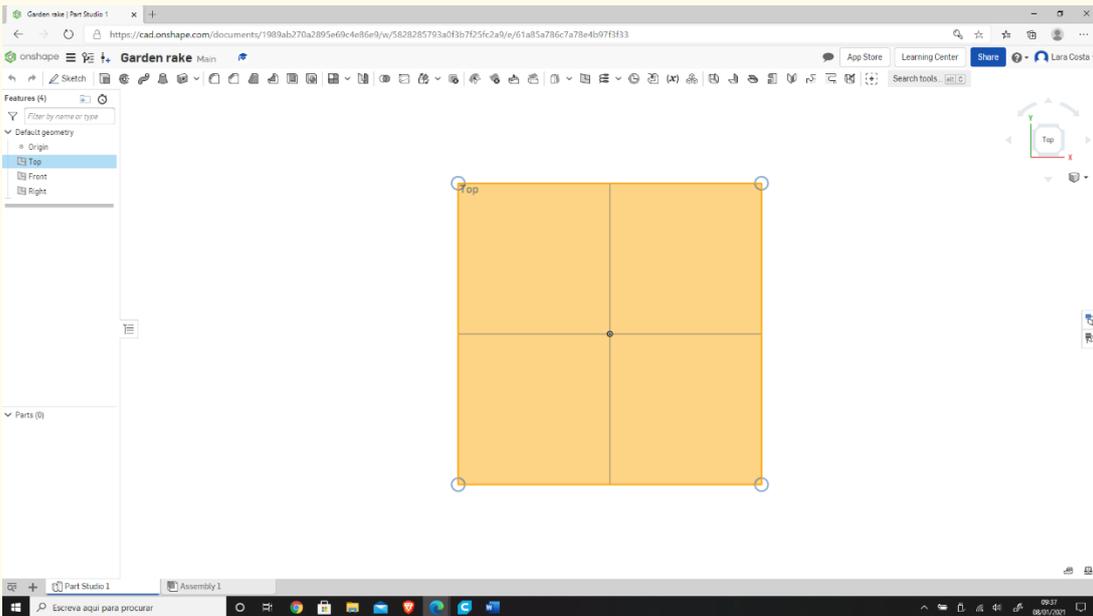
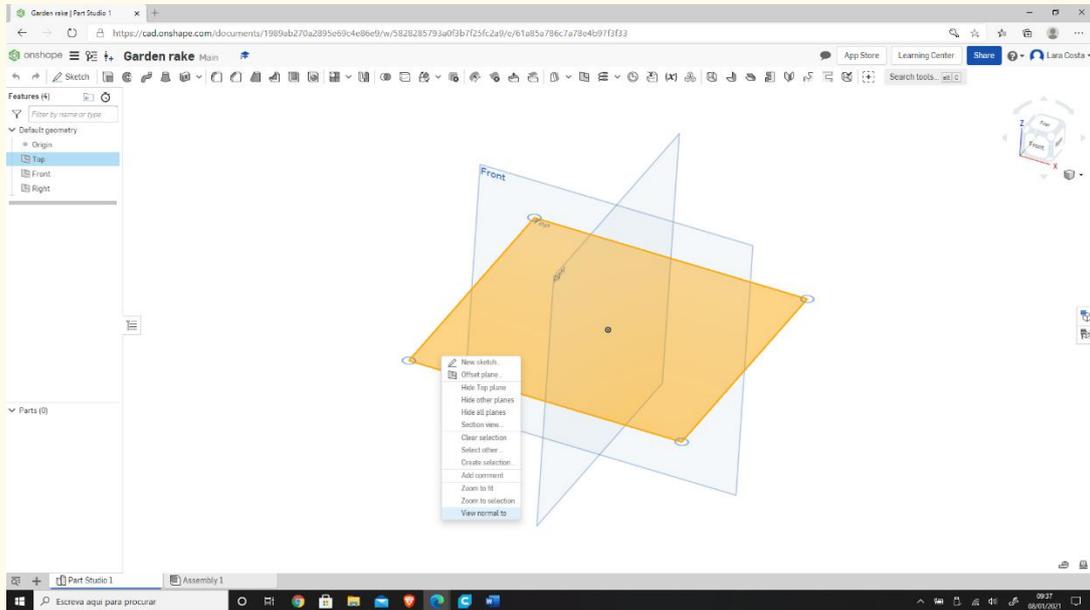
Step 4

Select the plane (top) to start drawing.



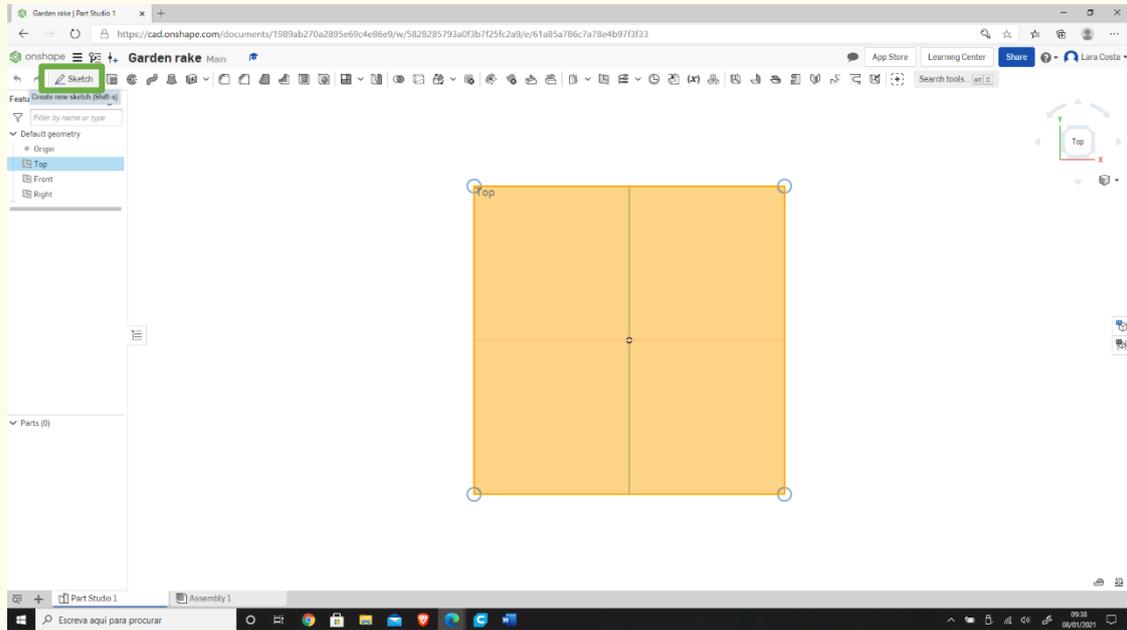
Step 5

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



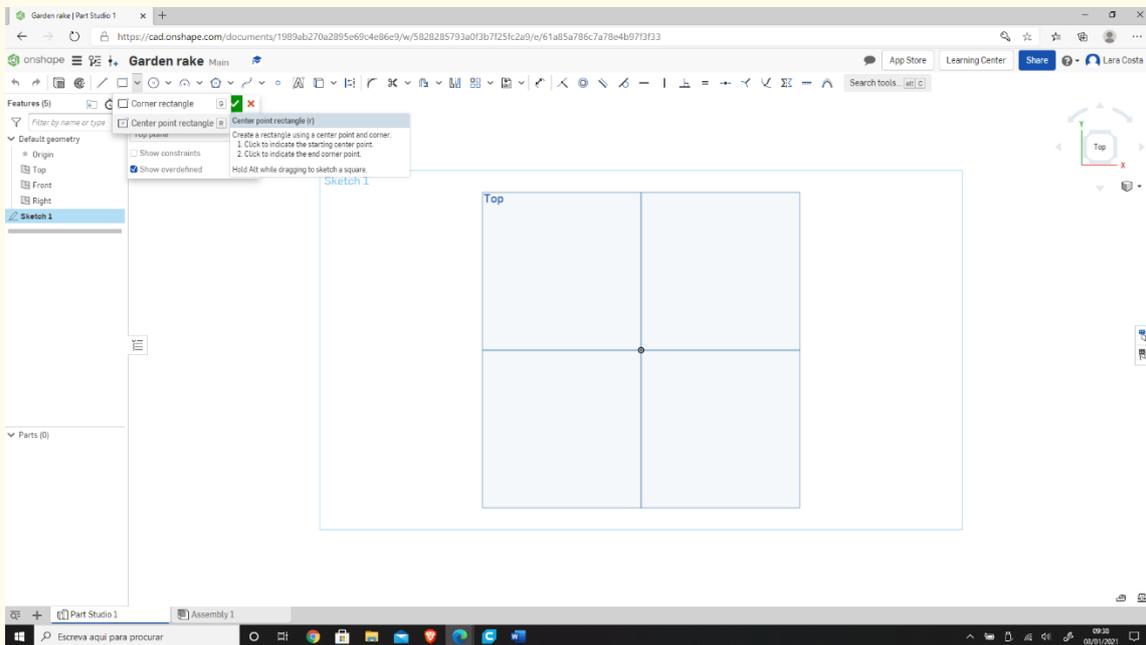
Step 6

Click "Sketch".



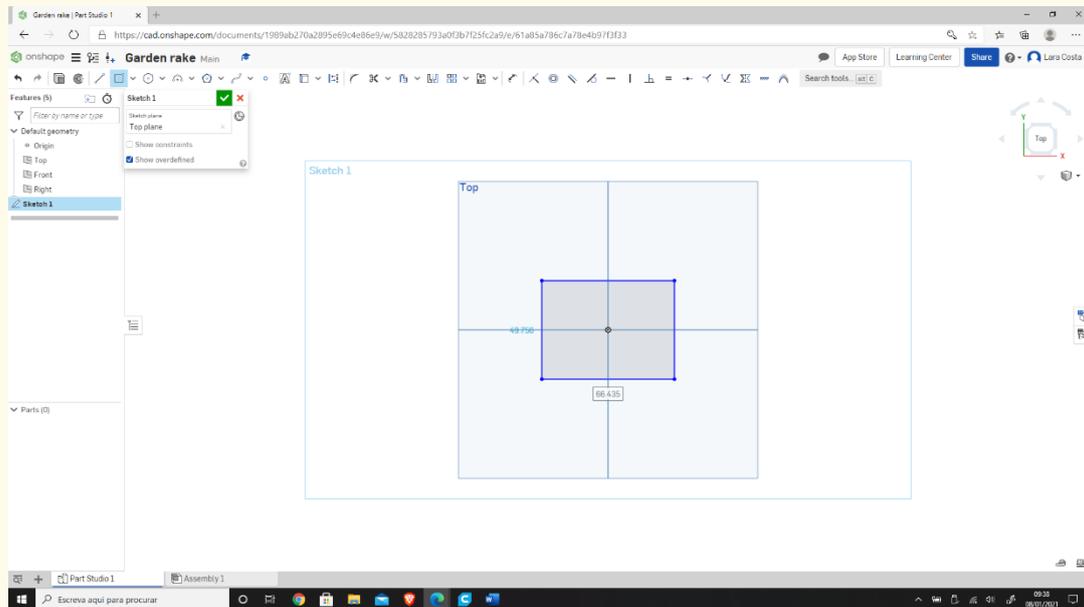
Step 7

Select the "Center point rectangle."



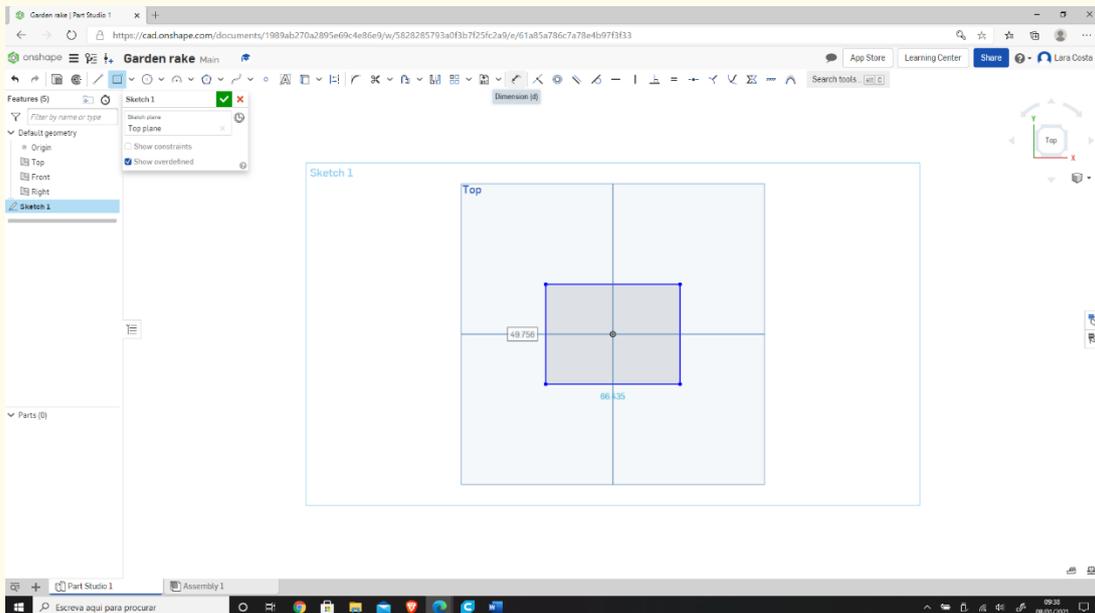
Step 8

Draw the rectangle.



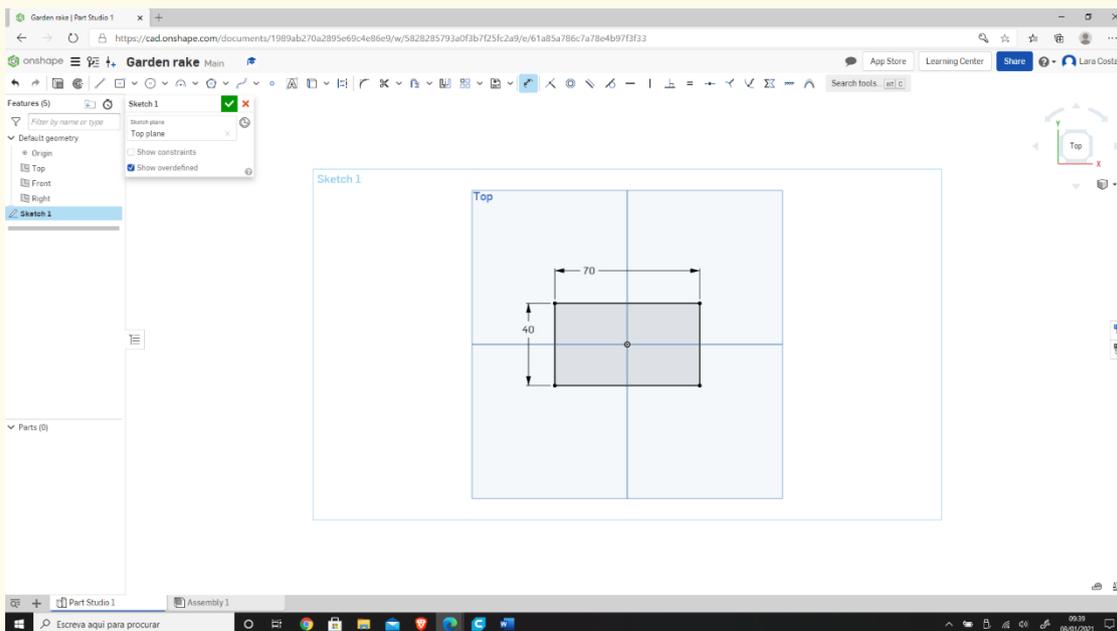
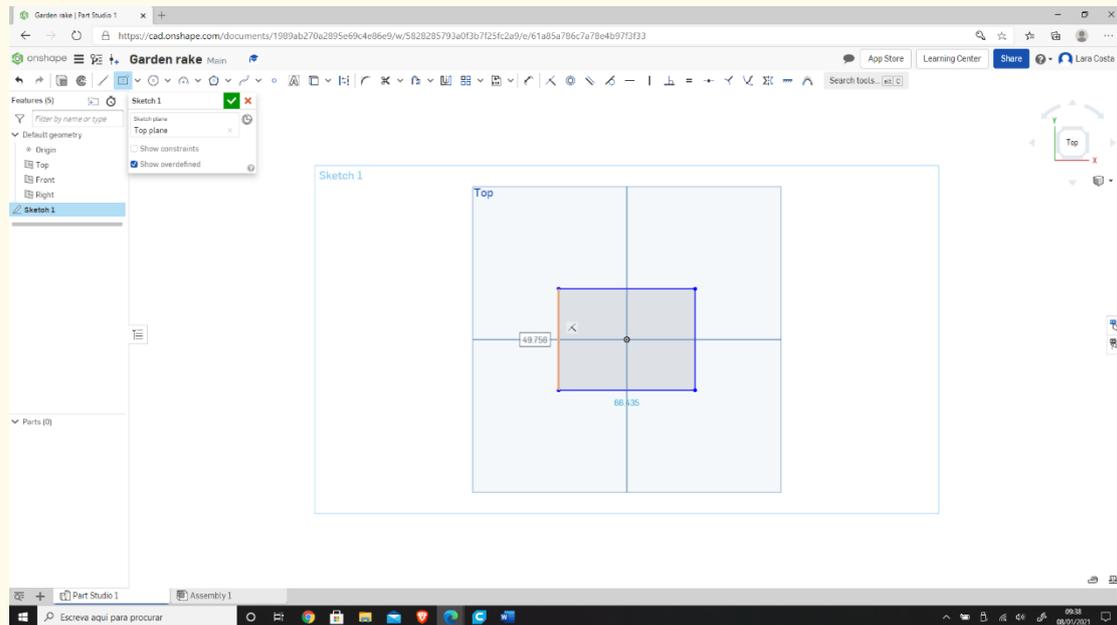
Step 9

Select "Dimension" to define the measures.



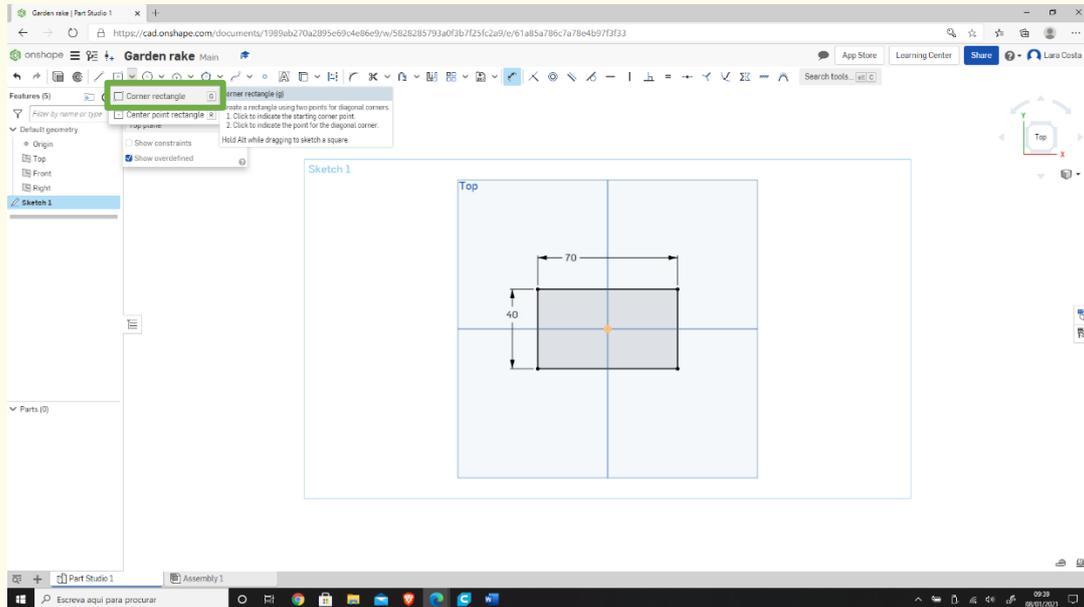
Step 10

Select the line (first left then top) and give the dimensions shown in the 2nd figure.



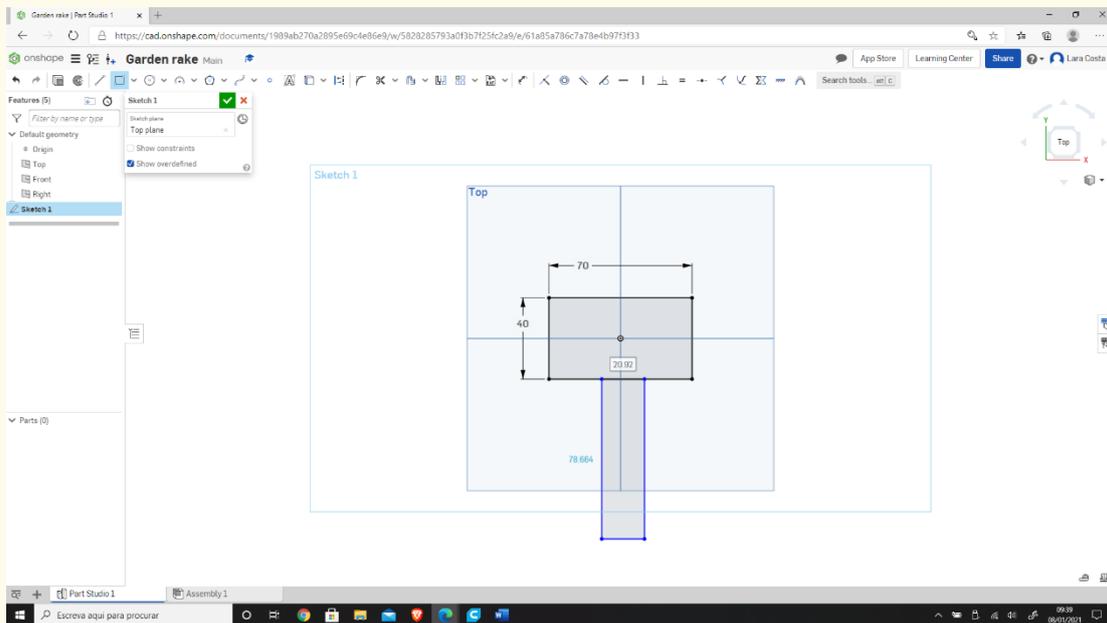
Step 11

Select "Corner rectangle."



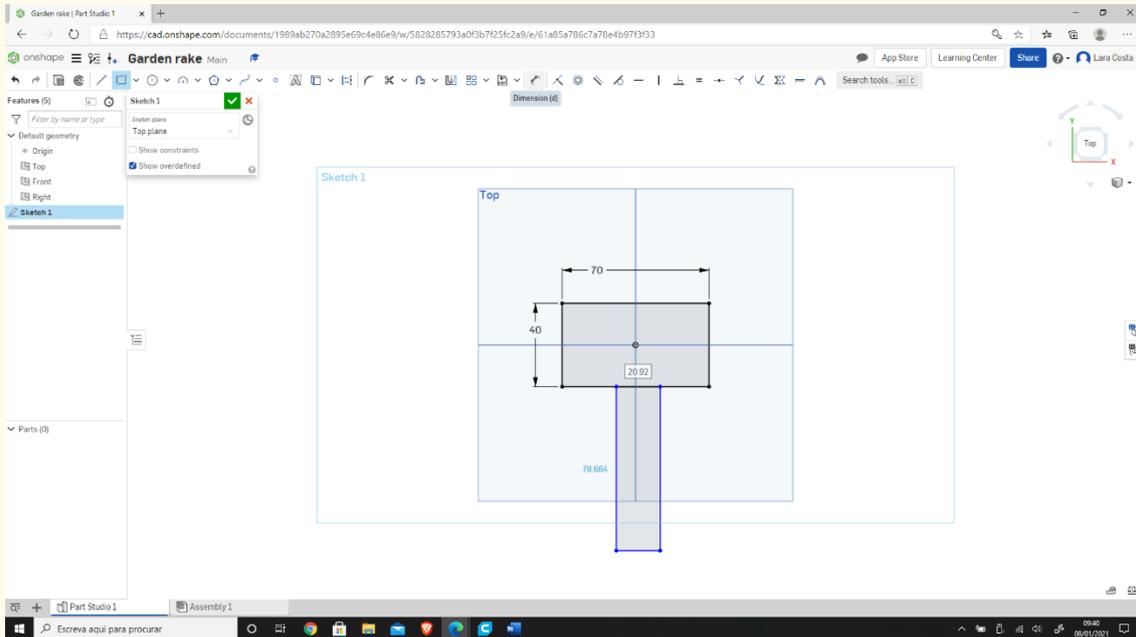
Step 12

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



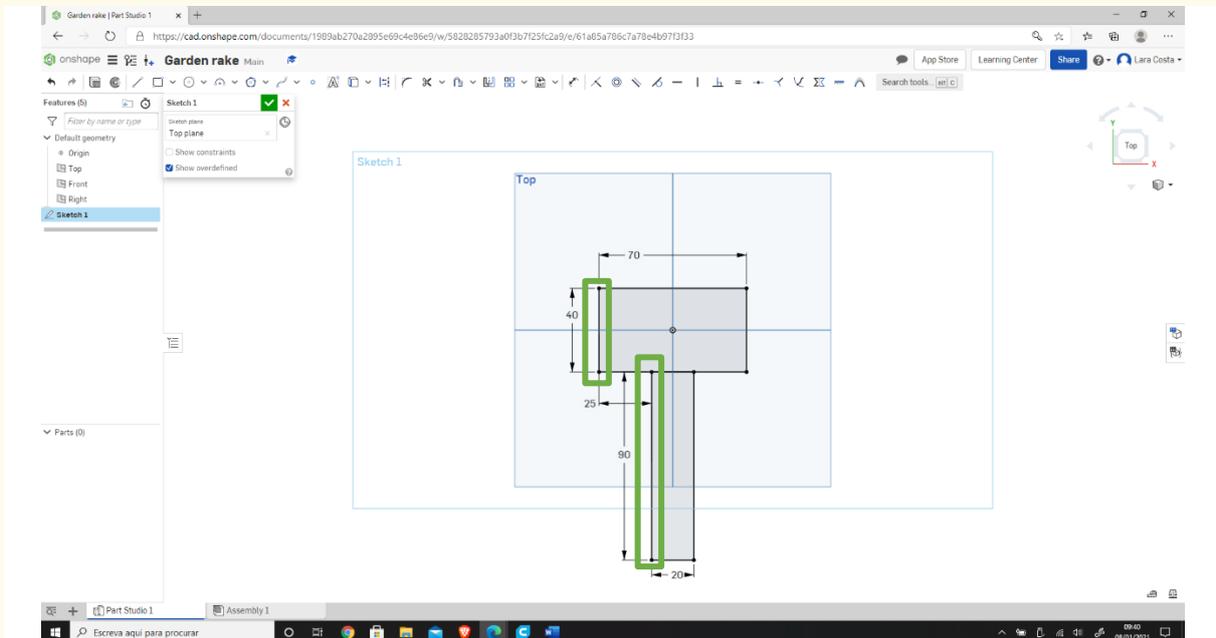
Step 13

Select "Dimension" to give the measures.



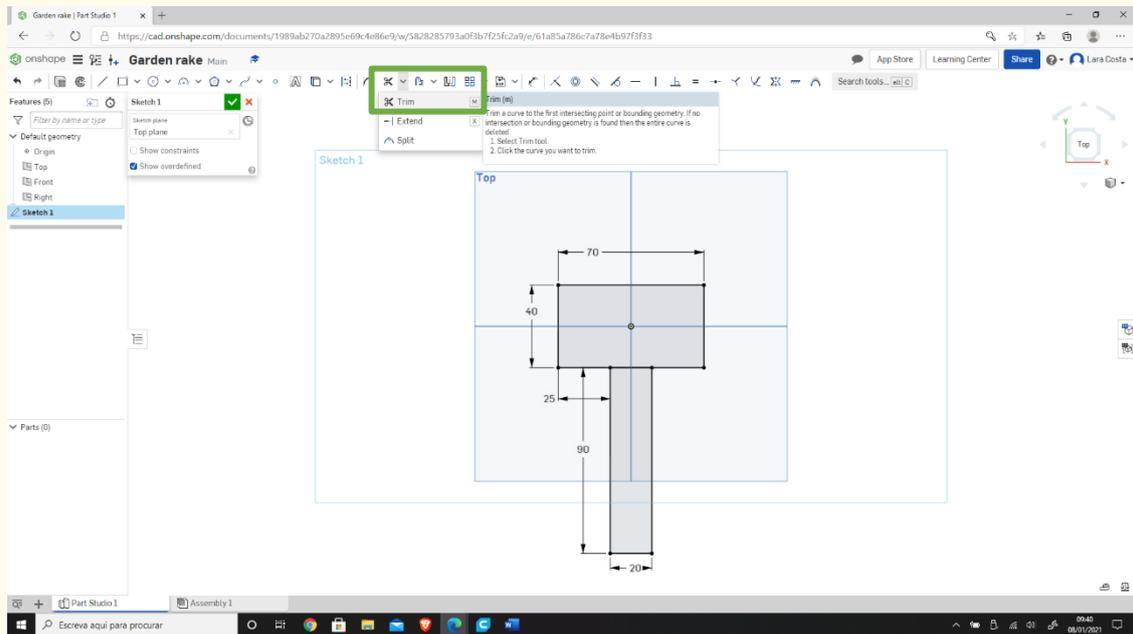
Step 14

Enter the dimensions 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).



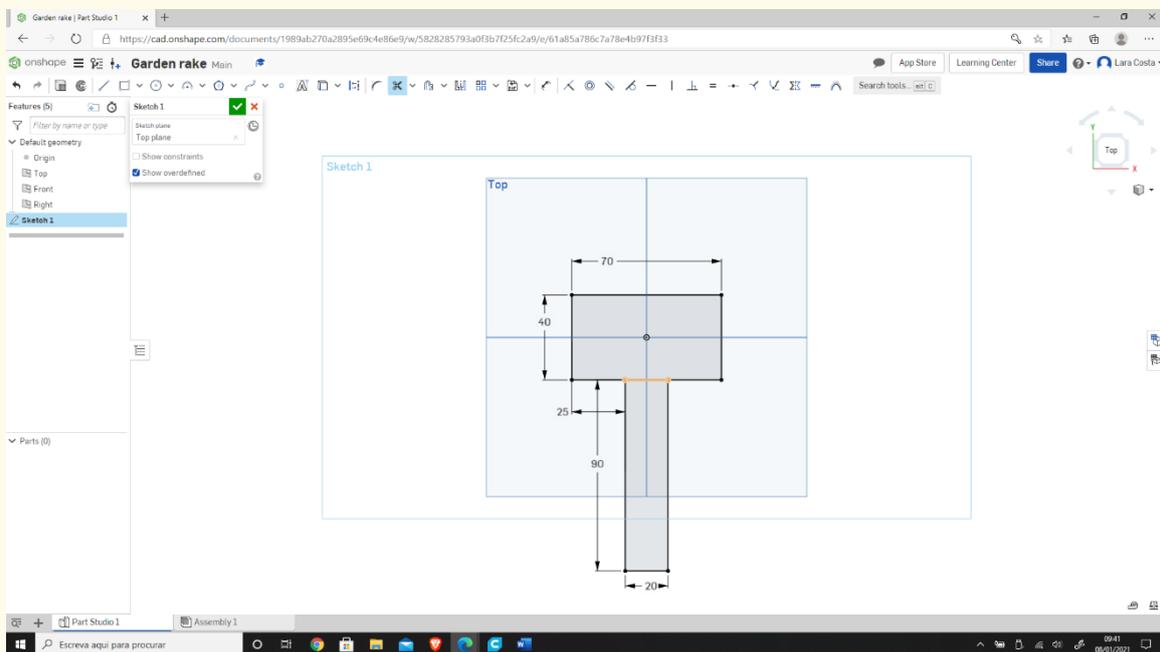
Step 15

Select "Trim."



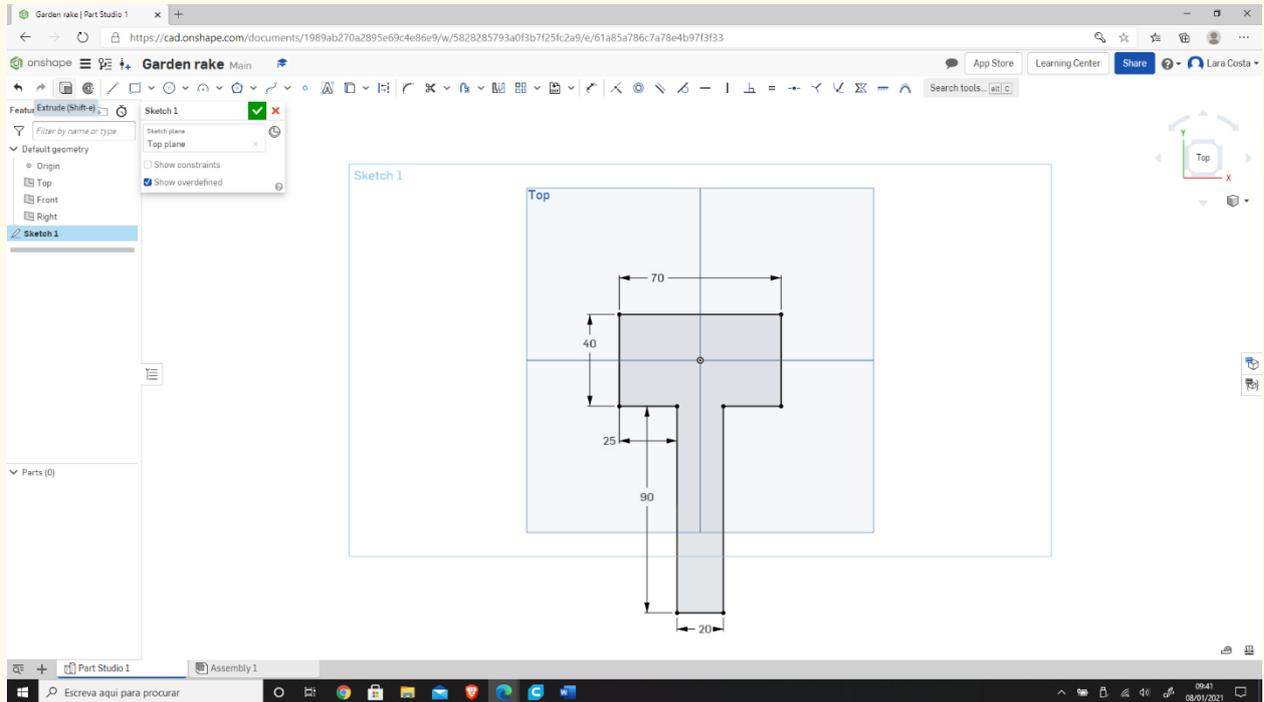
Step 16

Select the line in orange (after that the line will disappear).



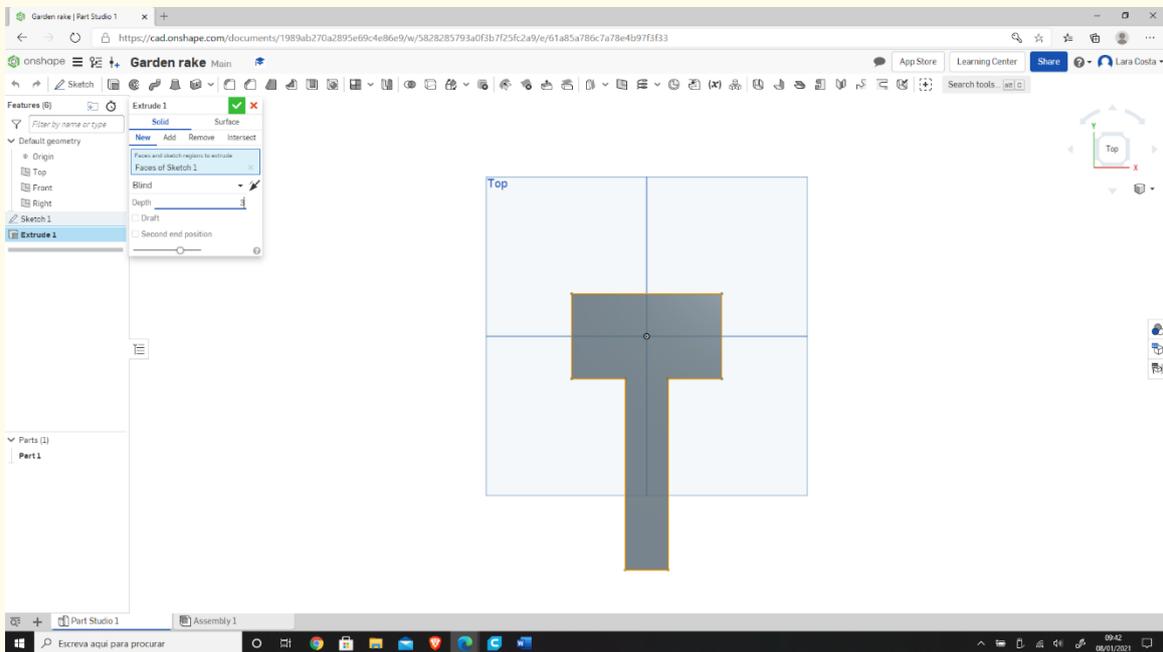
Step 17

Select "Extrude."



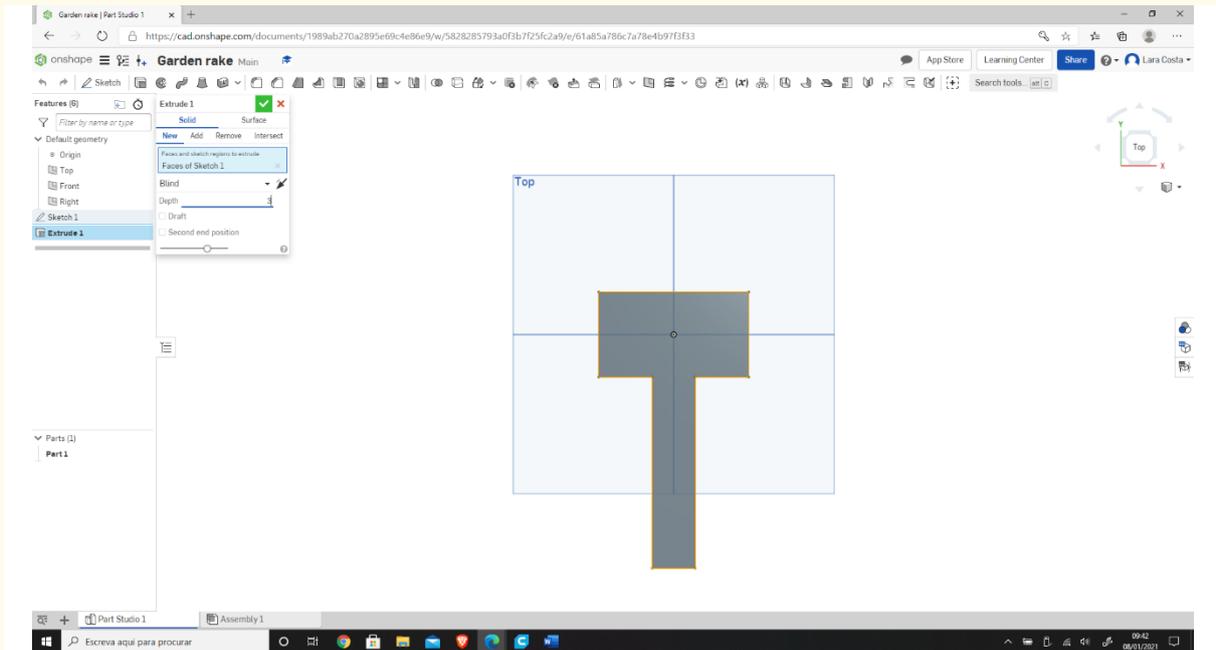
Step 18

Change the dimension to 3mm.



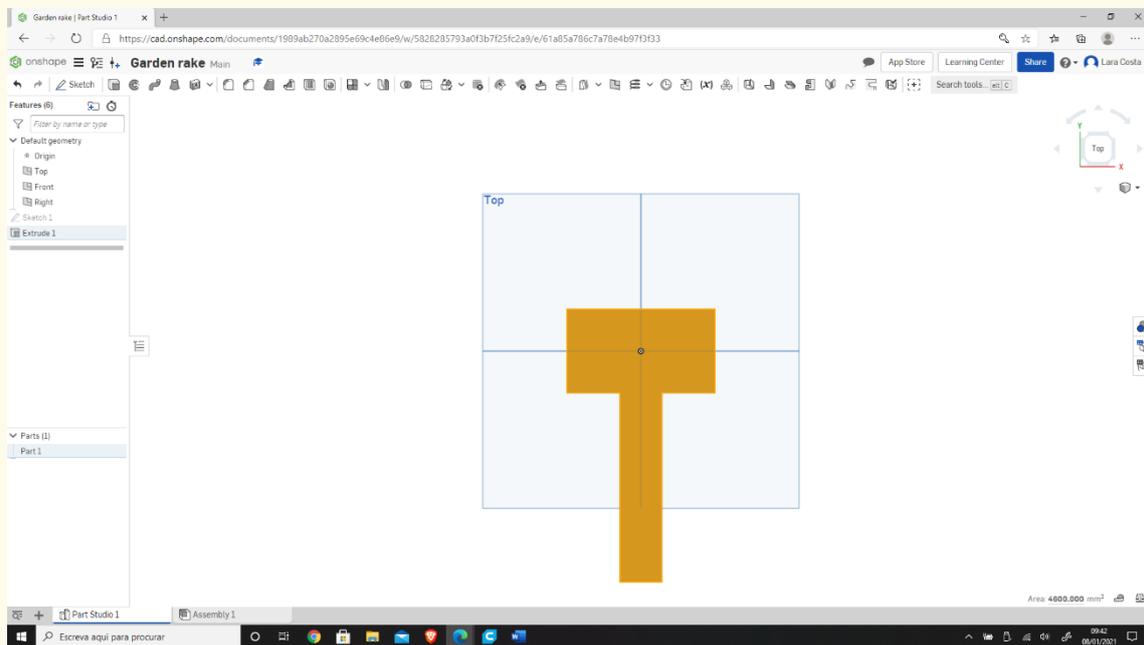
Step 19

Click on the green arrow (✓), and this should be the result:



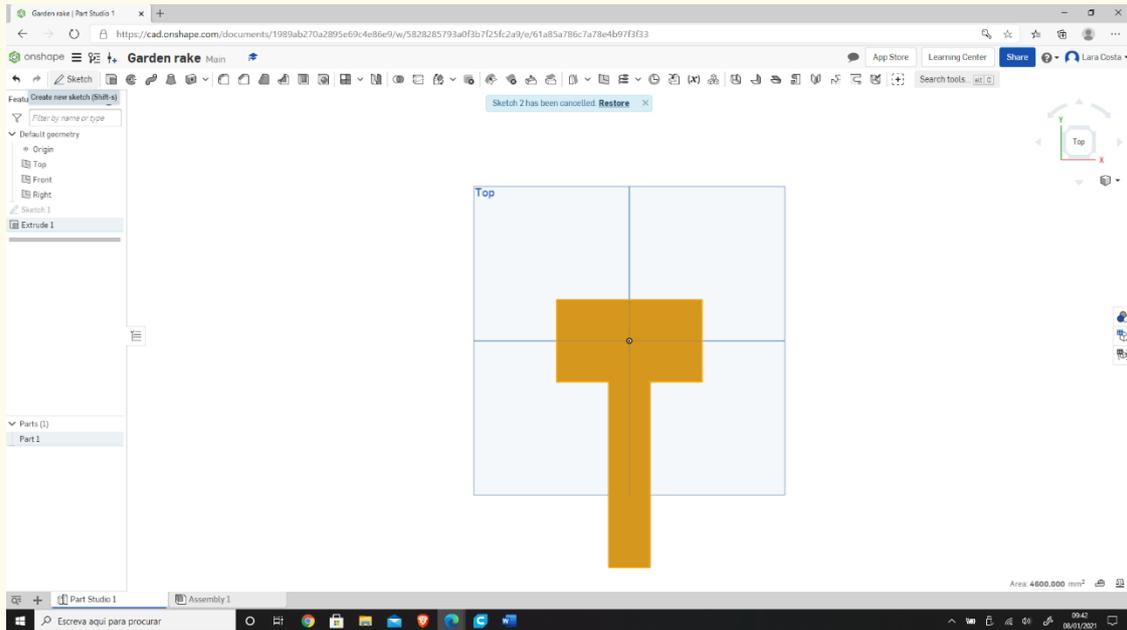
Step 20

Select the "Front."



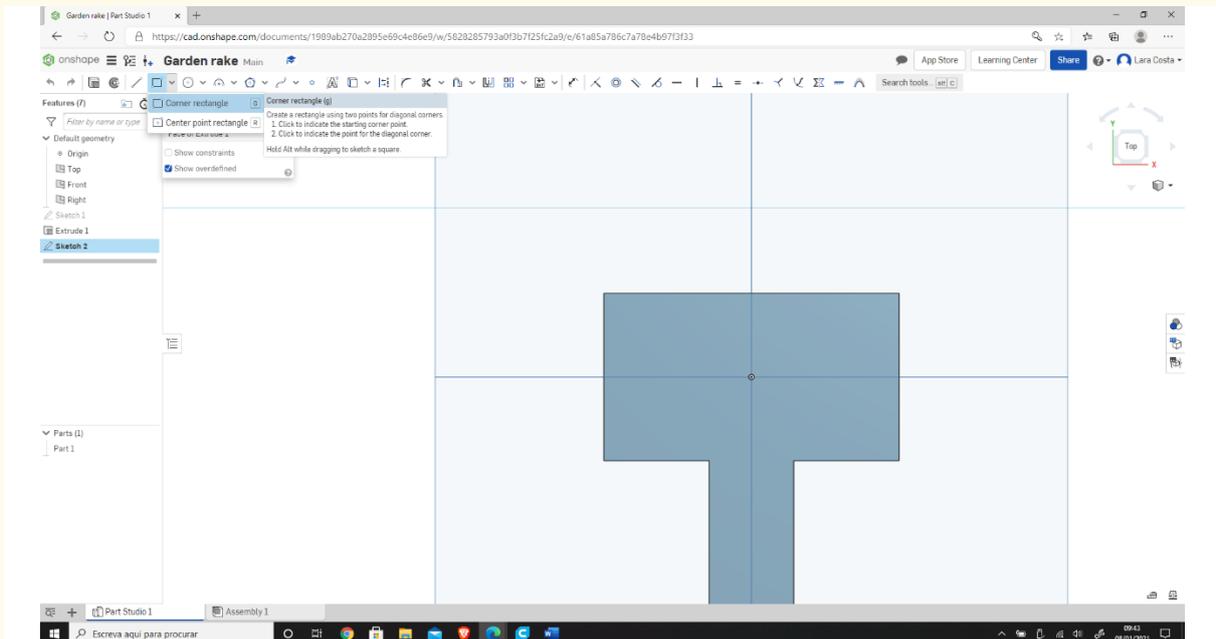
Step 21

Click "Sketch."



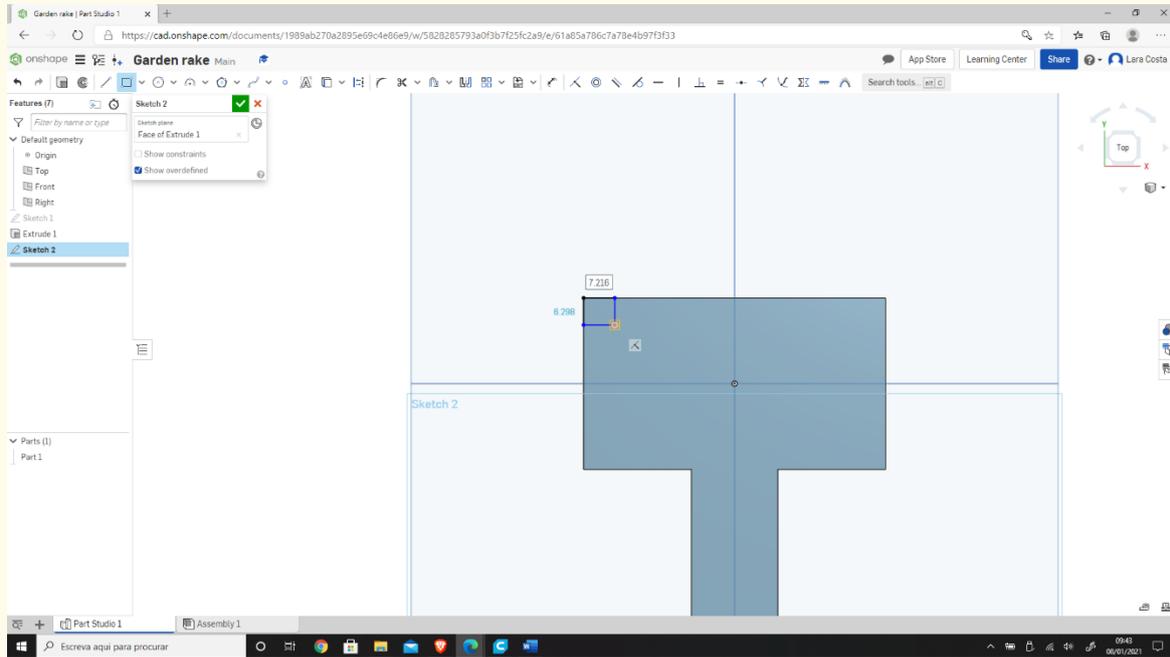
Step 22

Select "Corner rectangle."



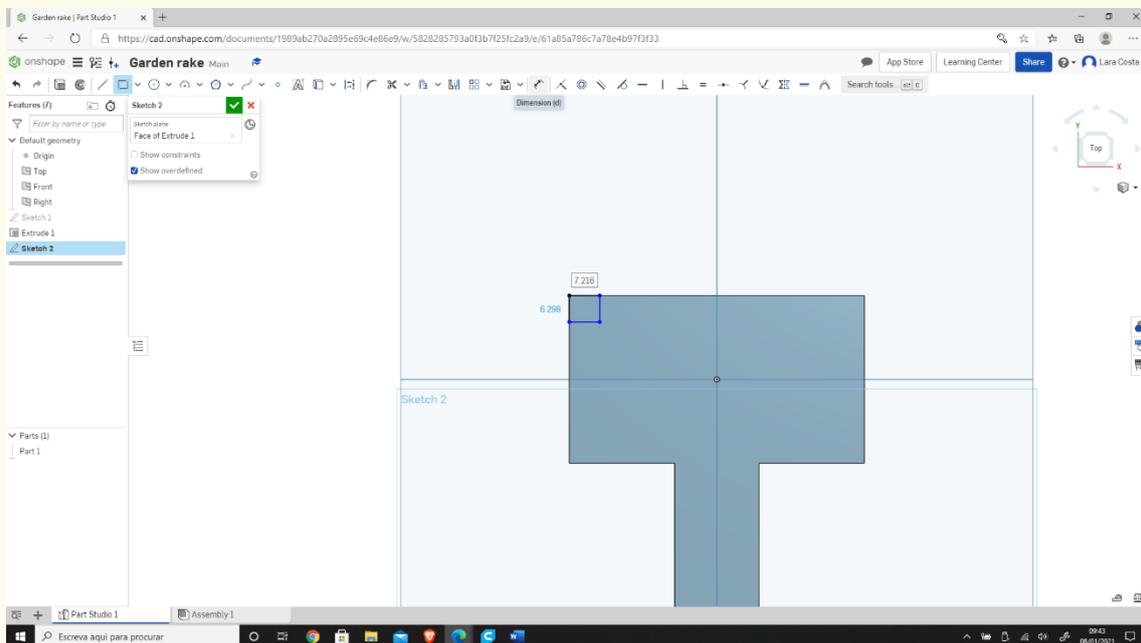
Step 23

Draw a similar rectangle as shown below.



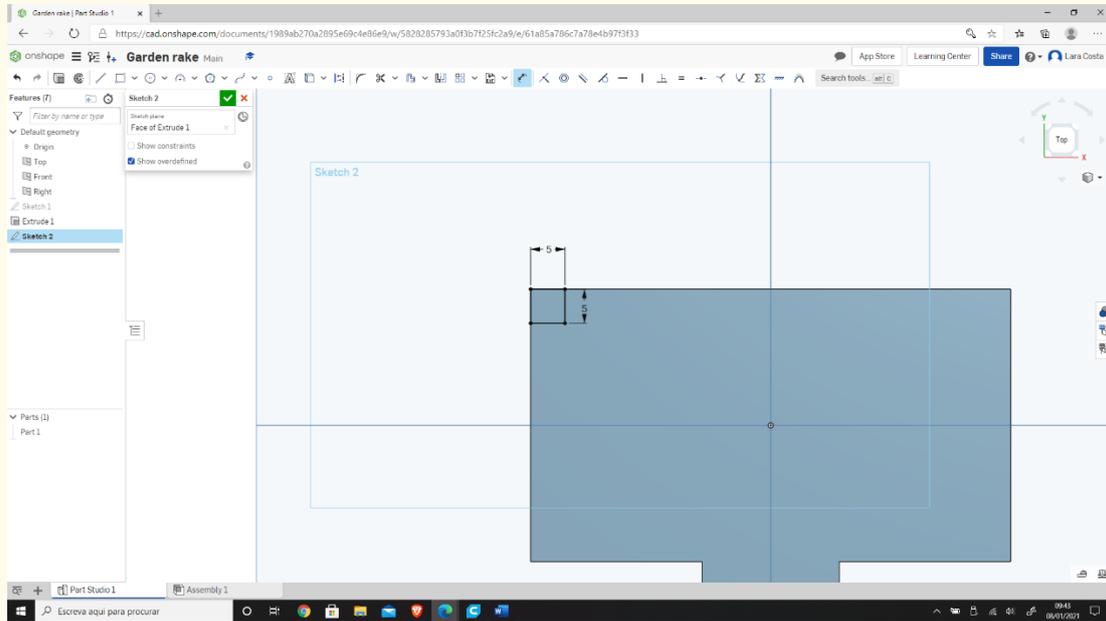
Step 24

Select "Dimension" to give the dimensions.



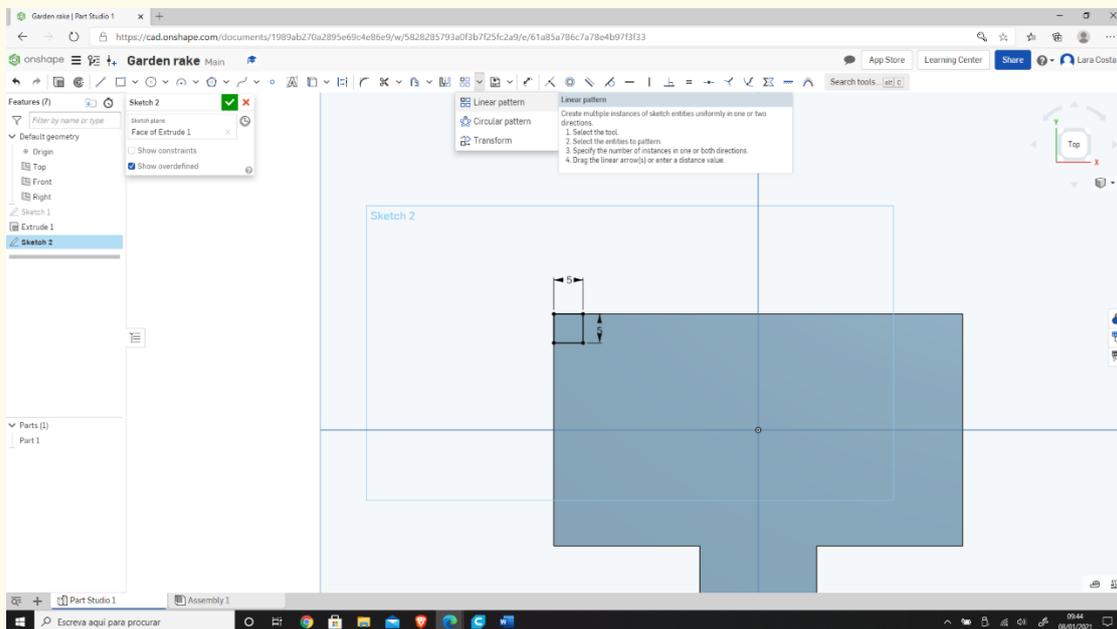
Step 25

Enter the dimensions as shown in the image below.

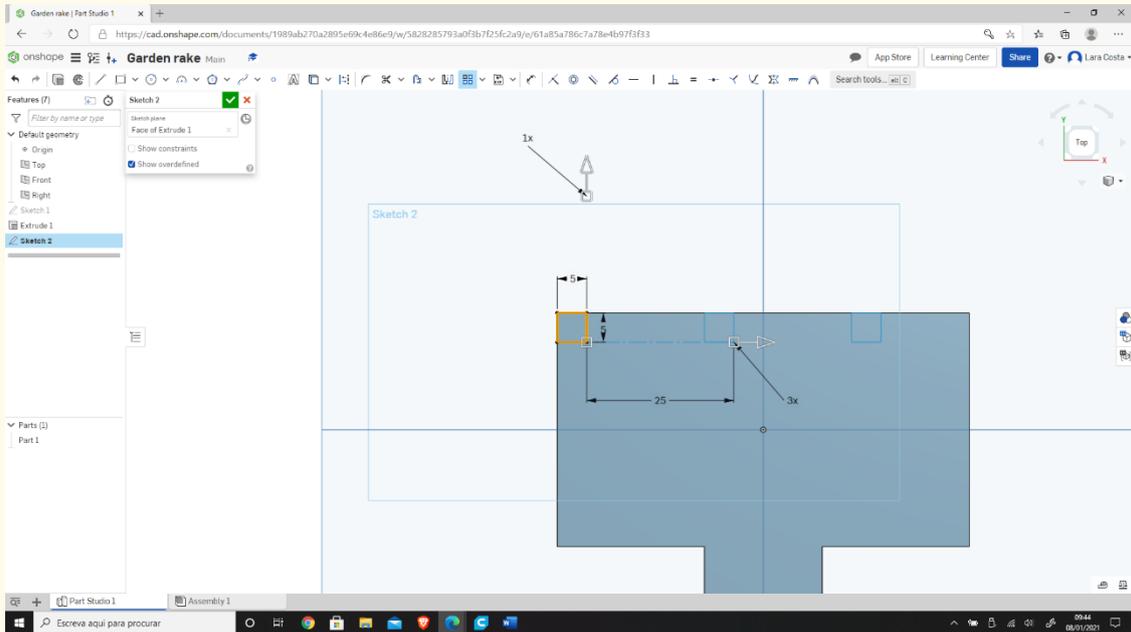


Step 26

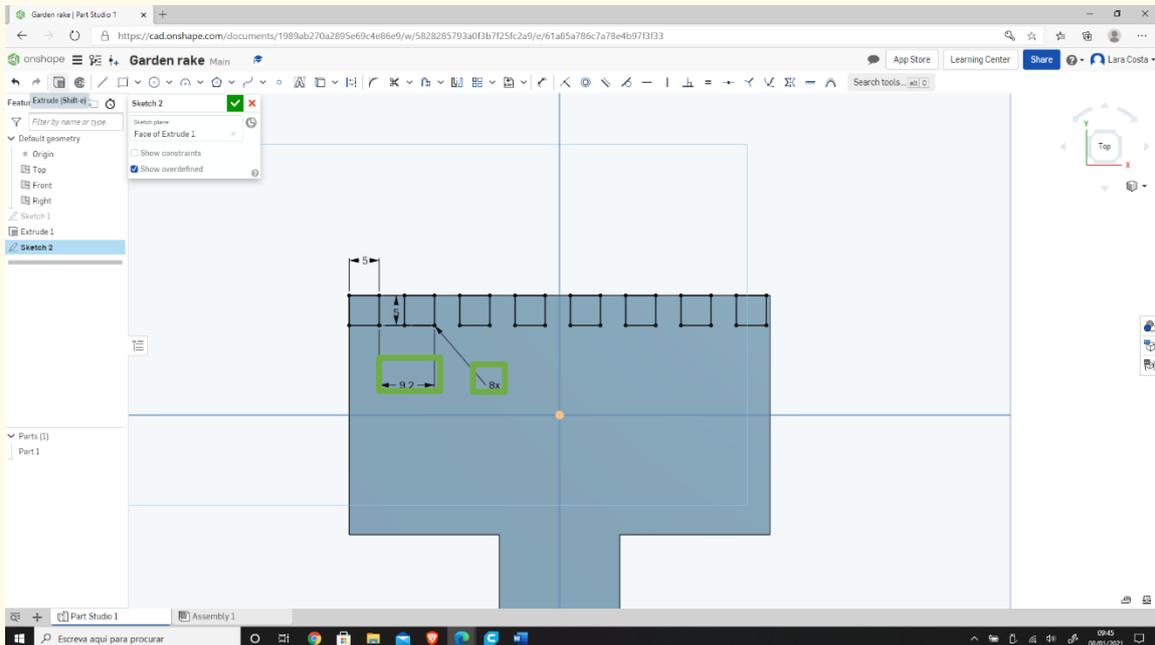
Then select the "Linear pattern."



Step 27 Select all the lines (starting from the right).

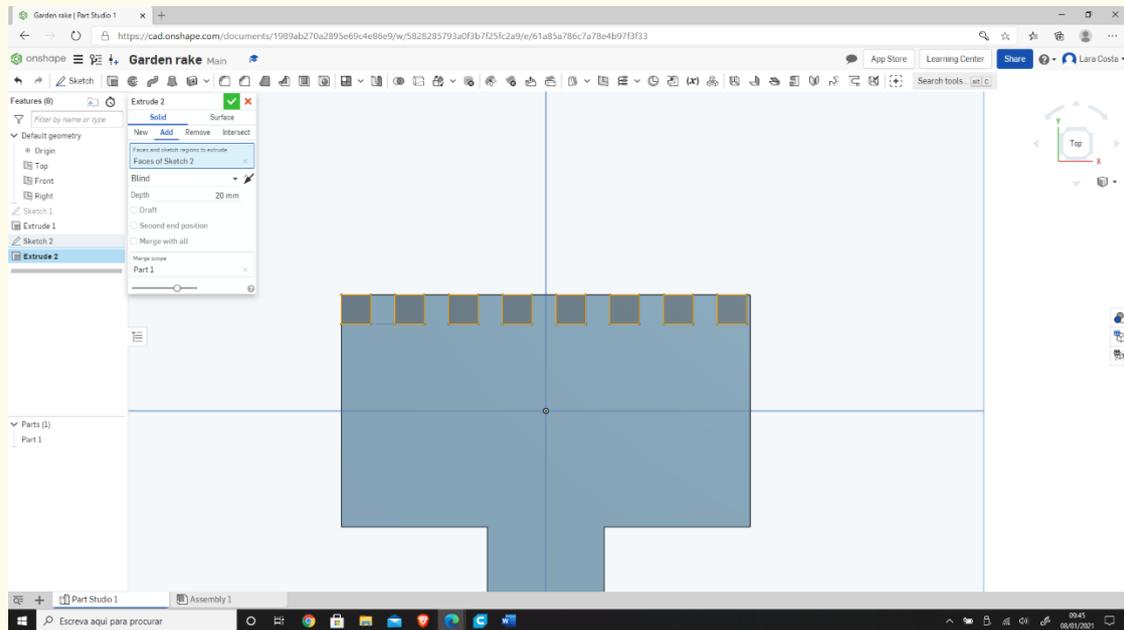


Step 28 Enter the parameters shown in the image below.



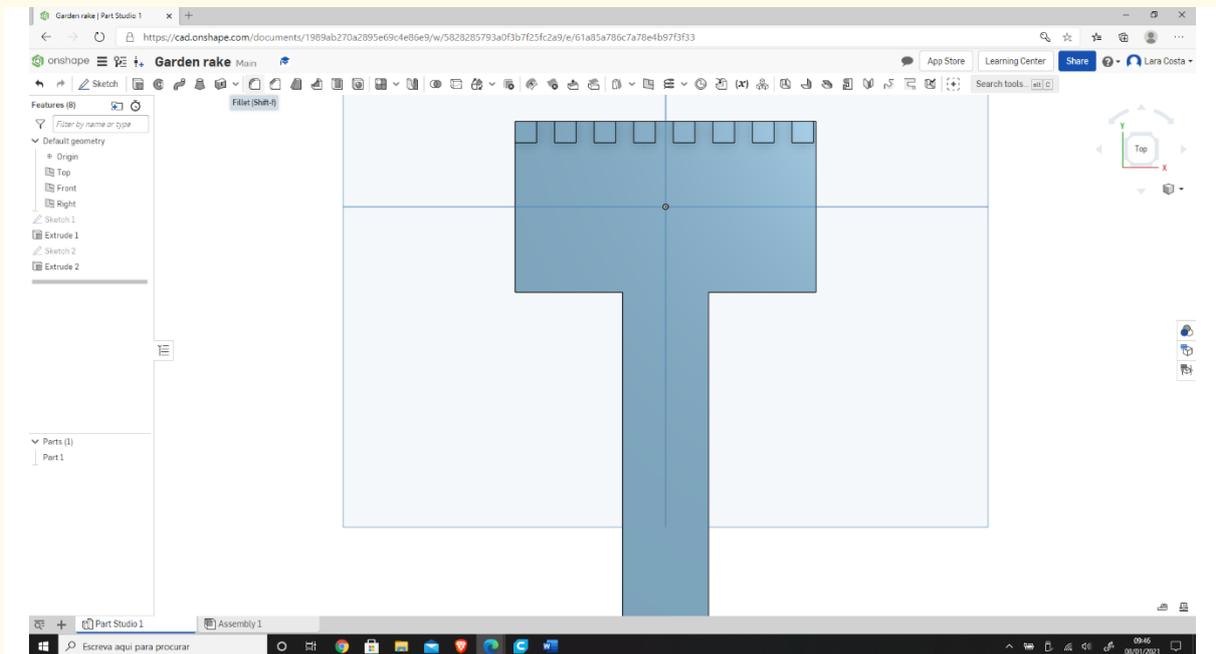
Step 29

Select "Extrude," and the depth should be 20mm.



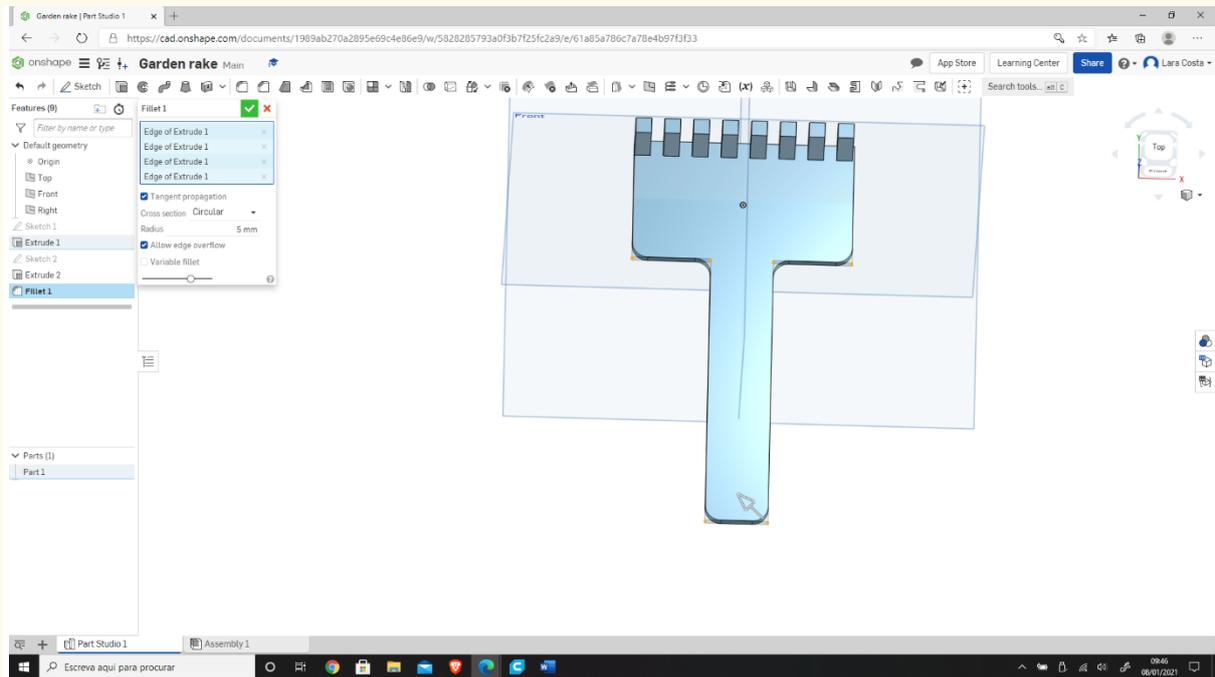
Step 30

Select "Fillet."



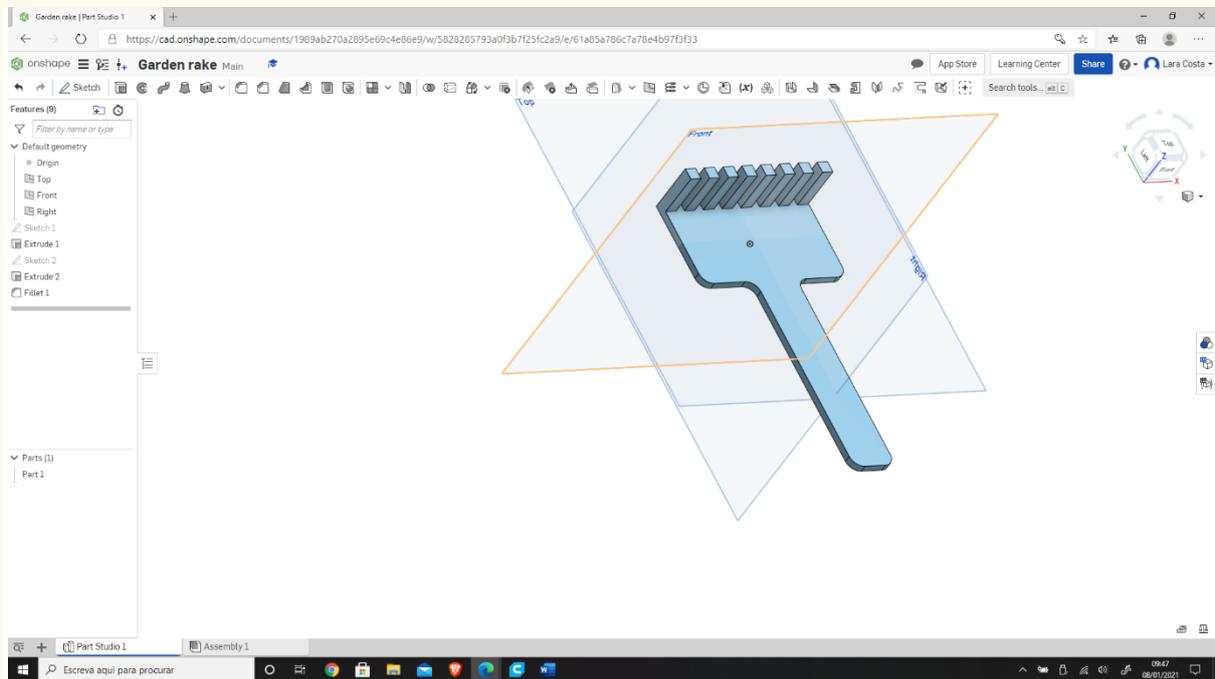
Step 31

As the image shows, select all the corners (edges) that we want to fillet.



Step 32

This should be the final result.



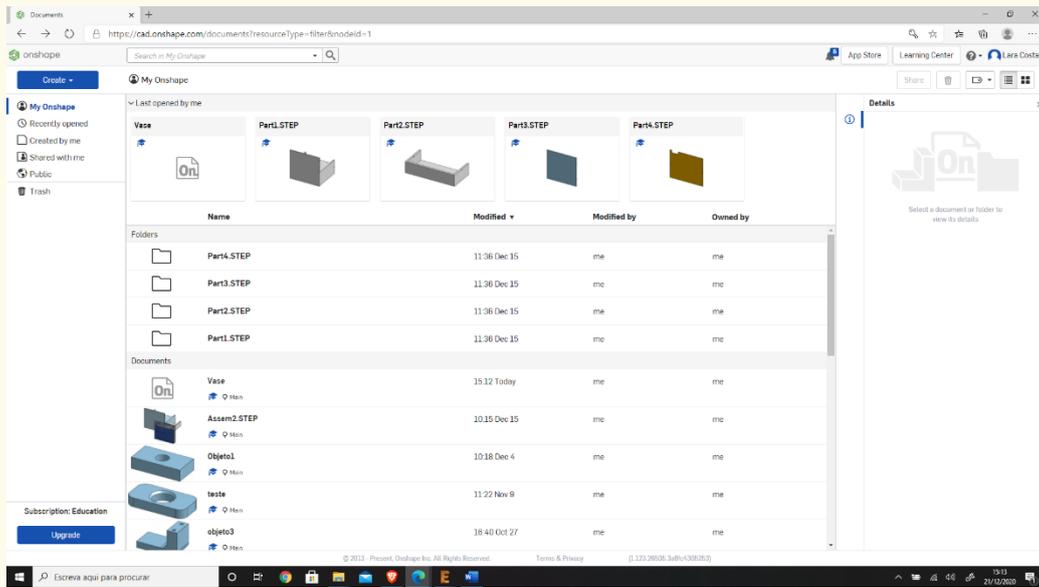
PROJECT: 3D DRAWING OF A VASE

- STEM field: Science, technology, and electronics.
- Indicative calendar: Any time of the year.
- Activity duration: 3 hours.
- Type of activity: Drawing of a vase.
- Educational objectives
- Learning outcomes and acquired competences:
 - How to do a vase on Onshape.
- Required material and resources:
 - 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:

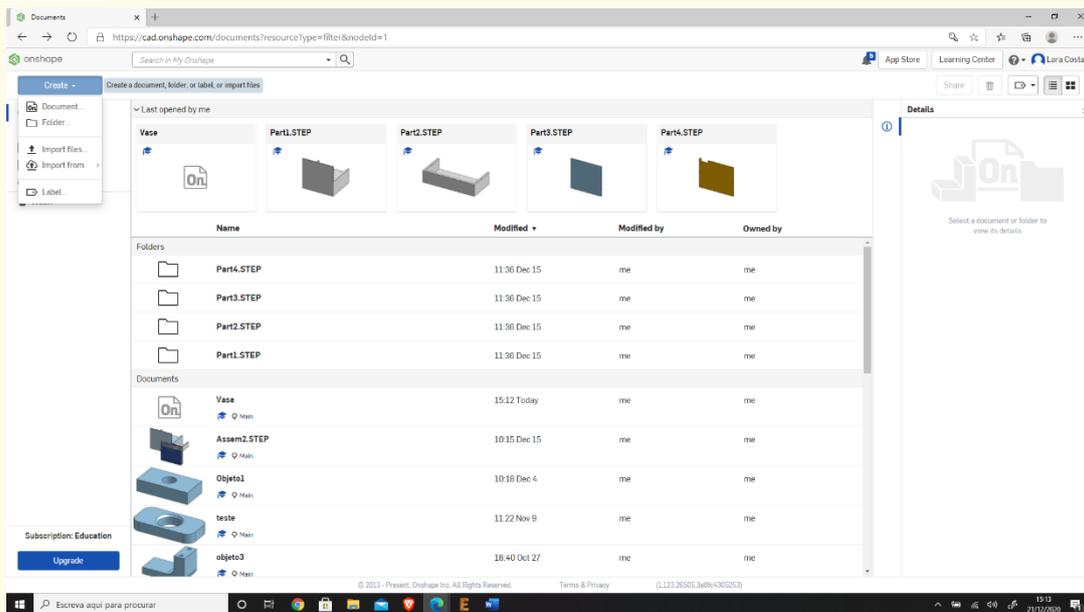
Step 1

Open the Onshape webpage.



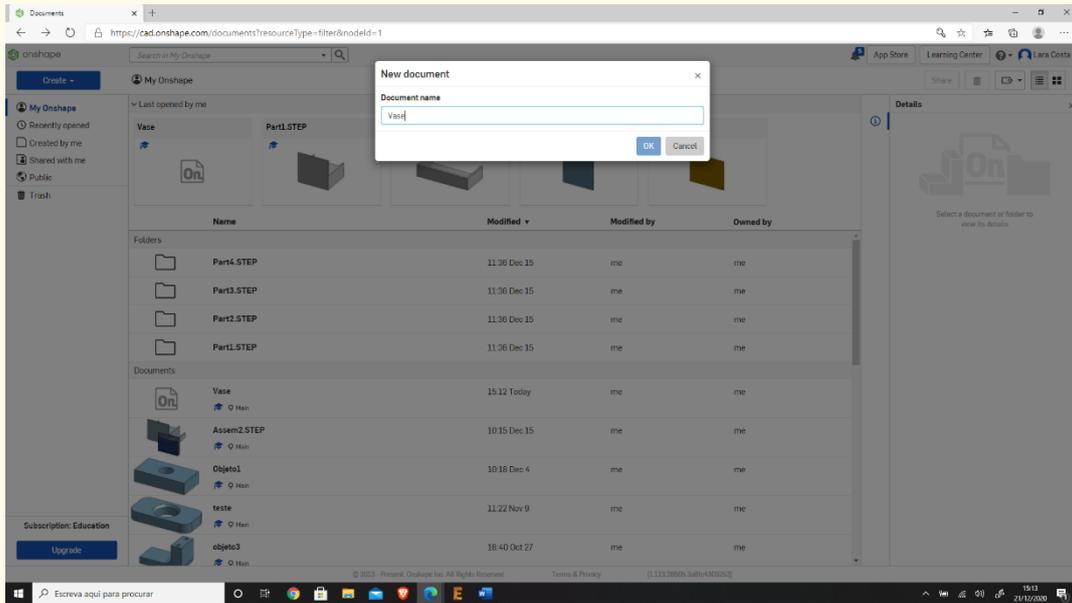
Step 2

Create a document.



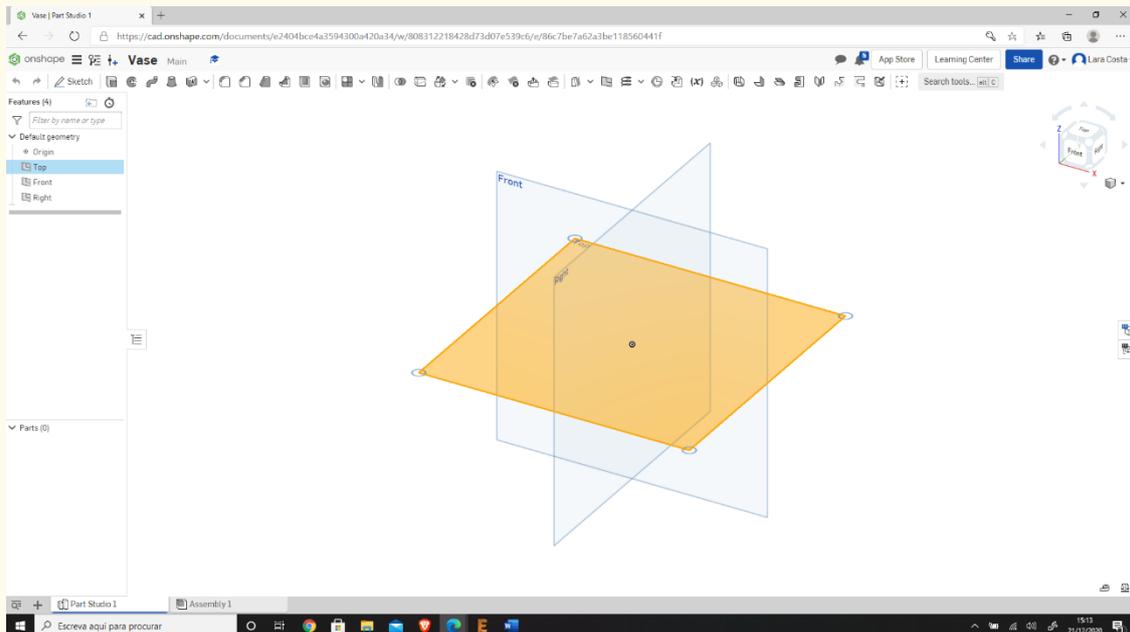
Step 3

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



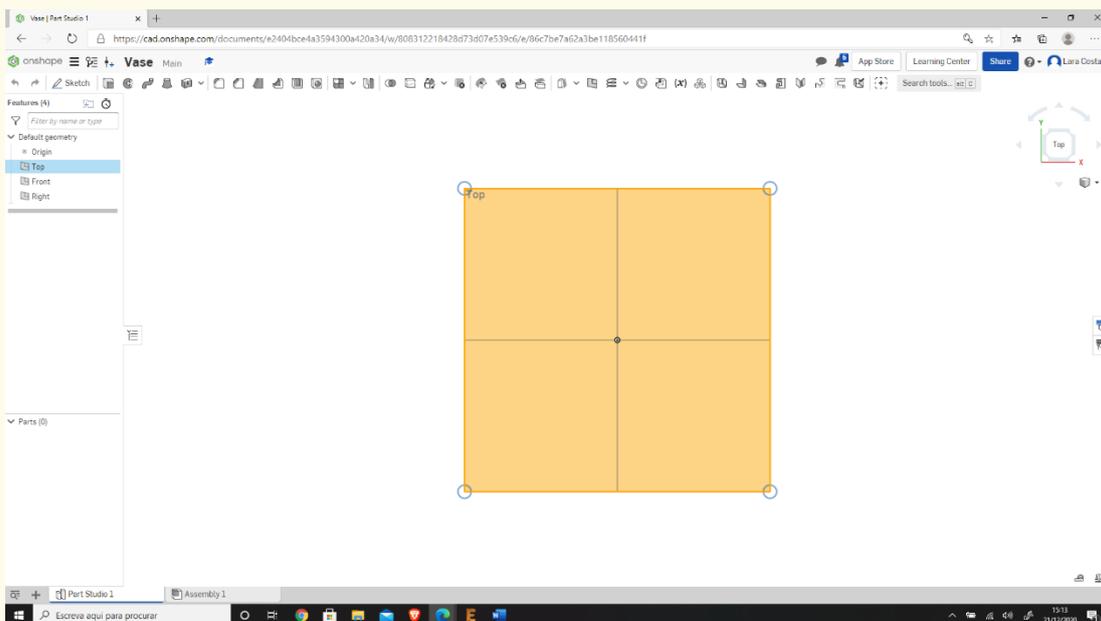
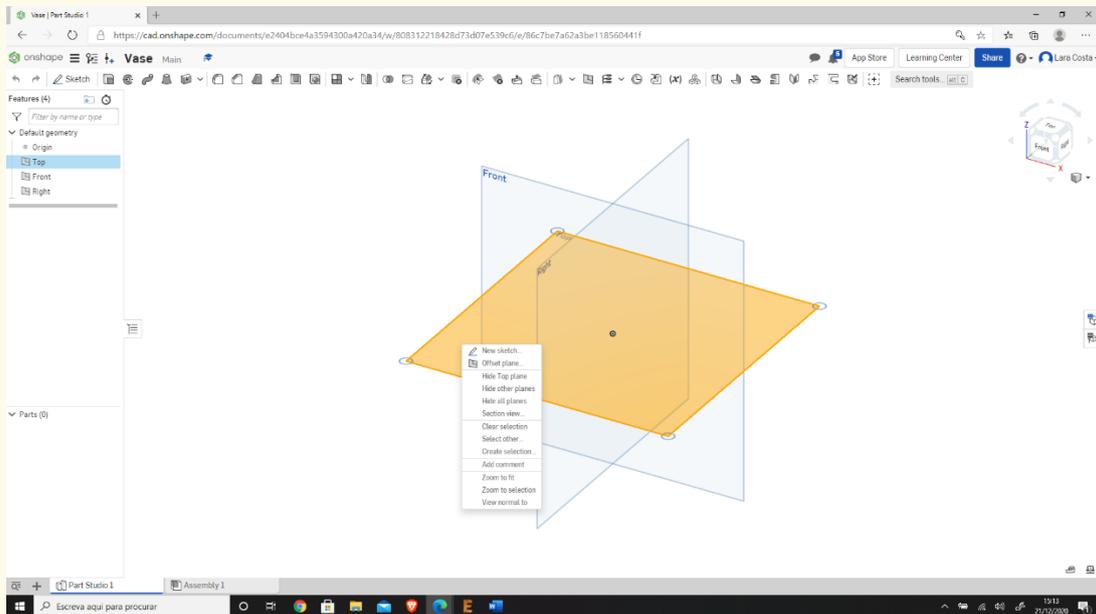
Step 4

Select the plane (top) to start drawing.



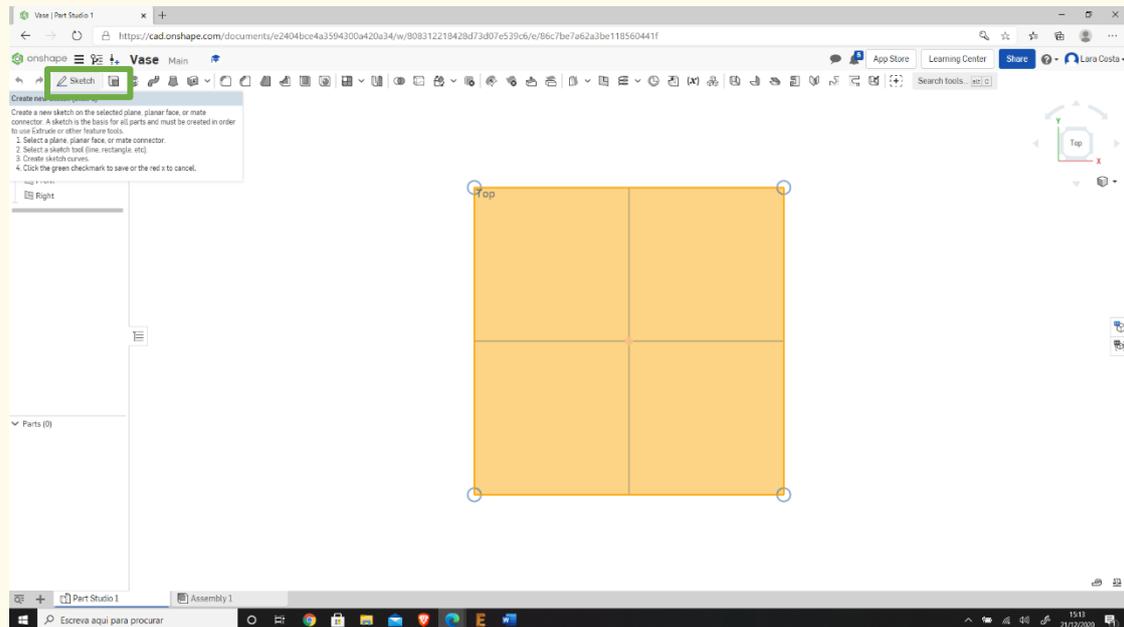
Step 5

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



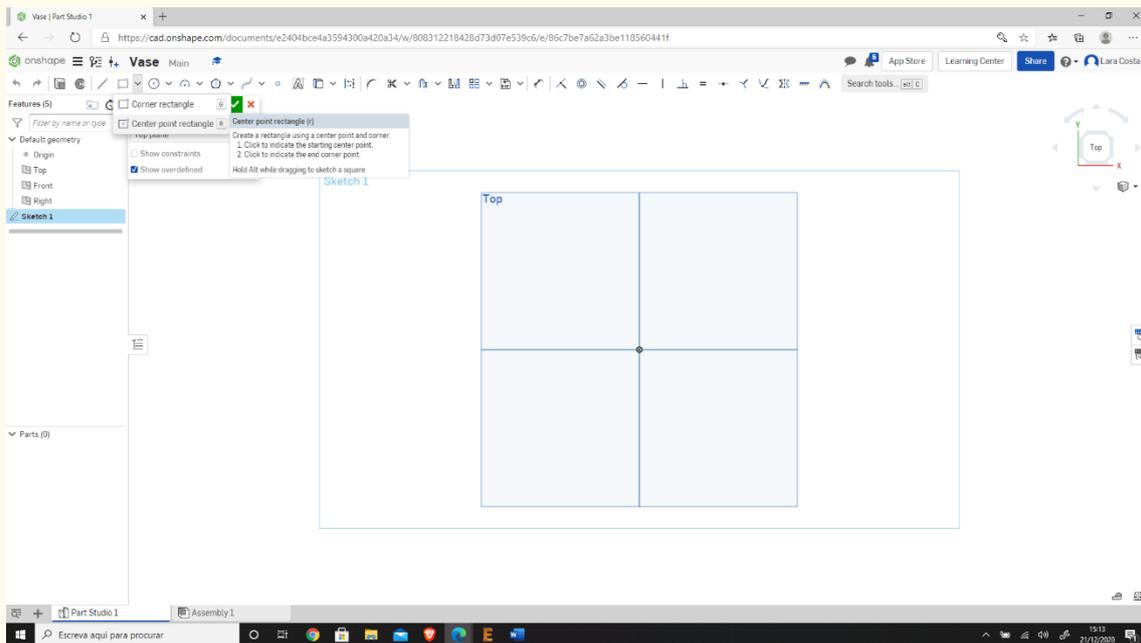
Step 6

Click "Sketch."



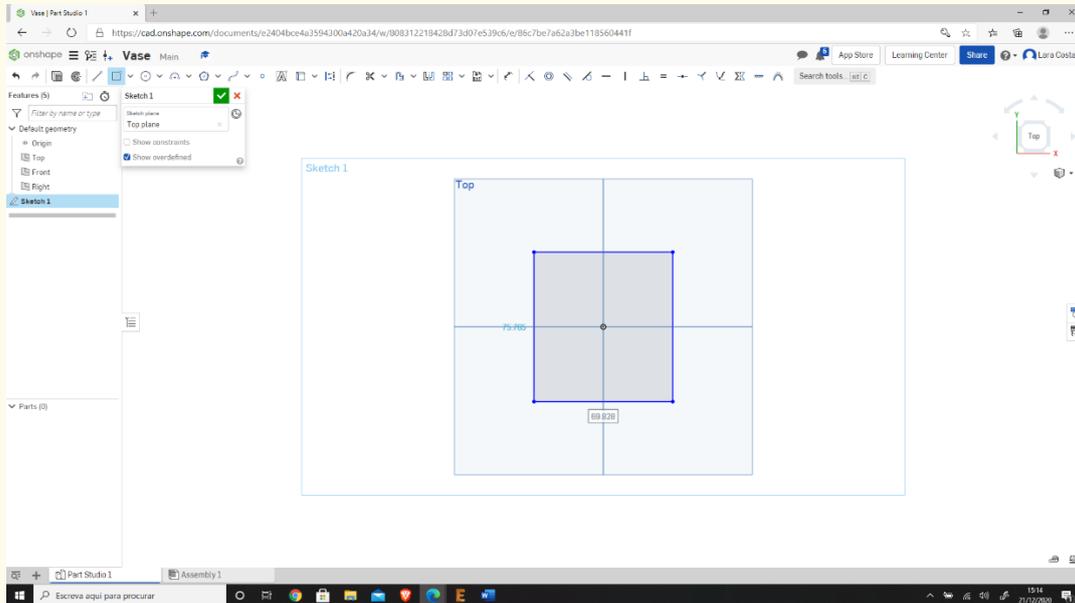
Step 7

Select "Center point rectangle."



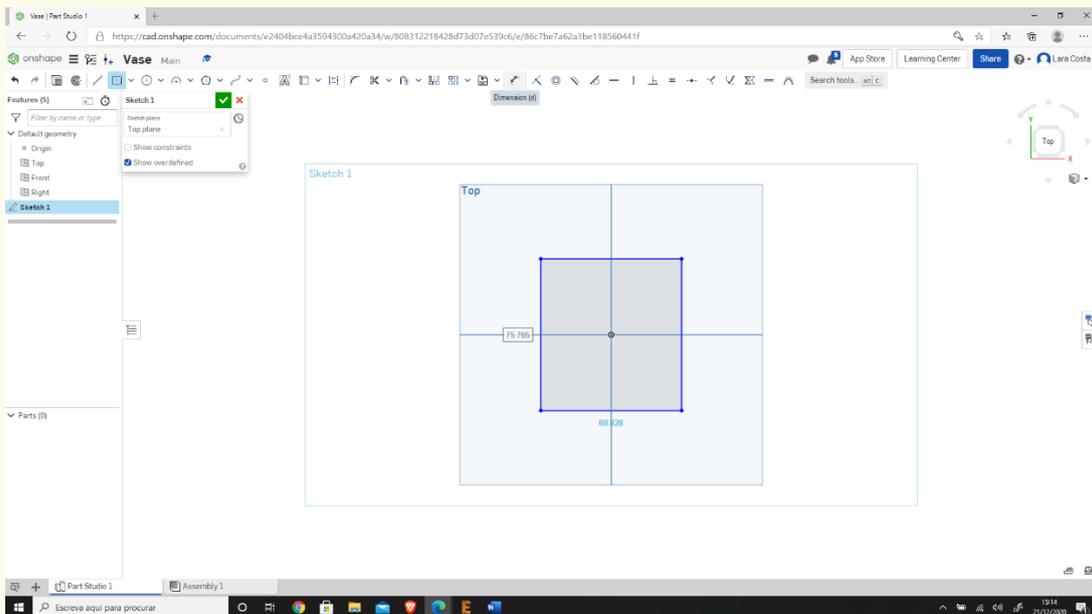
Step 8

Draw the rectangle.



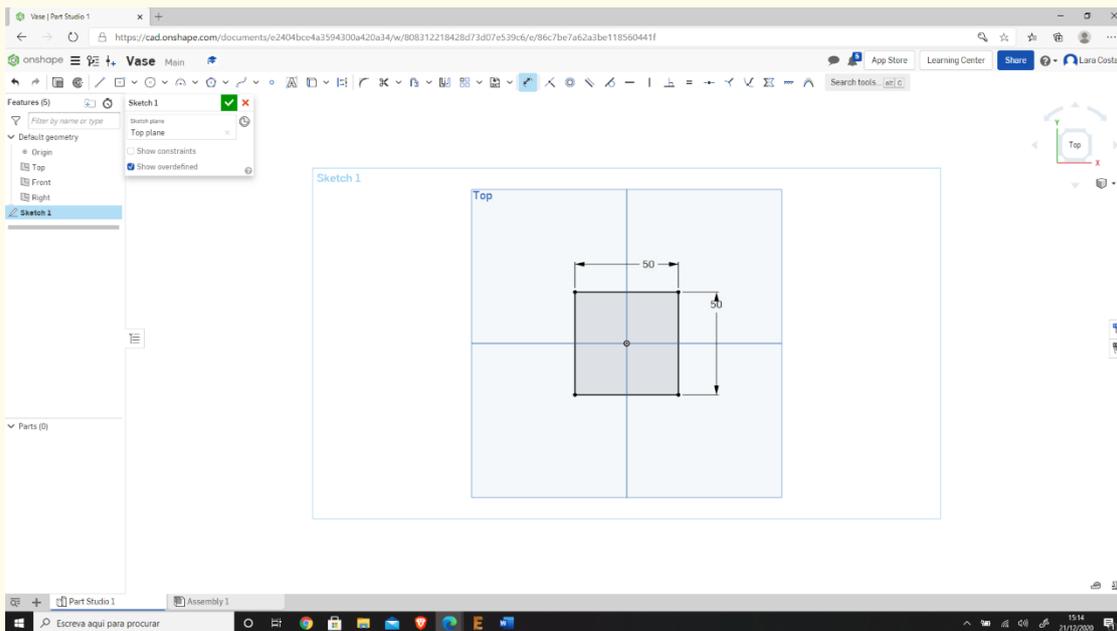
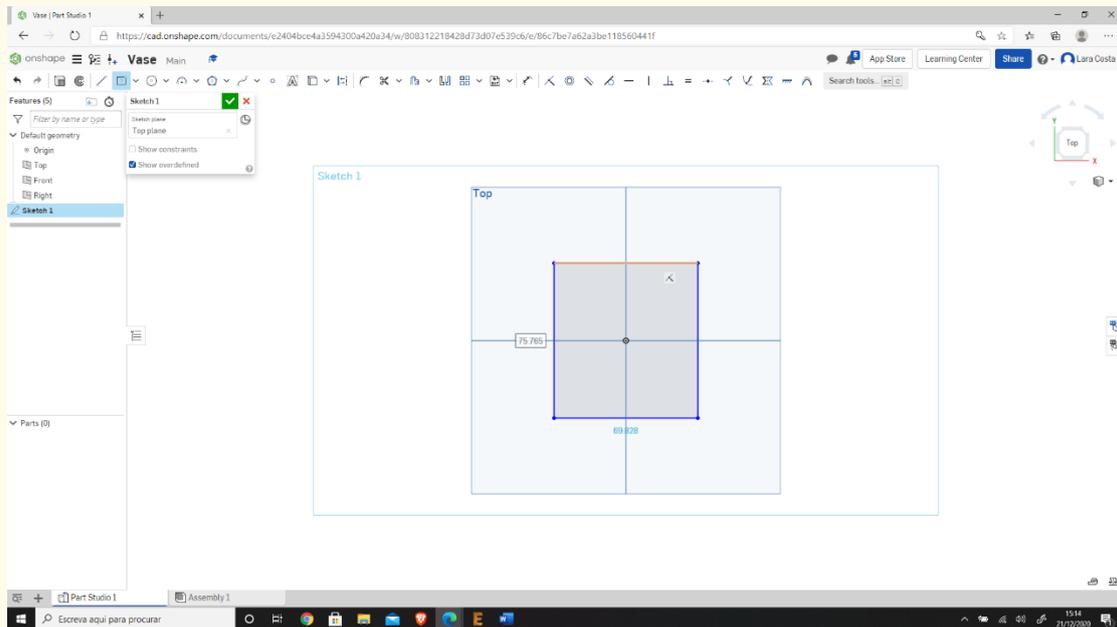
Step 9

Select "Dimension" to give the dimensions.

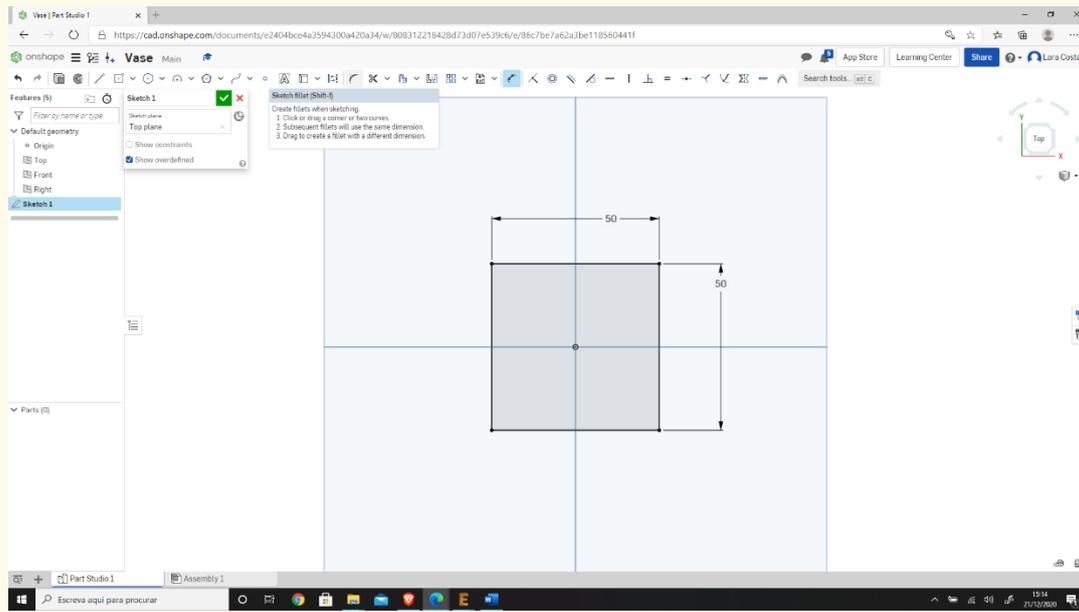


Step 10

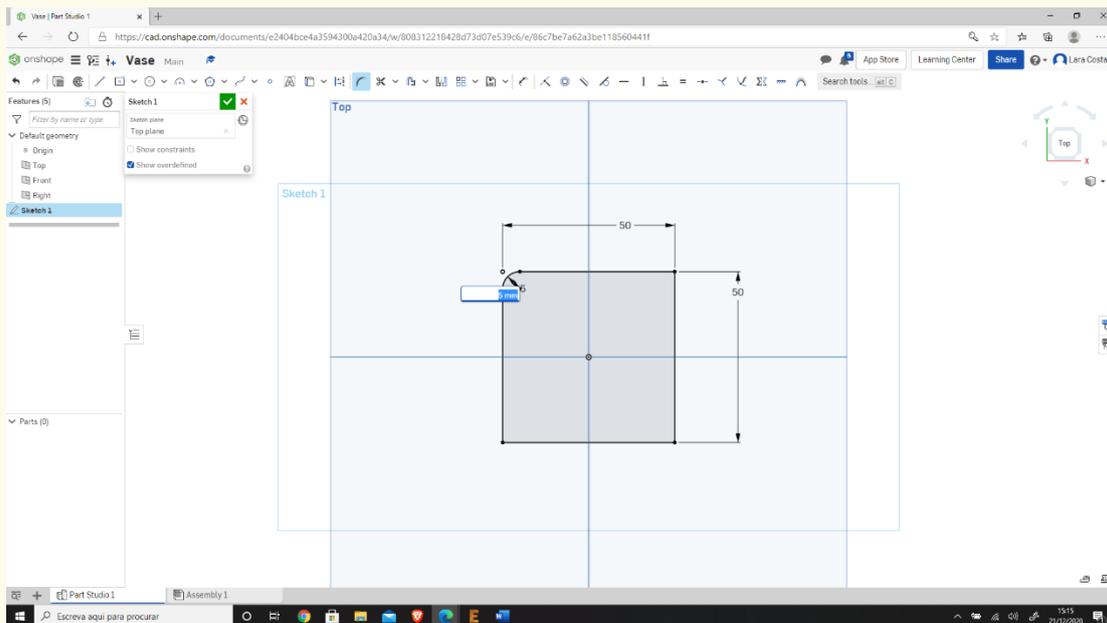
Select the line (first top then right) and give the dimensions as shown in the 2nd figure.



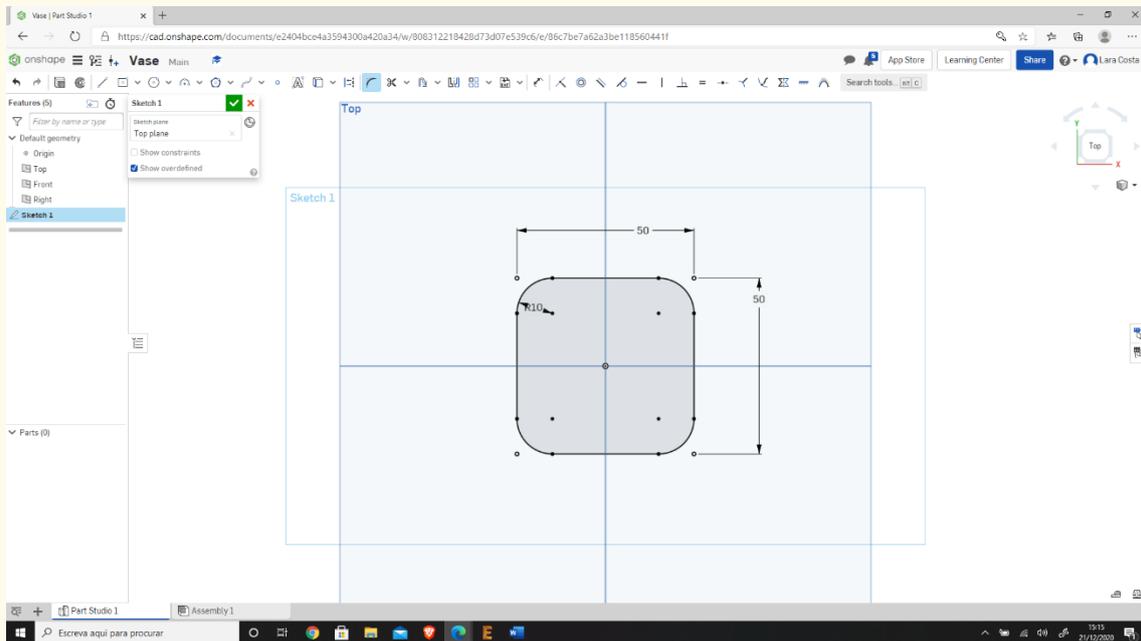
Step 11 Select "Sketch fillet."



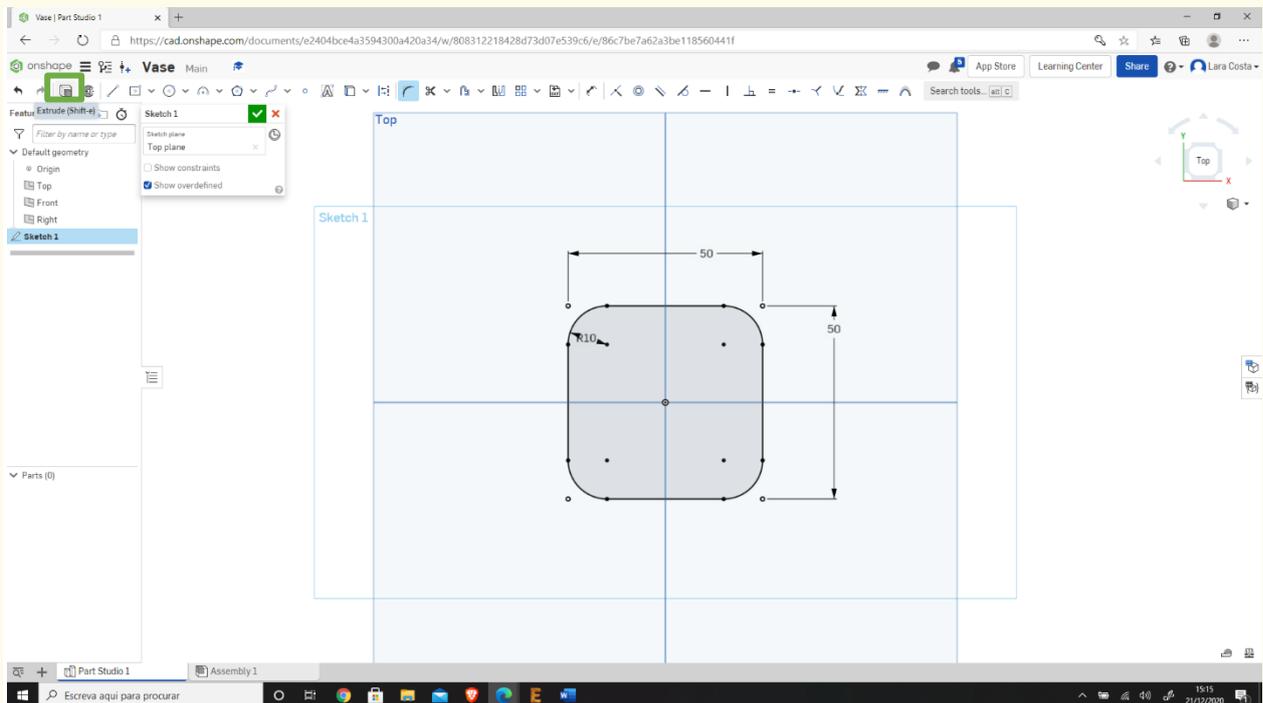
Step 12 Select two lines and click enter.



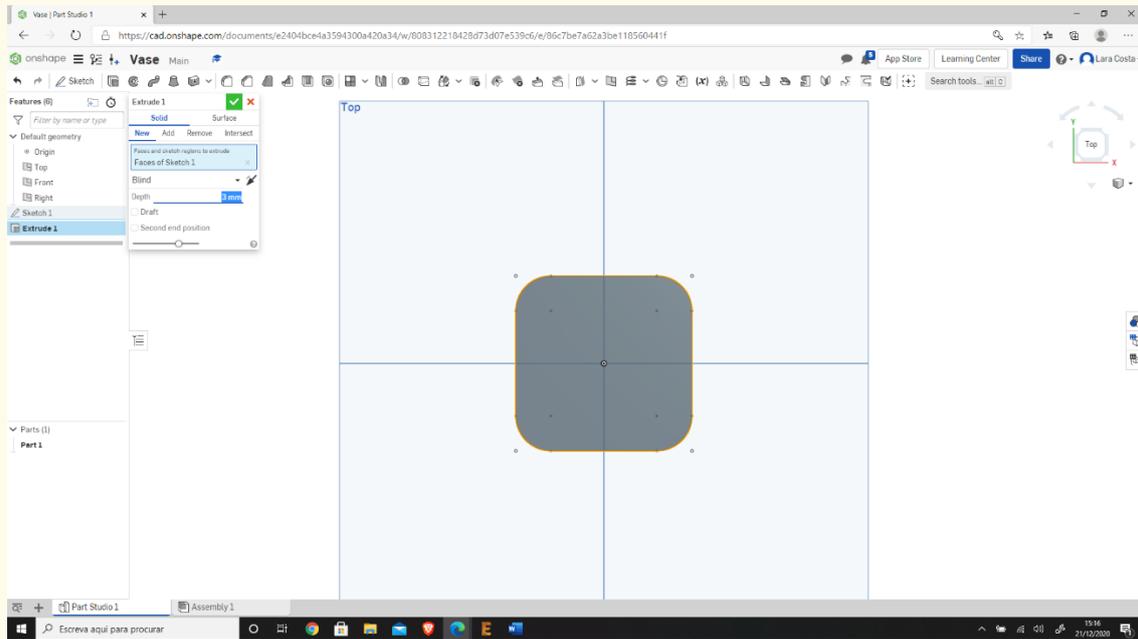
Step 13 Type 10mm as dimension and repeat the same process to every corner.



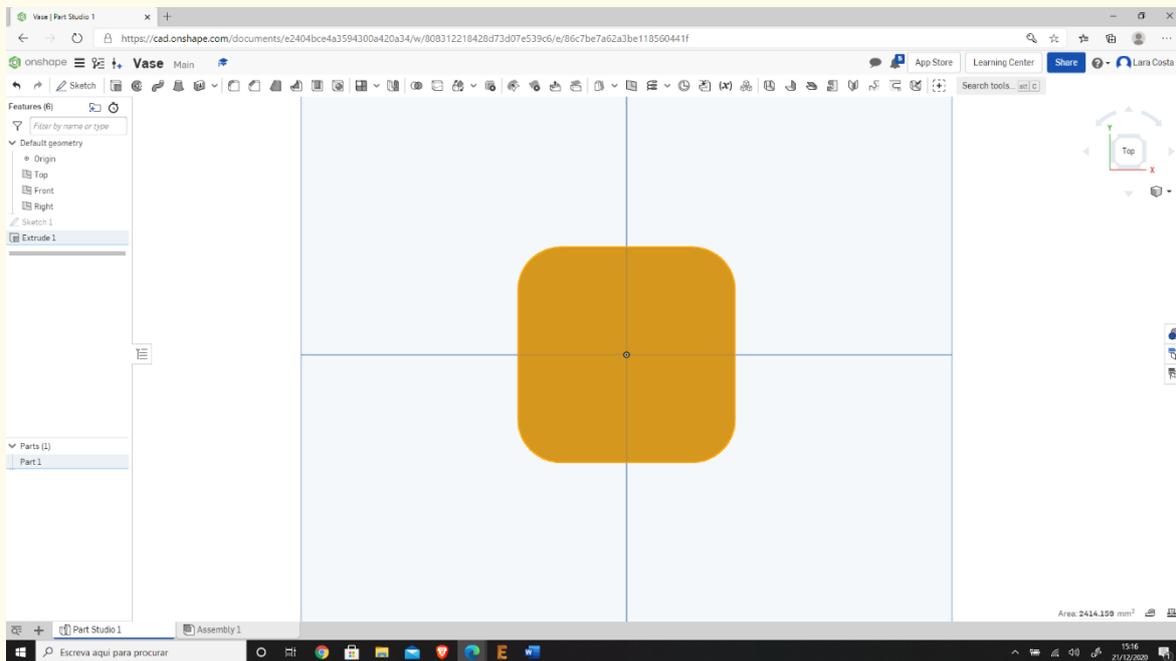
Step 14 Select "Extrude."



Step 15 Insert a Depth of 3mm and click .

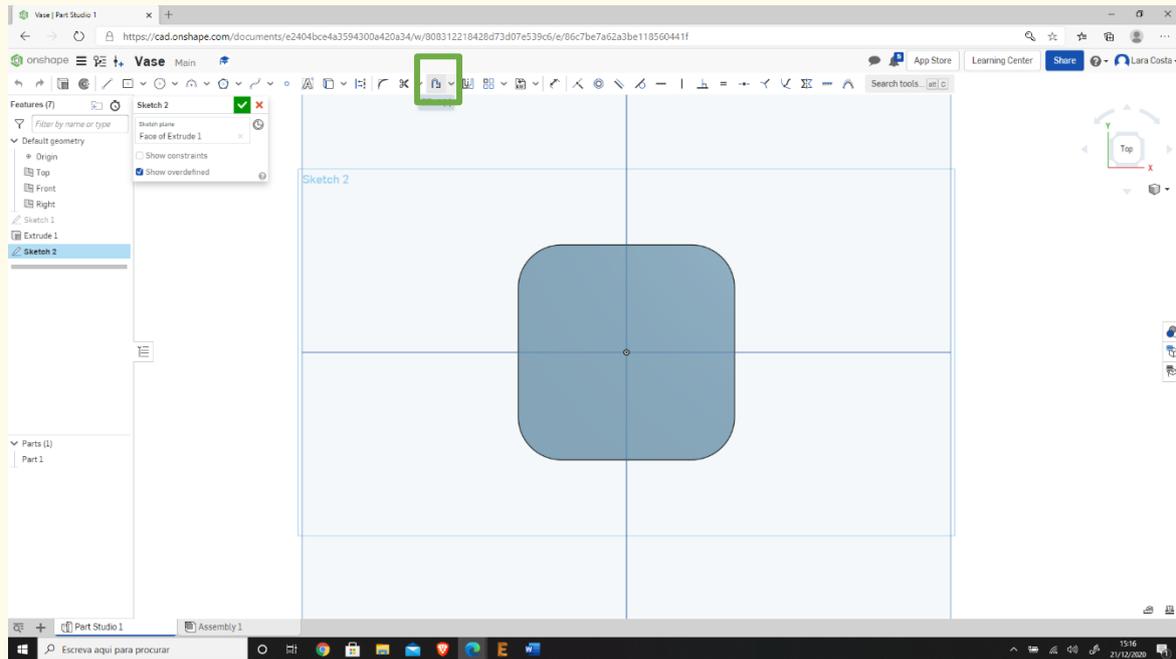


Step 16 Select the Front face.



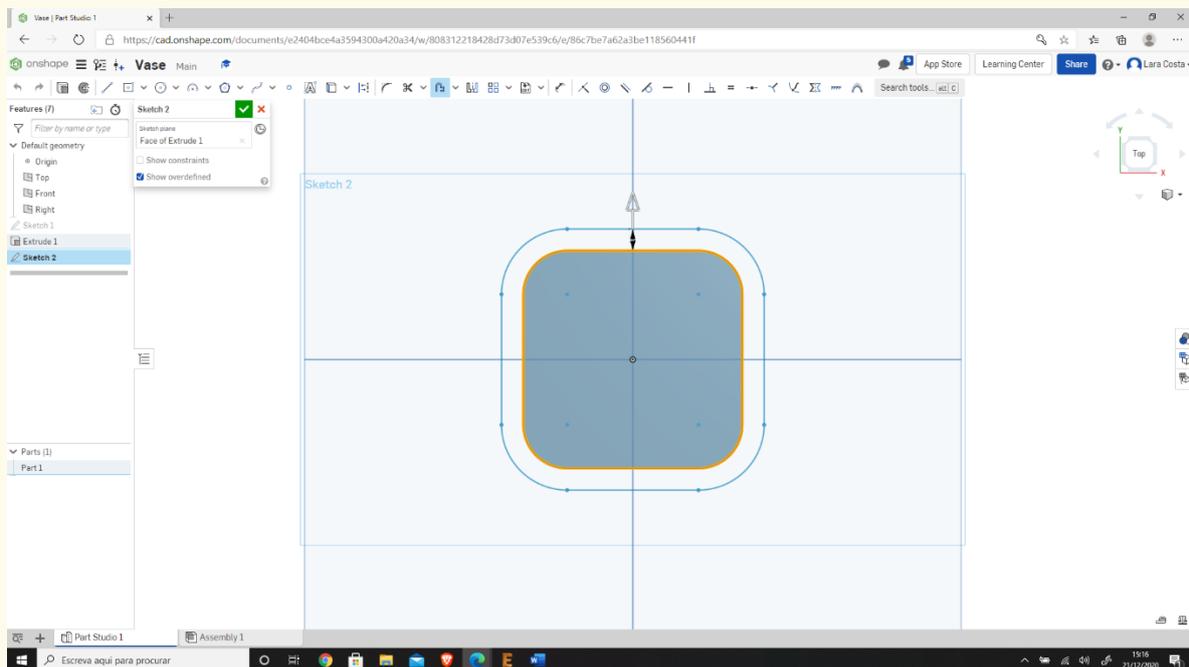
Step 17

First, select "Sketch" (like we did in step 6) and then "Offset."



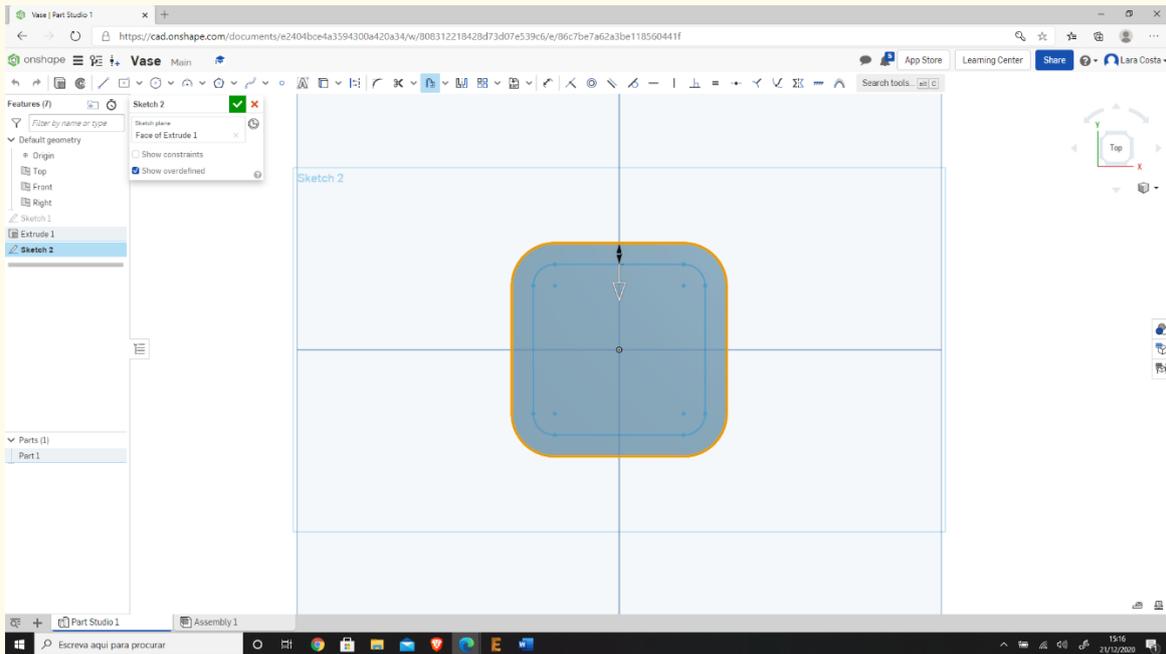
Step 18

Select all the edges.



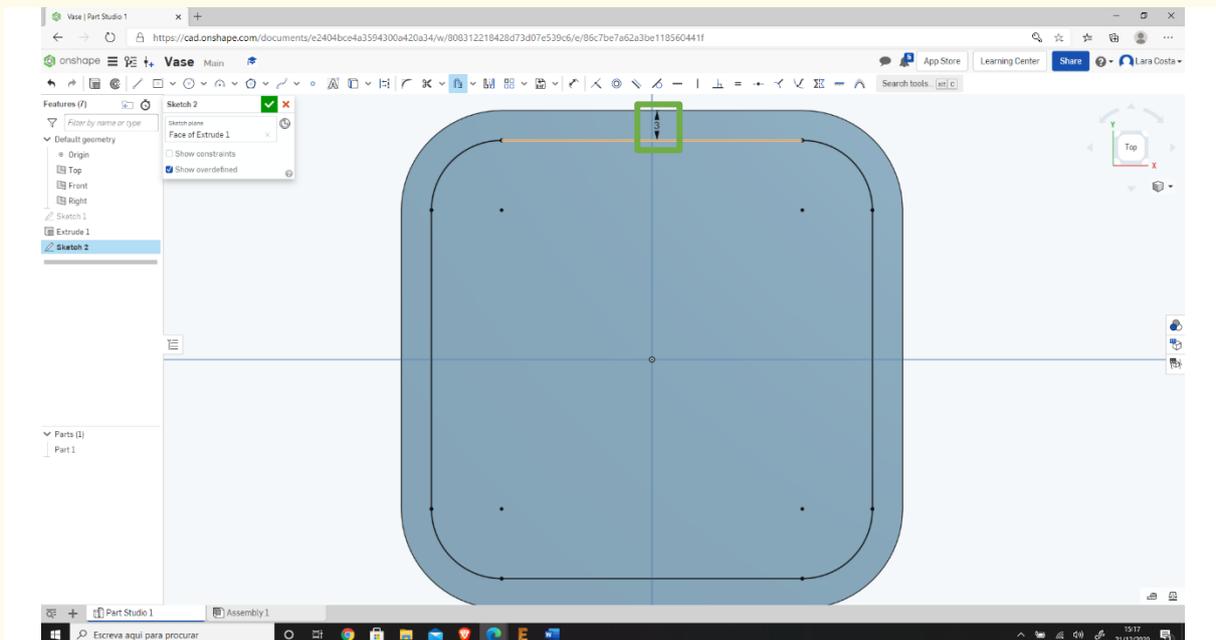
Step 19

Click on the arrow if necessary to have this result.



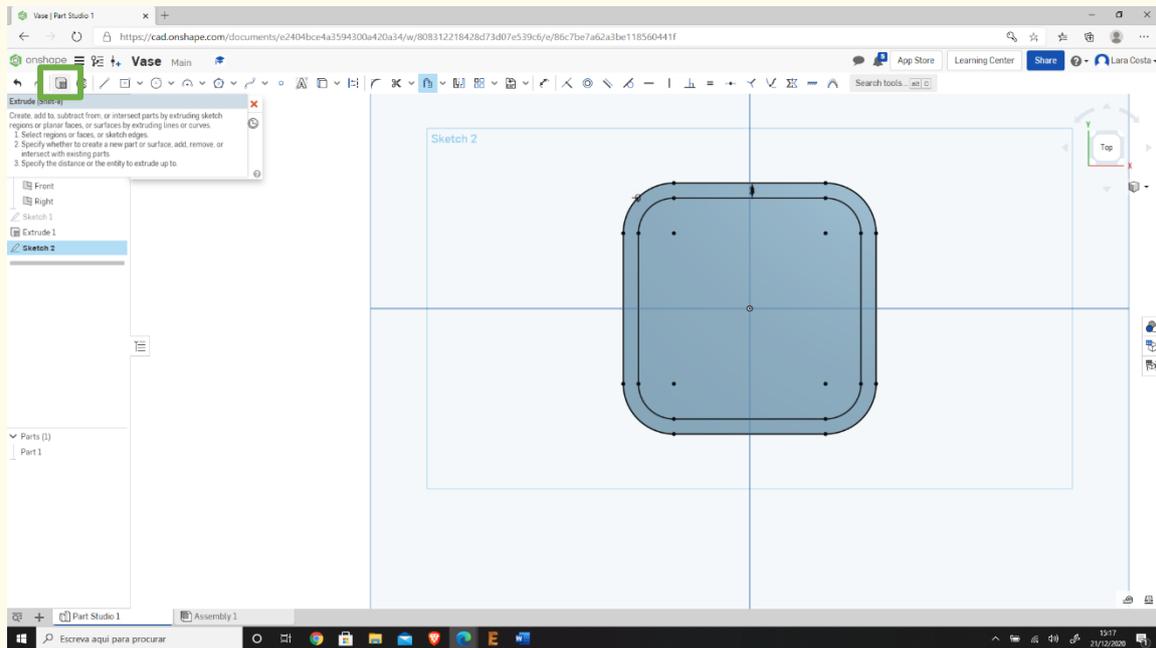
Step 20

Change the dimensions to 3mm.



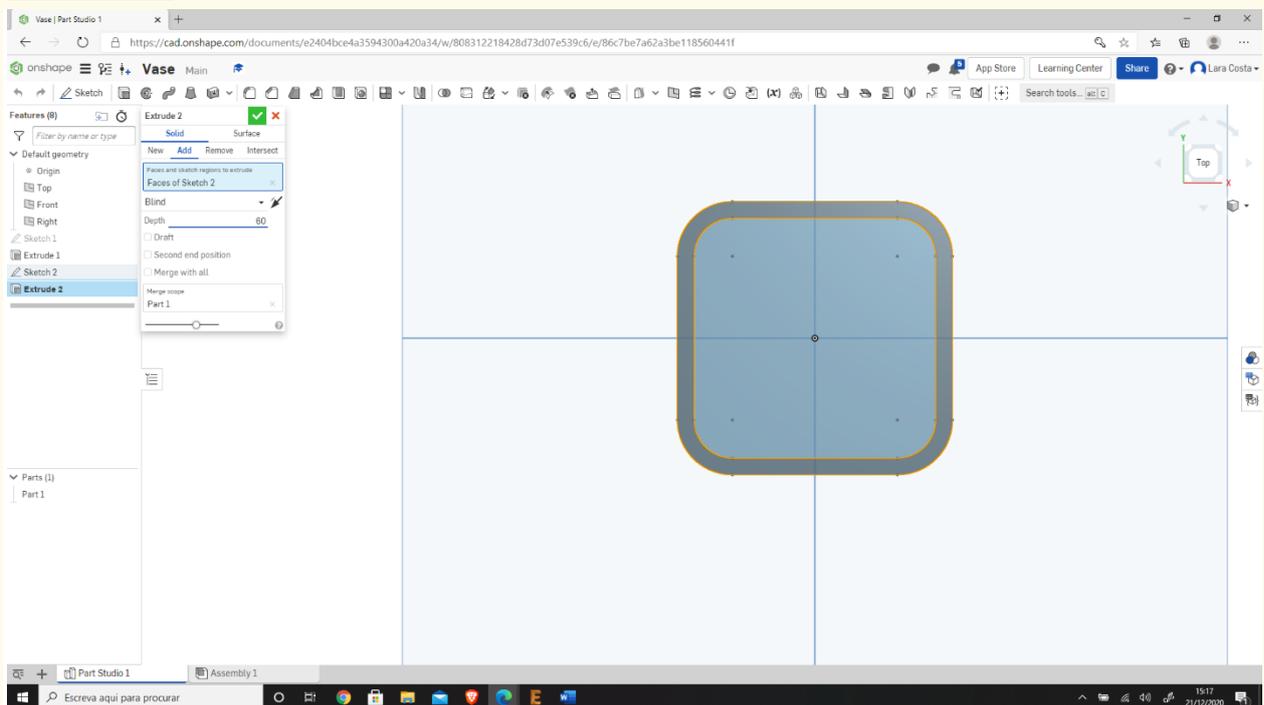
Step 21

Repeat steps 18,19 and 20, but now the dimension is 0.
Click "Extrude," select the outside lines (the original ones).



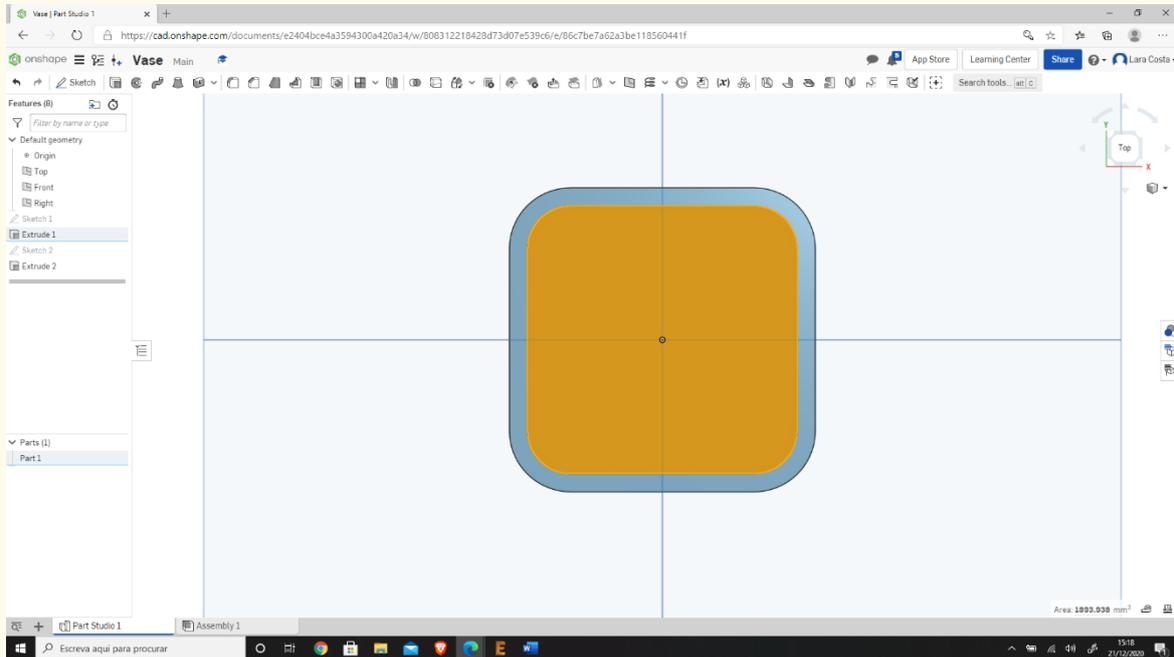
Step 22

Change the Depth to 60mm and click .



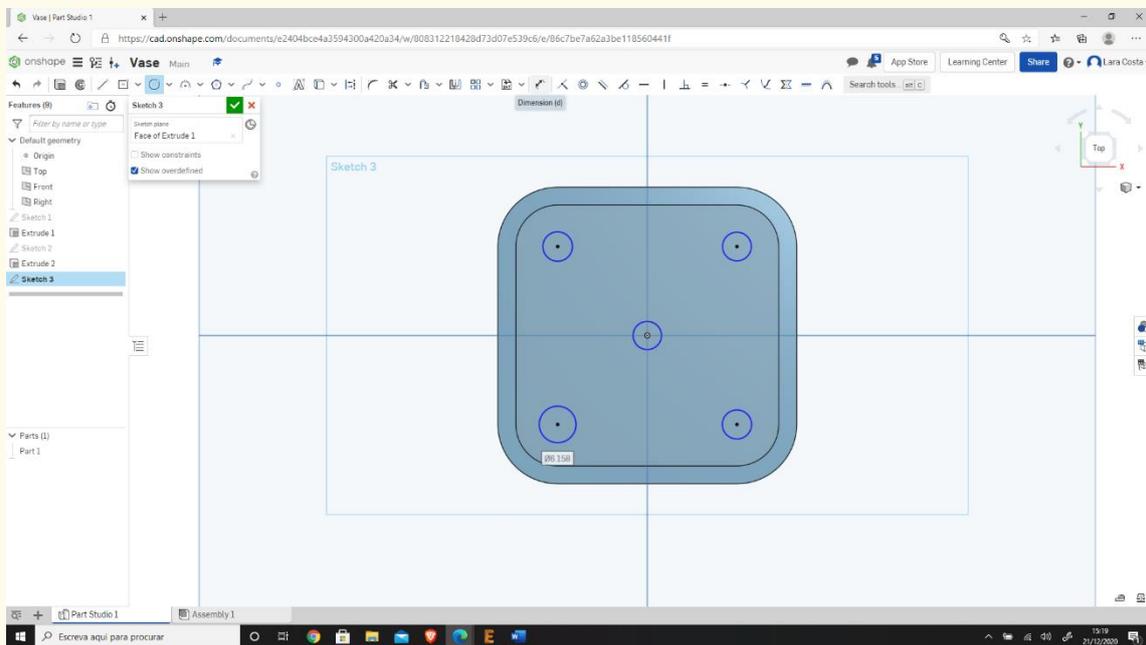
Step 23

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



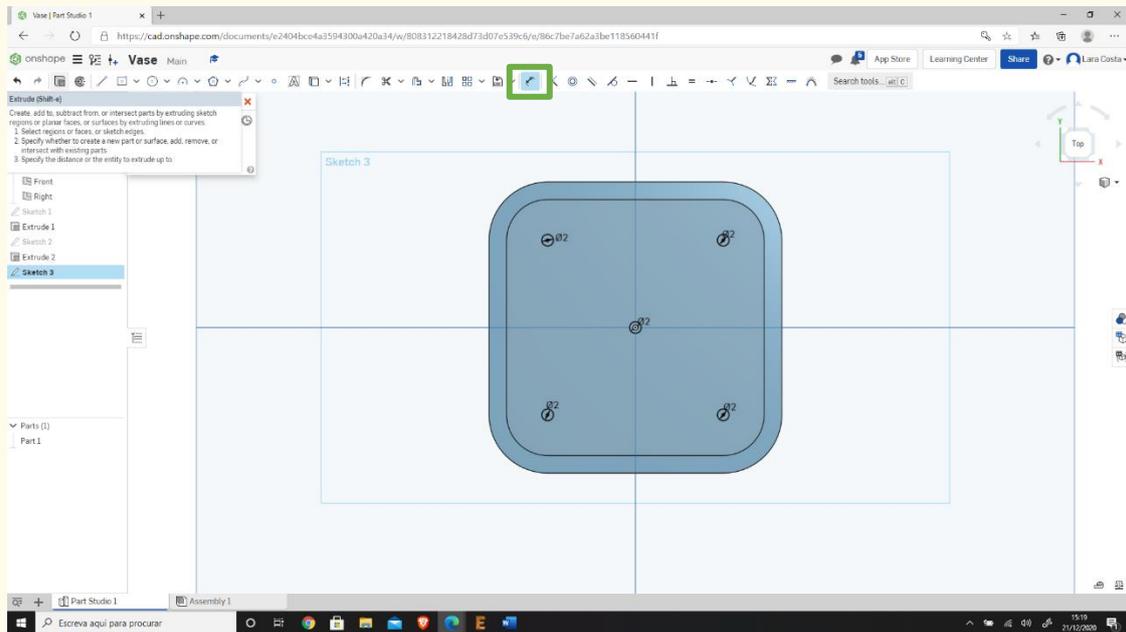
Step 24

Draw a circle in the center and one in each corner.



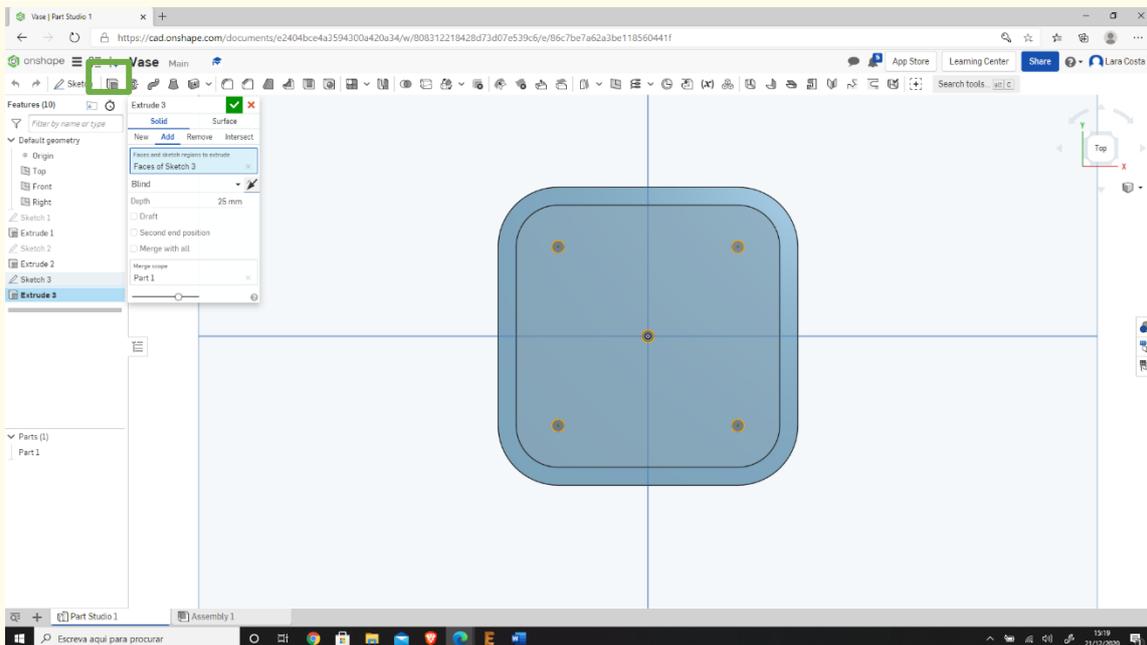
Step 25

Insert the dimensions (2mm) by clicking on "Dimension" and then in each circle.



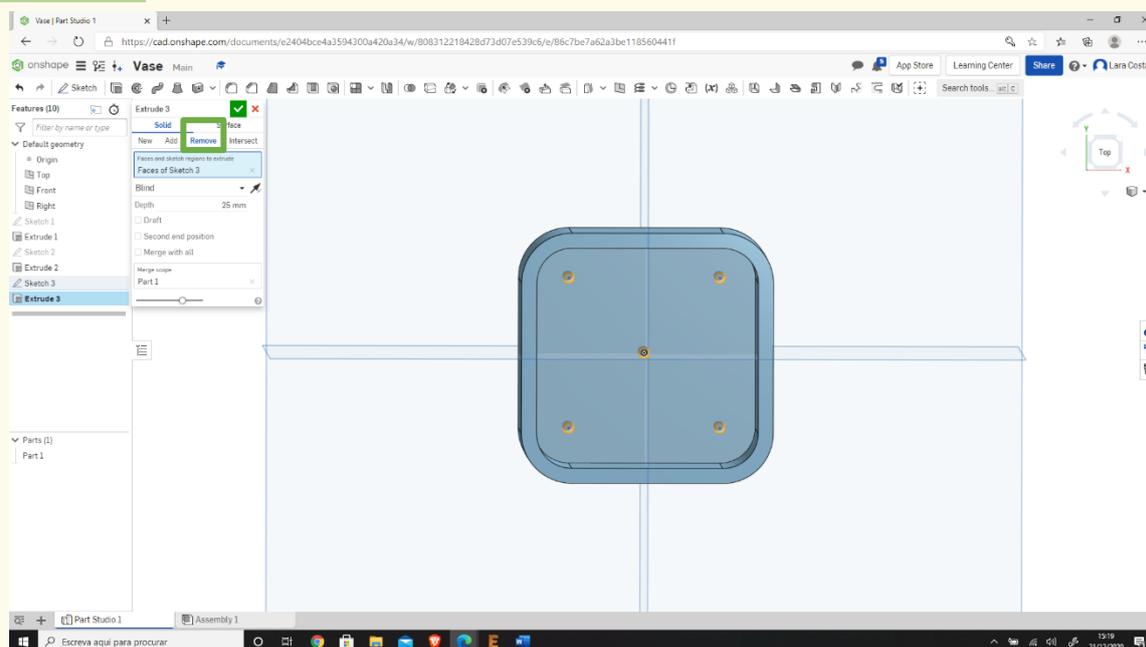
Step 26

Then select "Extrude."



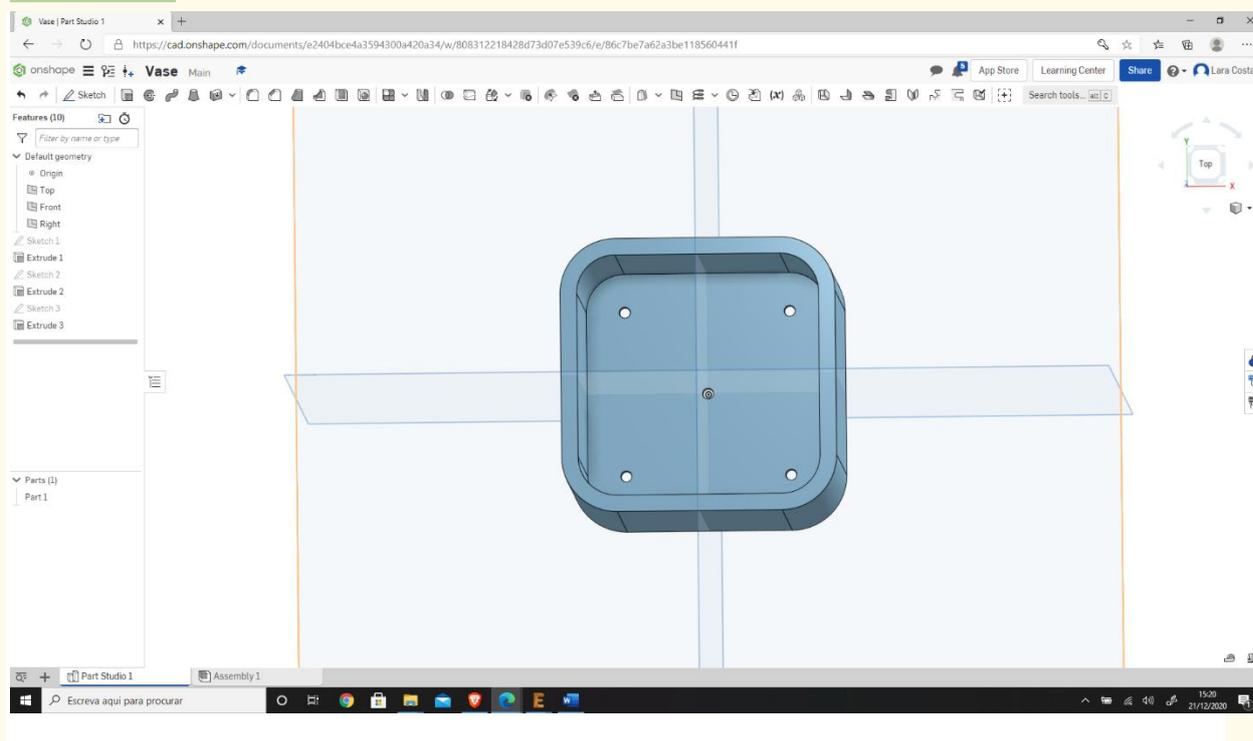
Step 27

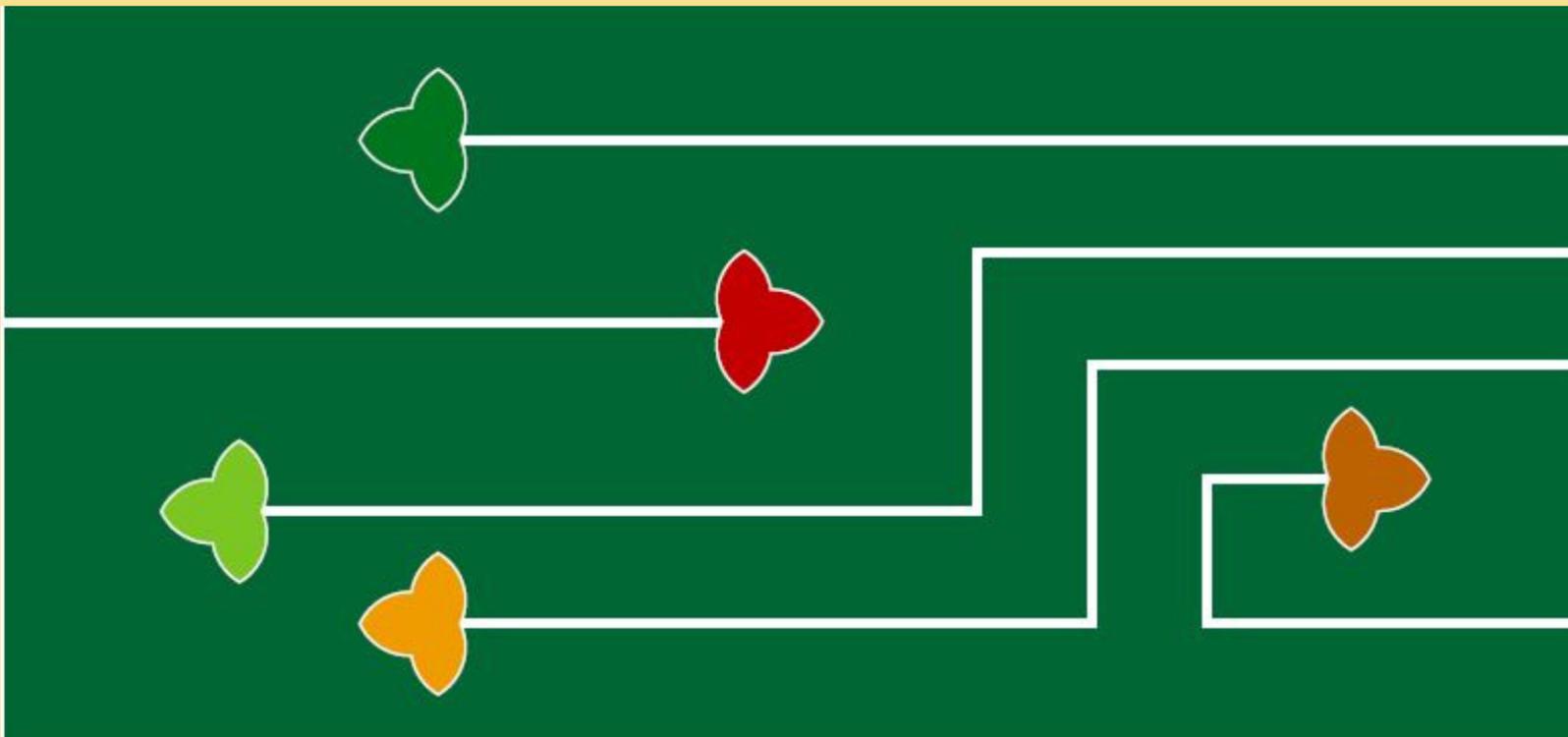
Click “Remove” and then click on .



Step 28

This is how your project should look like.





Erasmus+

The Green Steam Incubator project has been funded with support from the European Commission. The publication reflect the views only of the author, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

Project number: 2019-3-CY02-KA205-001692

