Green STEAM Incubator

Handbook on Environmental Projects using Onshape

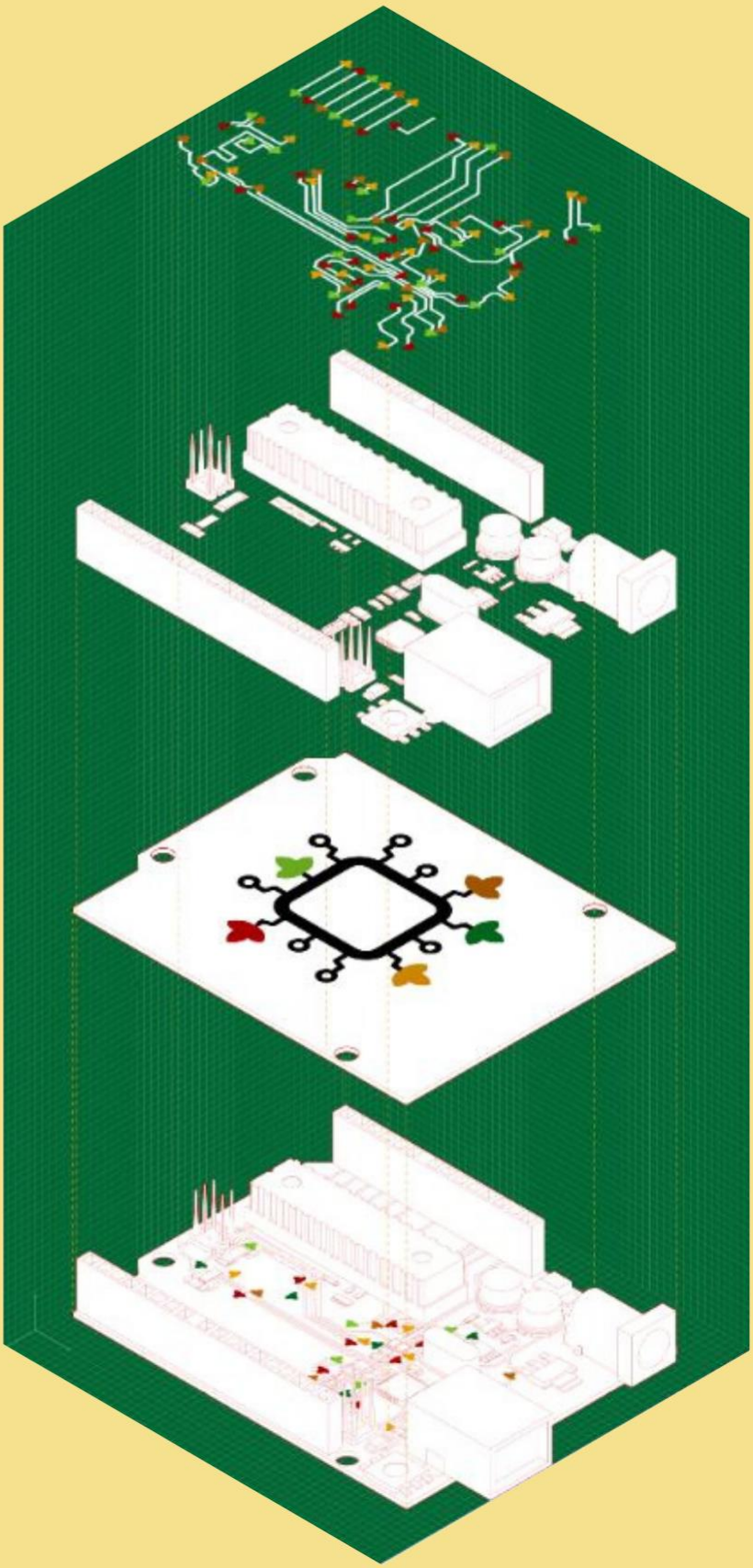

Table of Contents

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.

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 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.
- **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 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.

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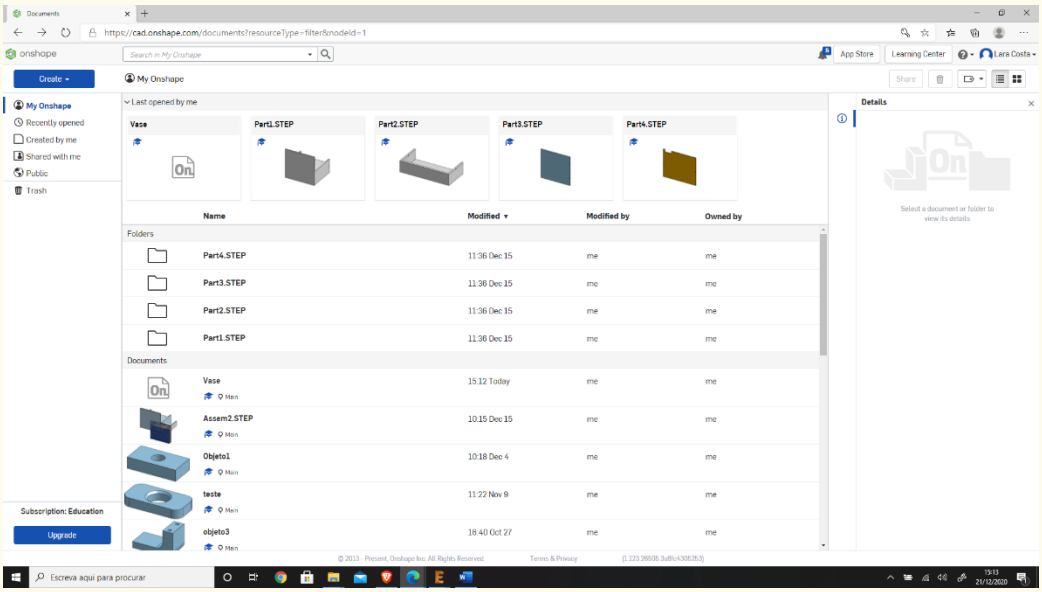
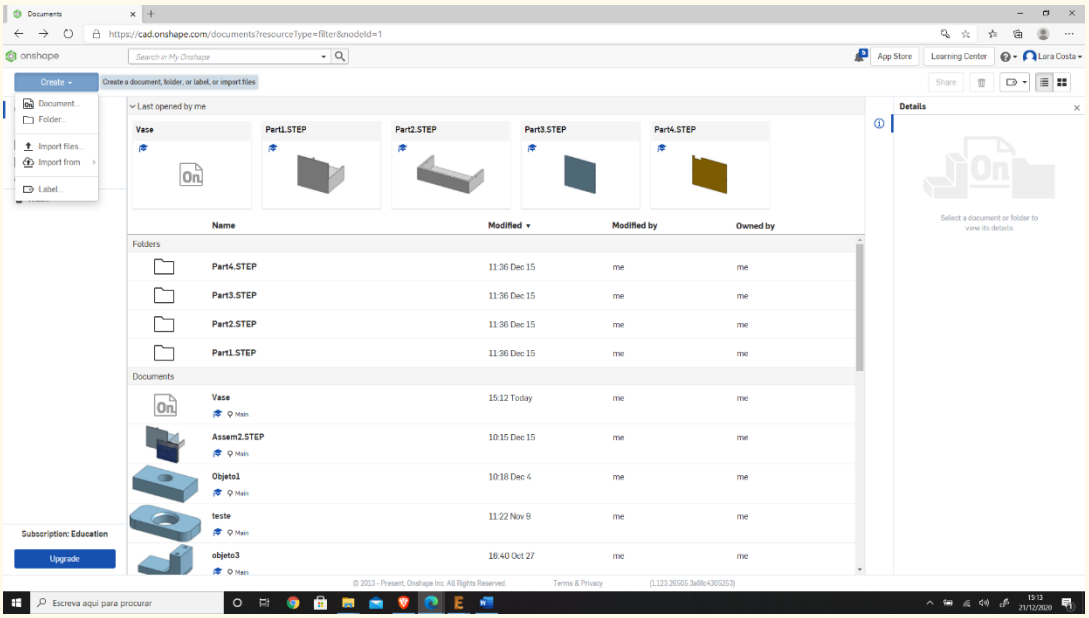
Figure 1 A bird feeder, source: https://www.theguardian.com/environment/2020/feb/27/country-diary-the-gatherings-at-the-bird-feeder-are-anything-but-random

Step 1 **Open Onshape.**



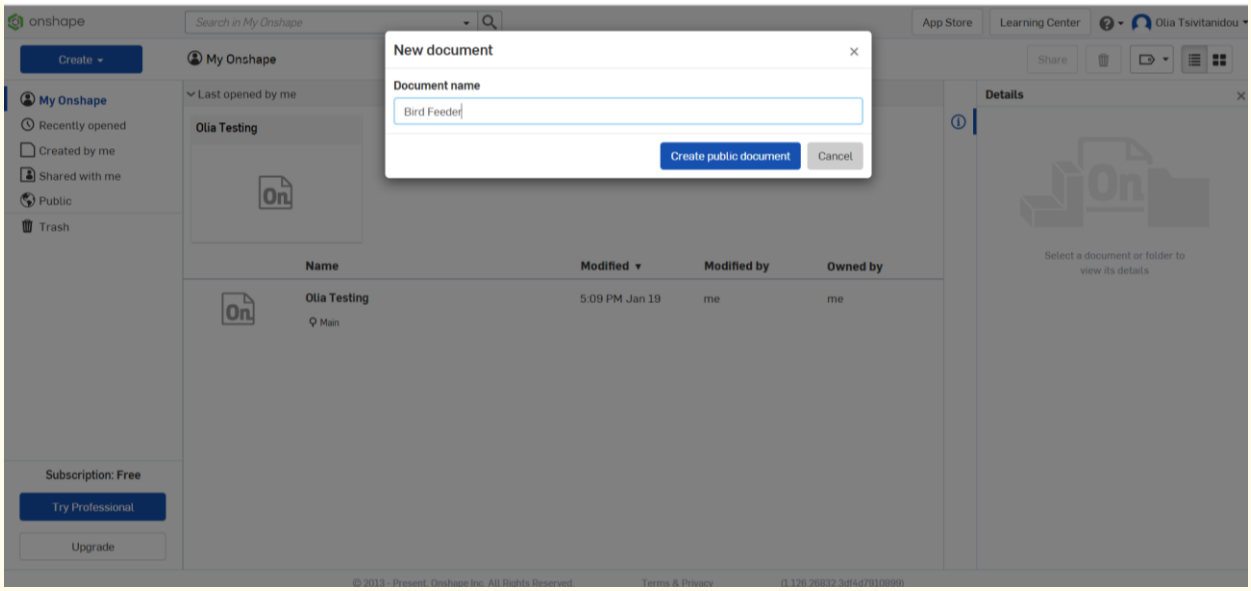
Step 2

Create a document.



Step 3

Give a name to your document such as “Bird Feeder”.



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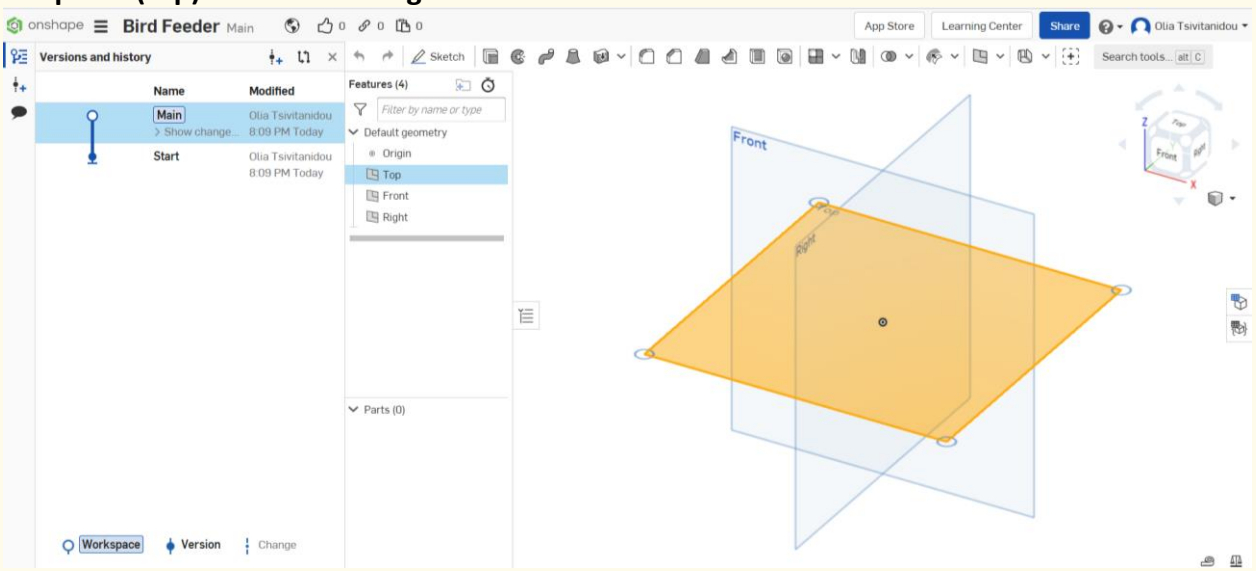
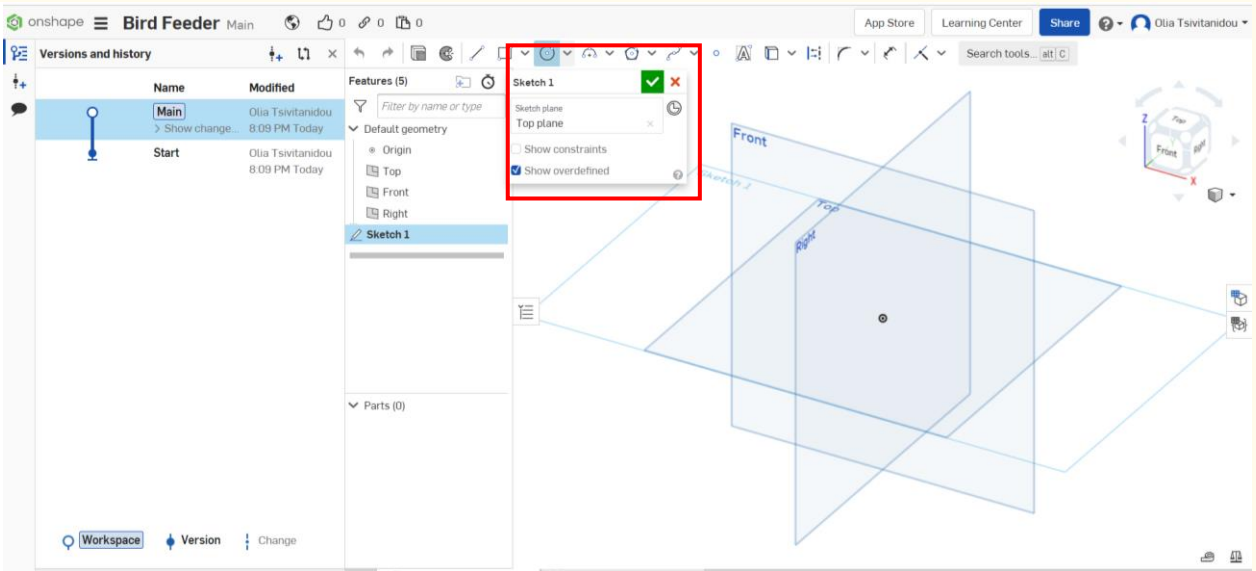
Step 4 **Start with the creation of the main cylinder. This can be done in multiple ways: we can create a circle, extrude it up, draw a rectangle, revolve it, etc. We will not worry about any dimensions right now; we will take care of those later. Select the plane (top) to start drawing.**



Step 5 **Select sketch, then circle.**



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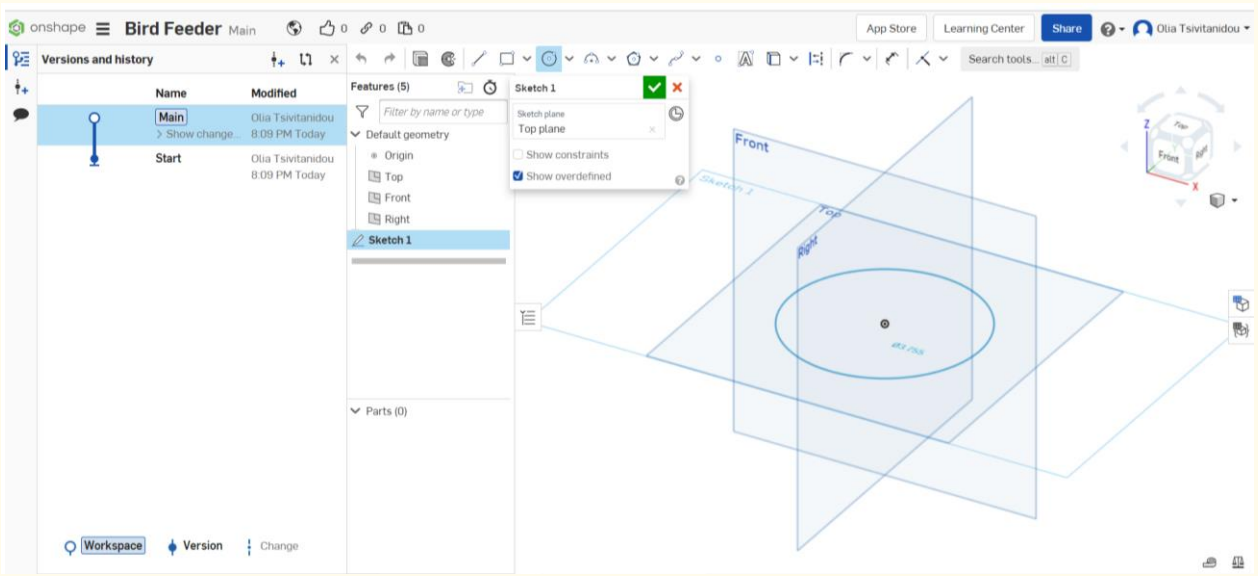
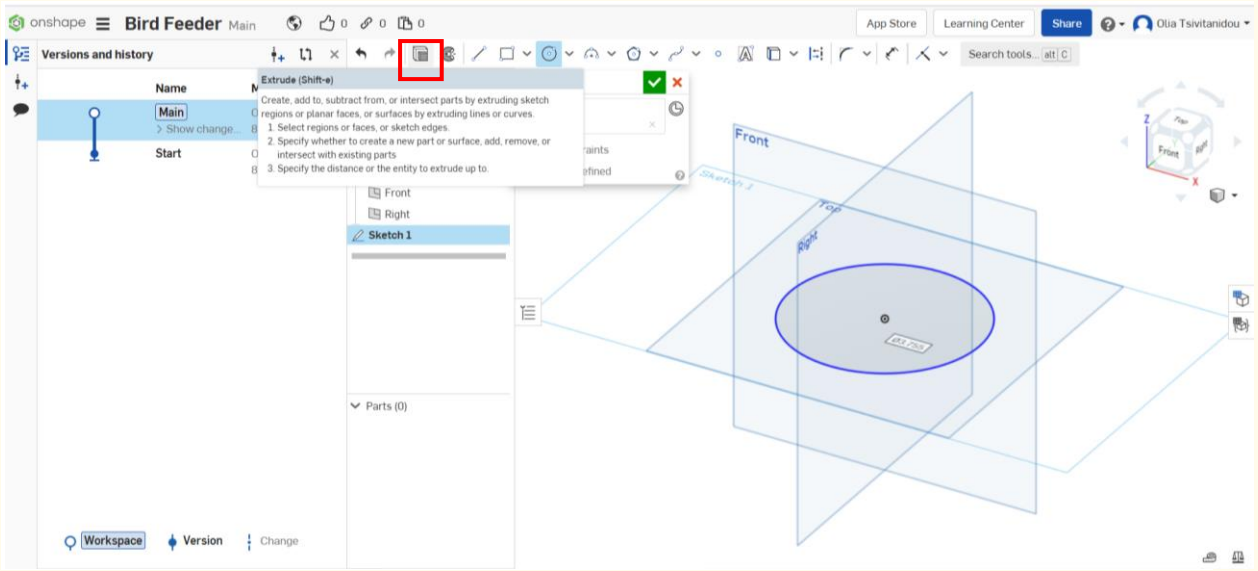
Step 6

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



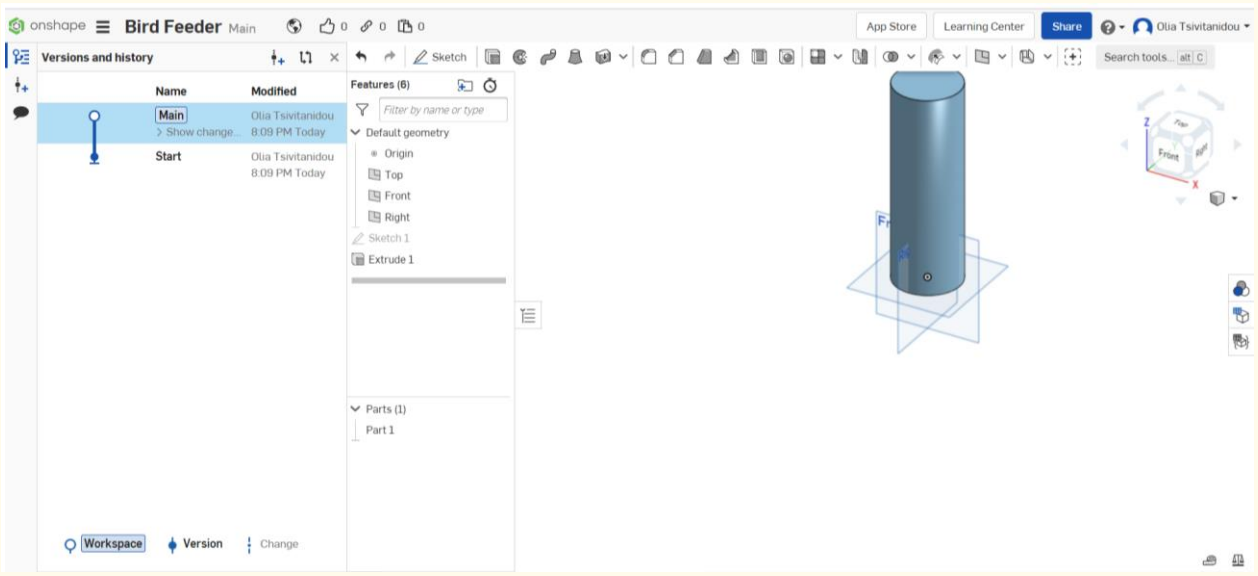
Step 7

Select extrude.



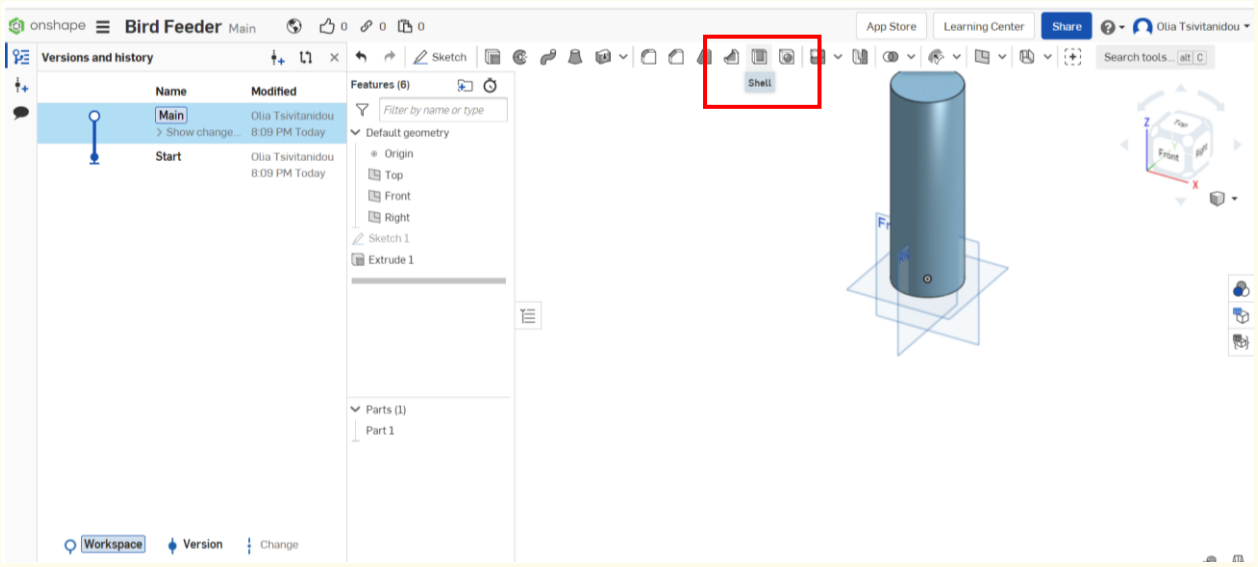
Step 8

Extrude that circle.



Step 9

Select Shell.



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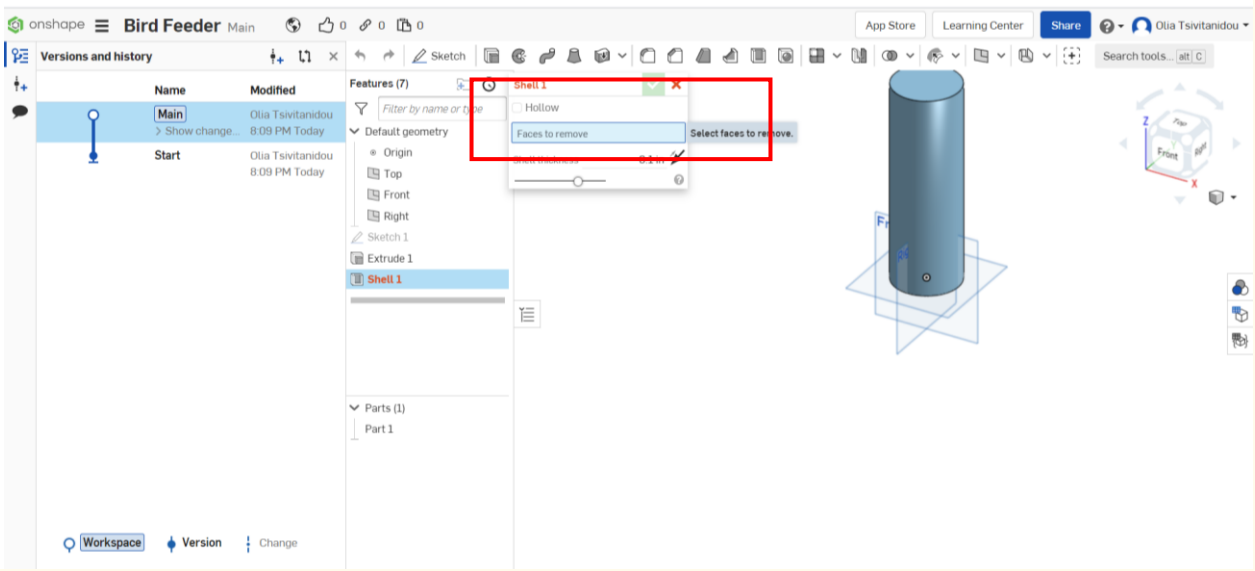
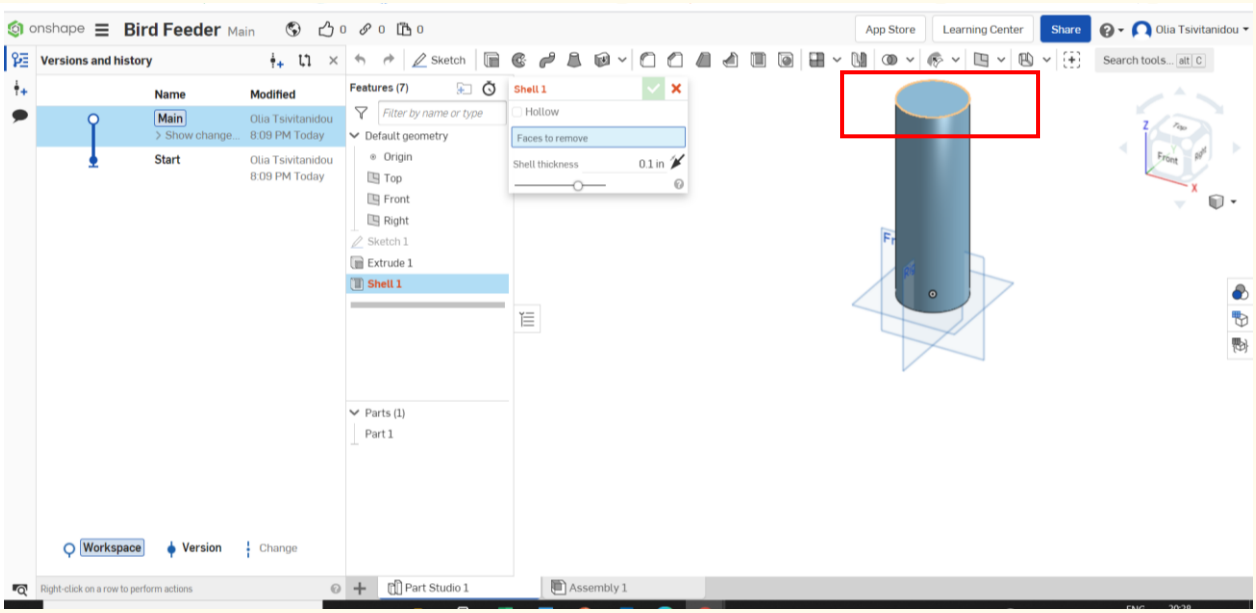
Step 10

Select Face to remove.



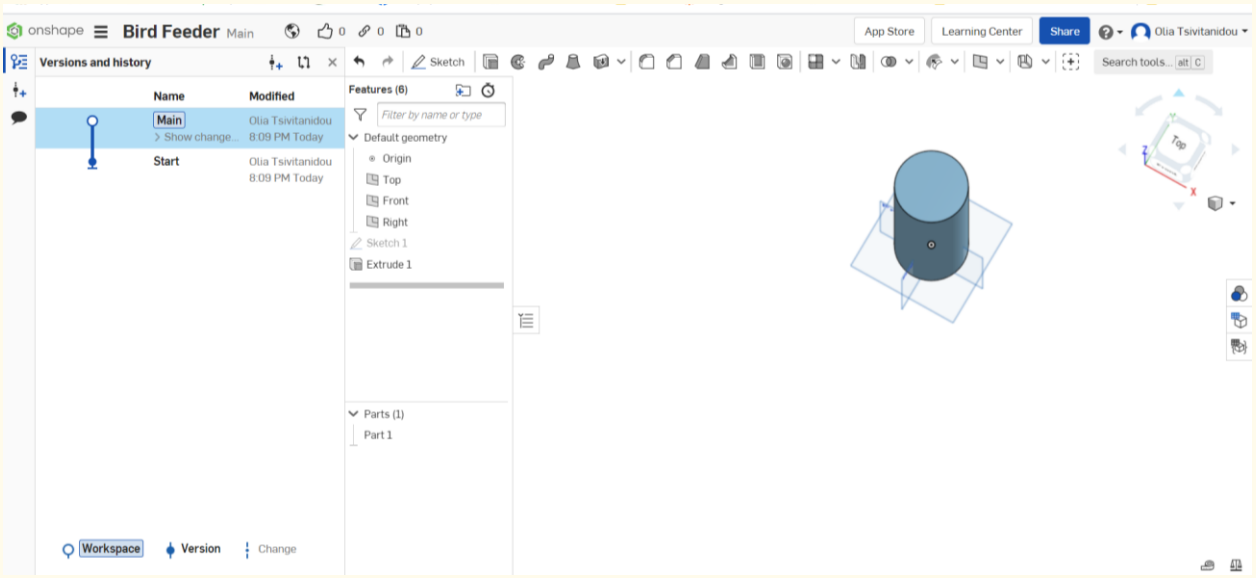
Step 11

Click on the top cylinder.



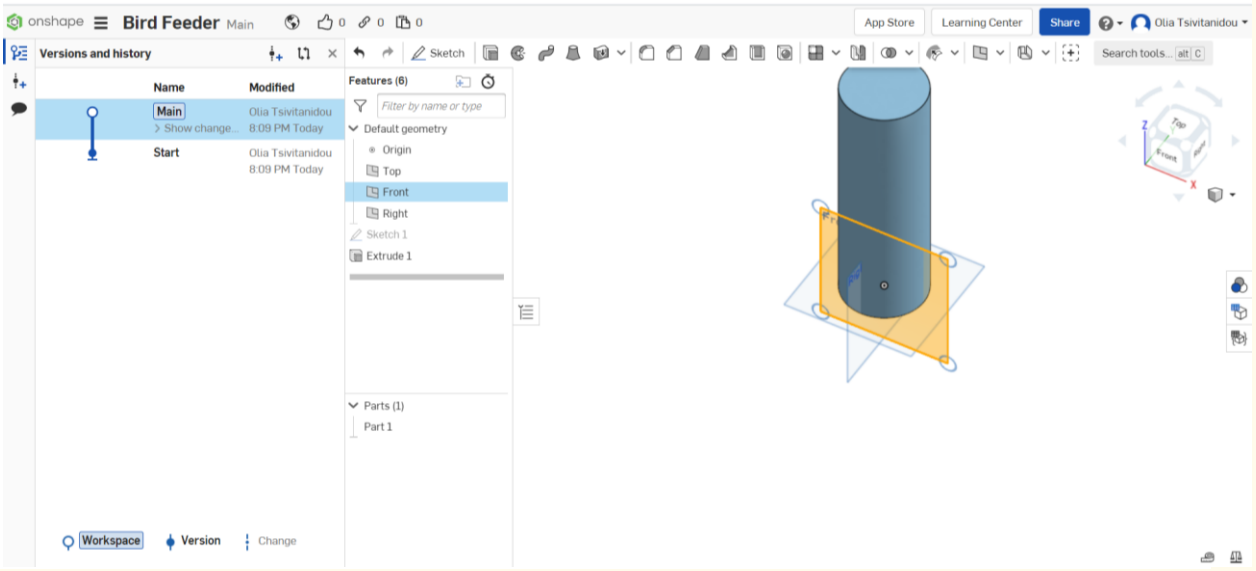
Step 12

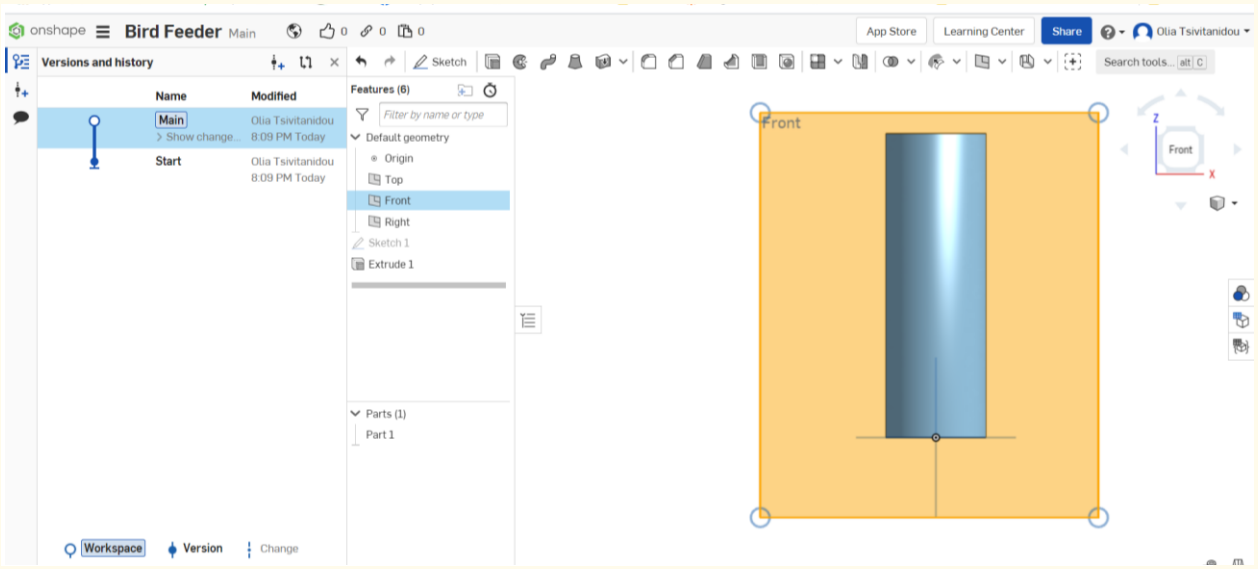
The top face has been removed. This is the result.



Step 13

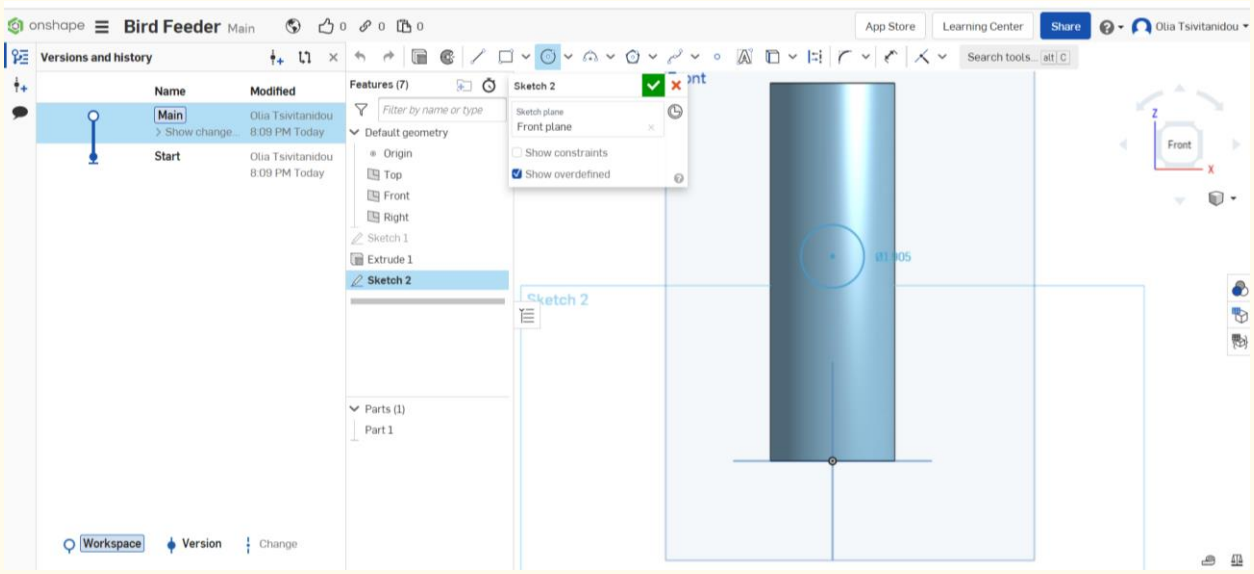
Next, we are going to put the large holes on the cylinder surface. Select the front plane and then sketch.





Step 14

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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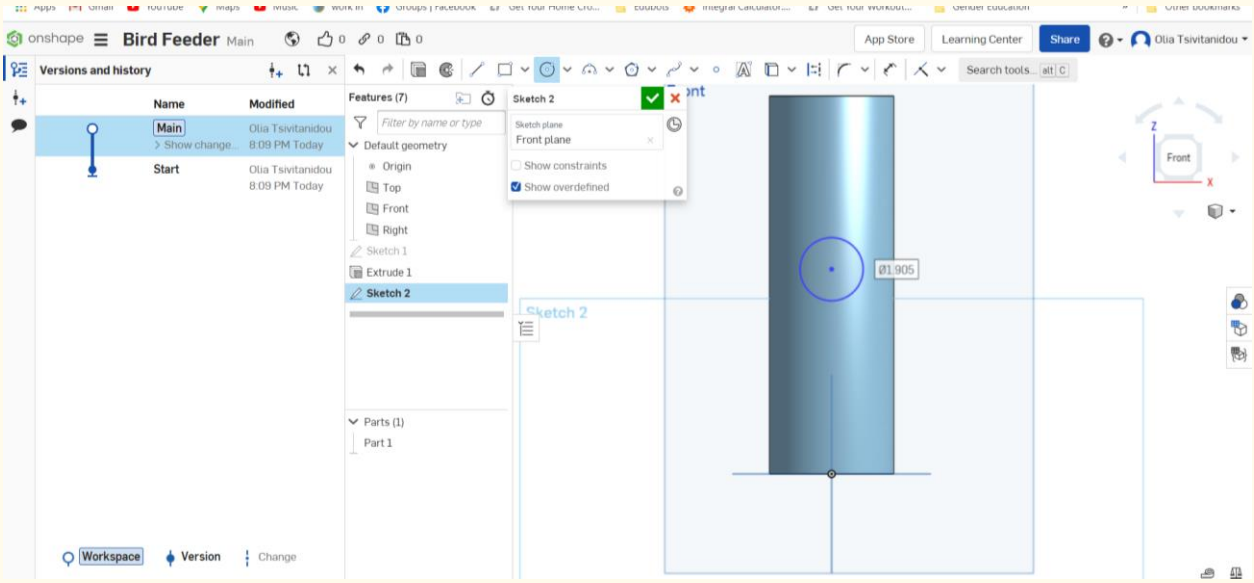
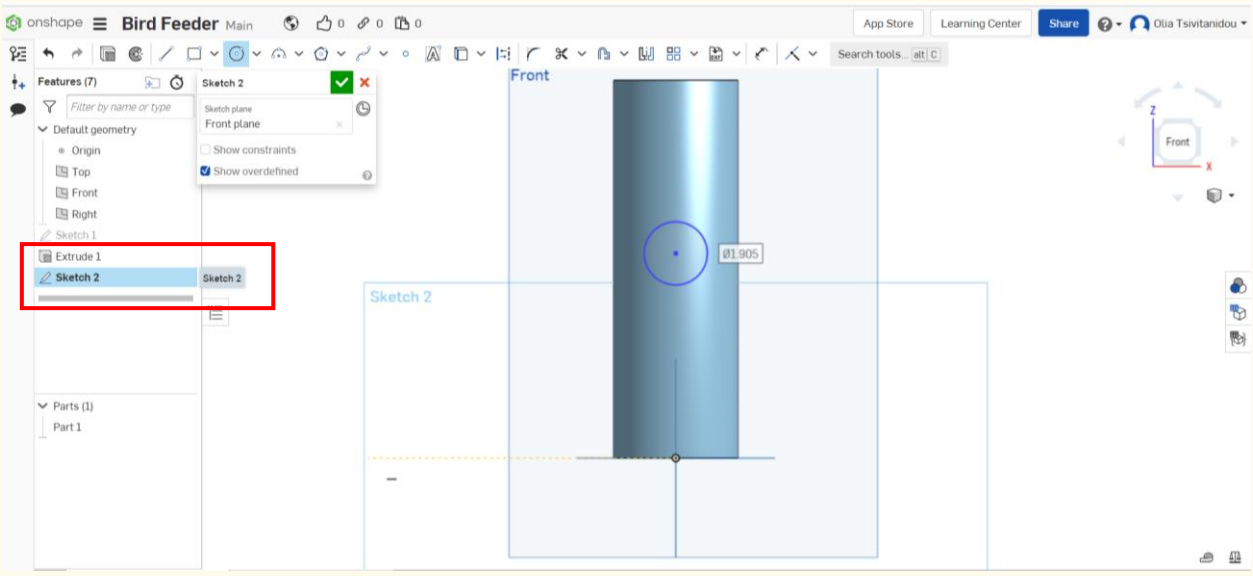


Step 15 **Go on the right side of your screen, where it says sketch 2 (see red box), and pick it from there.**



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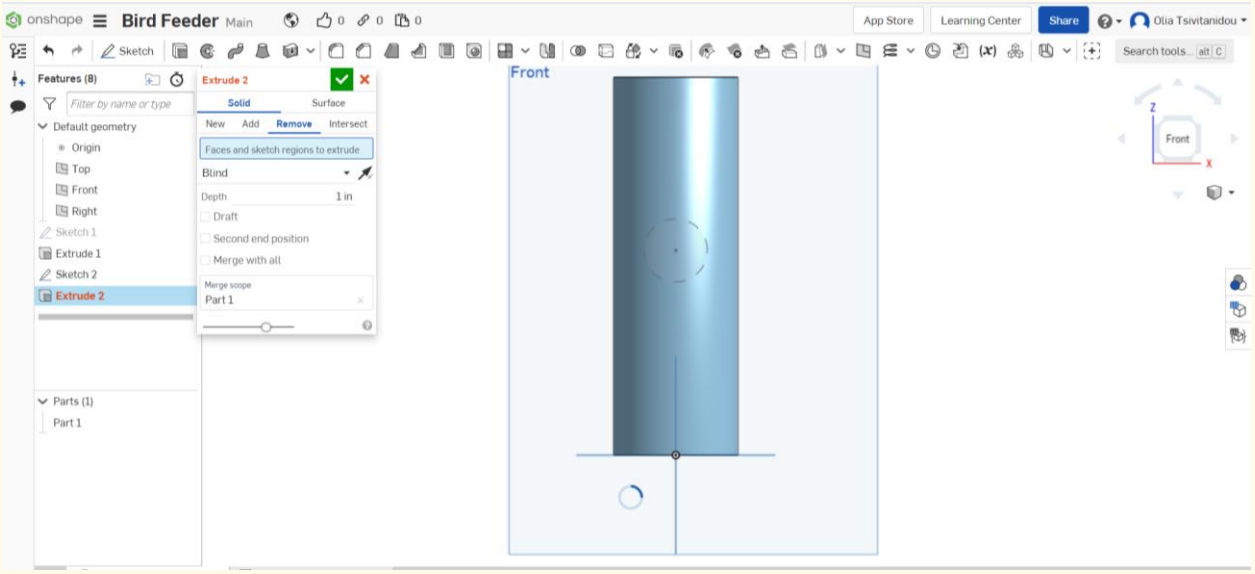
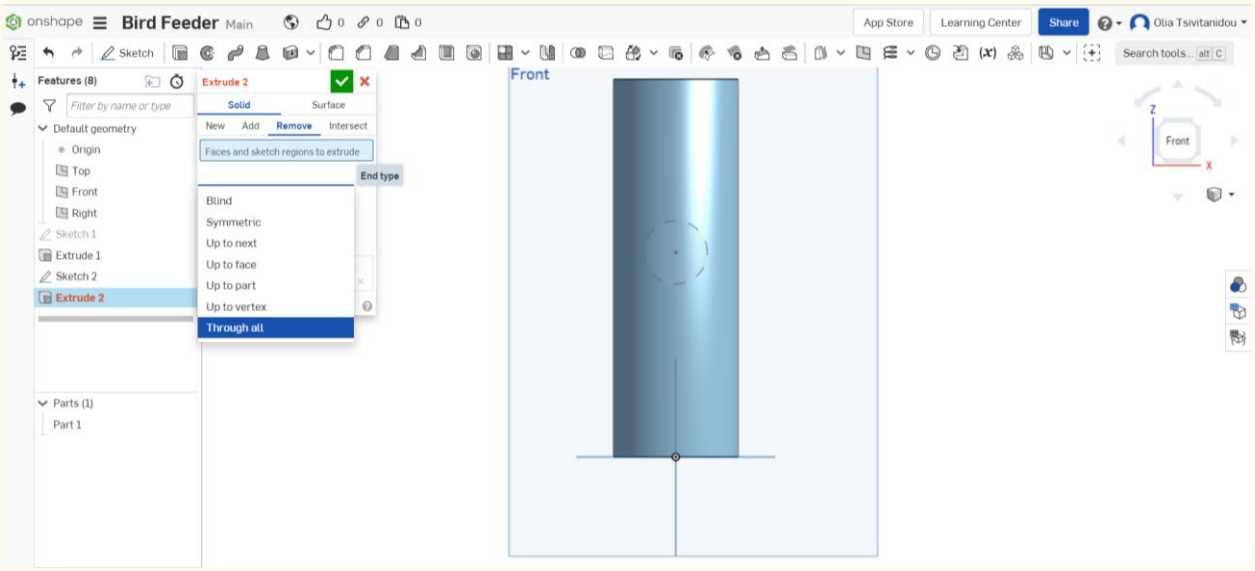
Step 16

We want this to be removed, so click on remove, as illustrated below.



Step 17

Select “through all.”



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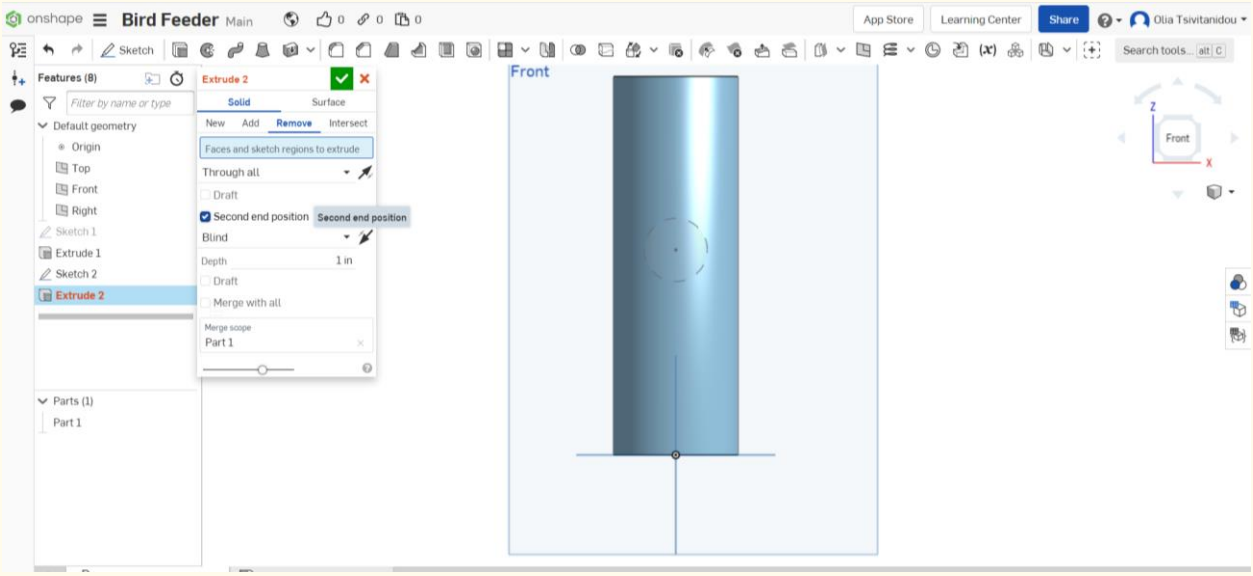
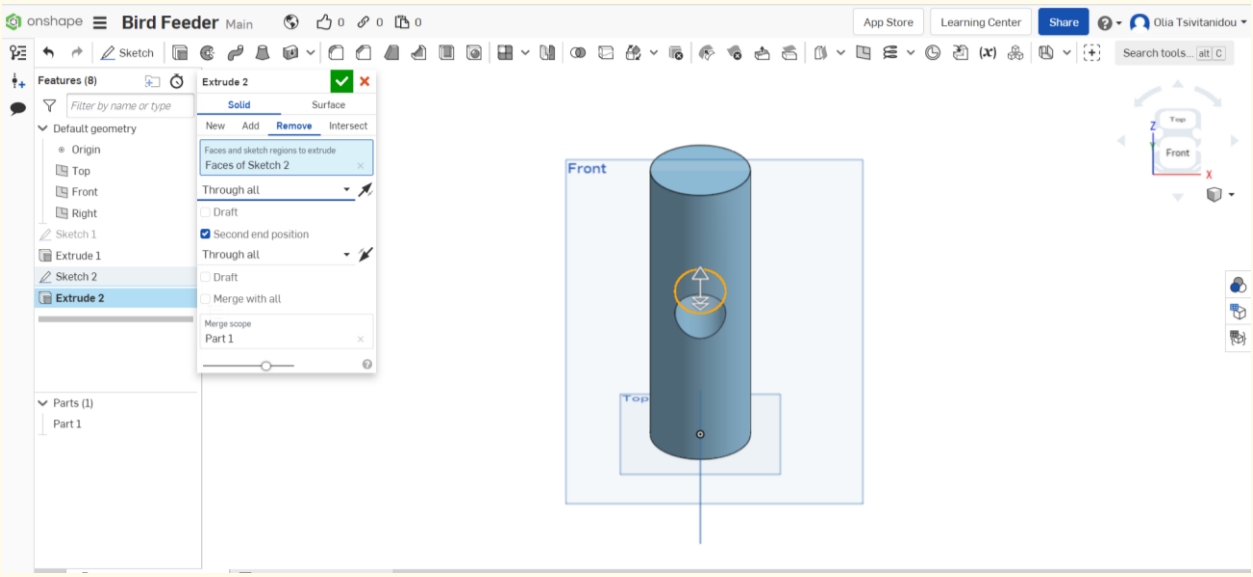
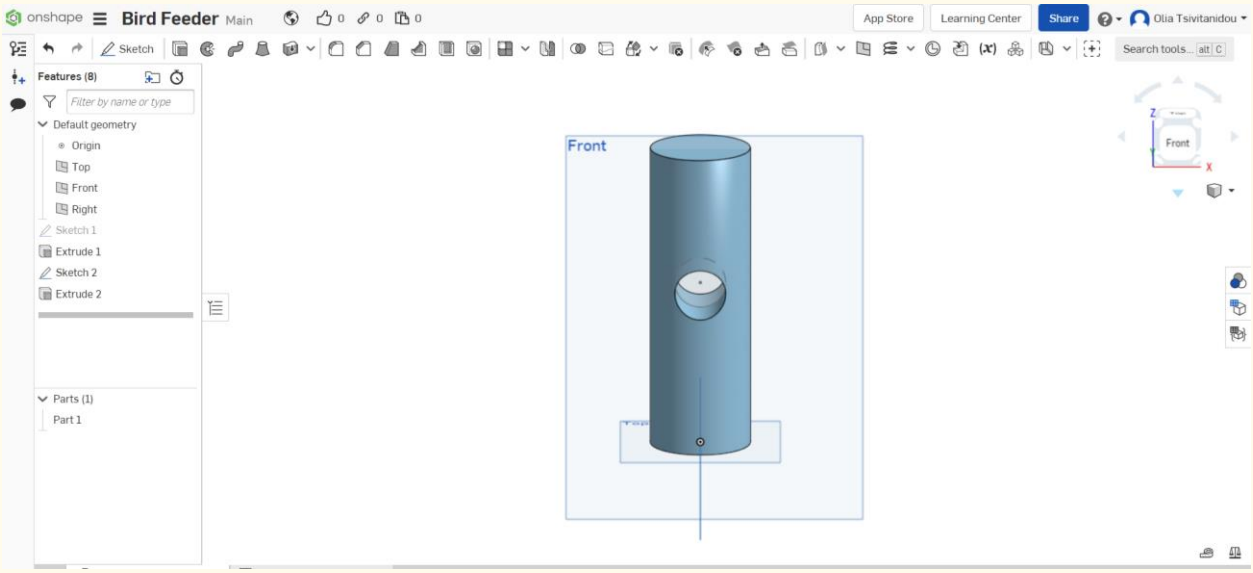
Step 18 **Select “second end position” to have the same exact hole on the opposite side of the inner cylinder surface.**



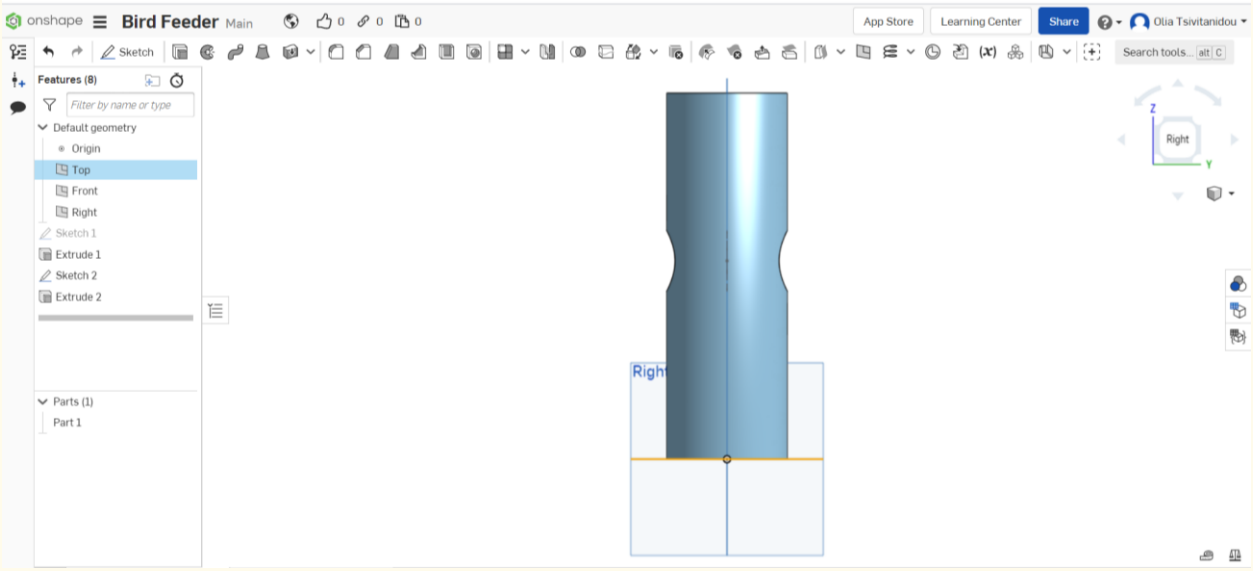
Step 19 **Select “second end position” and make that also a “through all.”**



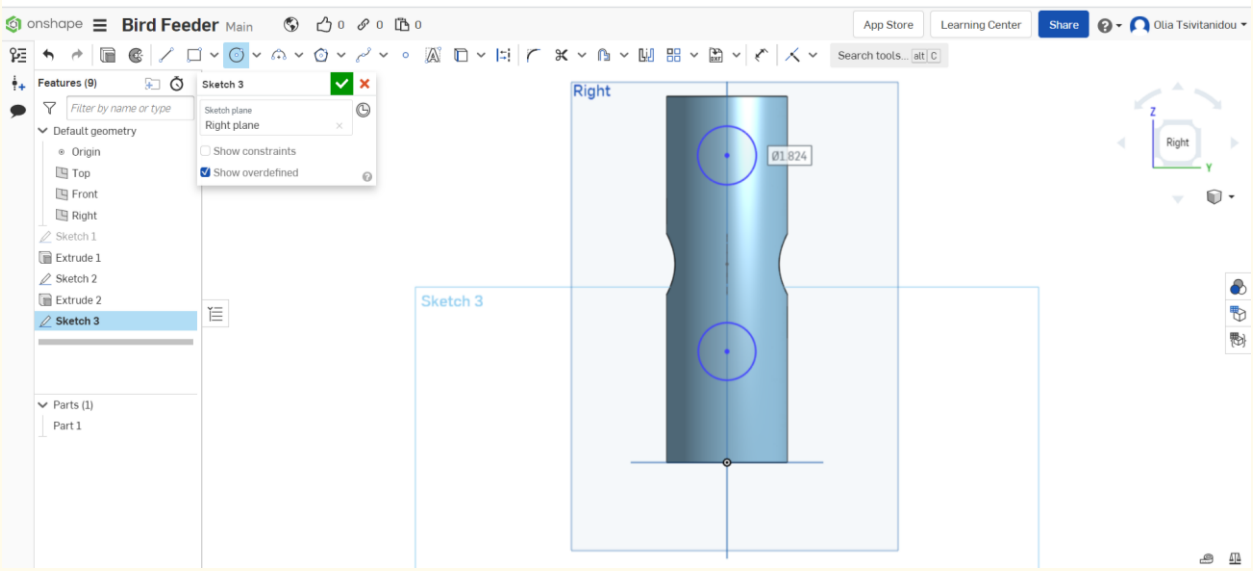
Step 20 **So, now we are one hole that goes all the way through the front face.**



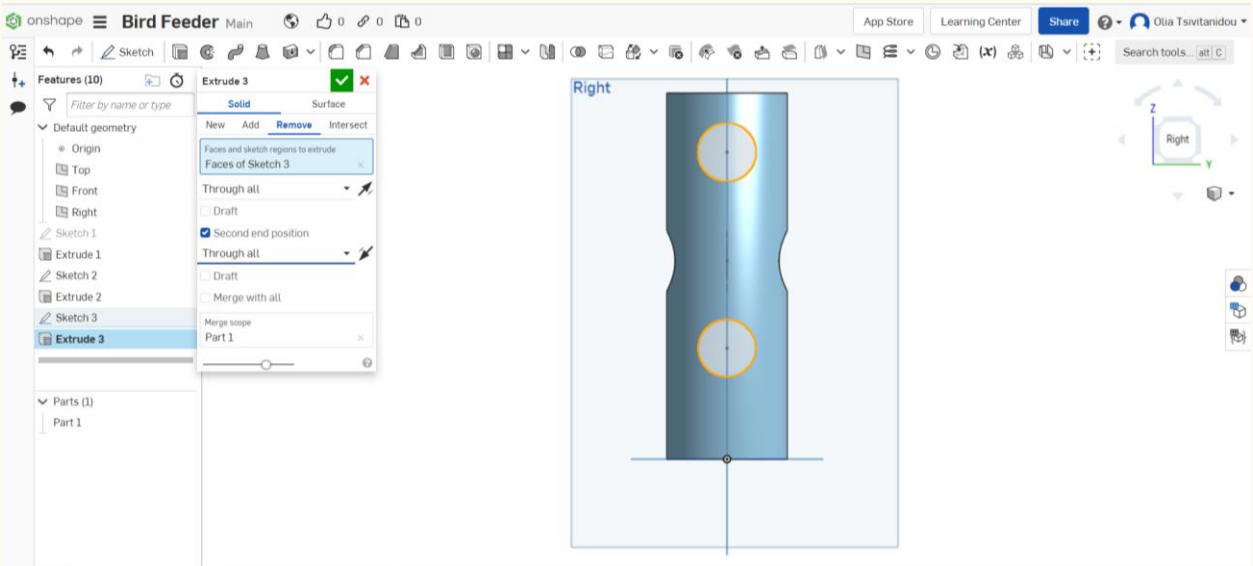
Step 21 **Let's go-ahead to create two more holes. We are going to do a new sketch on the right side face. Click on the right side plane, then sketch and circle.**



Step 22 **Let's go ahead and draw two new circle sketches, in the same way as done previously.**

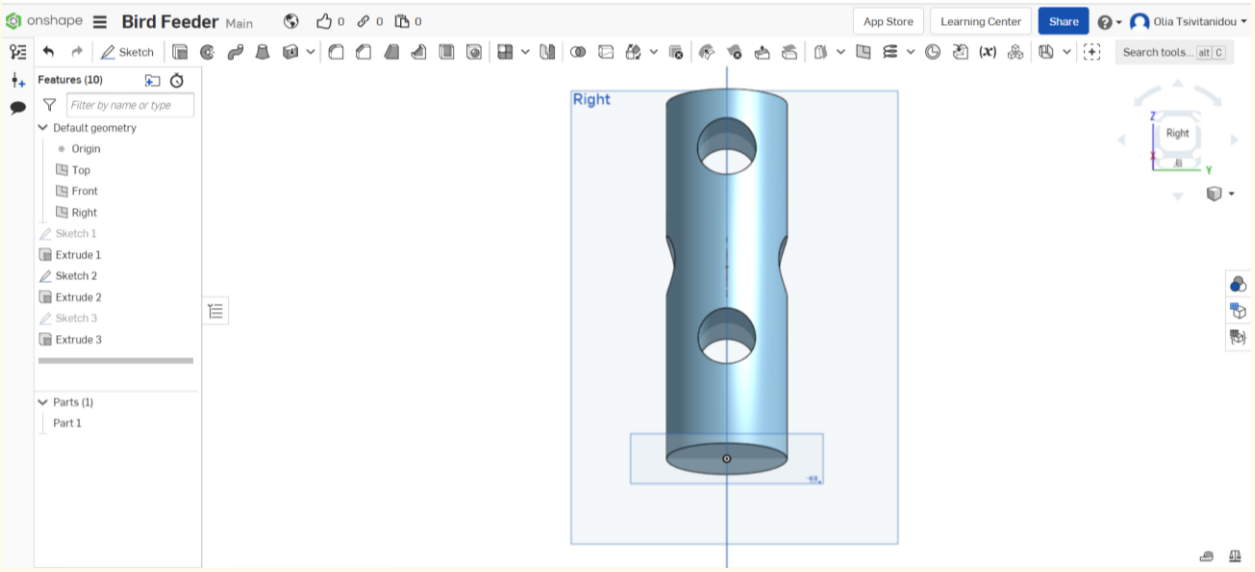


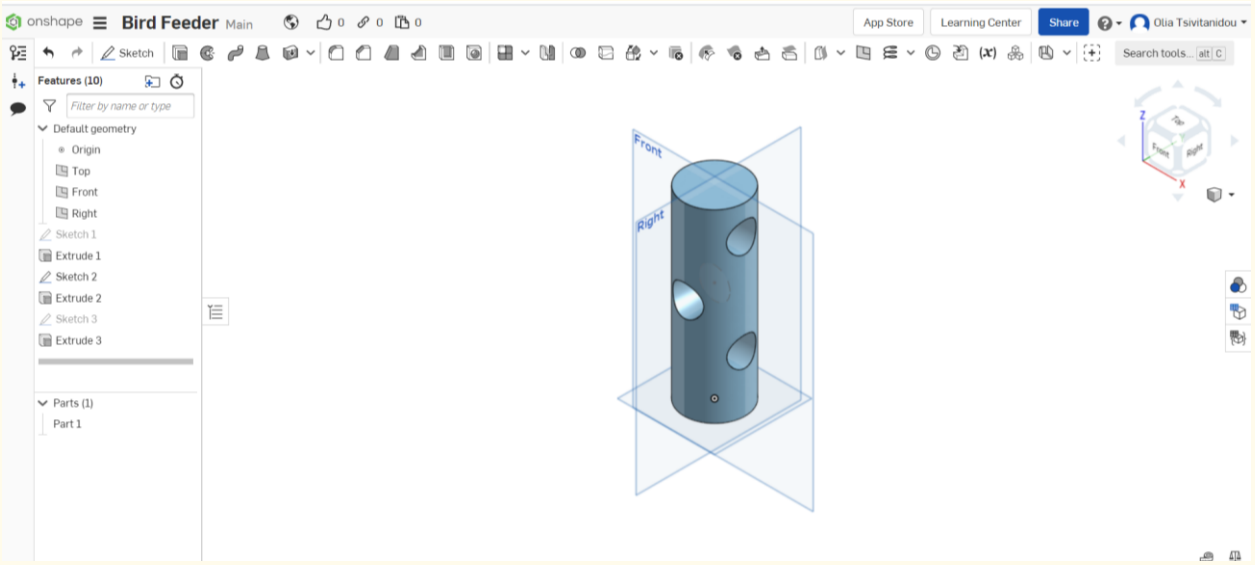
Step 23 **As before, we will extrude by selecting first by clicking on sketch 3, then extrude, then remove, through all, second end position, and again, through all.**



Step 24

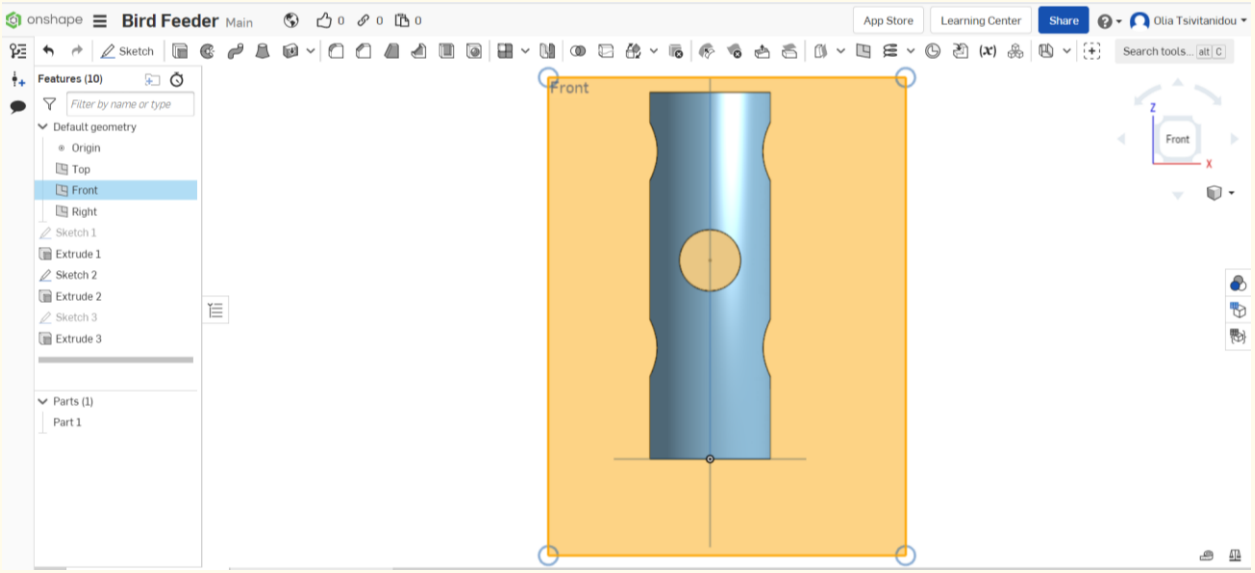
And now we have two more holes on the cylinder's surface.





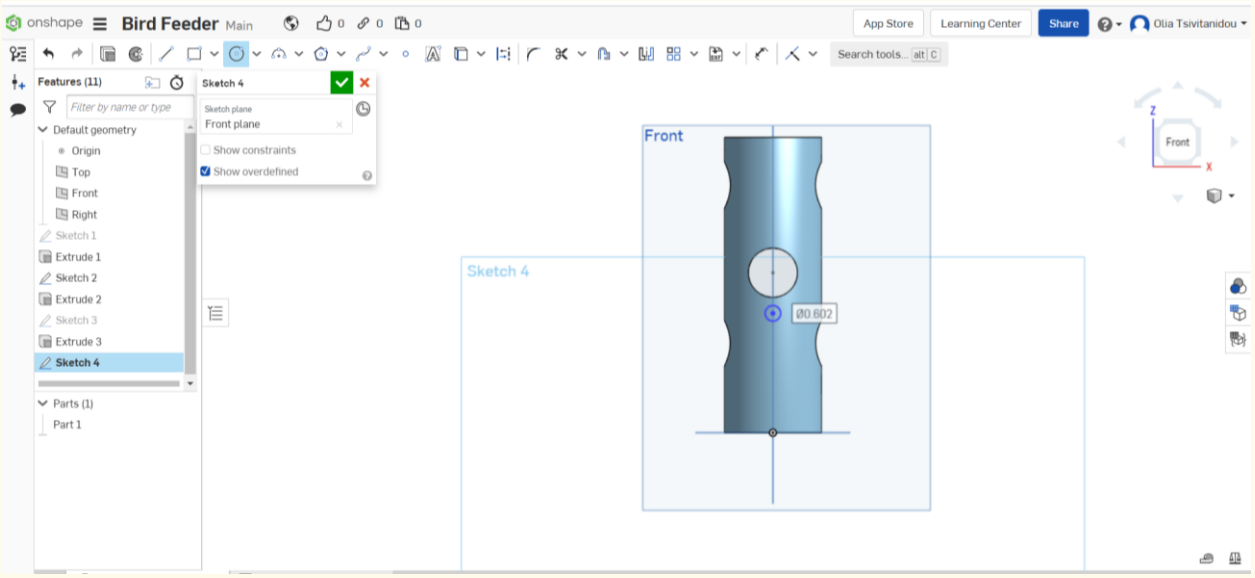
Step 25

Now, we go ahead and do a sketch on the front and a small perch on there.

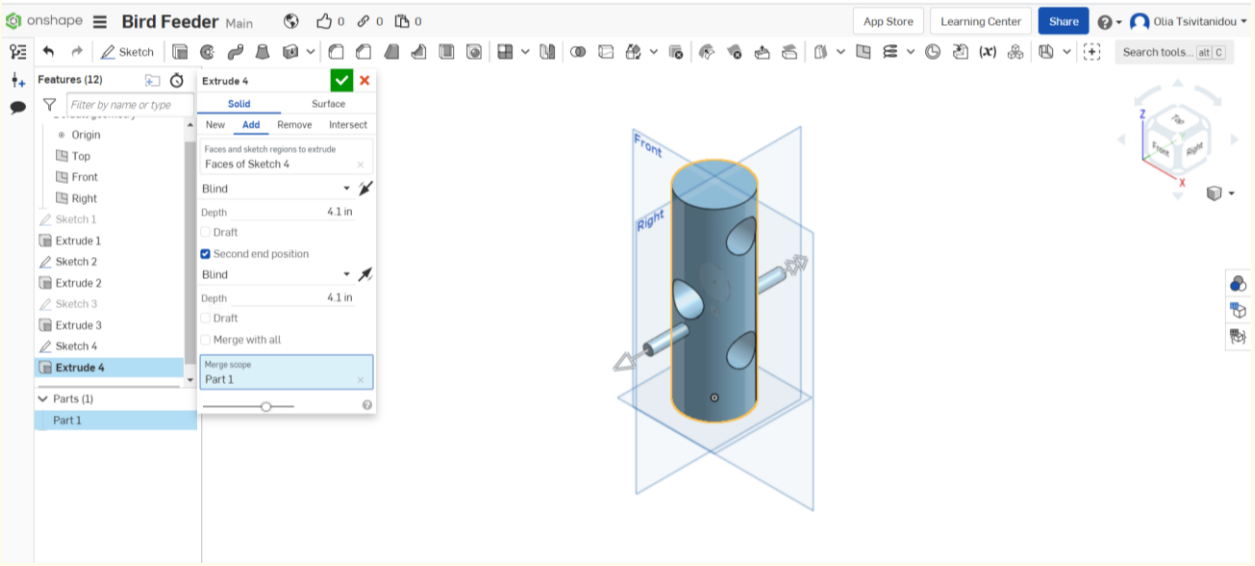


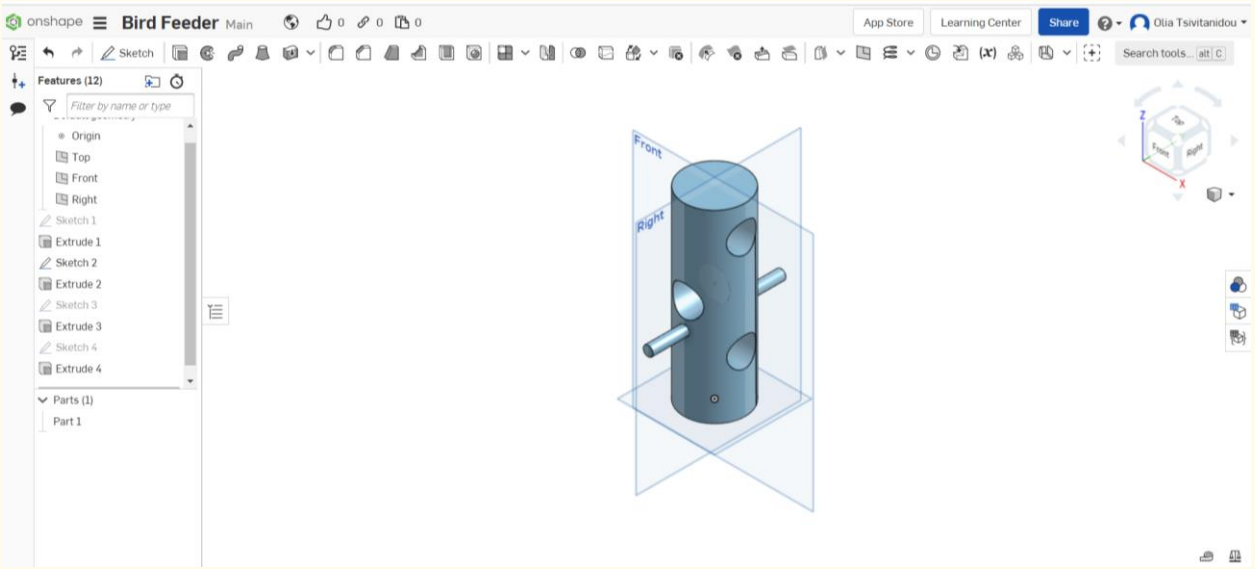
Step 26

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



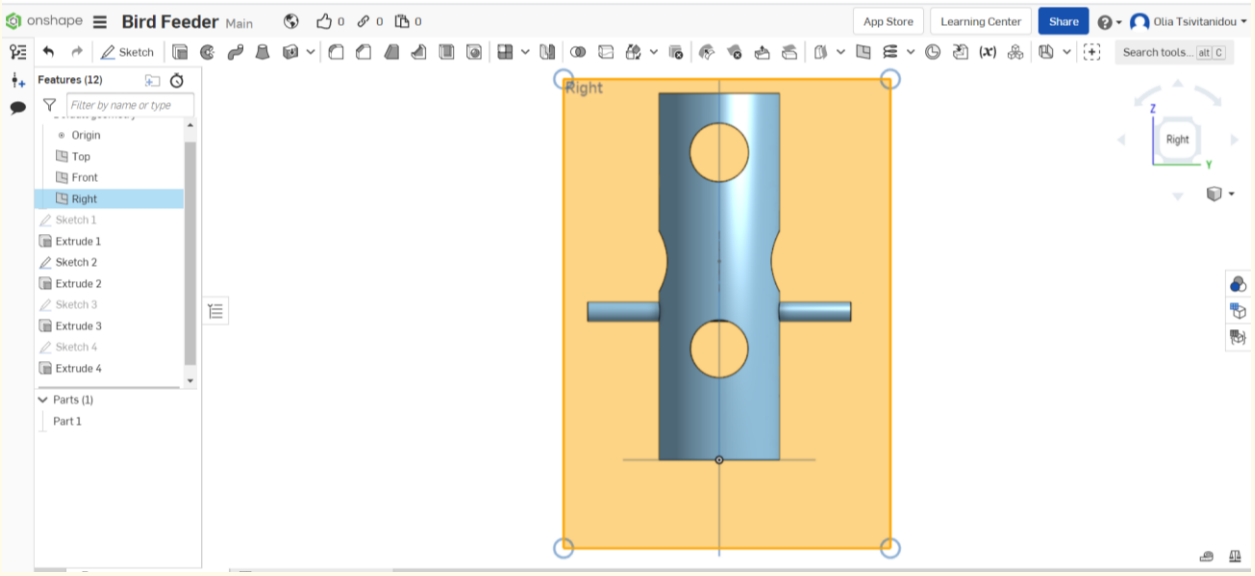
Step 27 **Now we are going to extrude. Select extrude and second end position. Keep the same distance (depth) for both perches.**

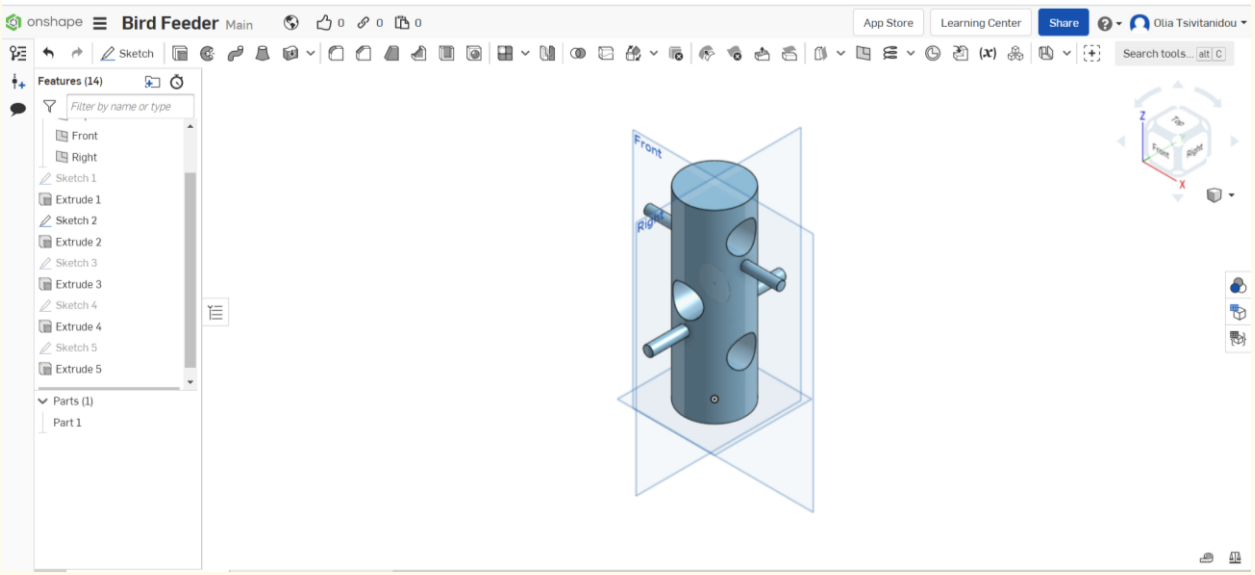




Step 28

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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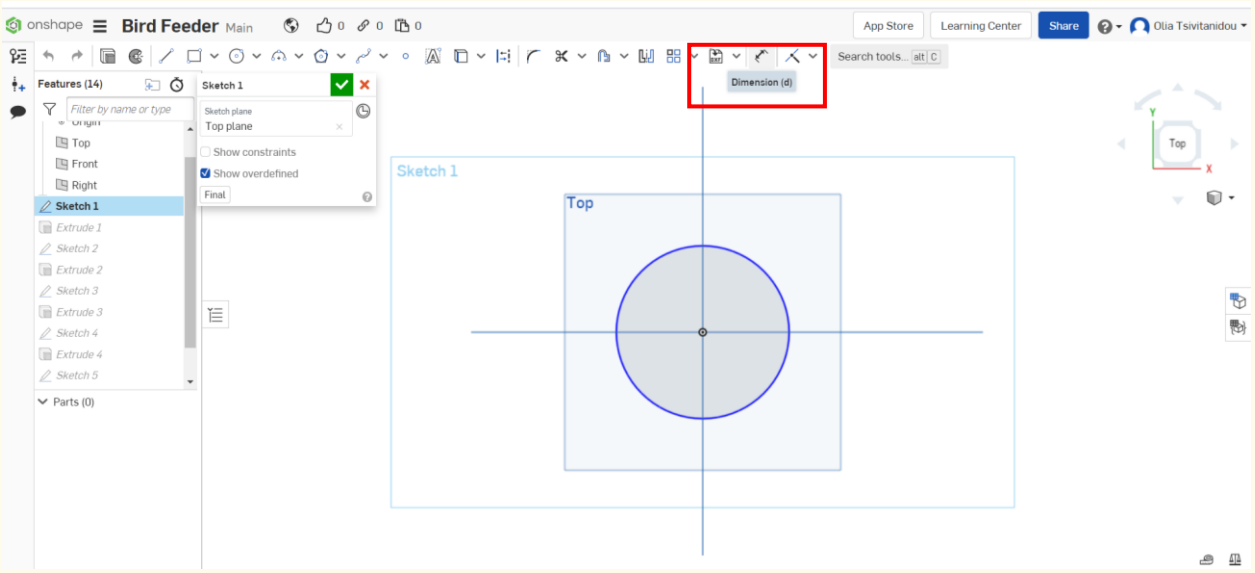
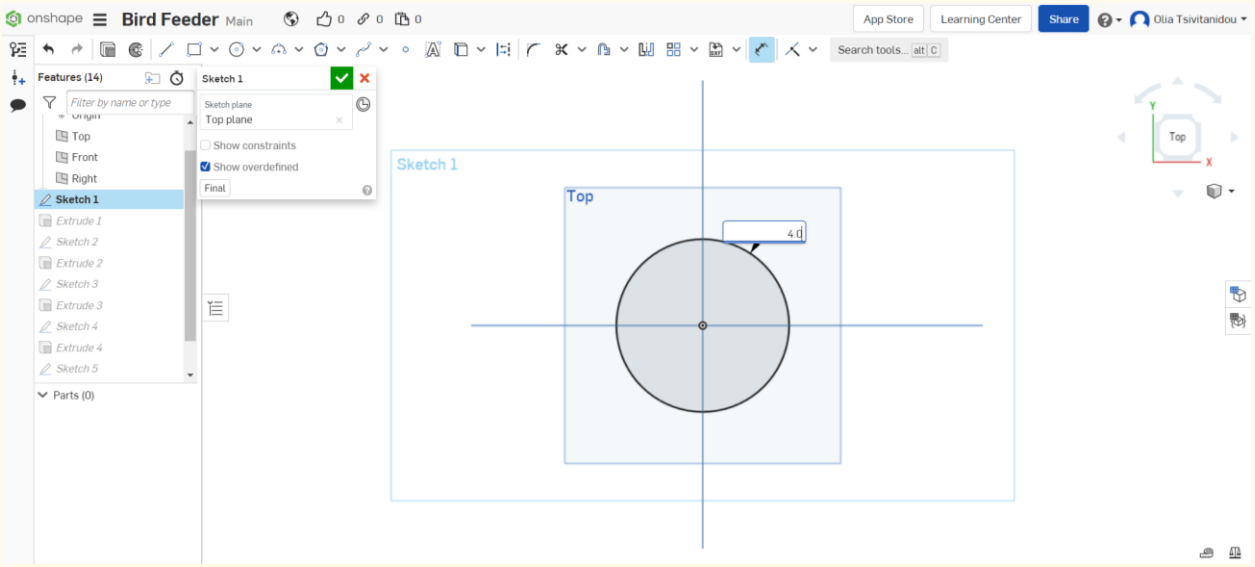
Step 29

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).

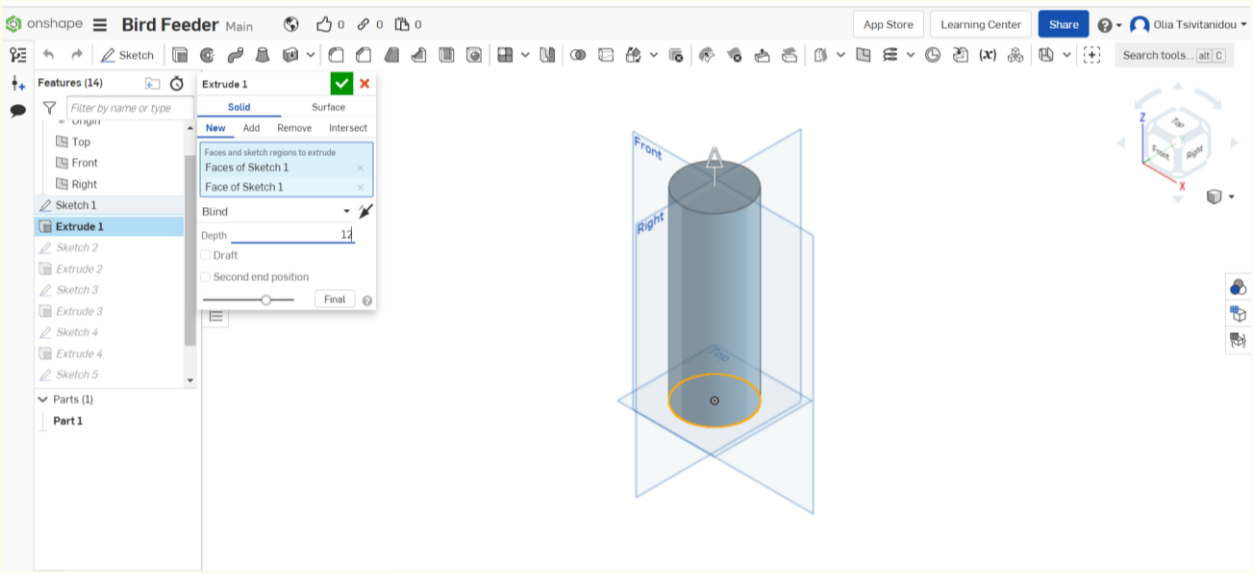


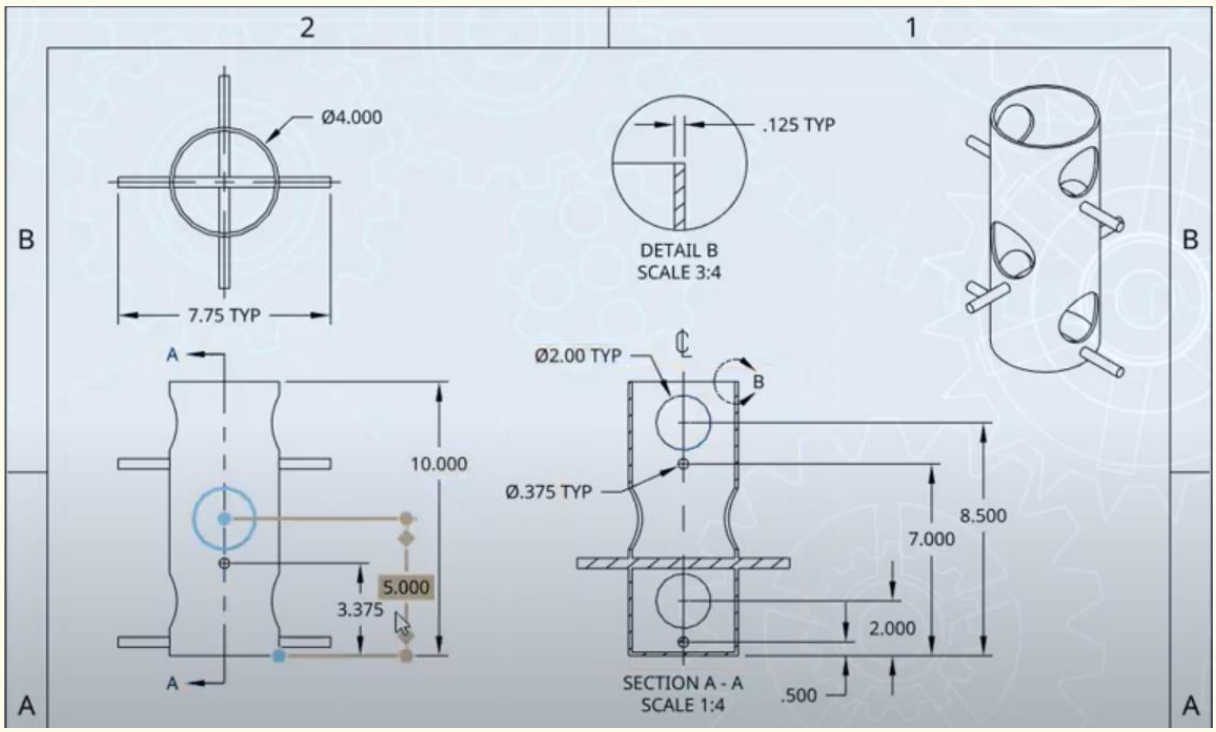
Step 30

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

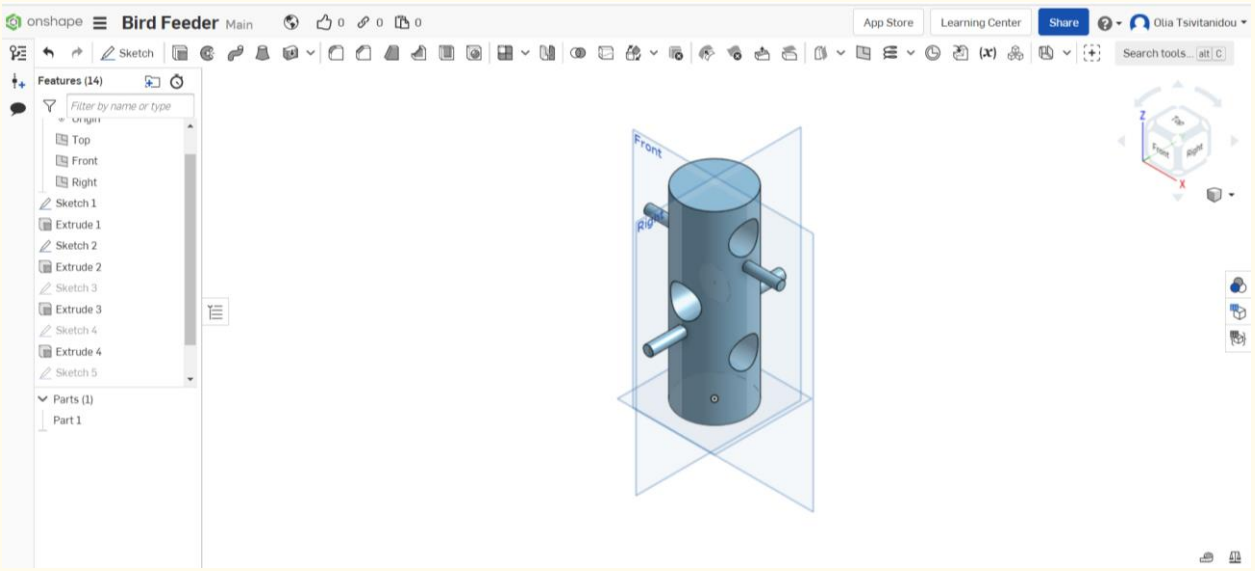


Step 31 **Set realistic sizes. Repeat step 30 for resizing all sketches to give the real dimensions (see figure below). In the same rationale, we can correct extrusion. For instance, select extrusion 1 from the features list. We set the depth into 12 inches.**





Step 32 **And finally, 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 competencies:**
    - 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**

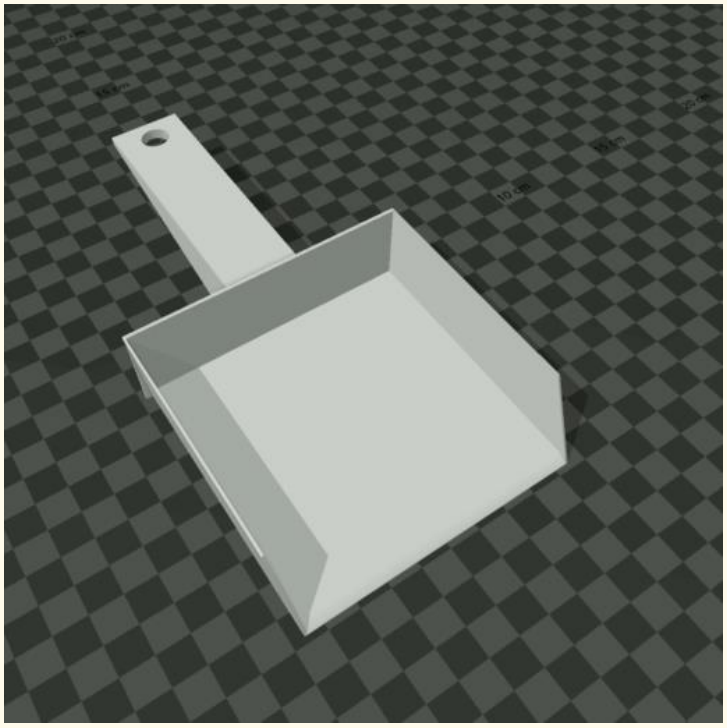
This project consists of the 3D design of a shovel, then we will present the step-by-step process of its elaboration.

PRELIMINARY STEPS:

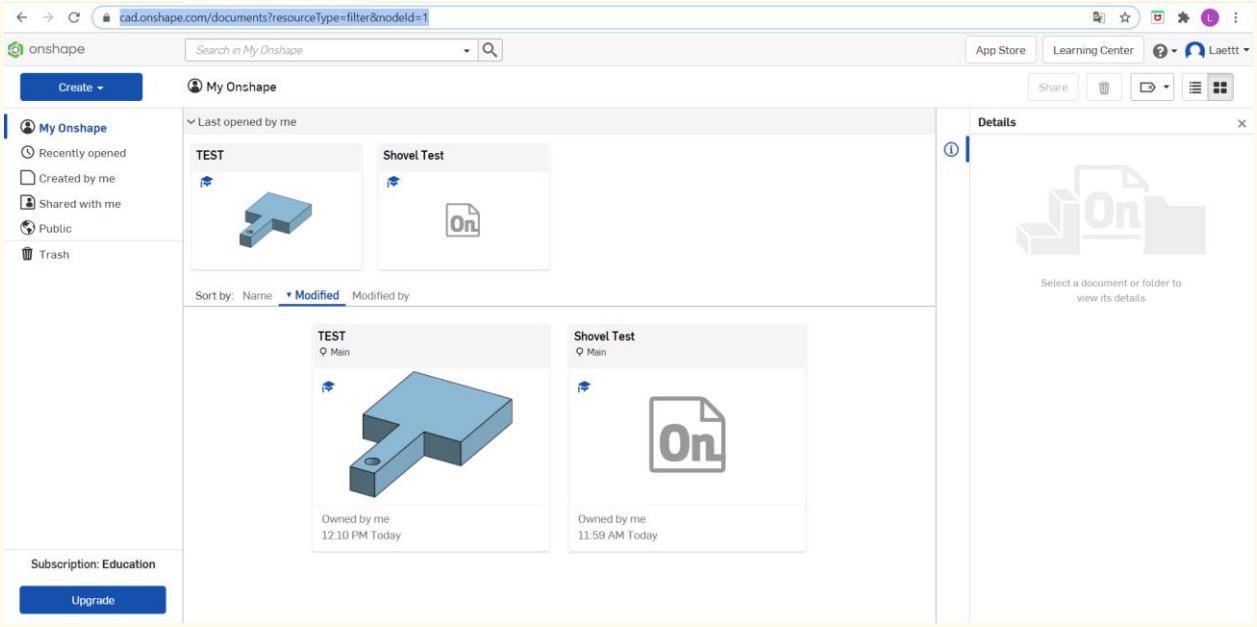
It is important to have an idea of the final object and make sure measures are relevant; you need to do some small research.

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

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

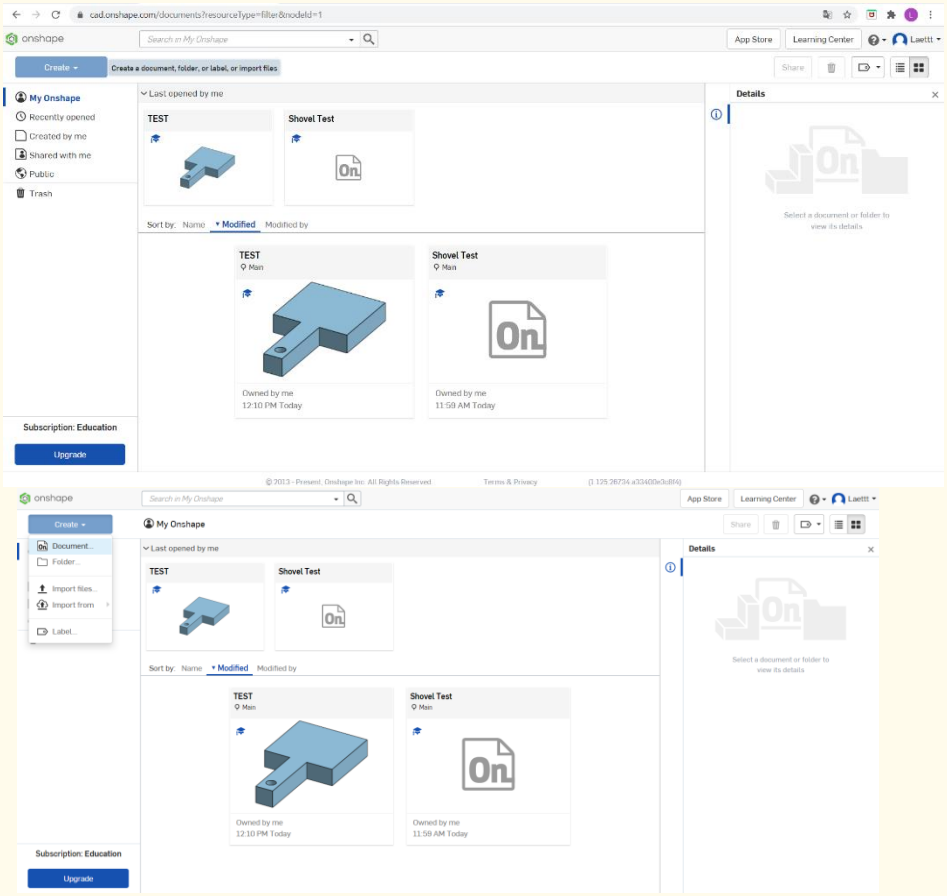


Step 1 **Open Onshape.**
https://cad.onshape.com/documents?resourceType=filter&nodeId=1



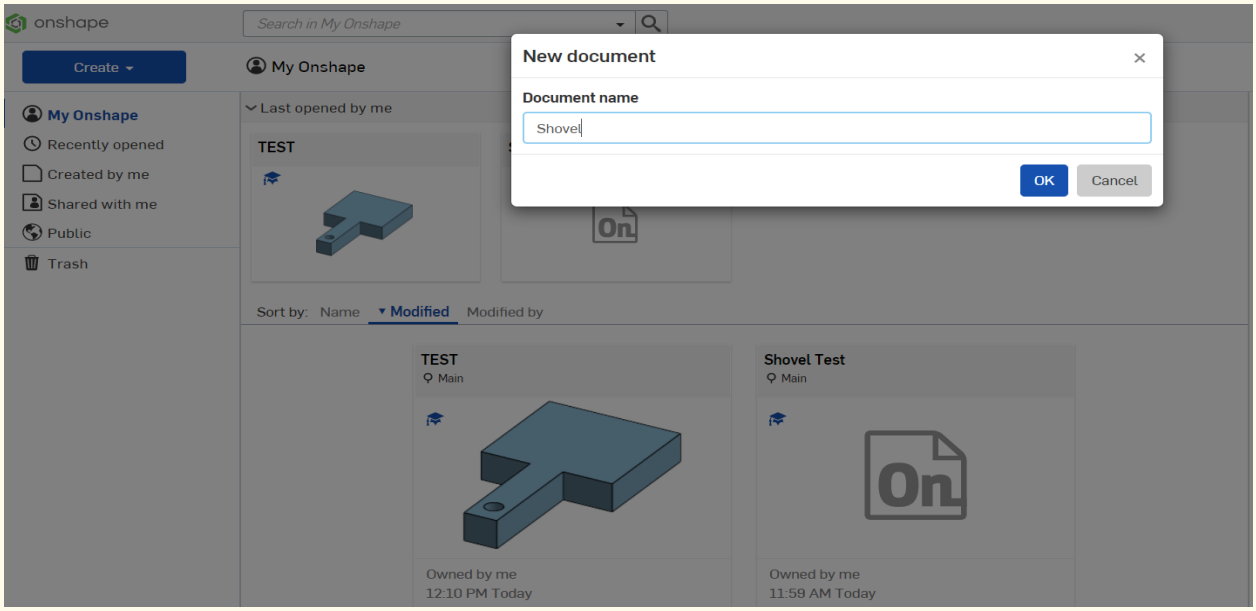
Step 2

Create a document.



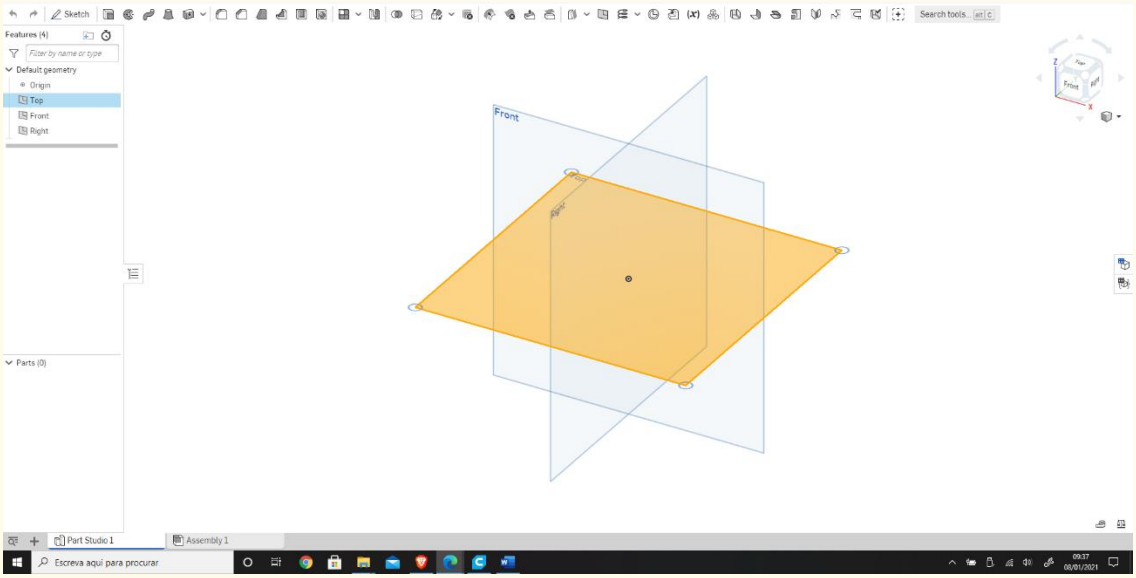
Step 3

Give a name to your document such as Shovel.



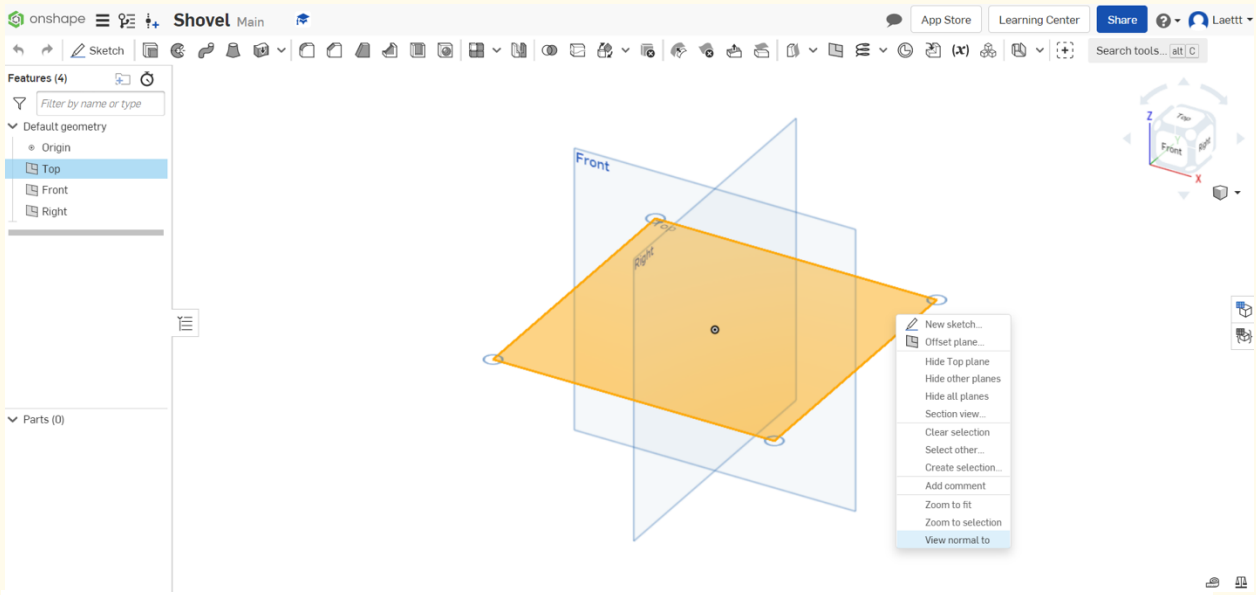
Step 4

Select the plane (top) to start drawing.



Step 5

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



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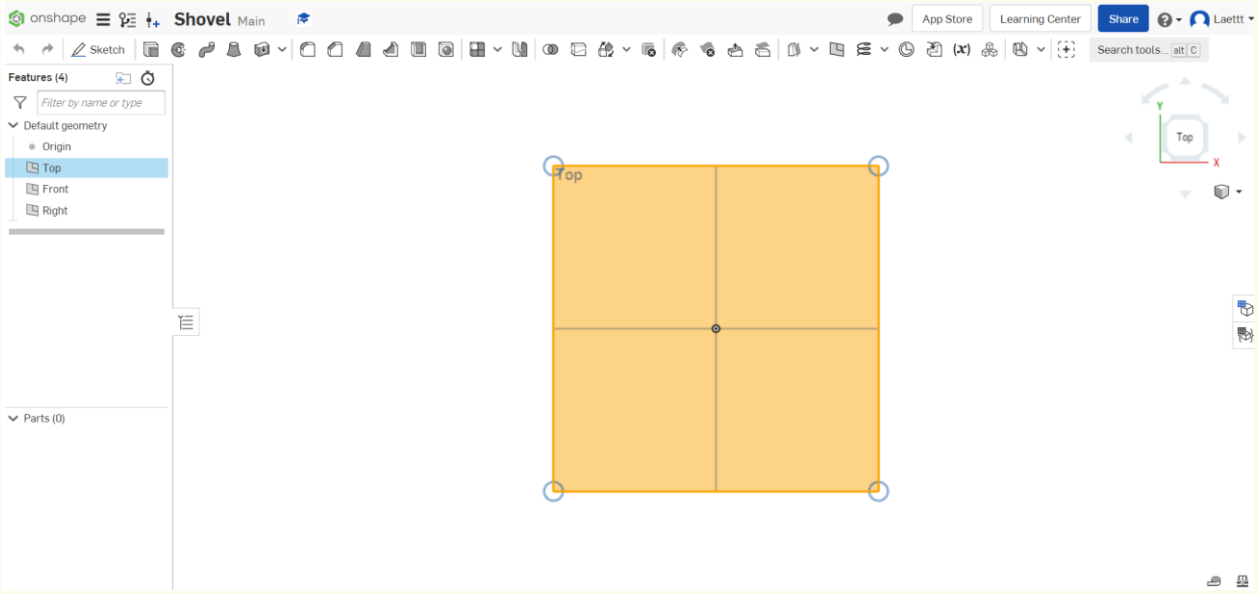
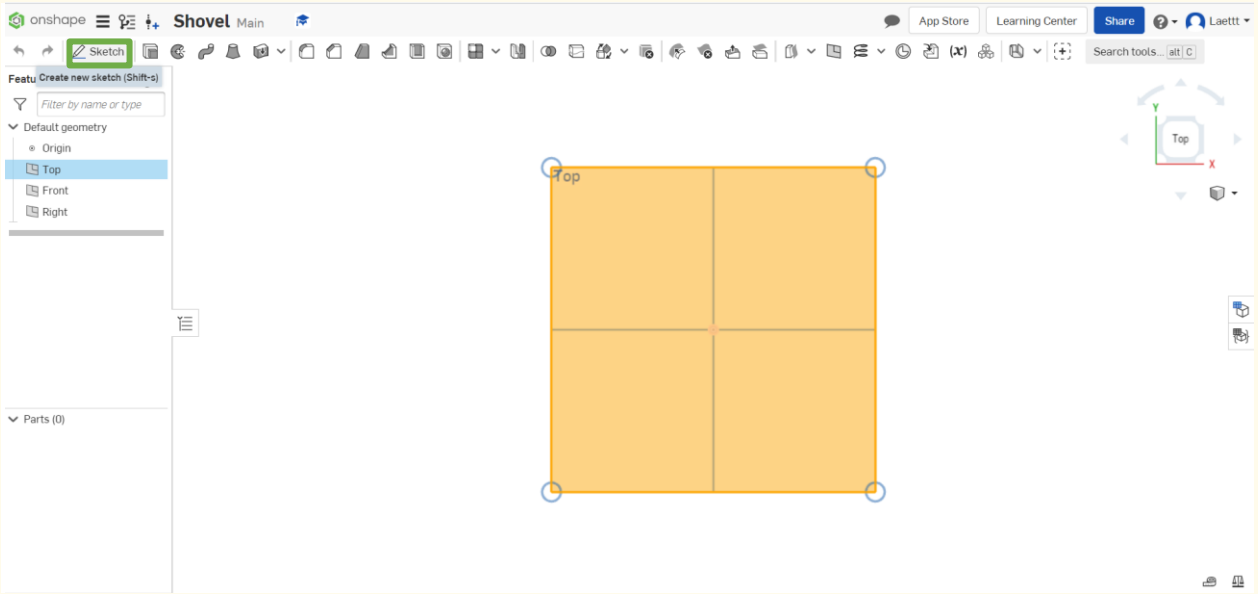


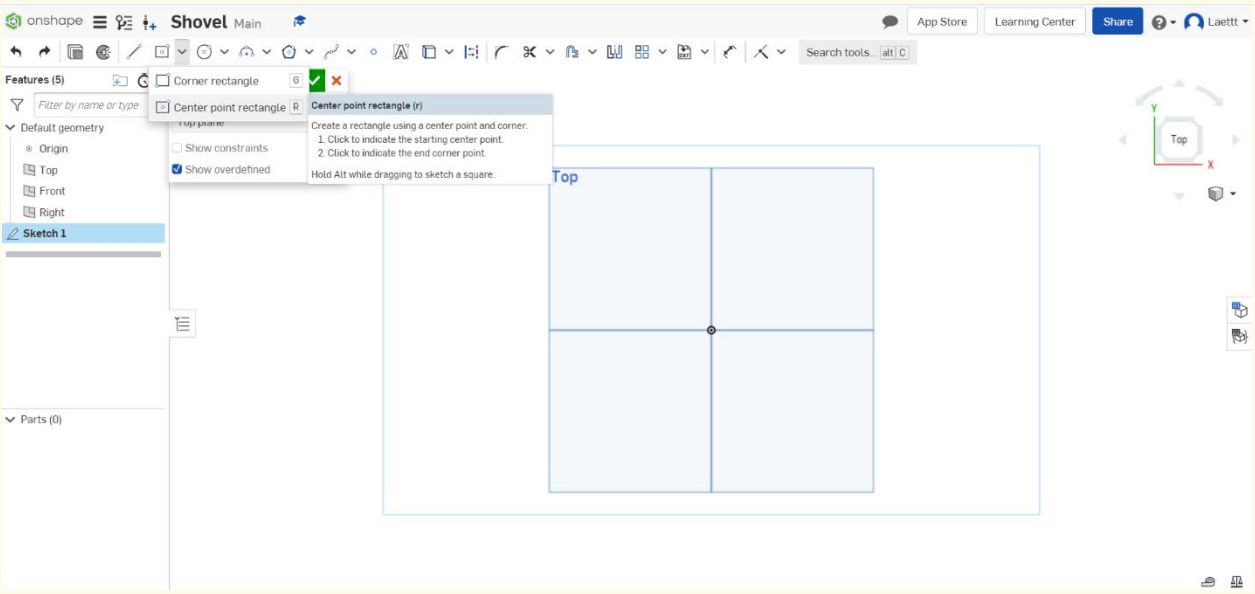
Step 6

Click on Sketch.



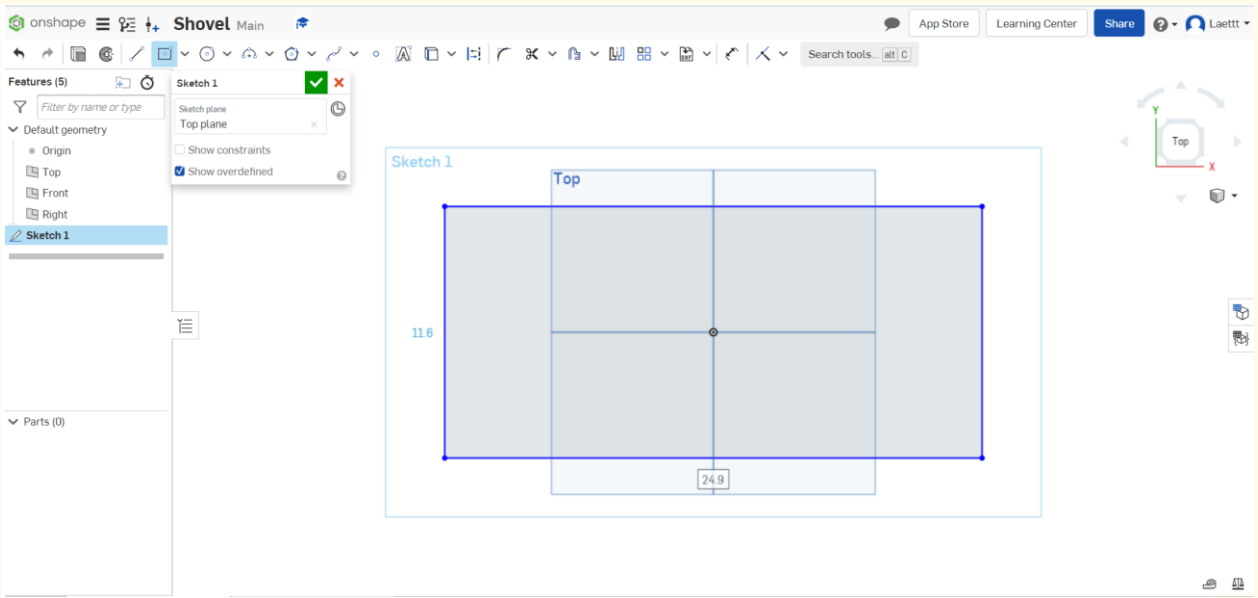
Step 7

Select the center point of the rectangle to draw.



Step 8

Draw the rectangle.



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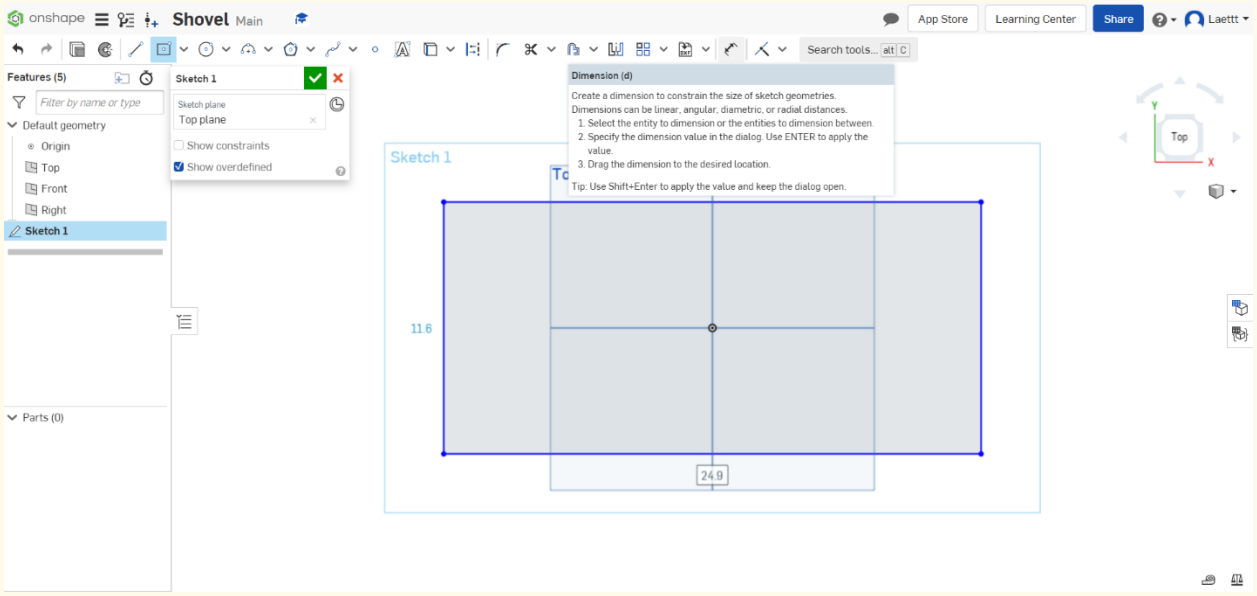
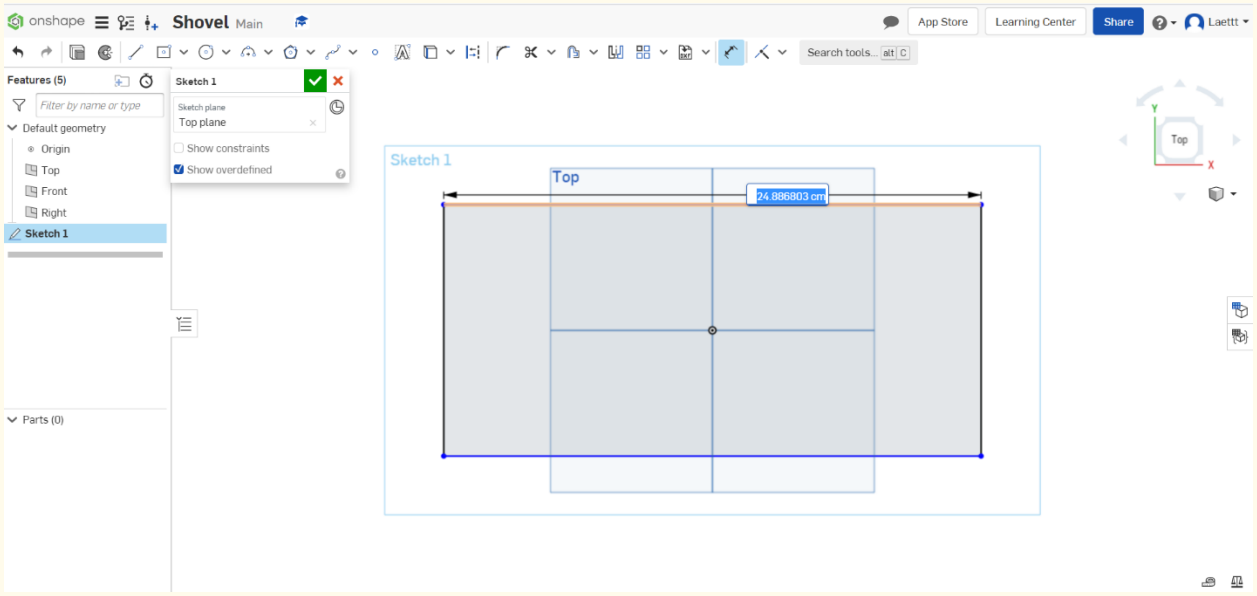
Step 9

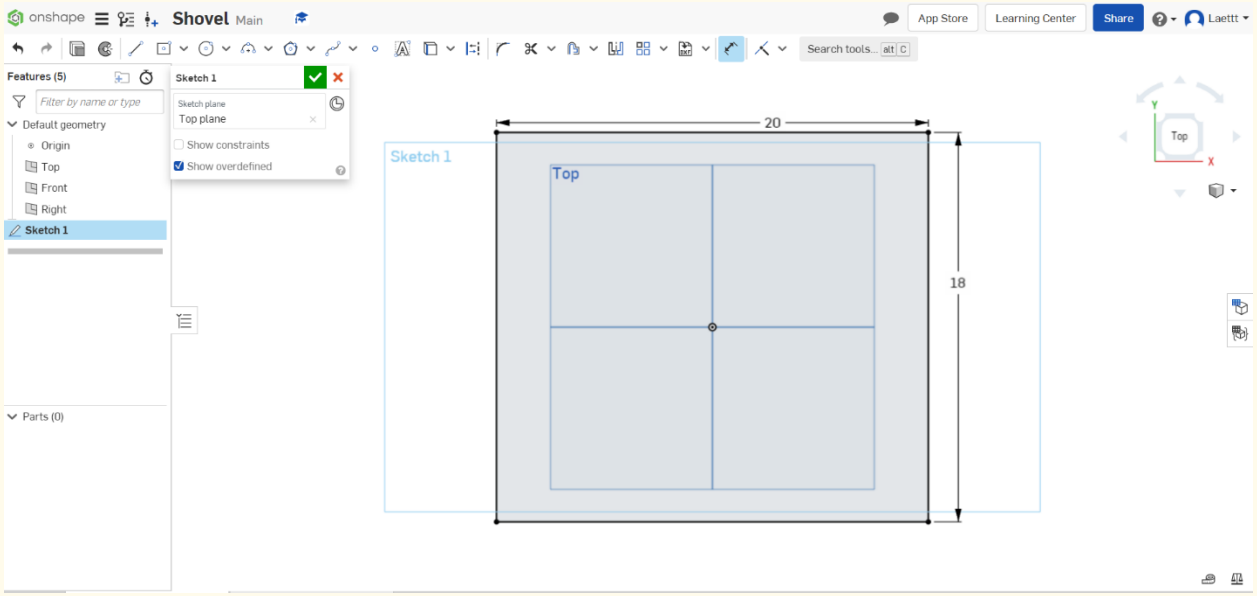
Select Dimension to give the measures.



Step 10

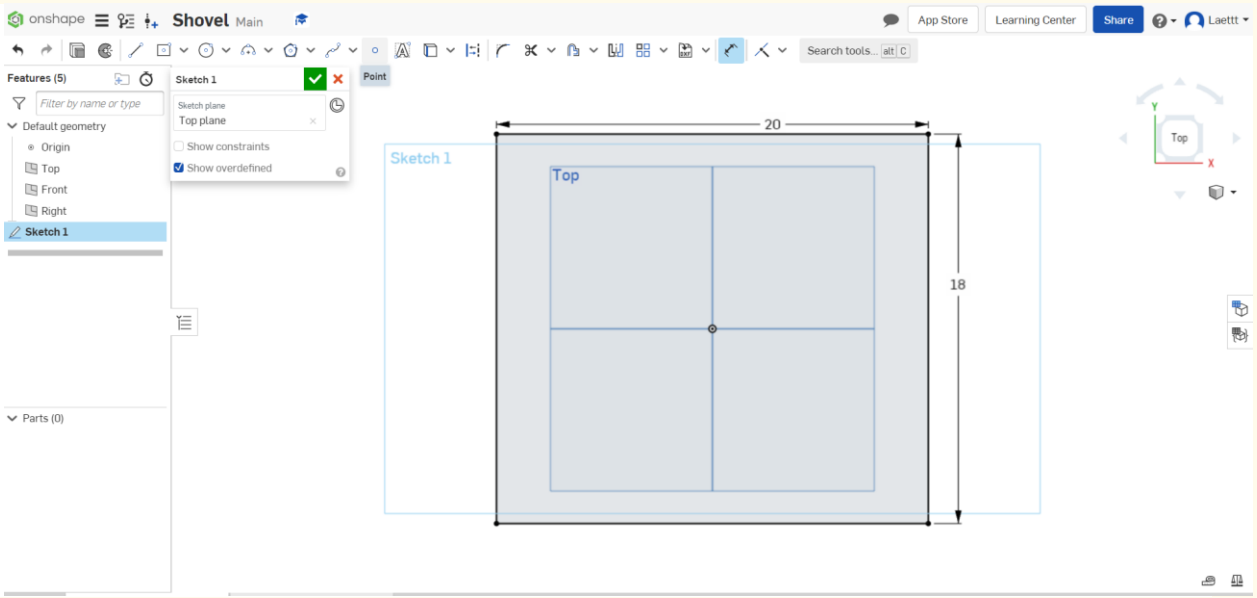
Select the line (first left then top) and give the measure 2nd figure.





Step 11

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



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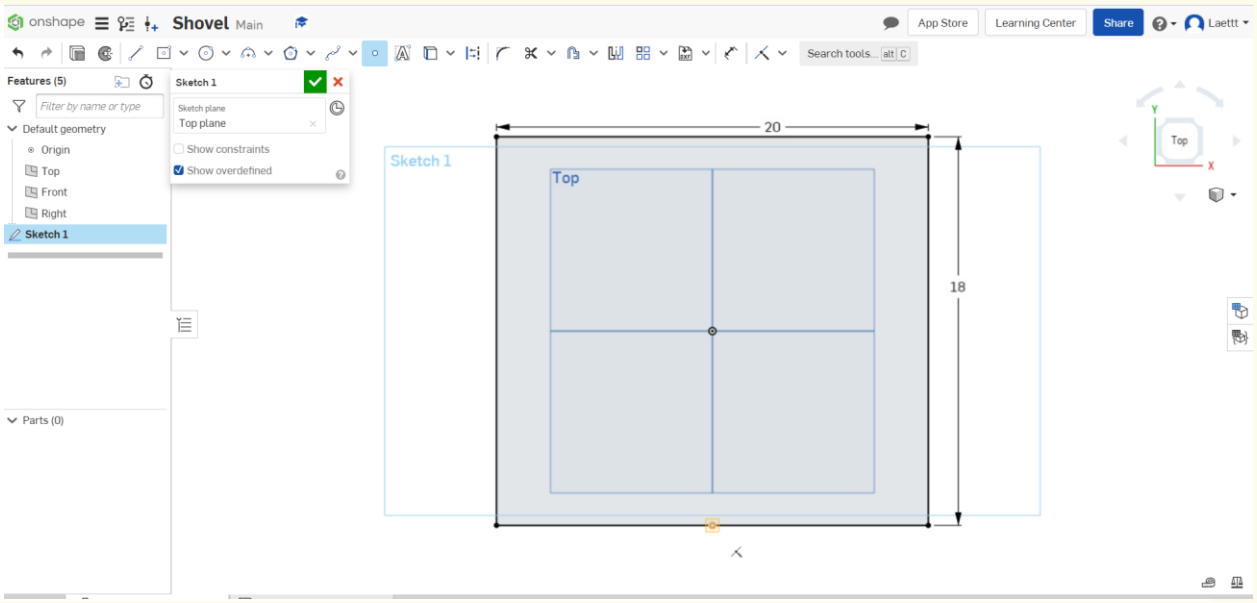
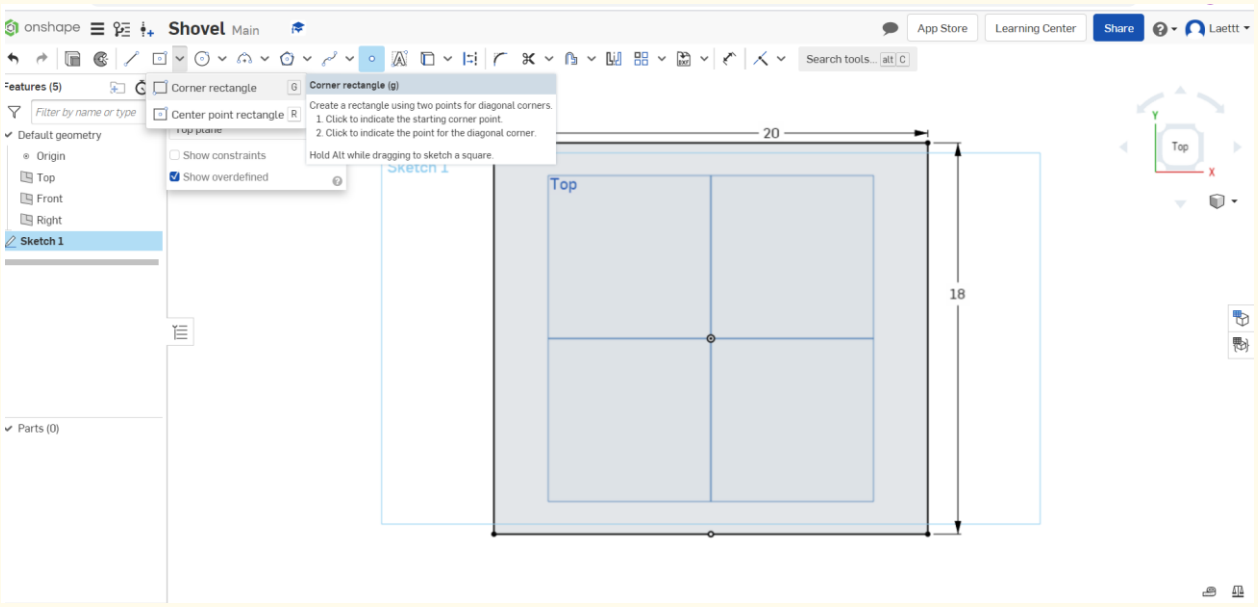
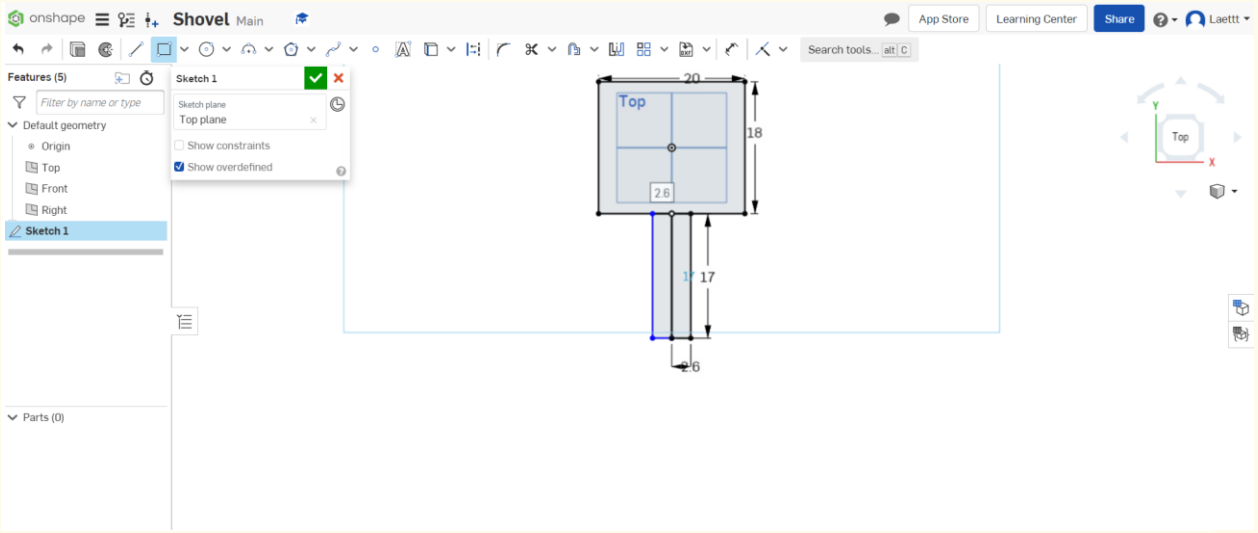


Step 12

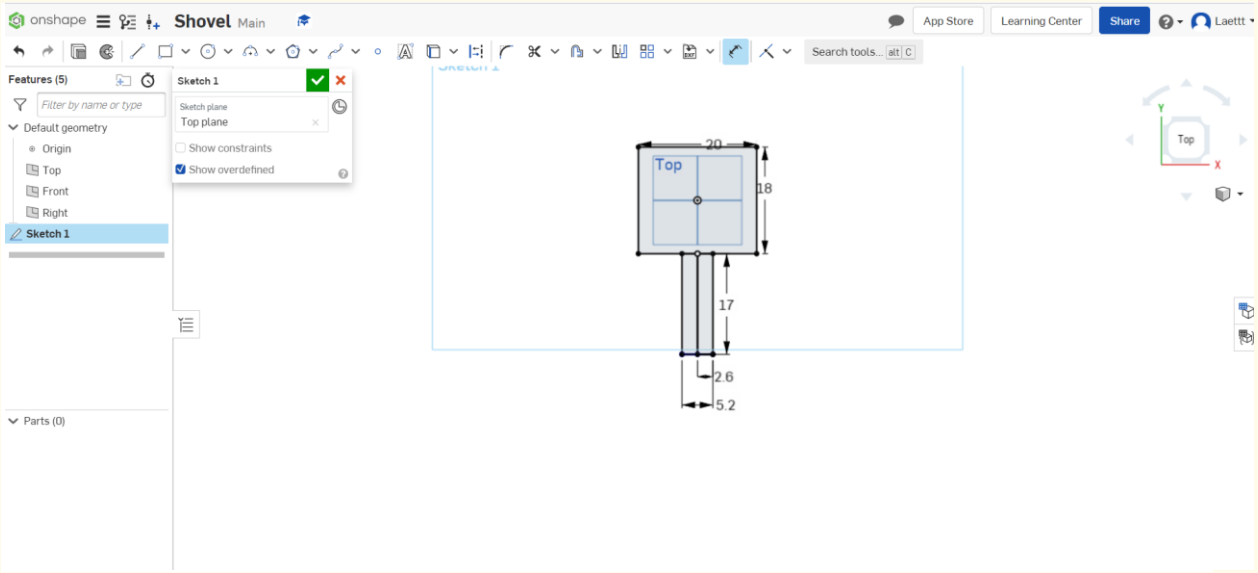
Draw 2 similar rectangles as shown.





Step 13

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



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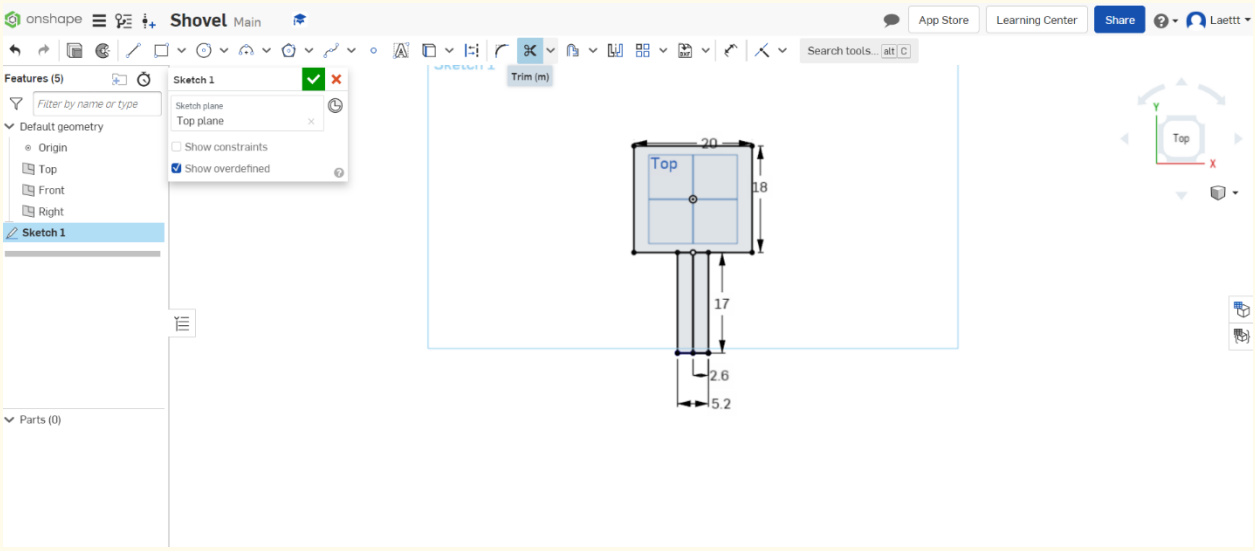
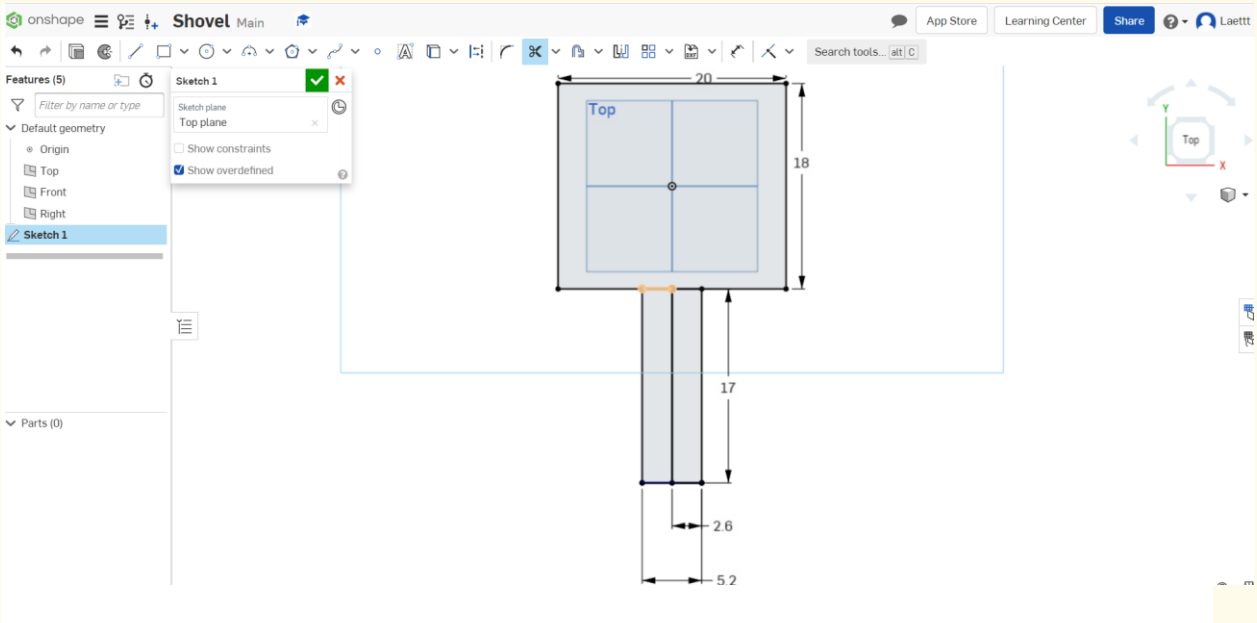
Step 14

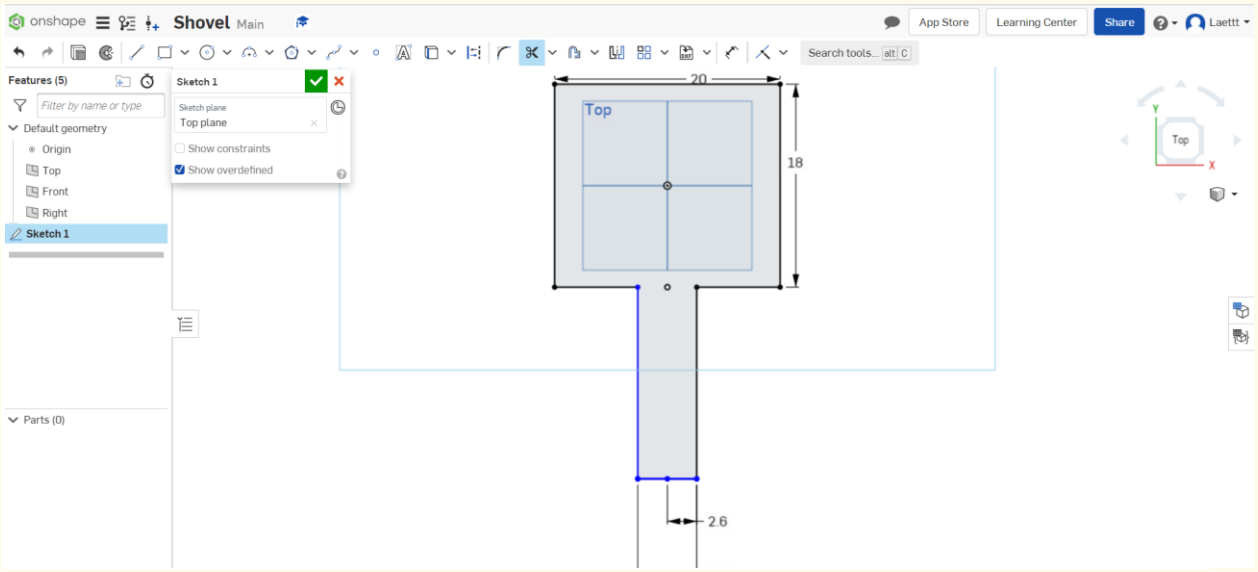
Select Trim.



Step 15

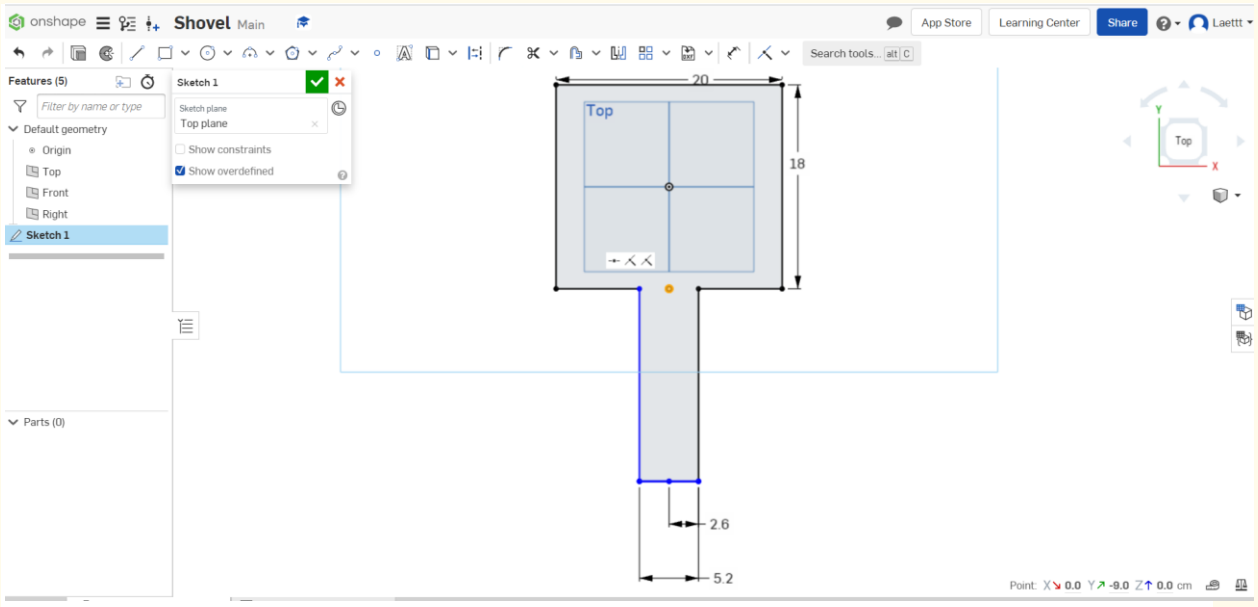
Select the orange lines (then the lines will disappear).

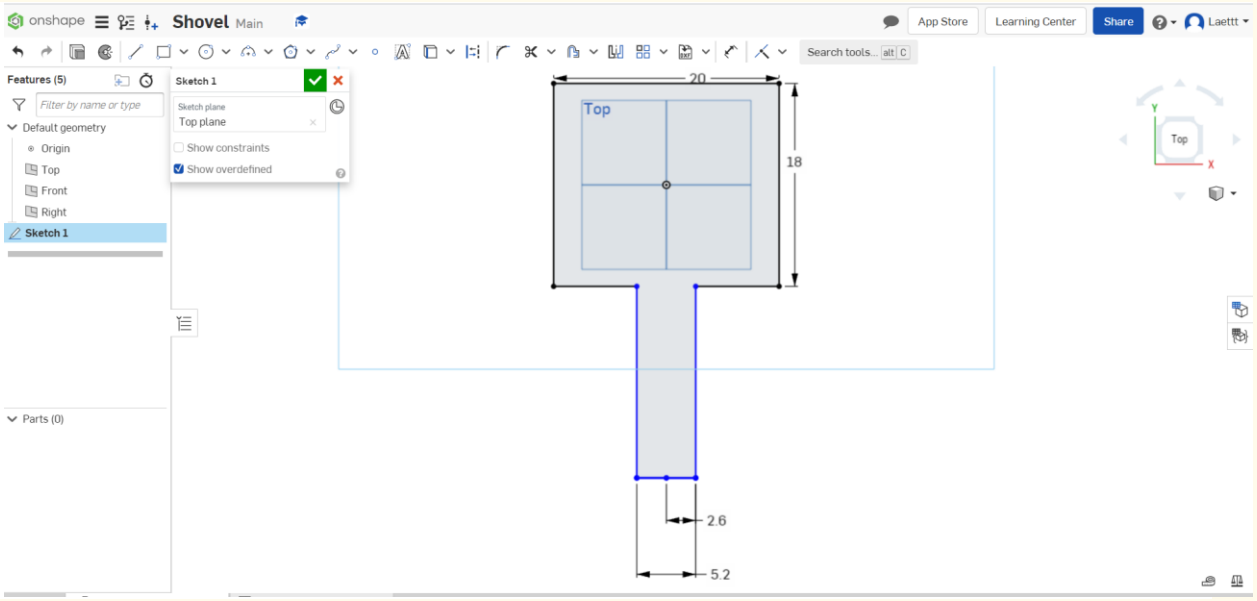




Step 16

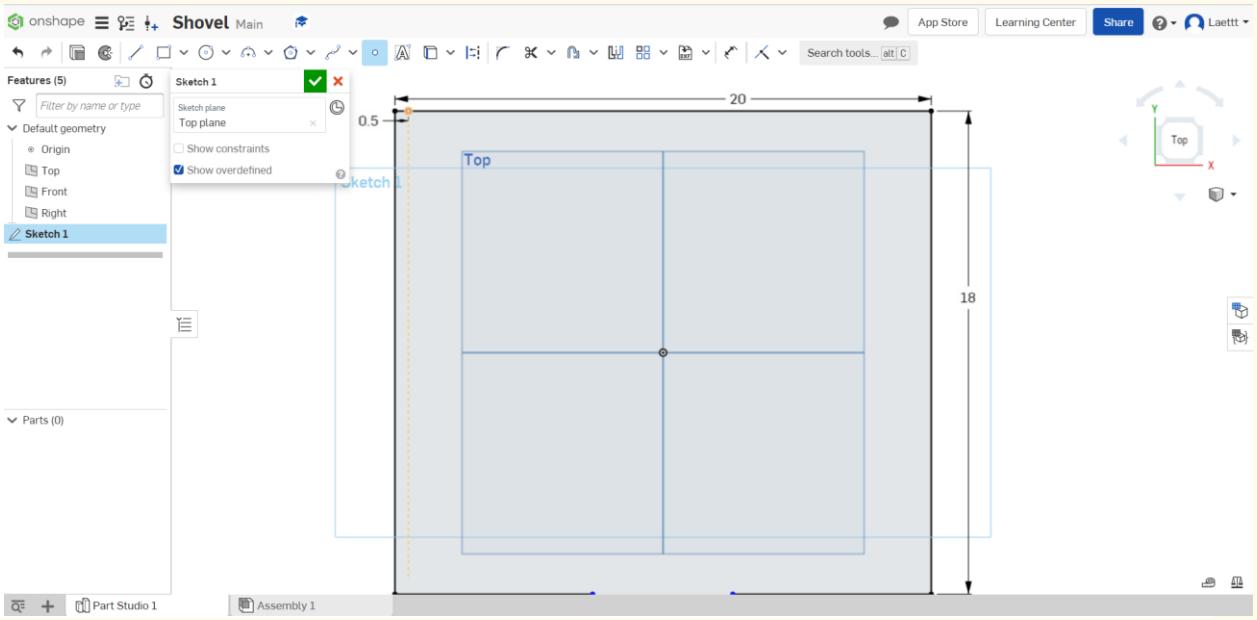
Remove the dot.

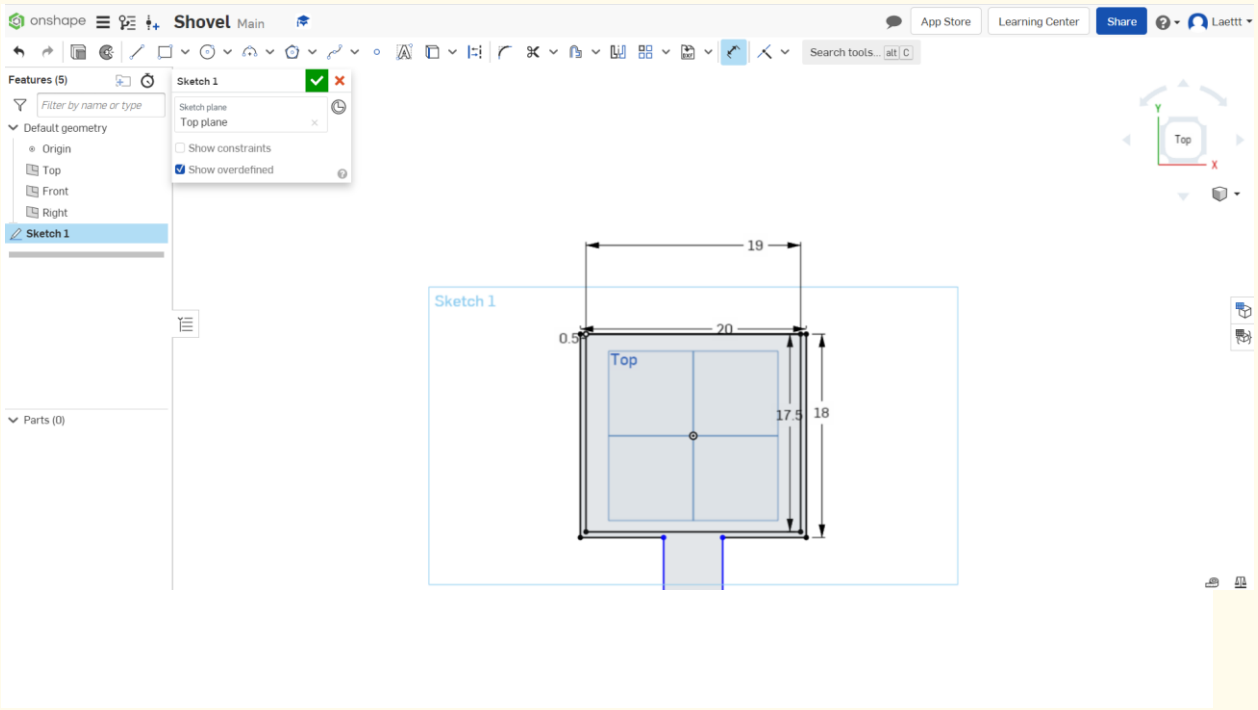




Step 17

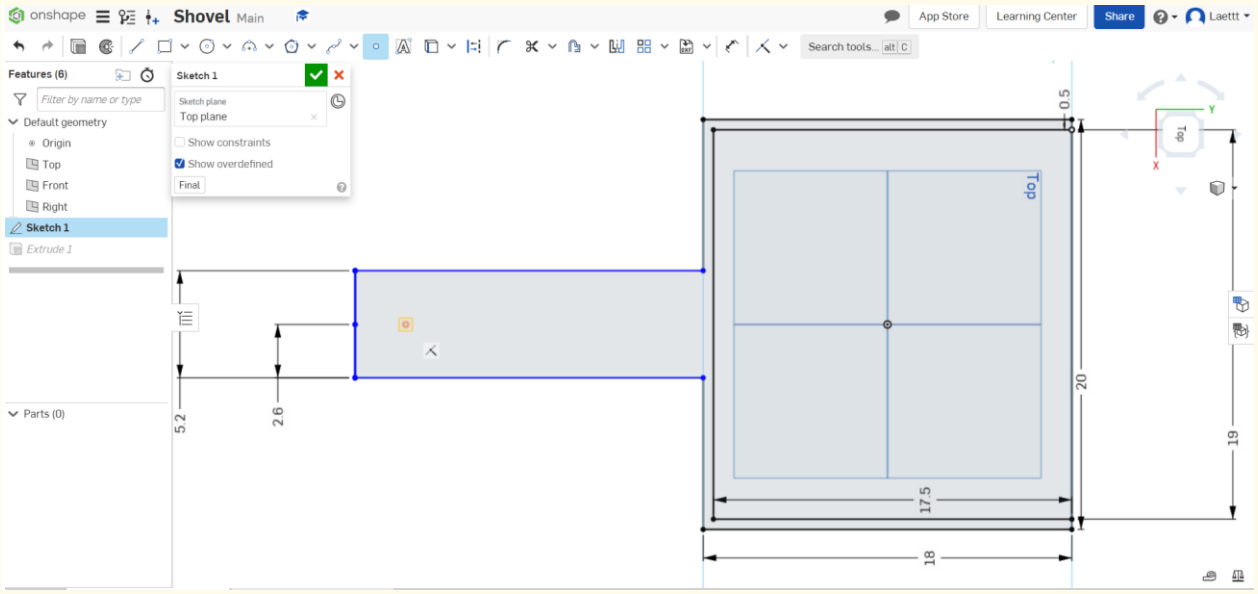
Place a dot at 0.5cm from the edge of the first rectangle and draw another rectangle inside (the rectangle is 19cm and 18.5cm). This will be the base of the shovel.





Step 18

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



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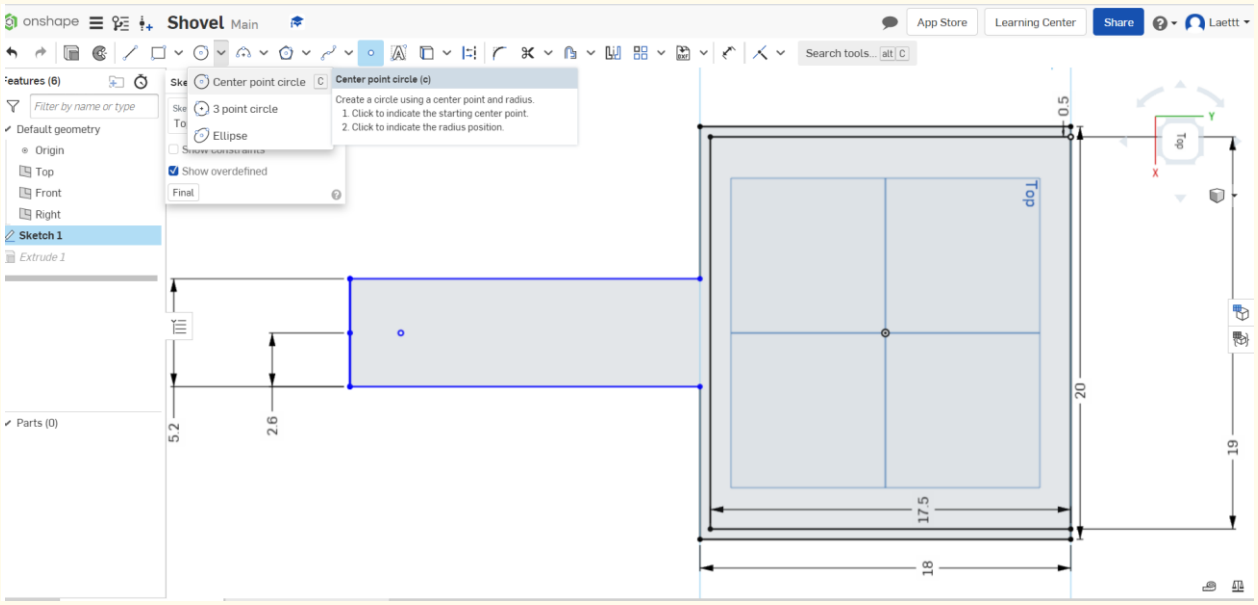
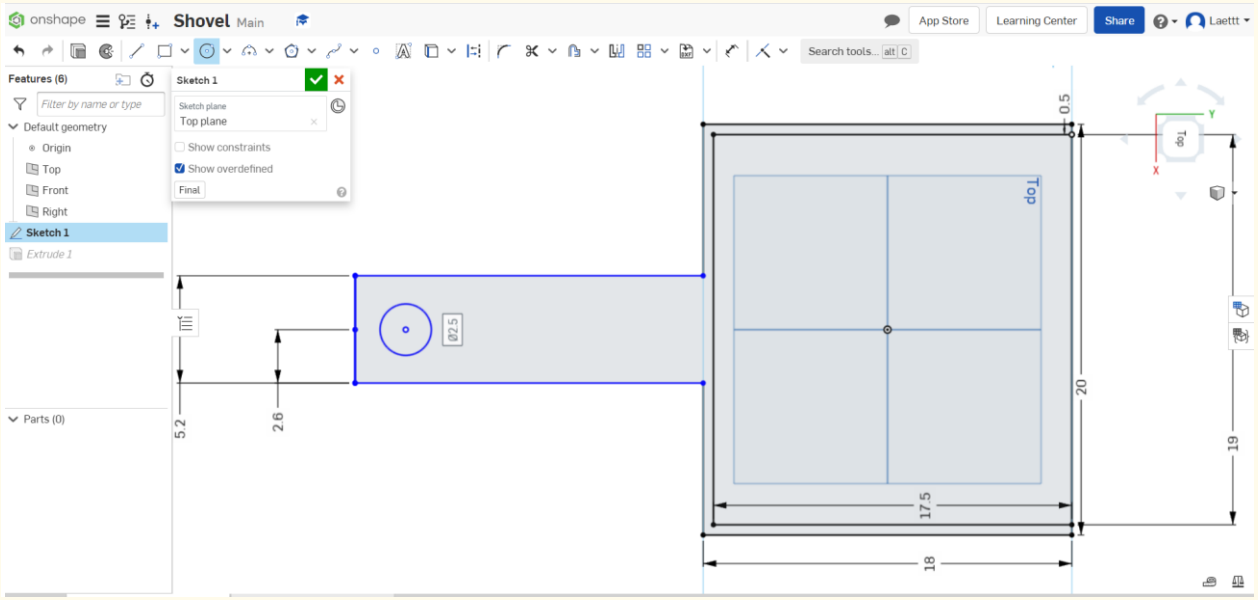
Step 19

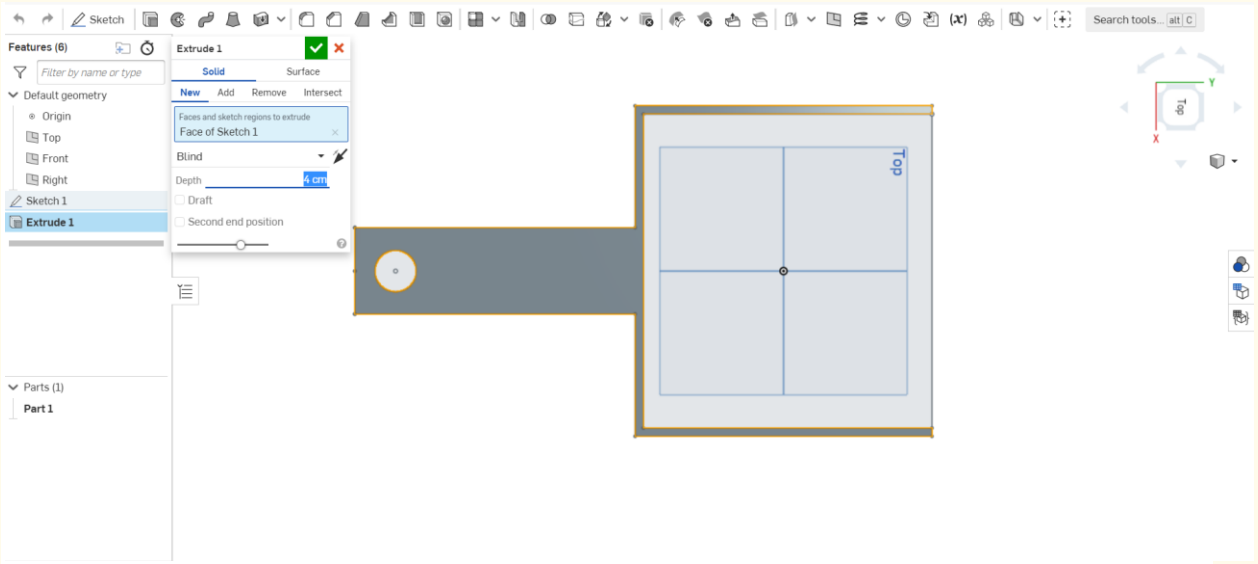
Draw a center dot circle.

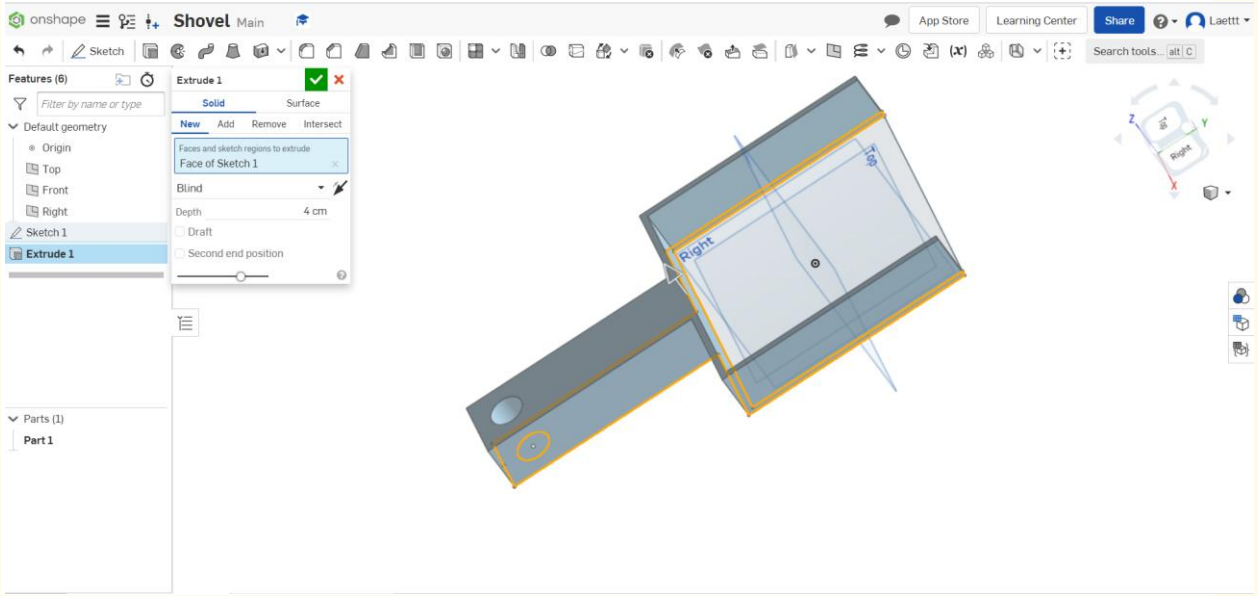


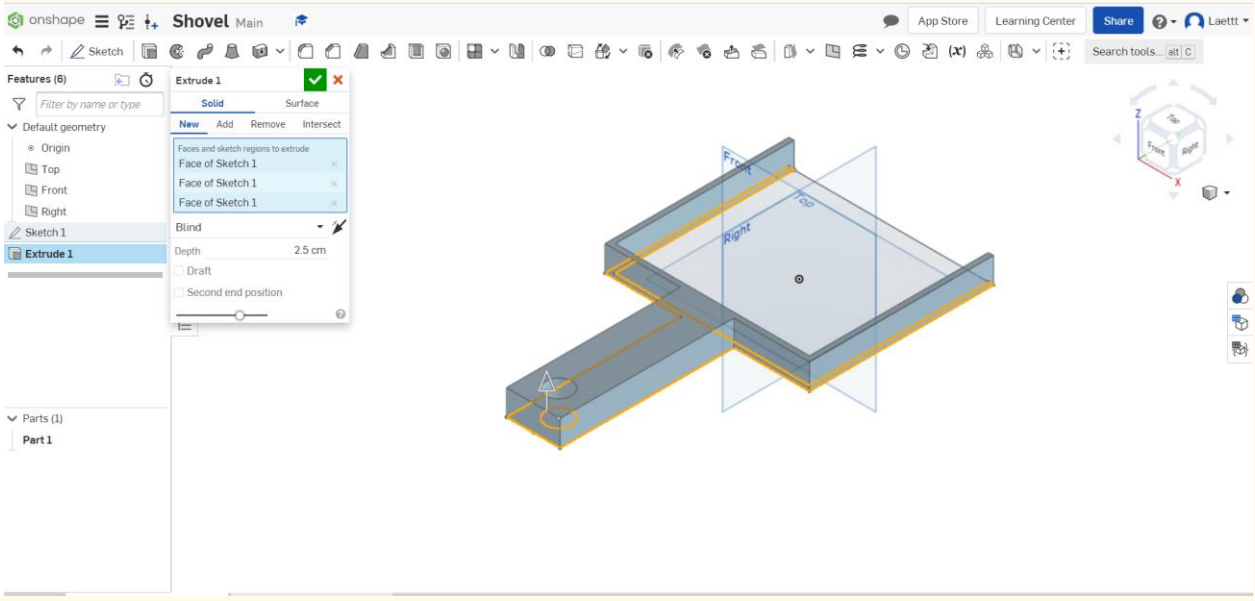


Step 20

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

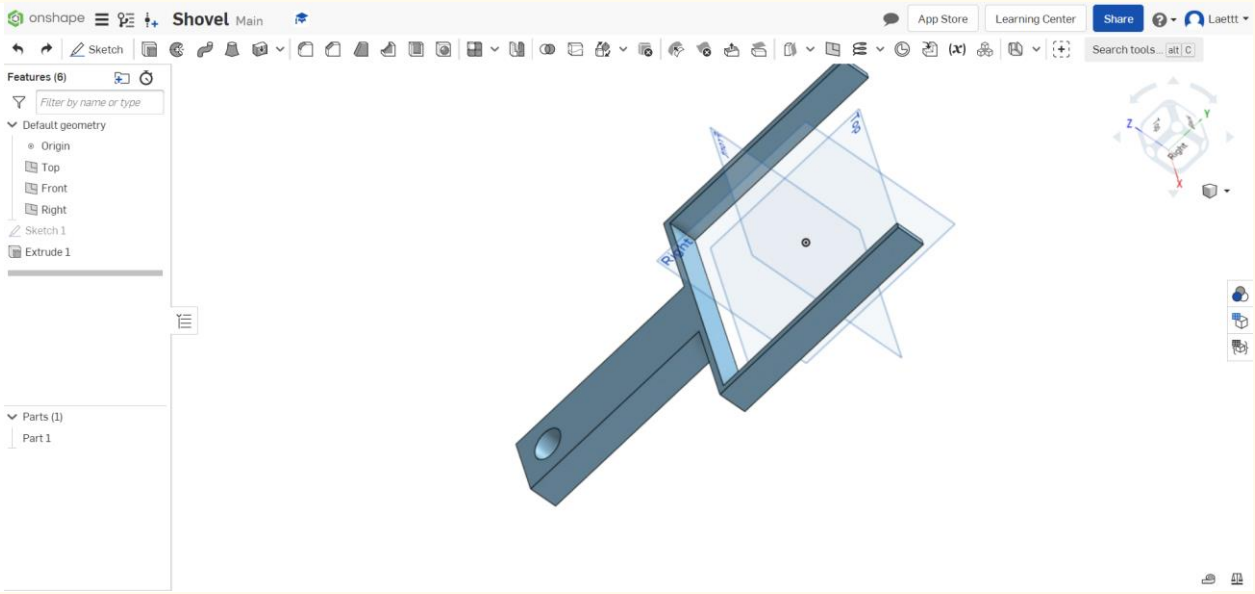






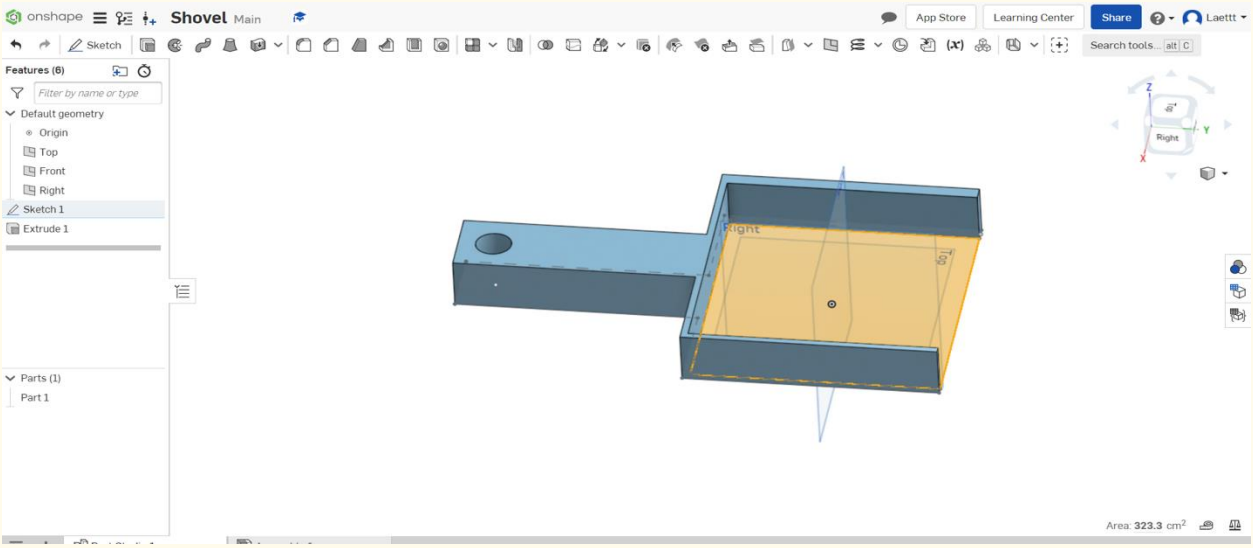
Step 21

Extrude as well the base of the shovel.
With isometric view, we obtain the following shovel, with no visible base.



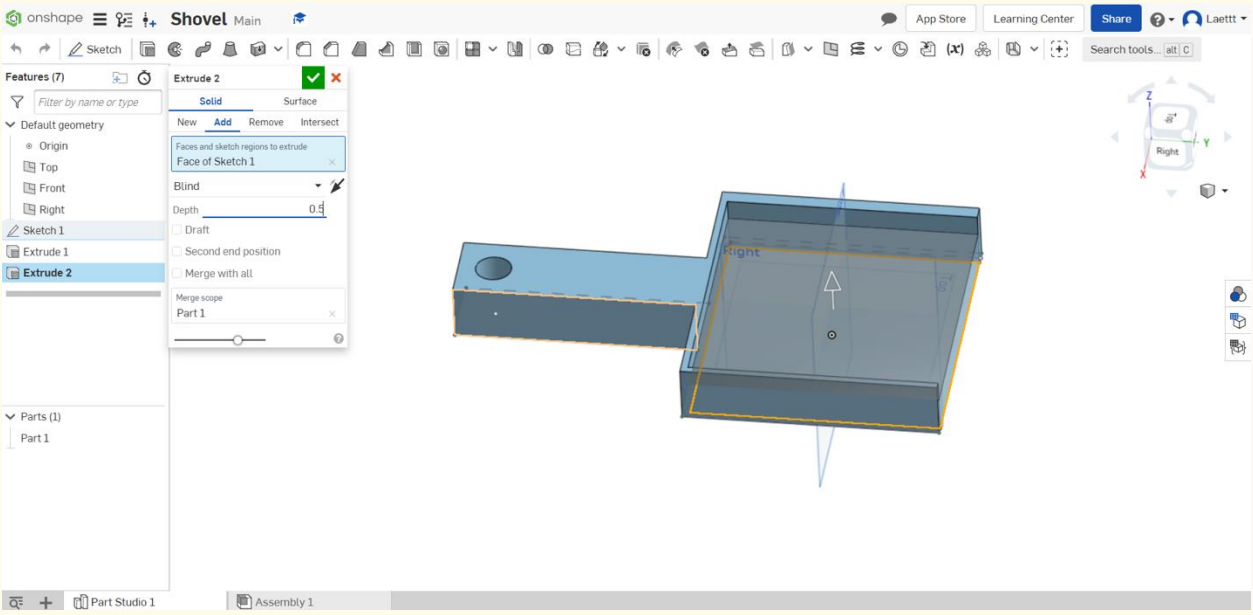
Step 22

Extrude the base
Select the base.

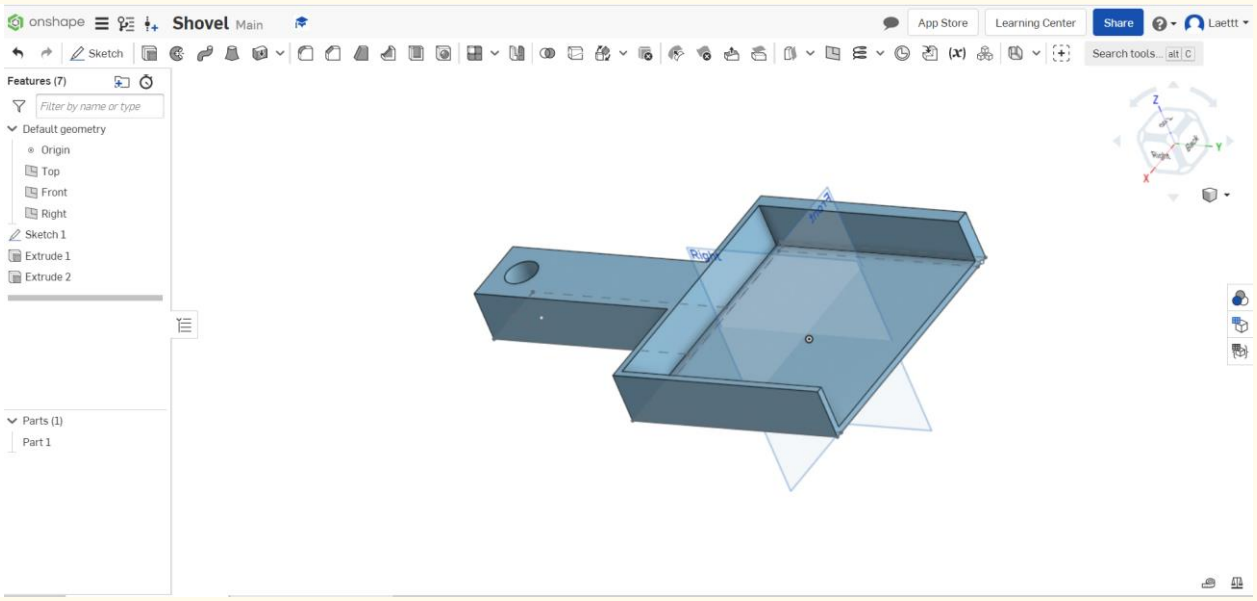


Step 23

Click on extrude and choose 0.5cm



Step 24 This should be your result.



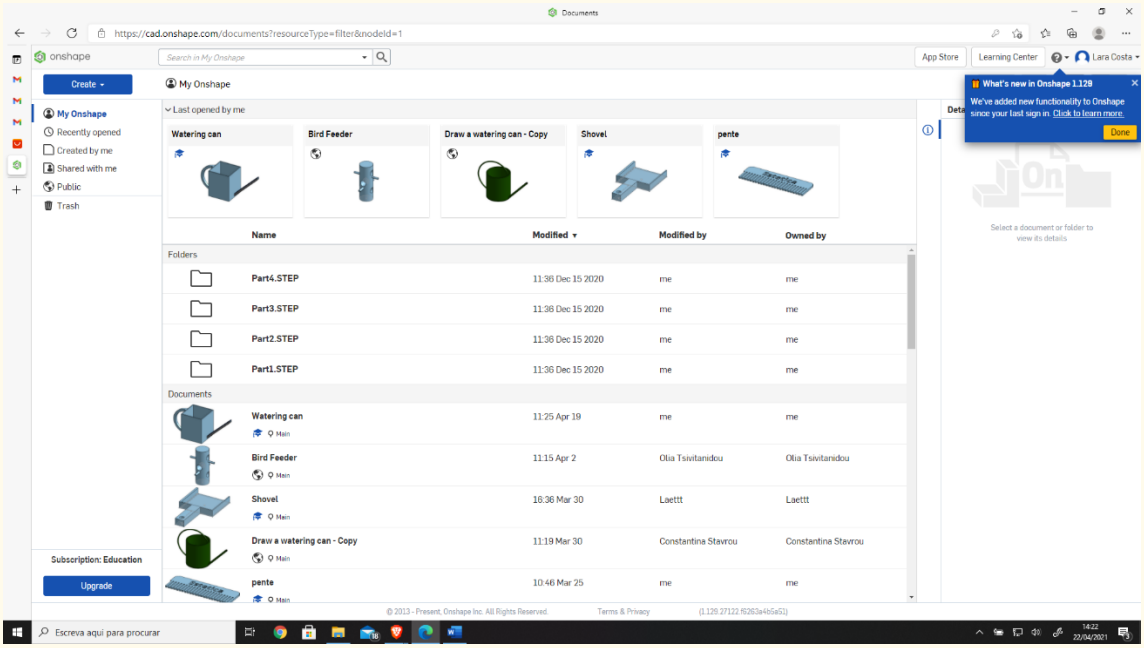
PROJECT: 3D DRAWING OF A WATERING CAN

- **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 competencies:**
    - 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:

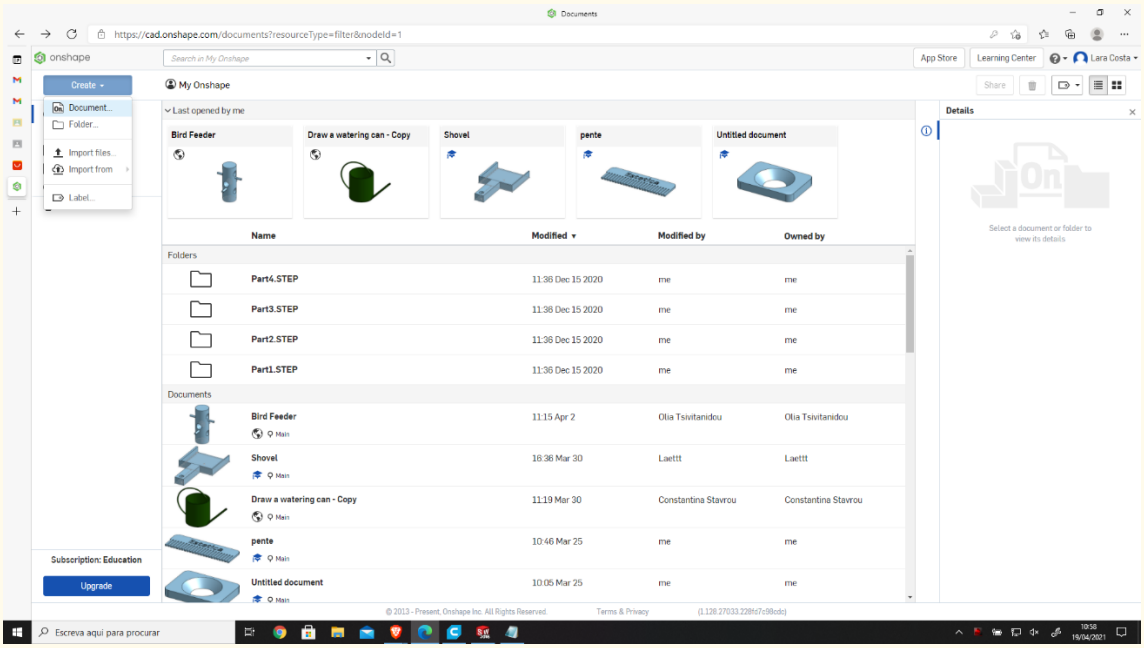
Step 1

Open Onshape (the free version).



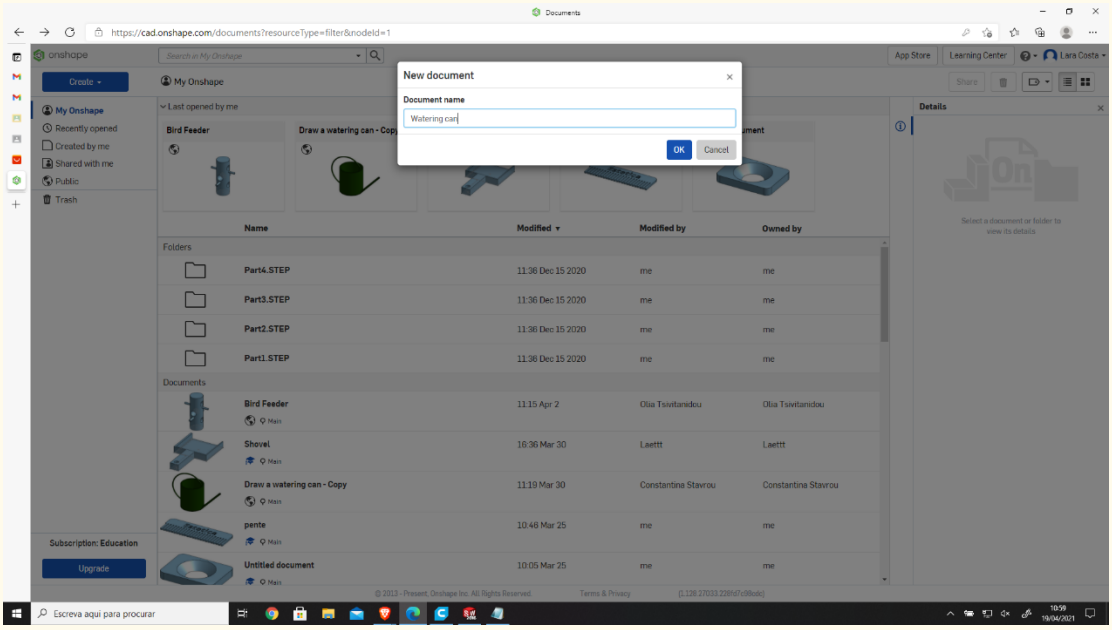
Step 2

Create a document.



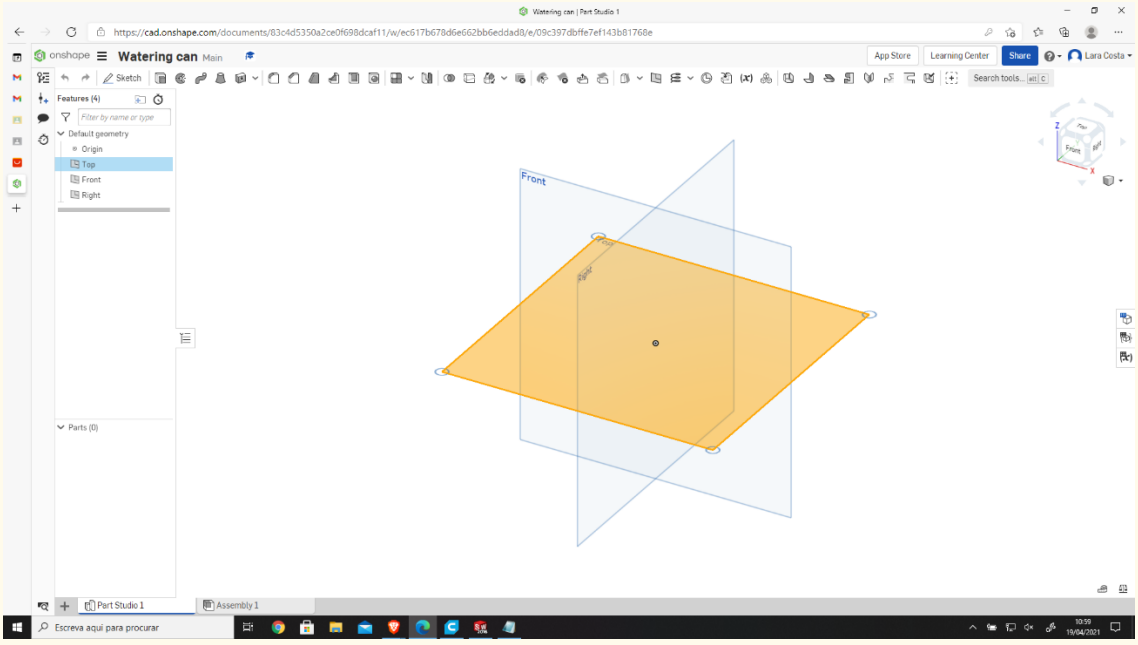
Step 3

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



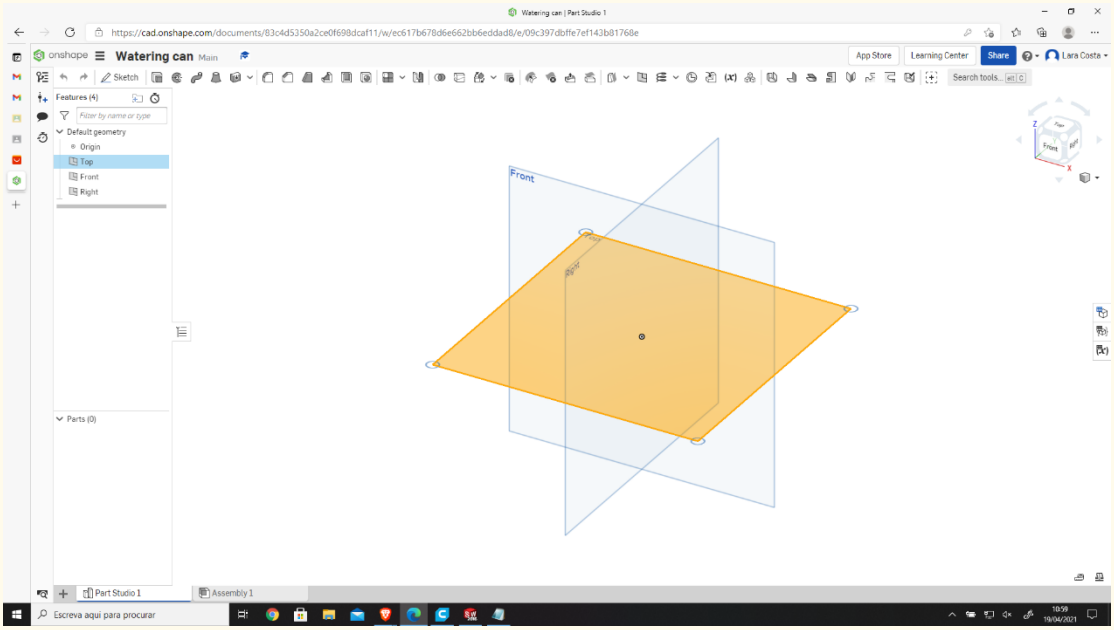
Step 4

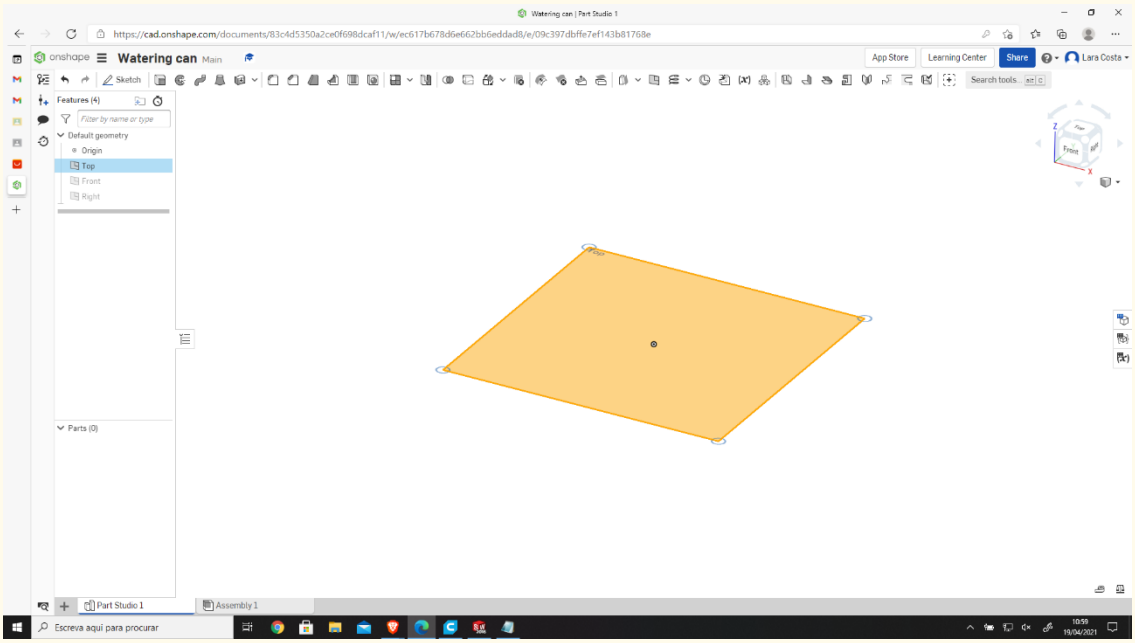
Select the Front to start drawing.



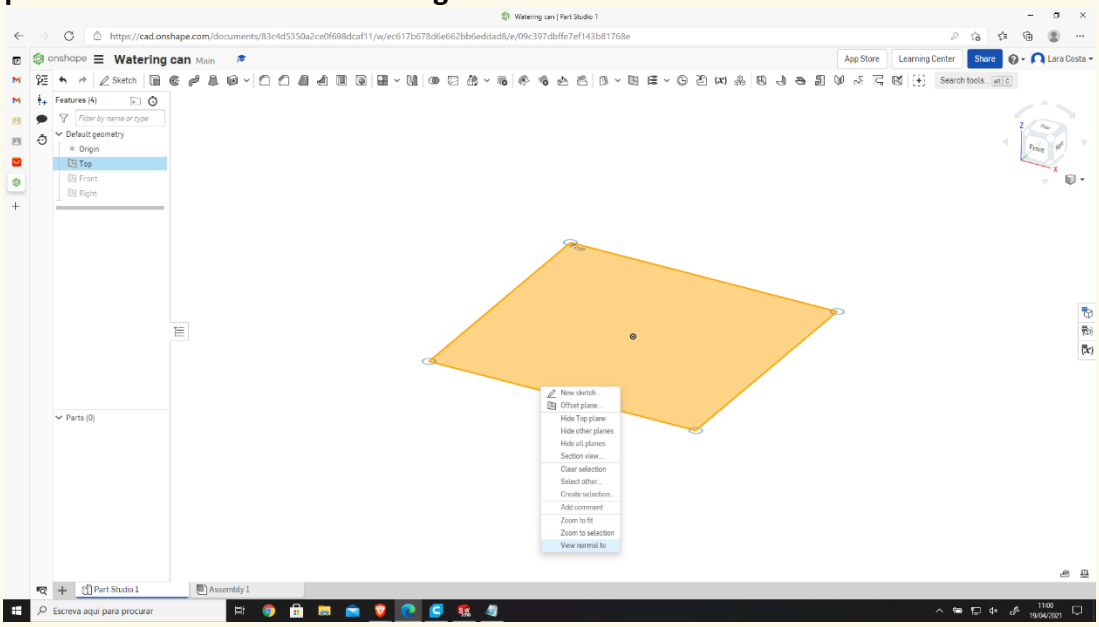
Step 5

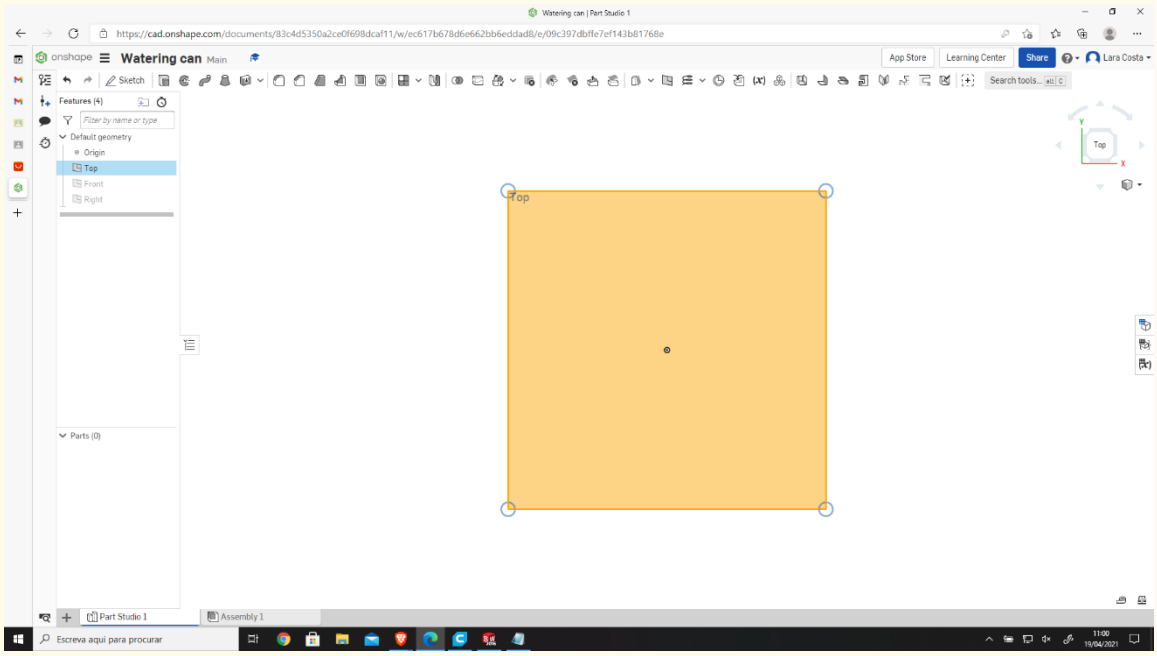
Right-click and select “Hide other planes.”
The plan should look like the 2nd image.





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





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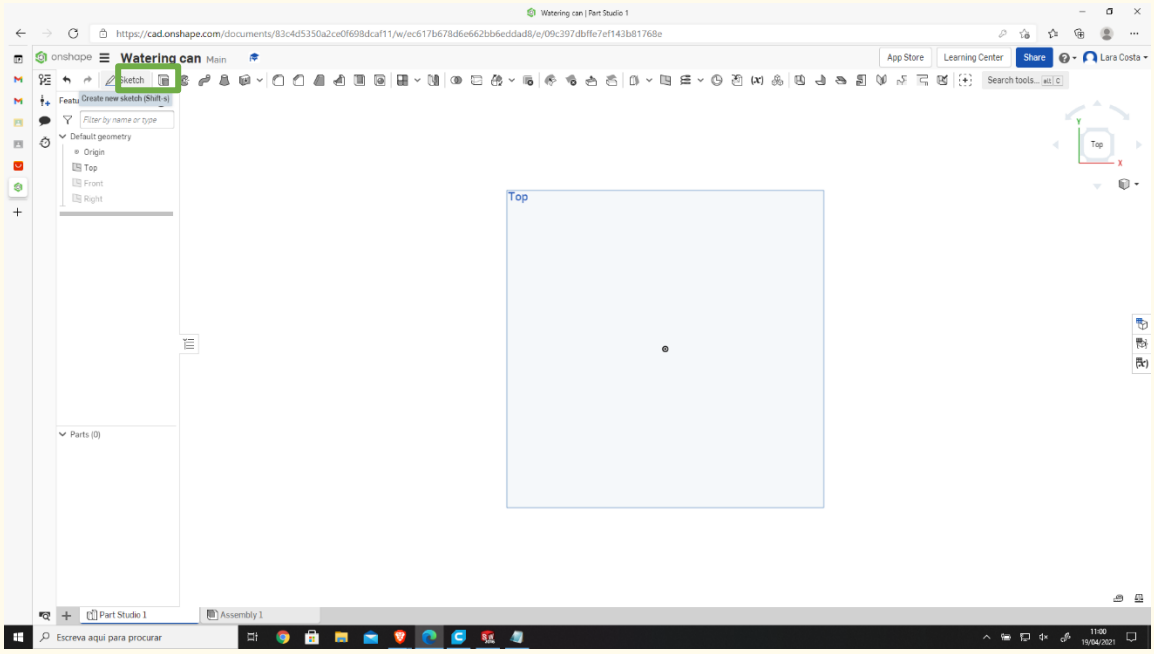
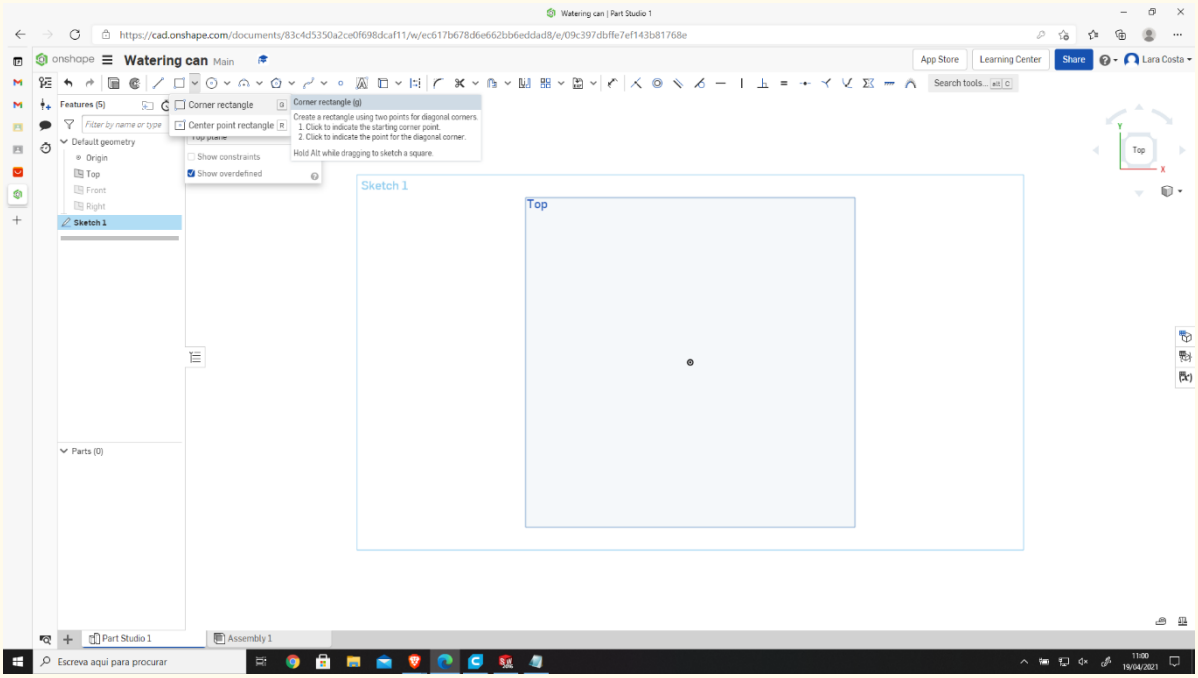
Step 6

Click Sketch.



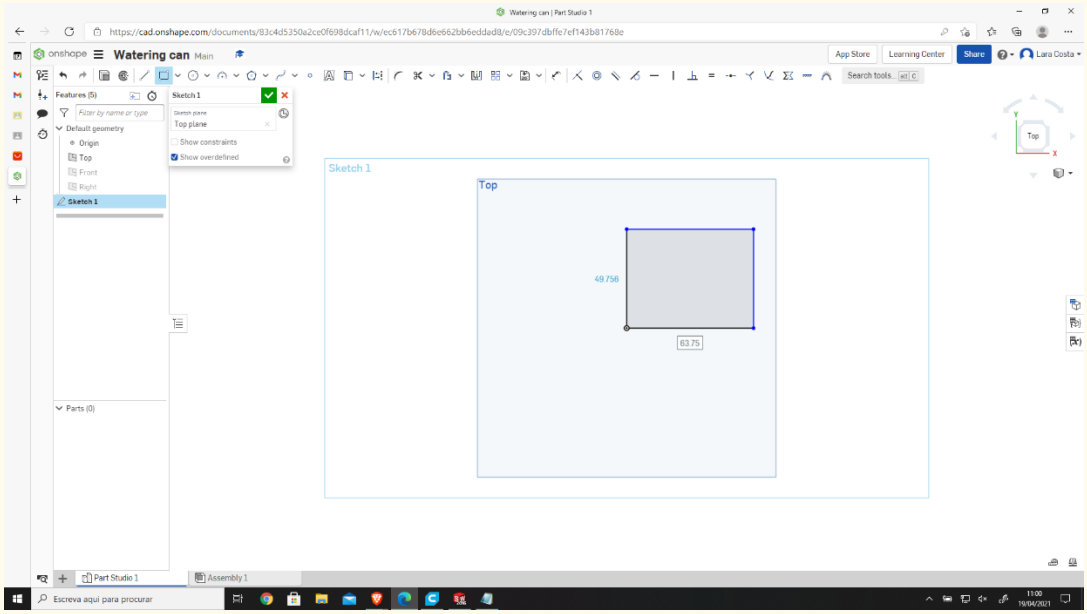
Step 7

Select corner rectangle to draw.



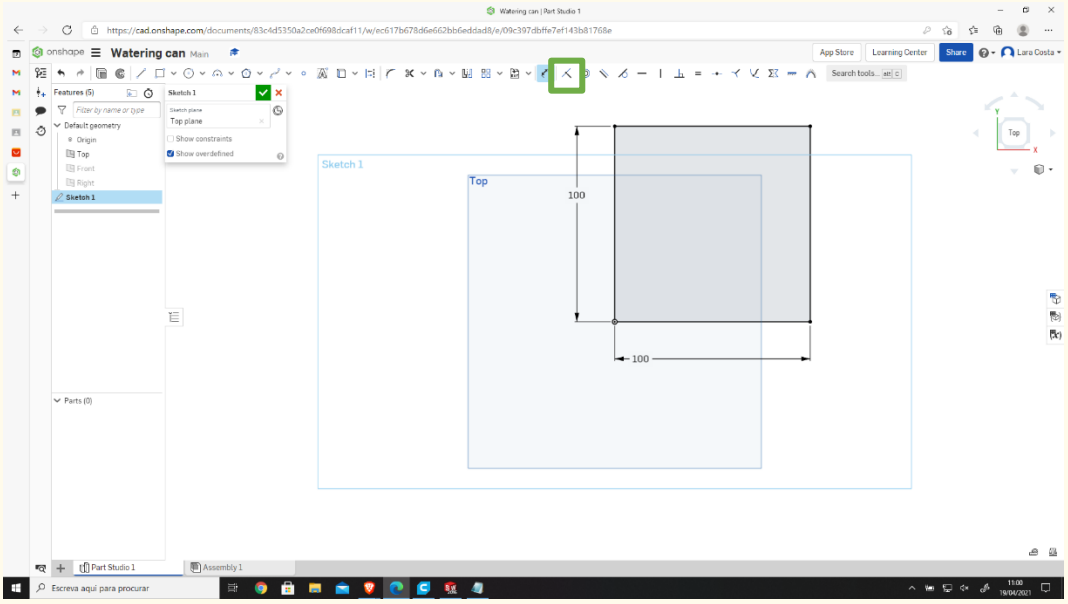
Step 8

Draw the rectangle.



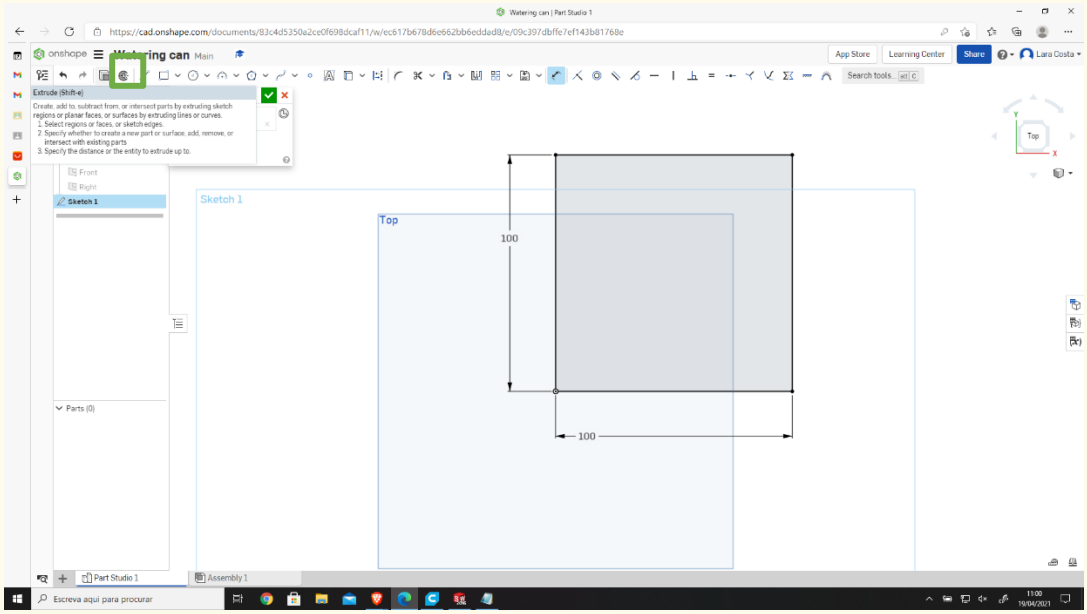
Step 9

Click on Dimension and give 100mm to each side.



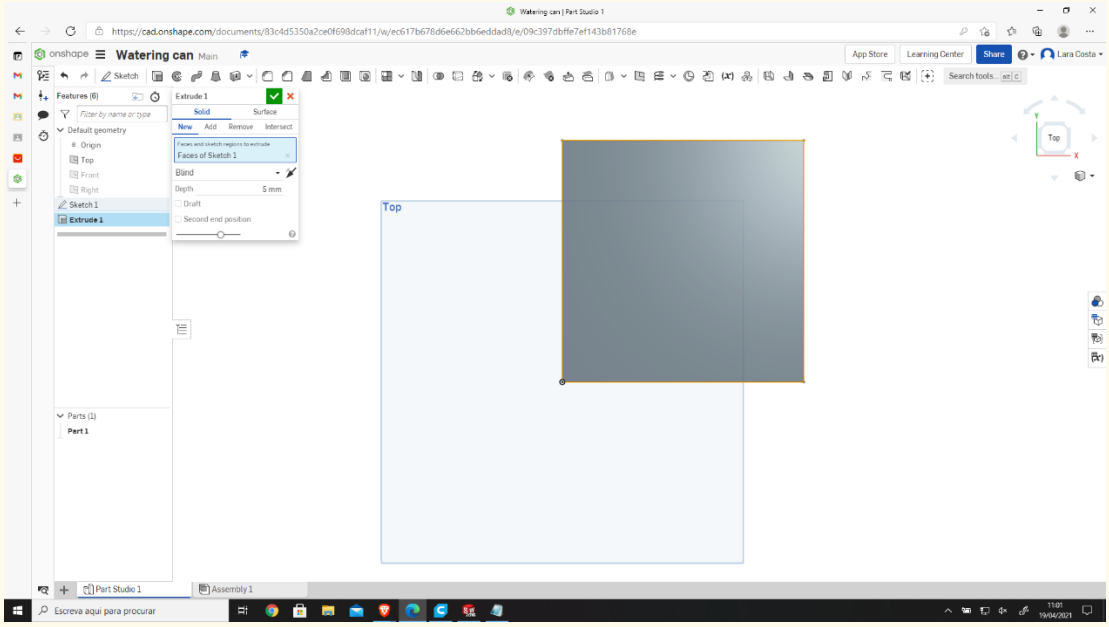
Step 10

Select extrude.



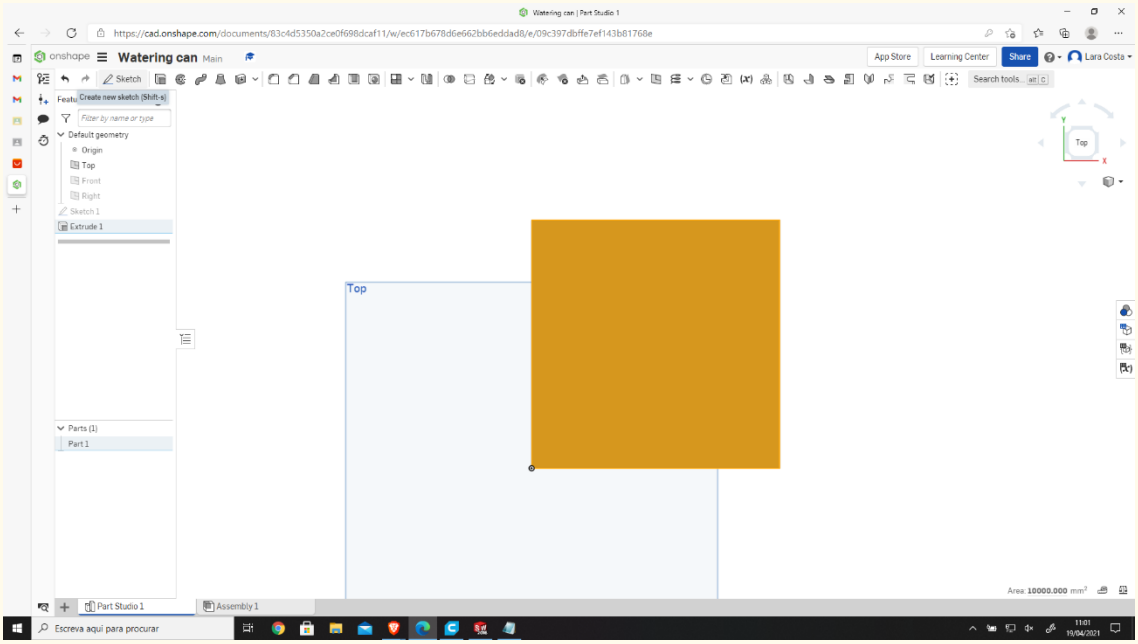
Step 11

The Depth should be 5mm.



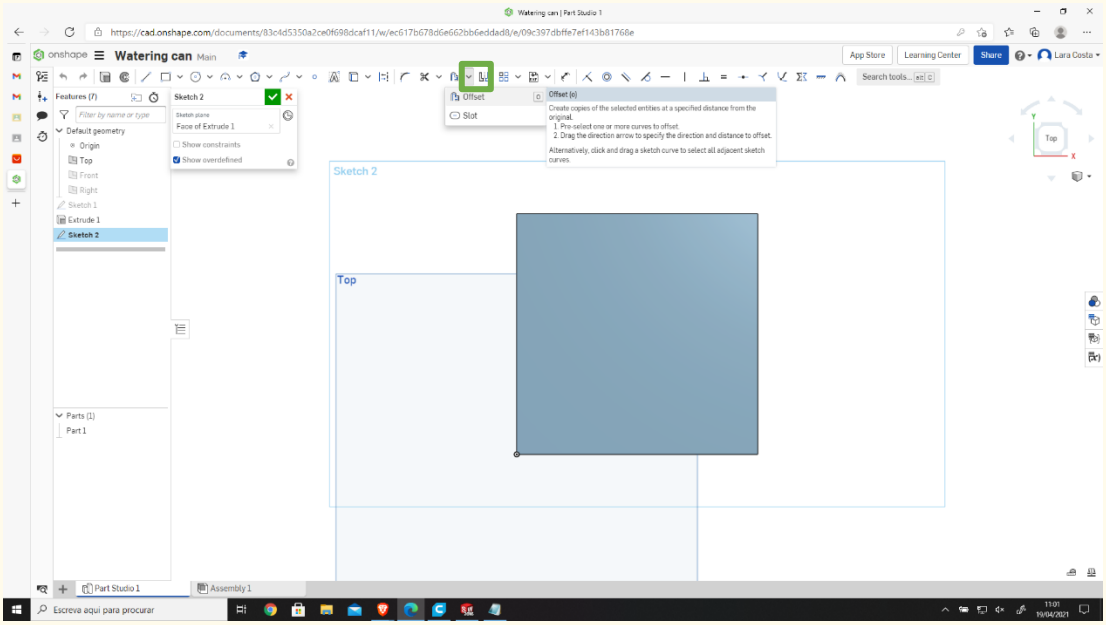
Step 12

Select the plan and sketch.



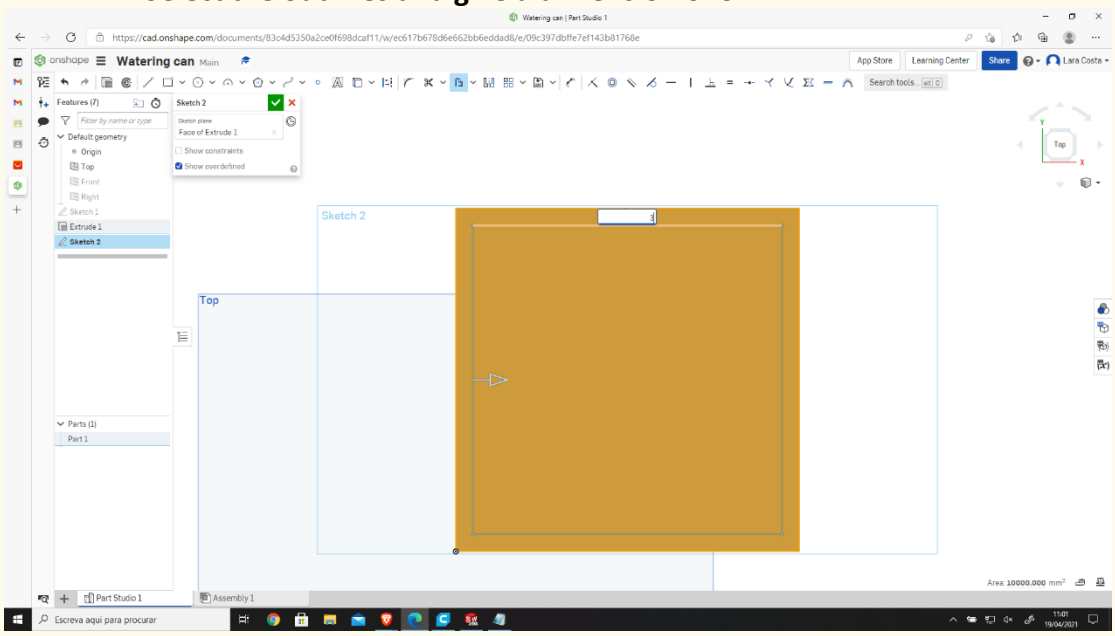
Step 13

Select OFFSET.



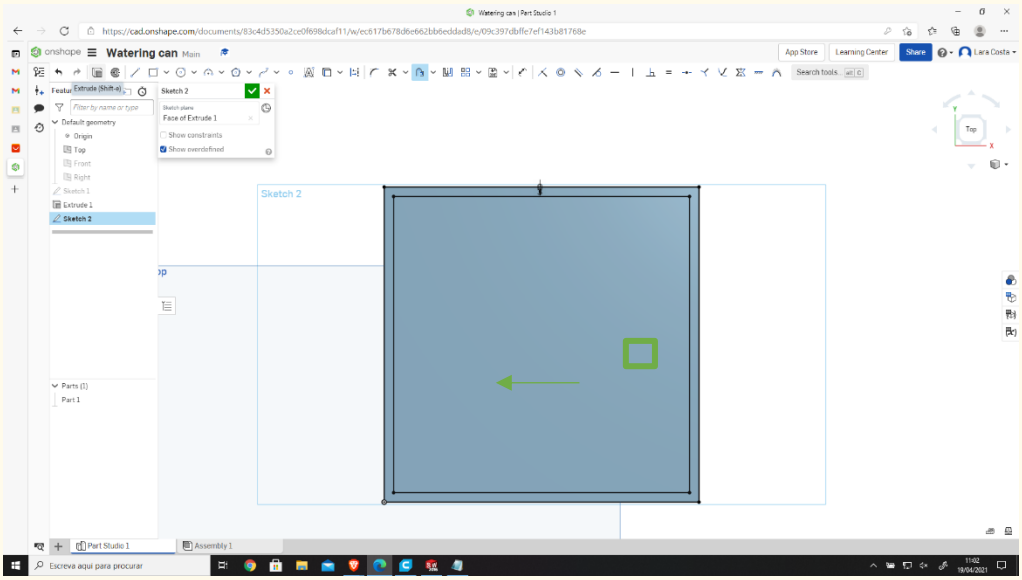
Step 14

Select the outlines and give a dimension of 3mm.



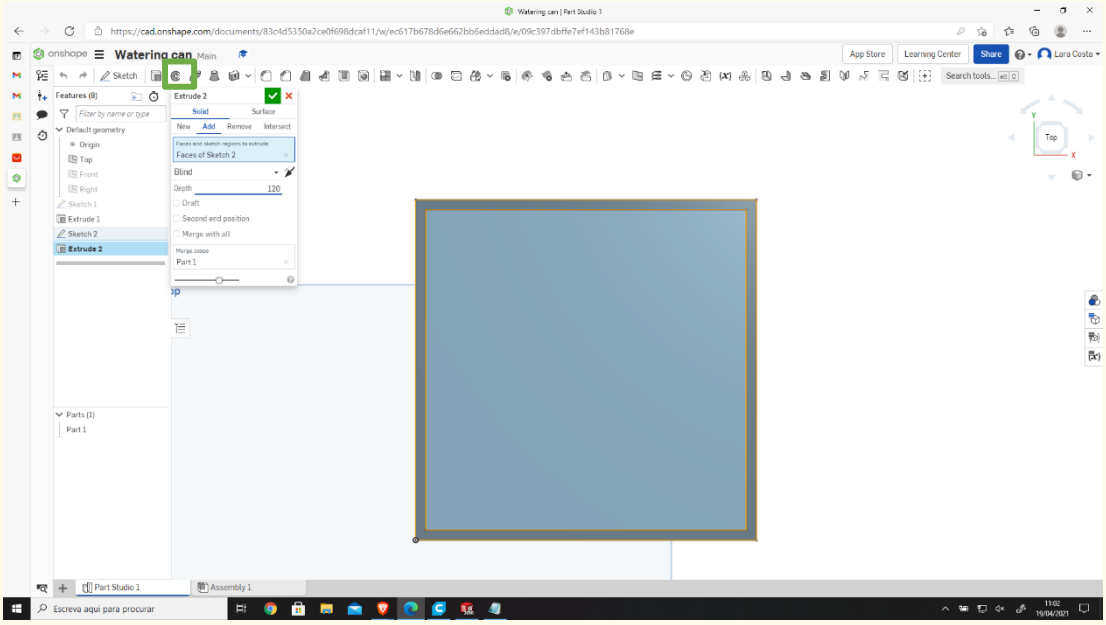
Step 15

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



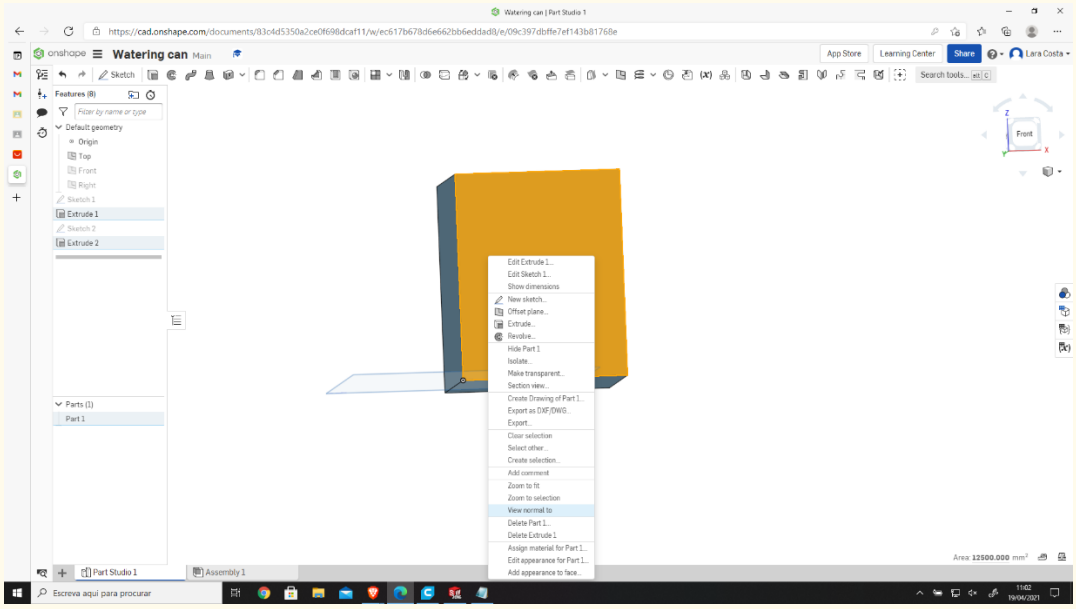
Step 16

Select Extrude and give the measure of 120mm.



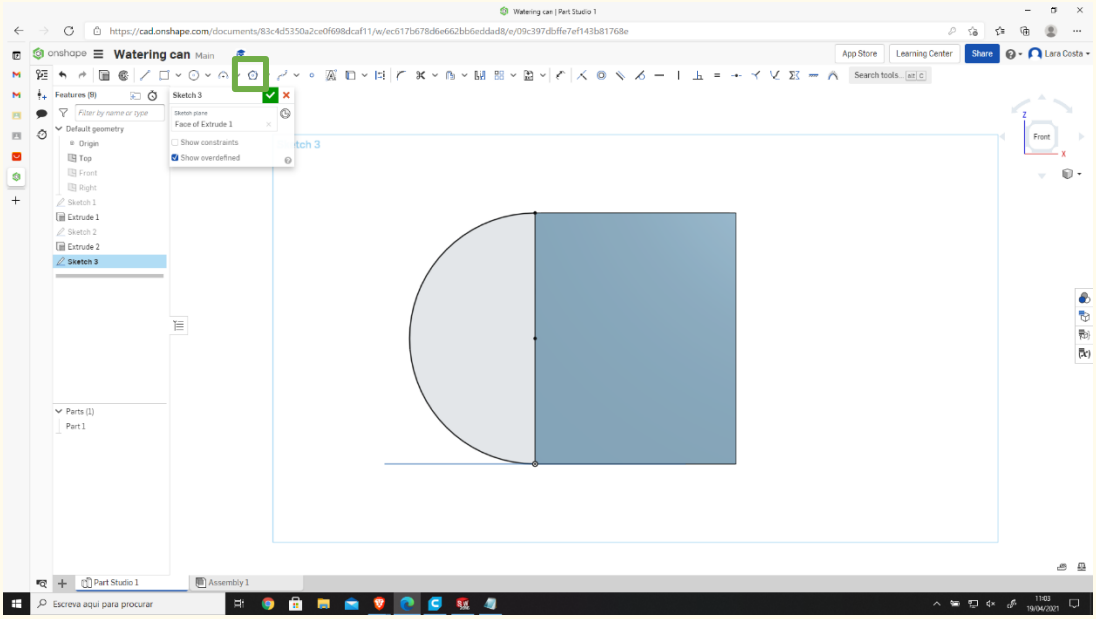
Step 17

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



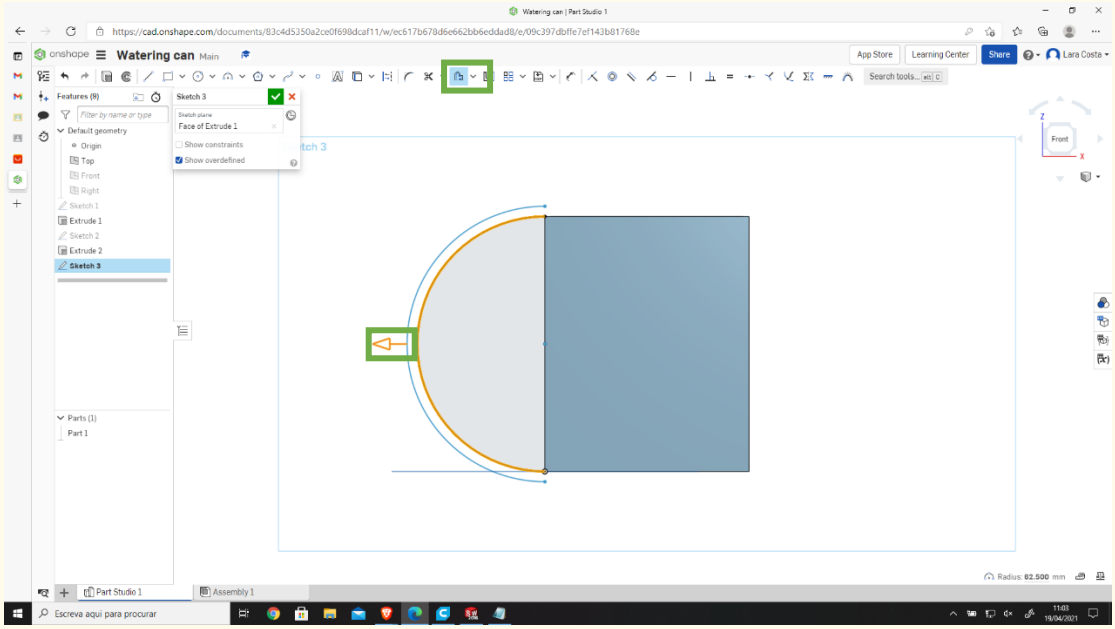
Step 18

Select 3 arc points and do it like the image.



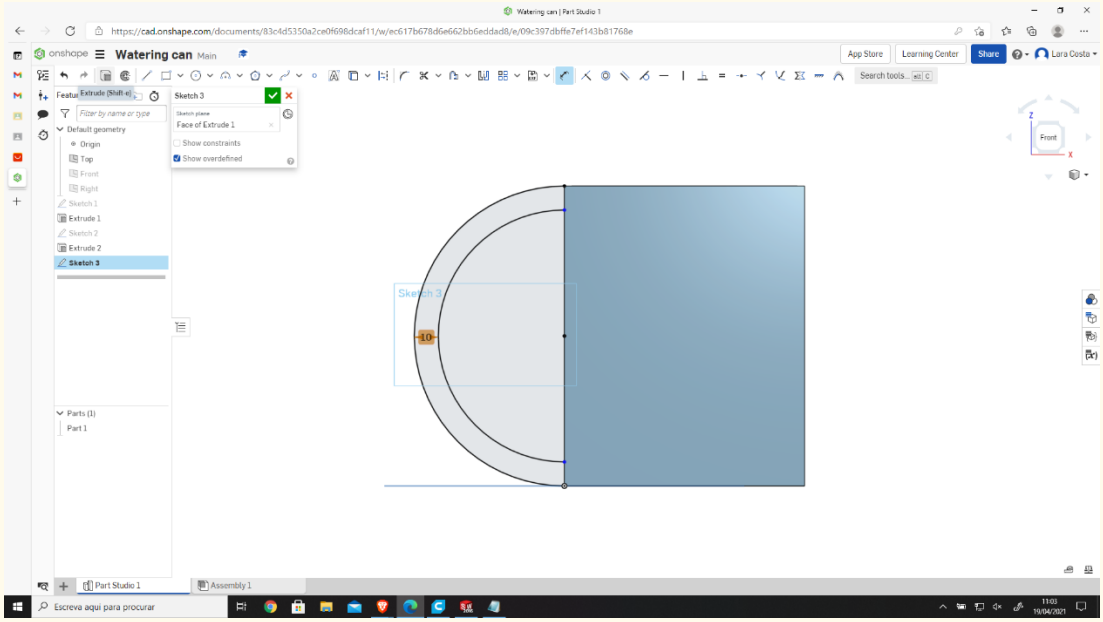
Step 19

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

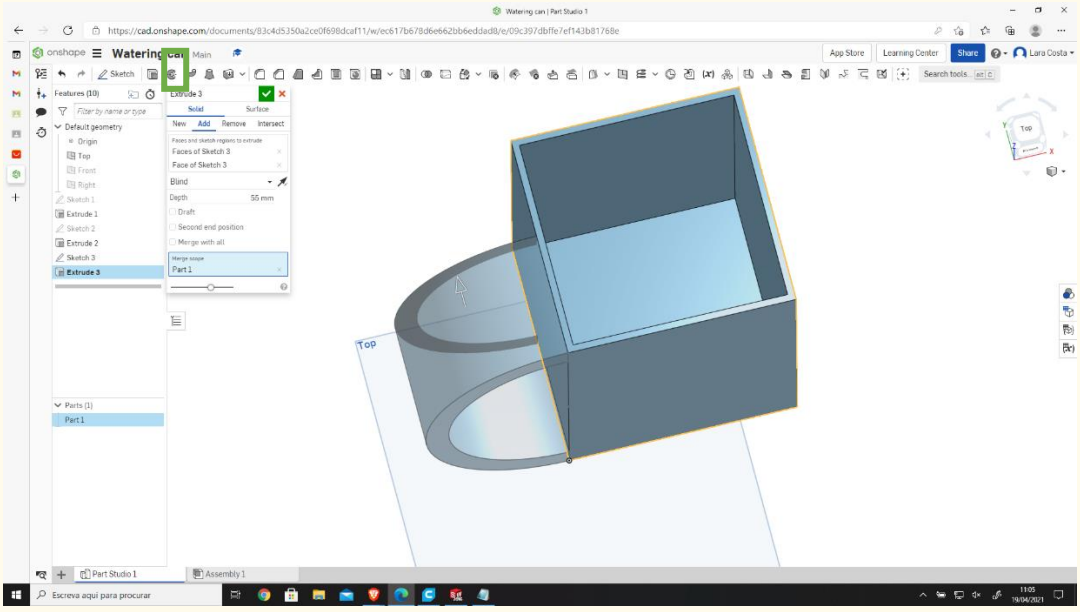


Step 20

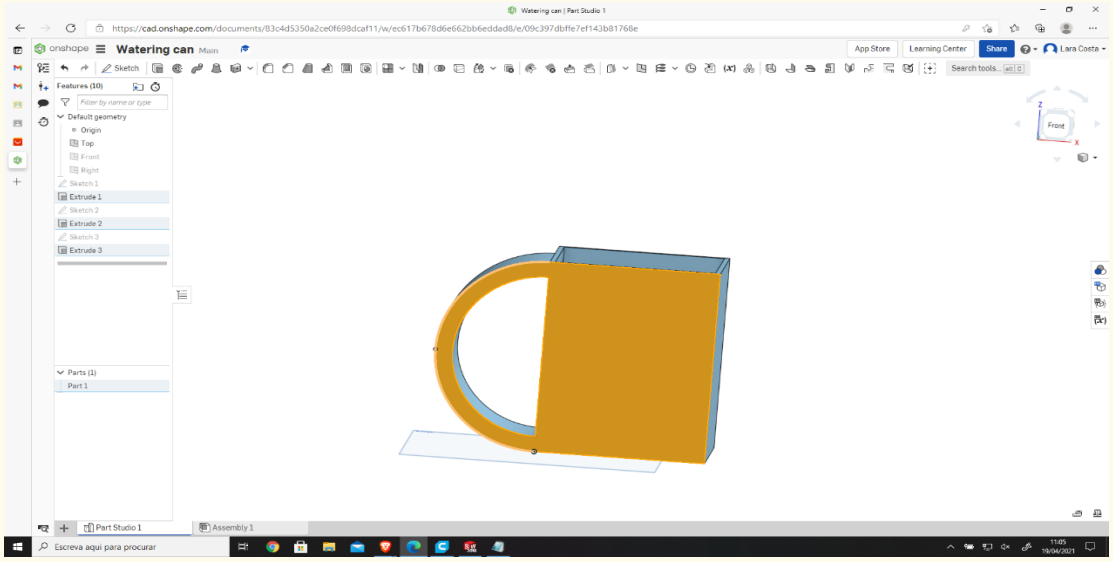
Give 10 mm as a measure.



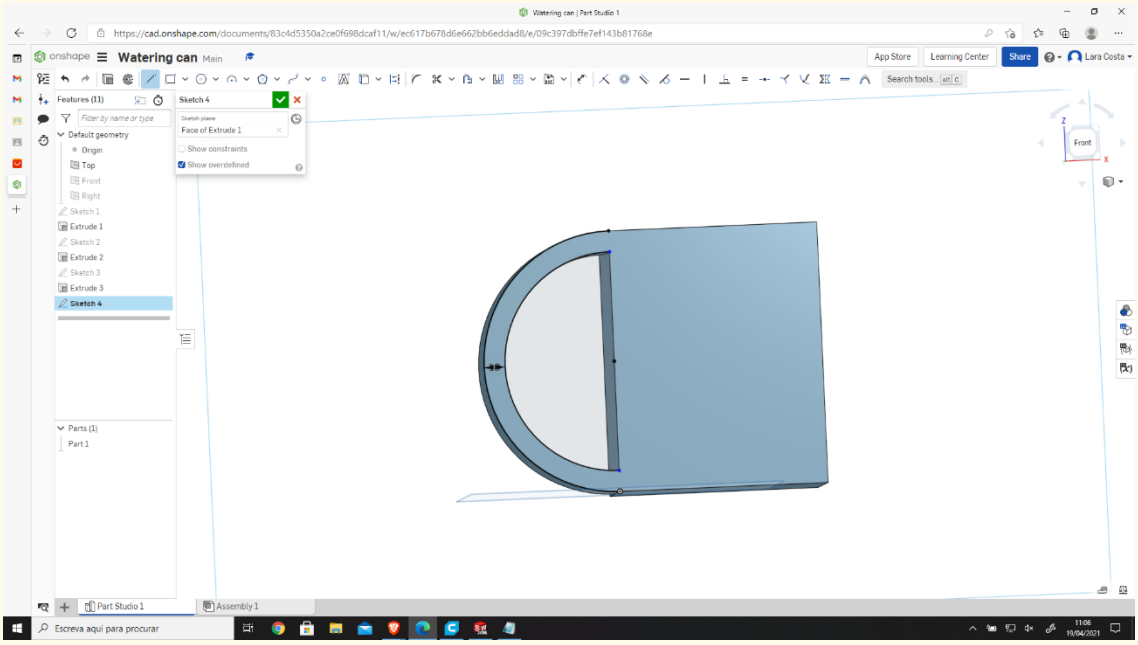
Step 21 **Click on Extrude and give 55 mm as Depth.**



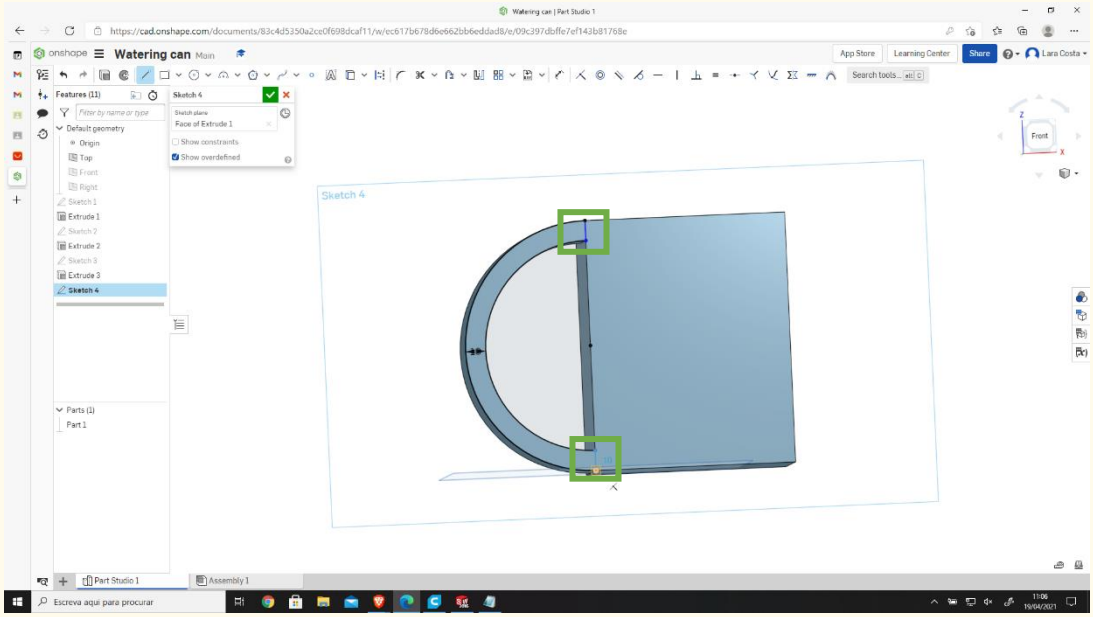
Step 22 **Select the front side again and sketch.**



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

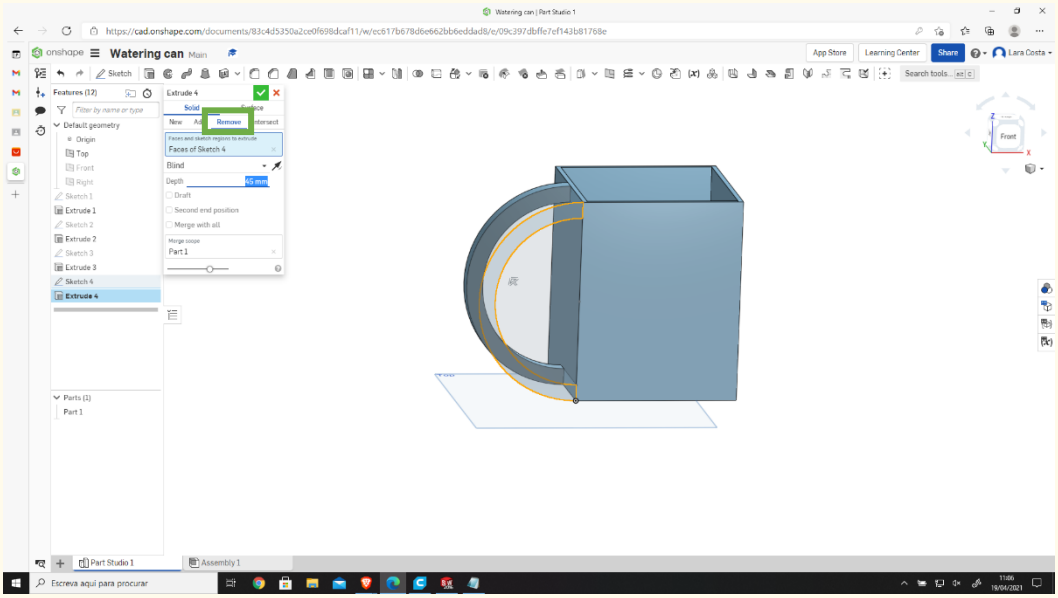


Step 24 **Select line and do it as the image shows (blue lines).**



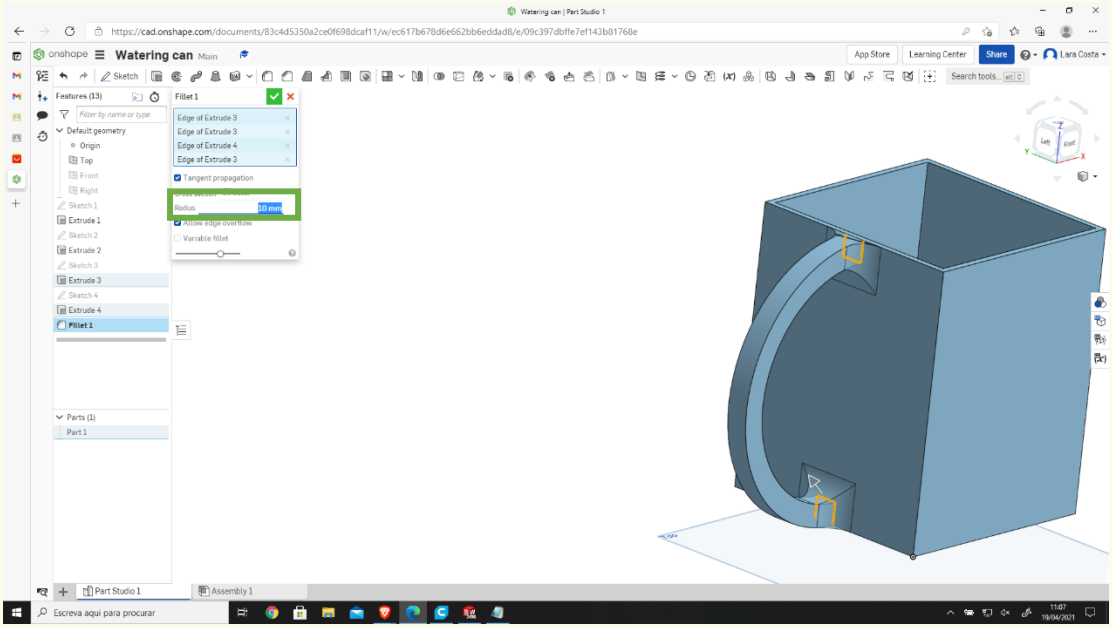
Step 25

Select Extrude – remove – change Depth to 45mm.



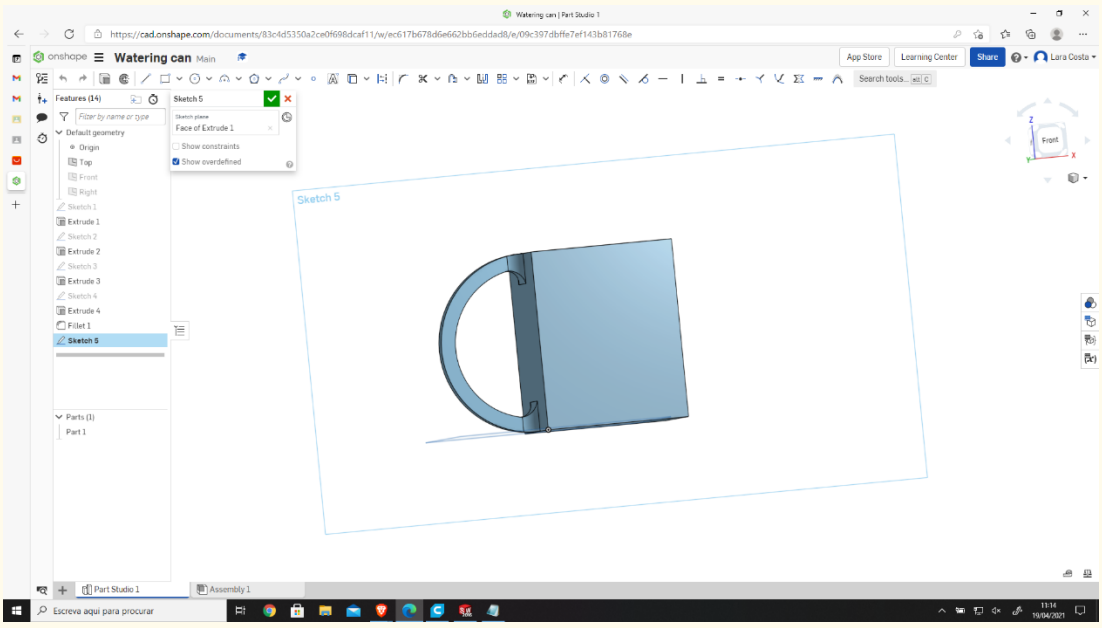
Step 26

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



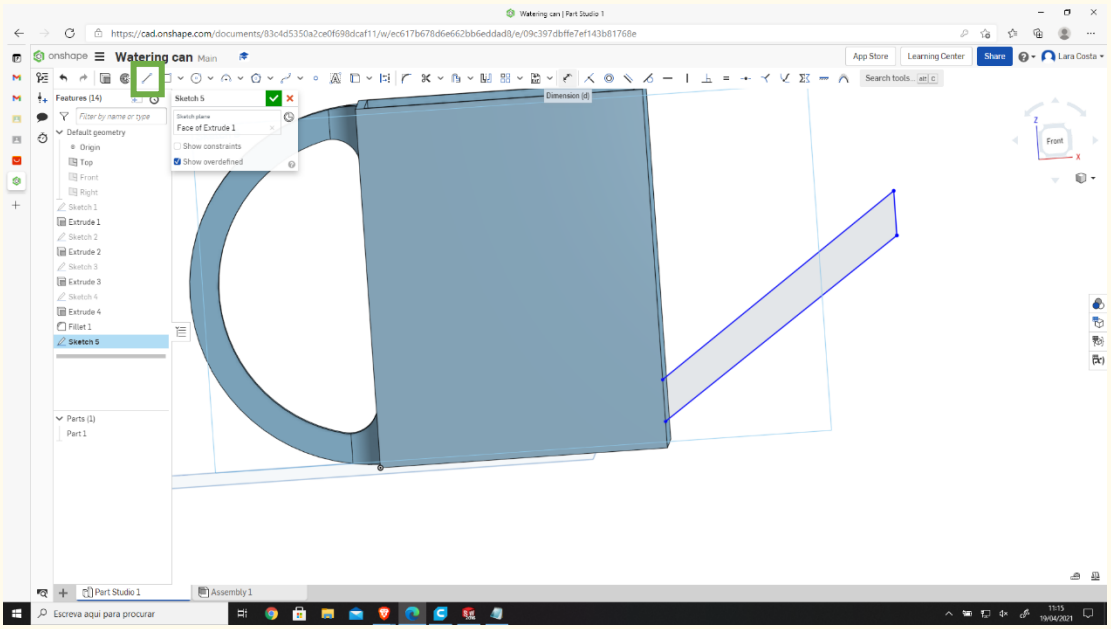
Step 27

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



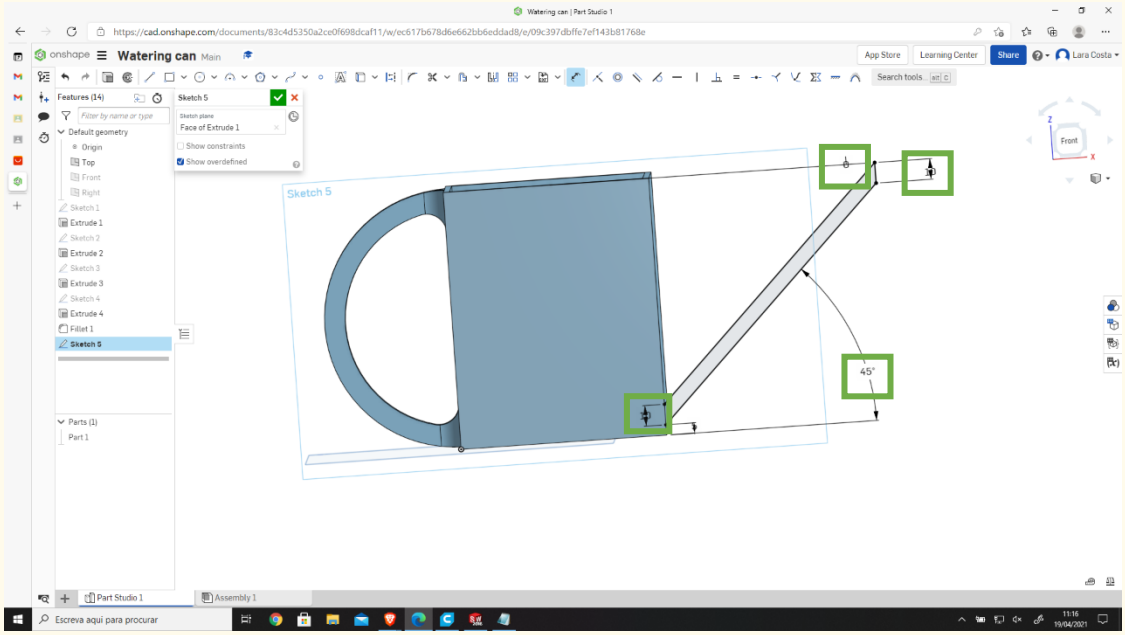
Step 28

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



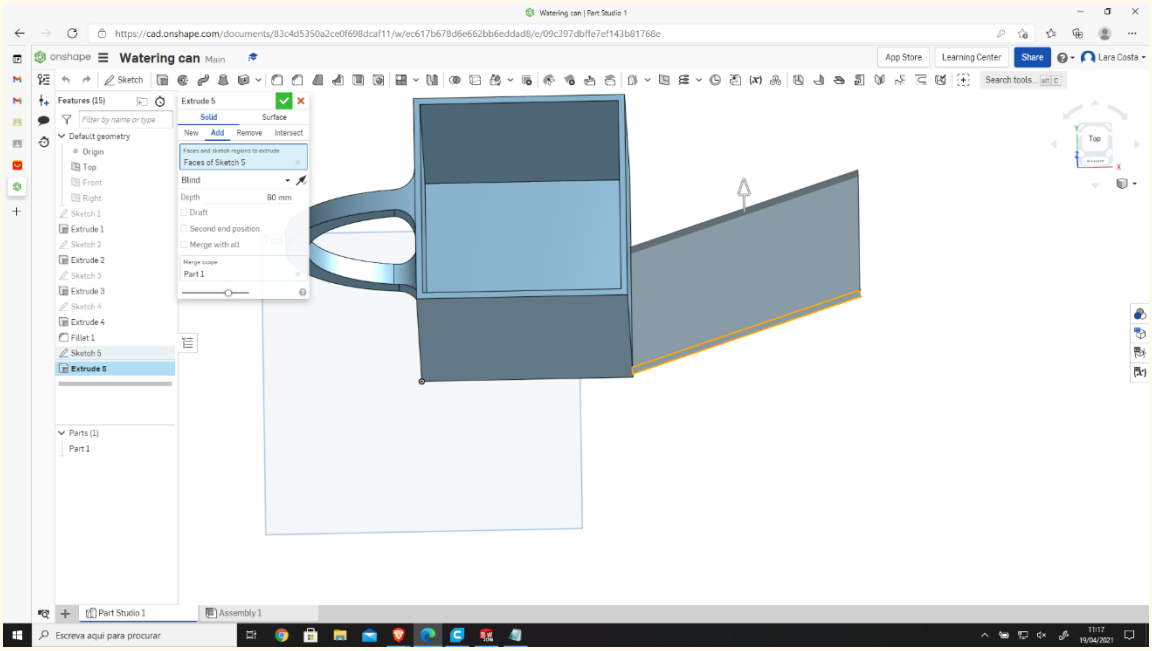
Step 29

Select Dimension and put the right dimensions.



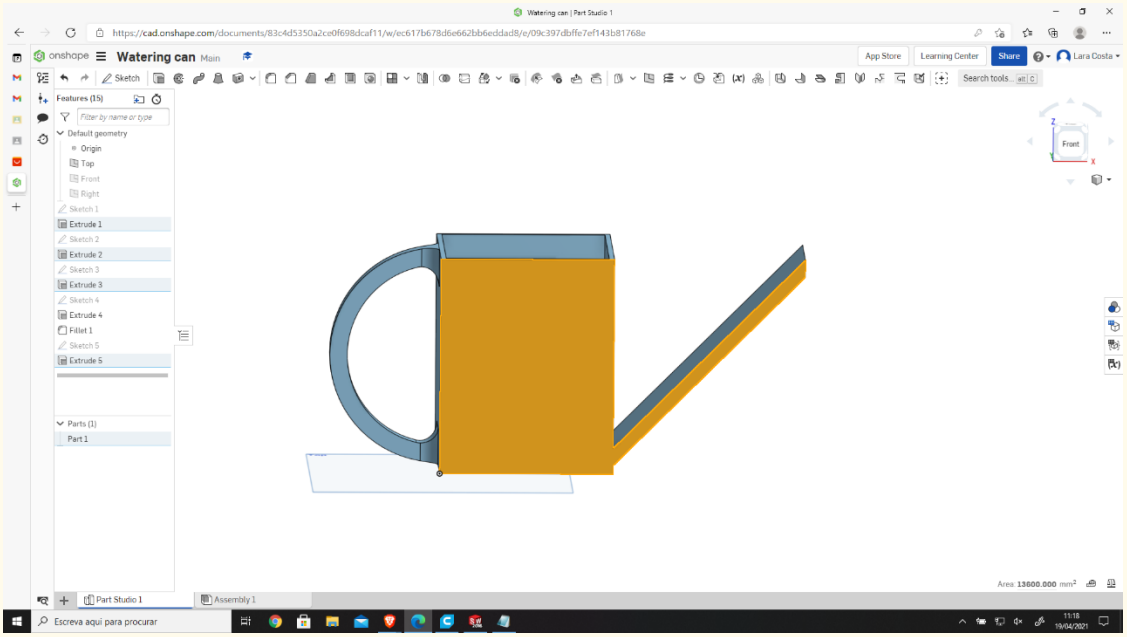
Step 30

Select Extrude and give 60 mm as Depth.



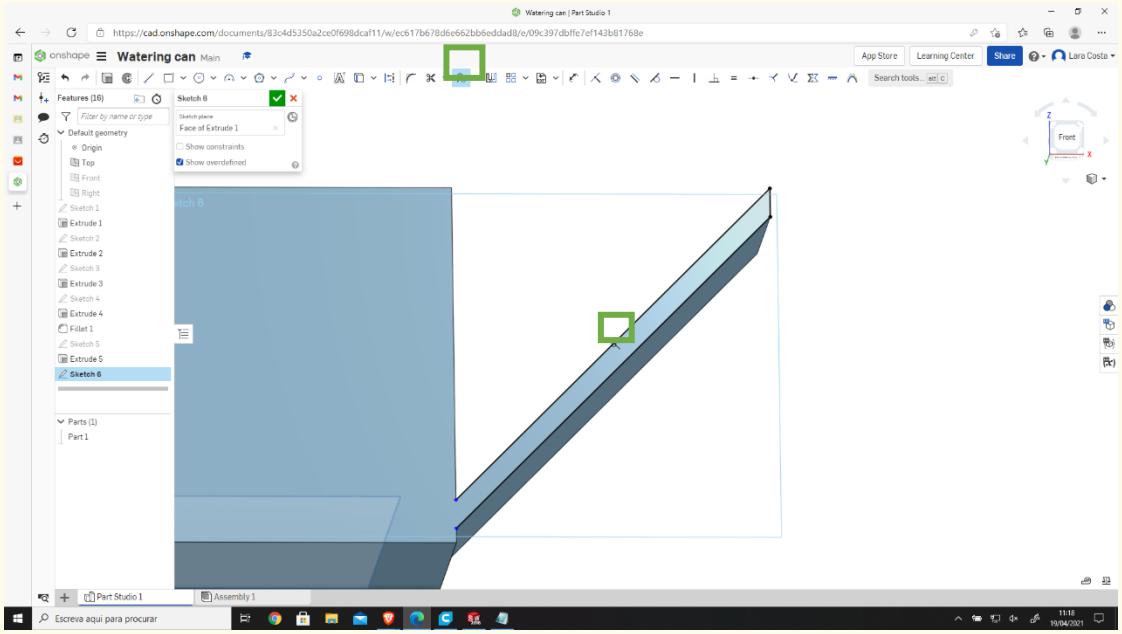
Step 31

Select the front plan and then sketch.



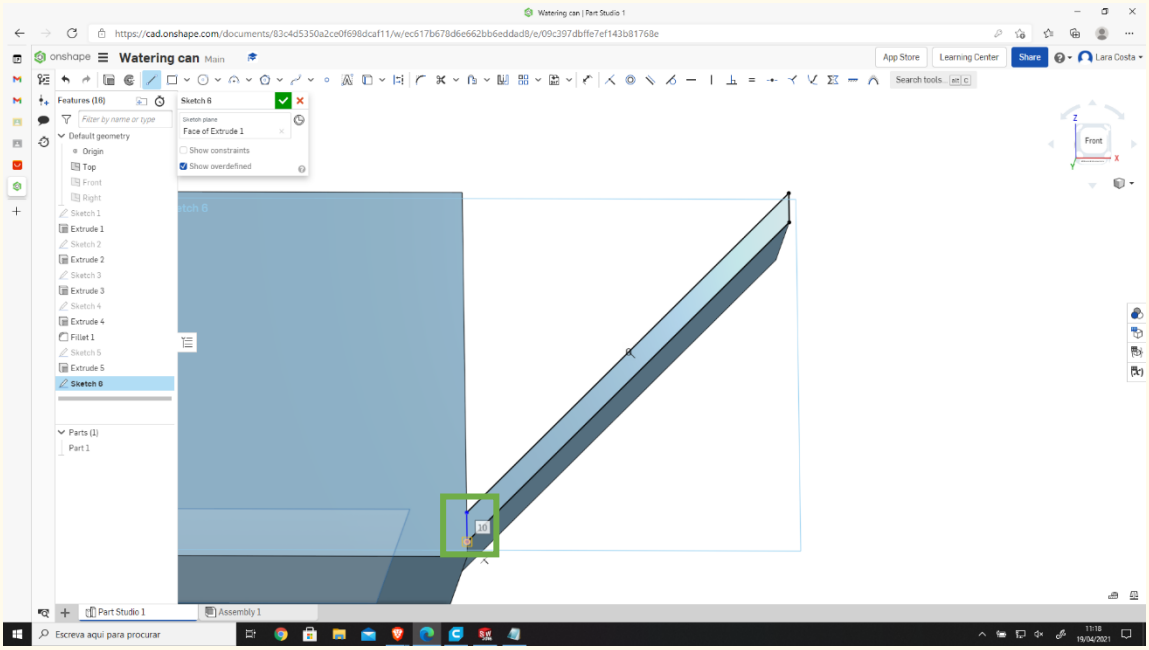
Step 32

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



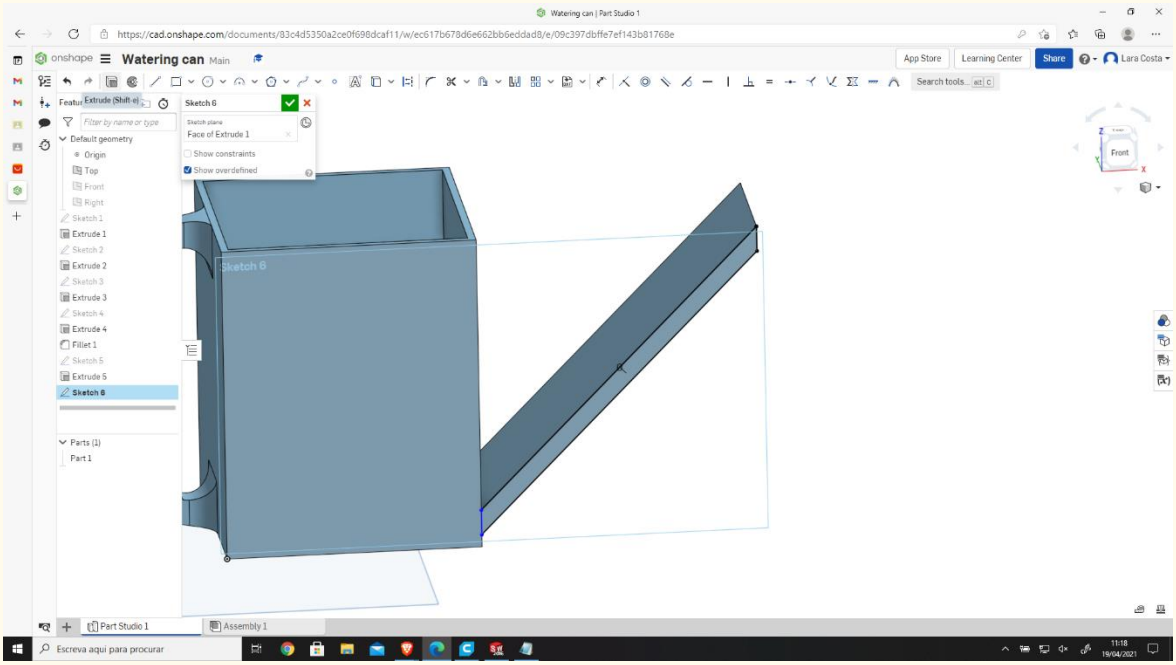
Step 33

Draw the line represented in blue.



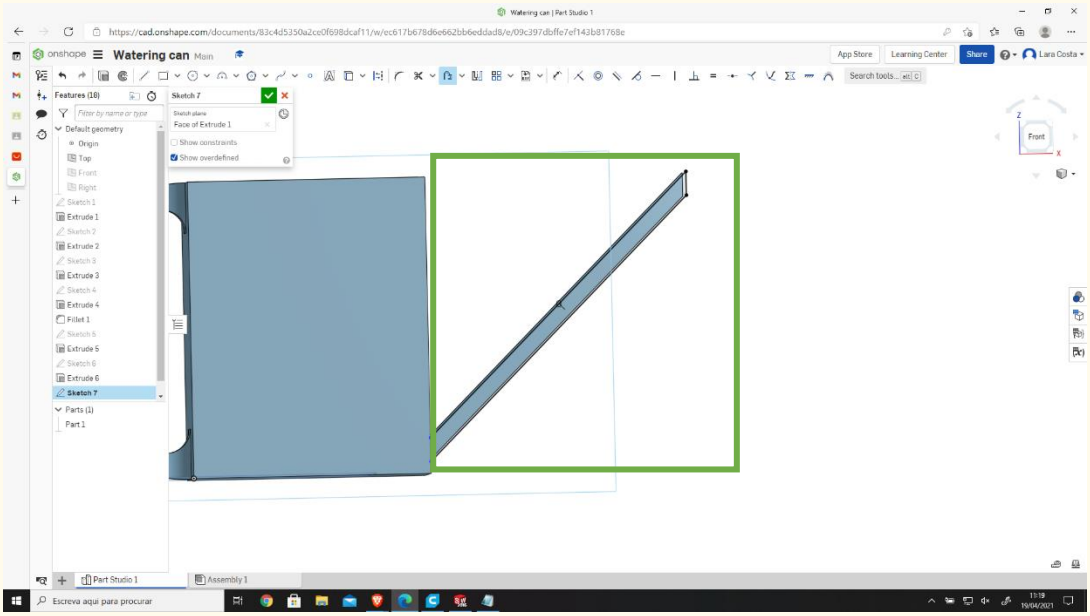
Step 34

Select Extrude.



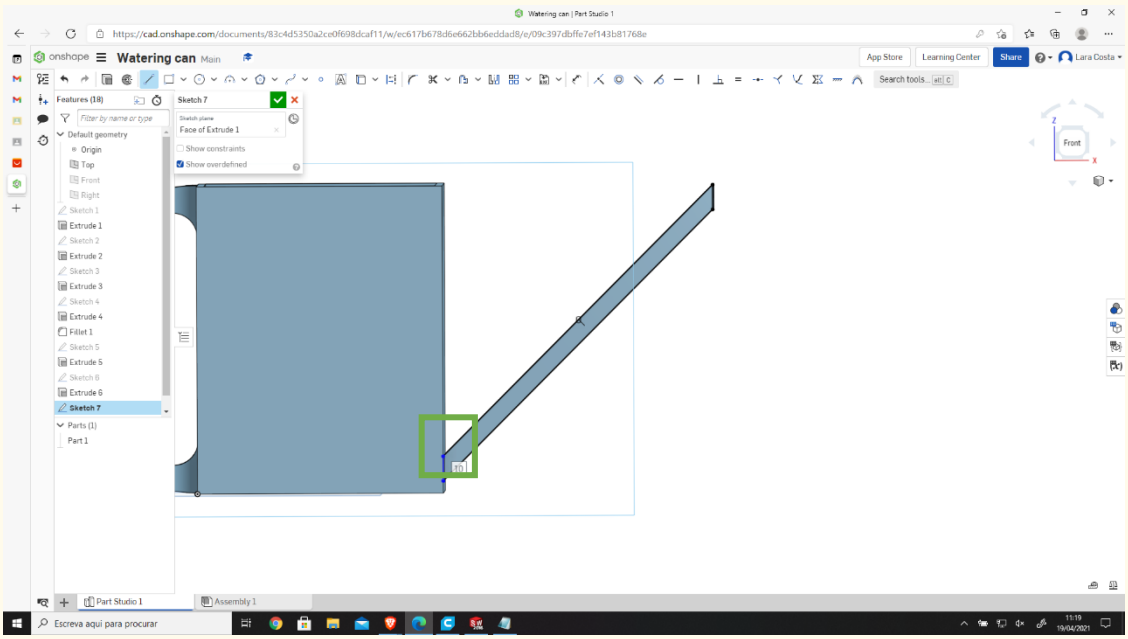
Step 35

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



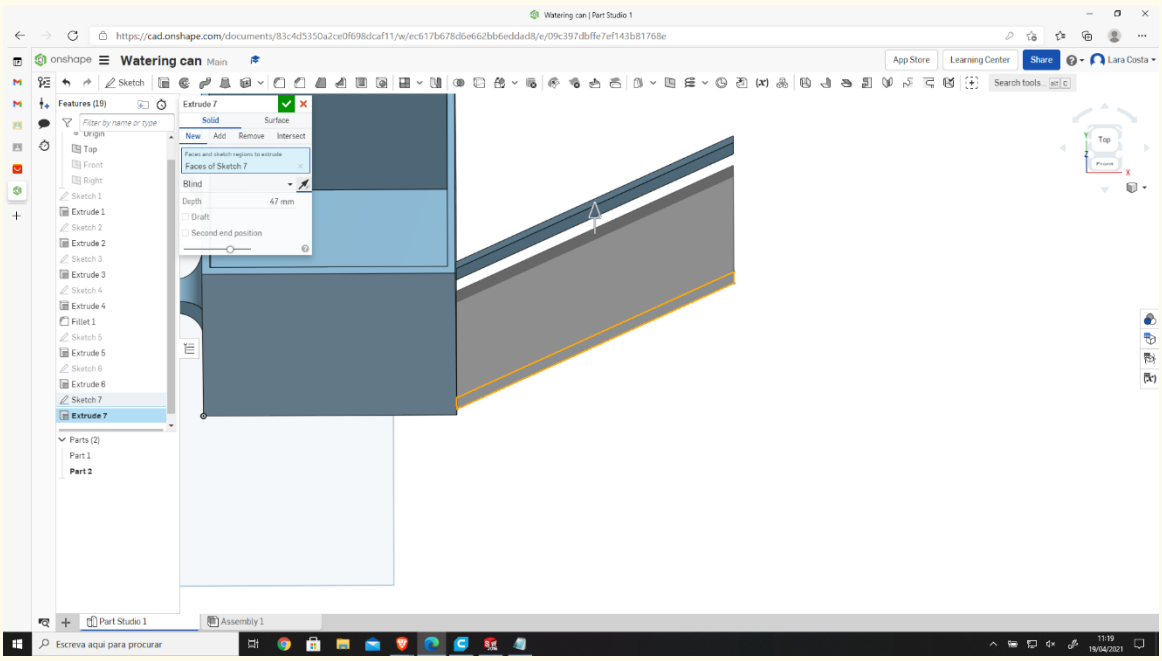
Step 36

Do the offset of black lines (the measure is 0mm) and draw the blue one.



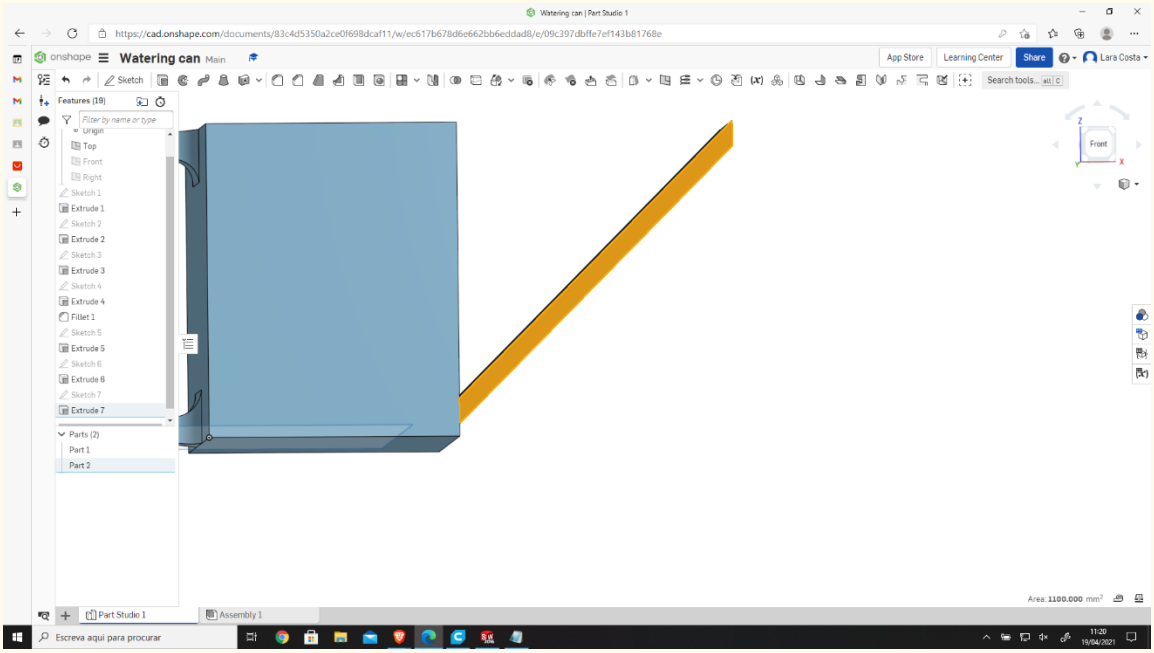
Step 37

Select extrude with 47mm of Depth.



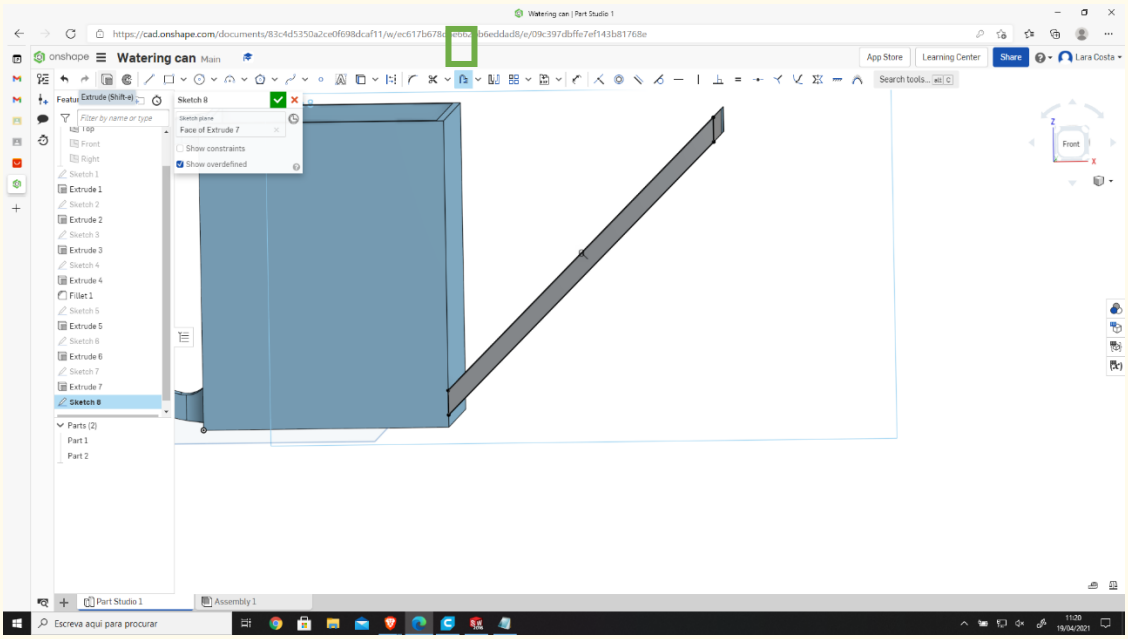
Step 37

Select the orange face.



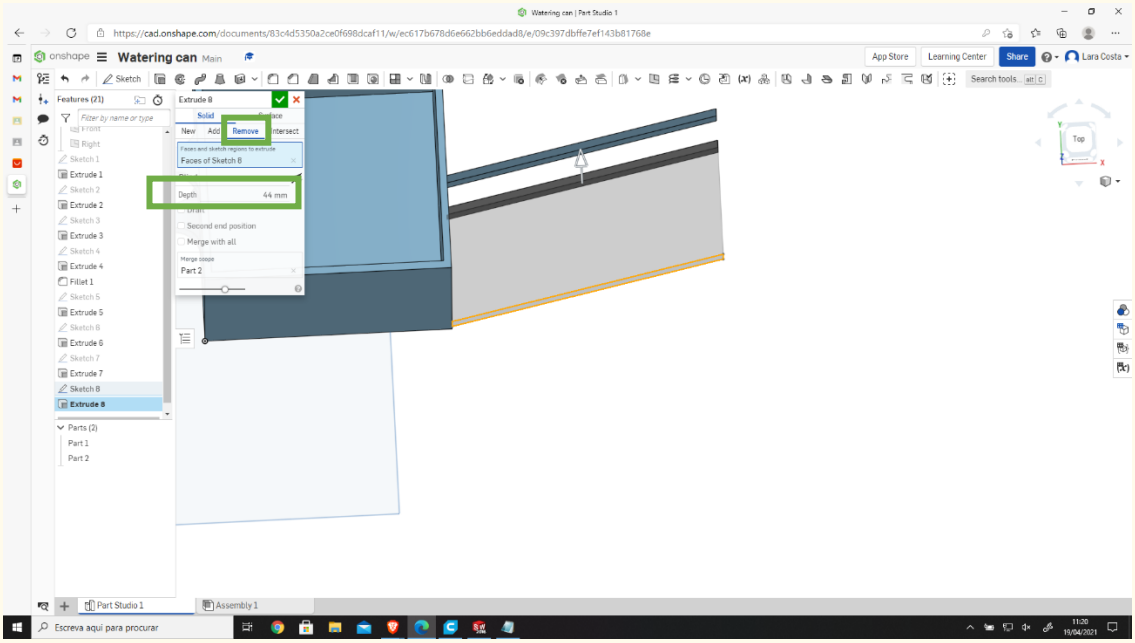
Step 38

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



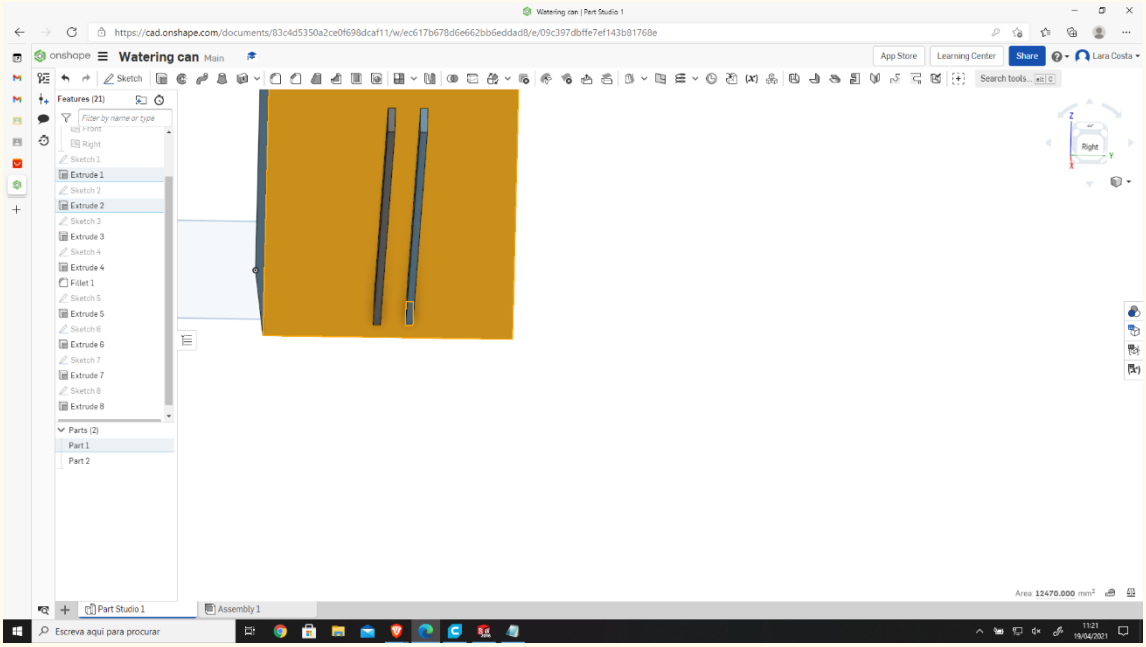
Step 39

Select extrude – remove with 44 of Depth.



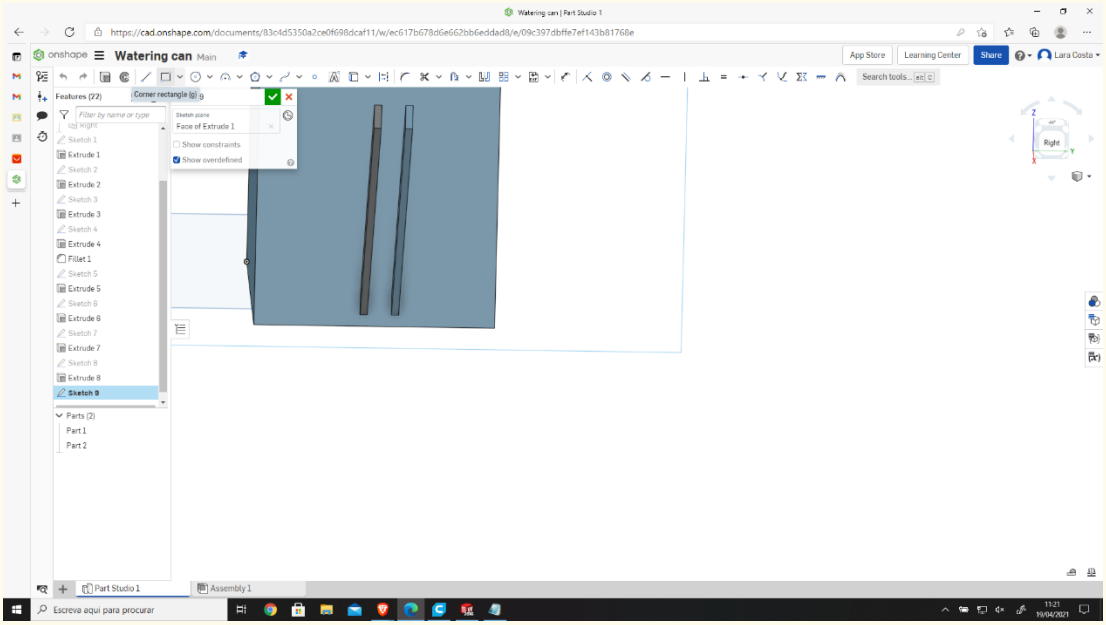
Step 40

Select the orange face.



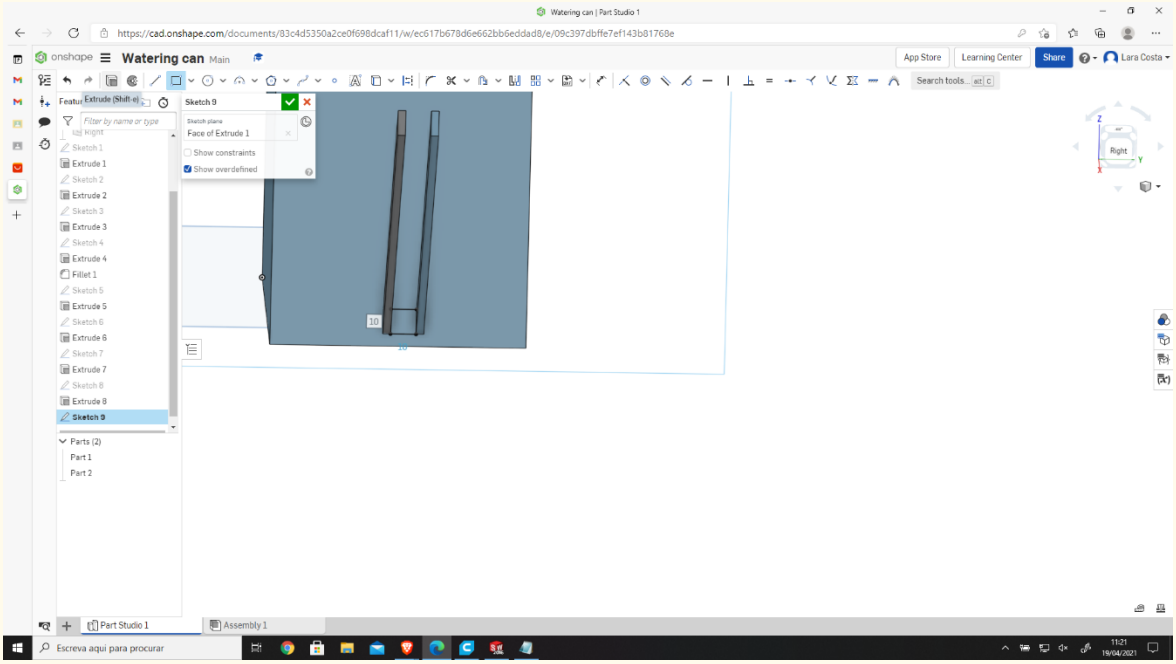
Step 41

Select corner rectangle.



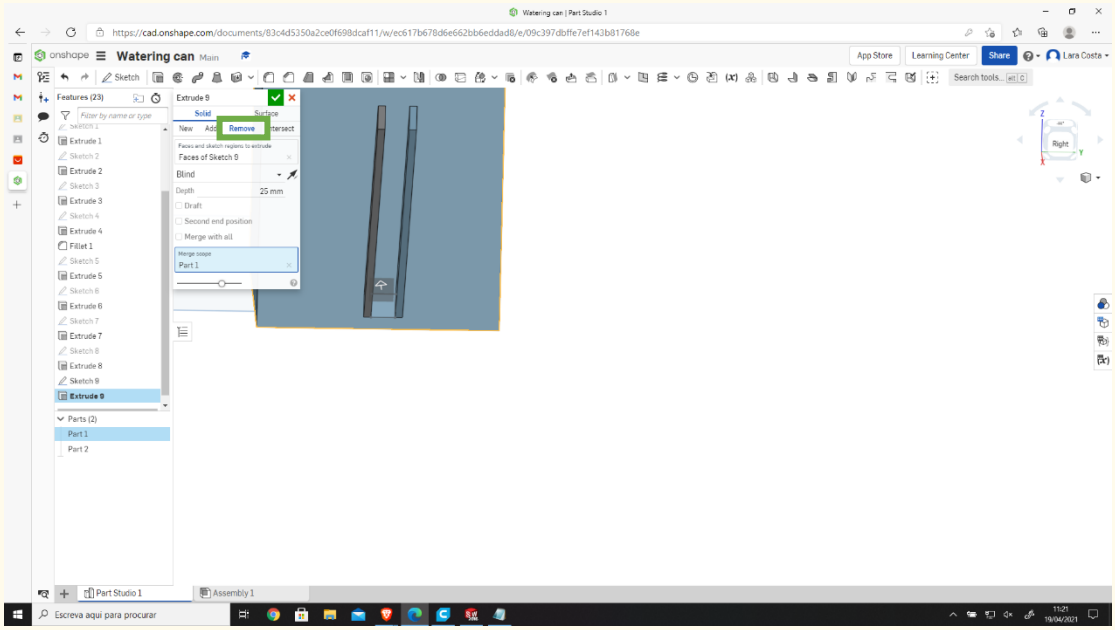
Step 42

Do the rectangle as represented.



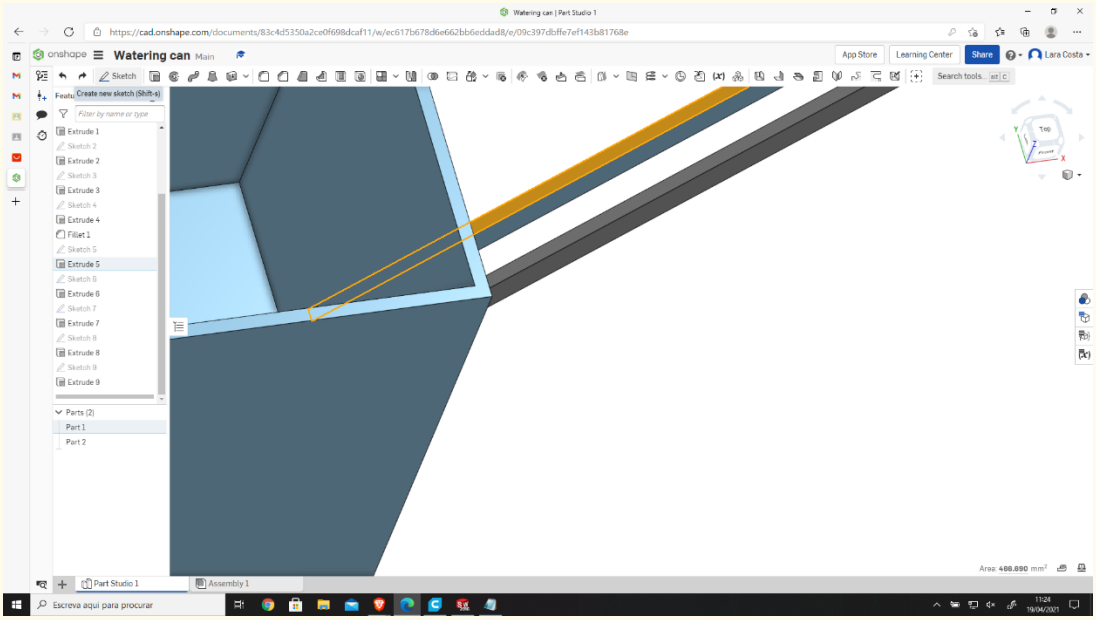
Step 43

Select – Extrude and remove (the depth is 25mm).



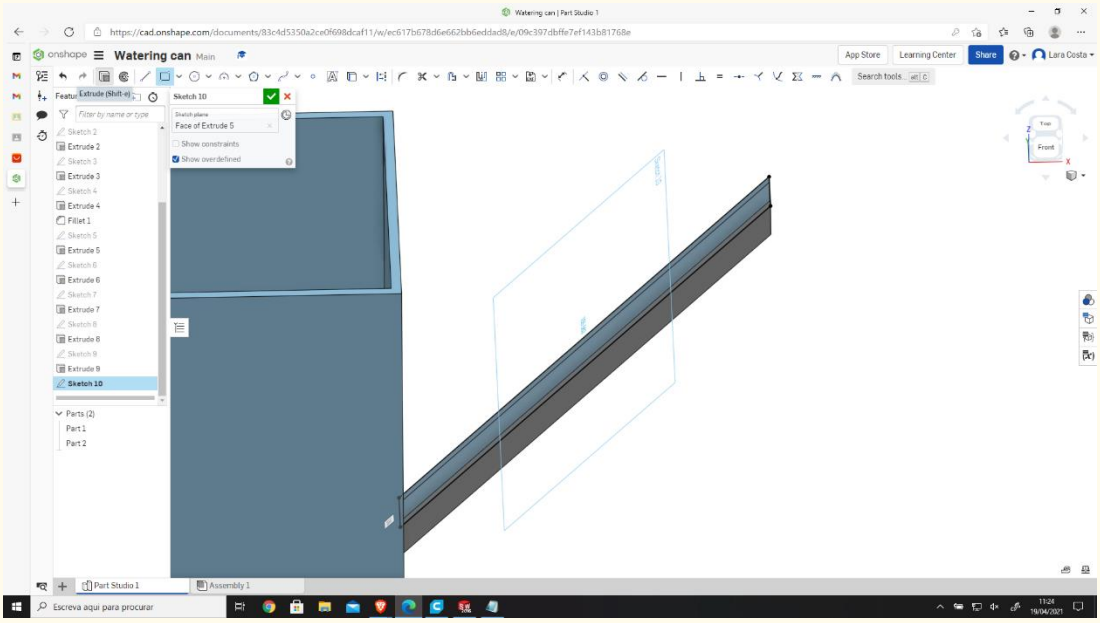
Step 44

Select the face in orange.



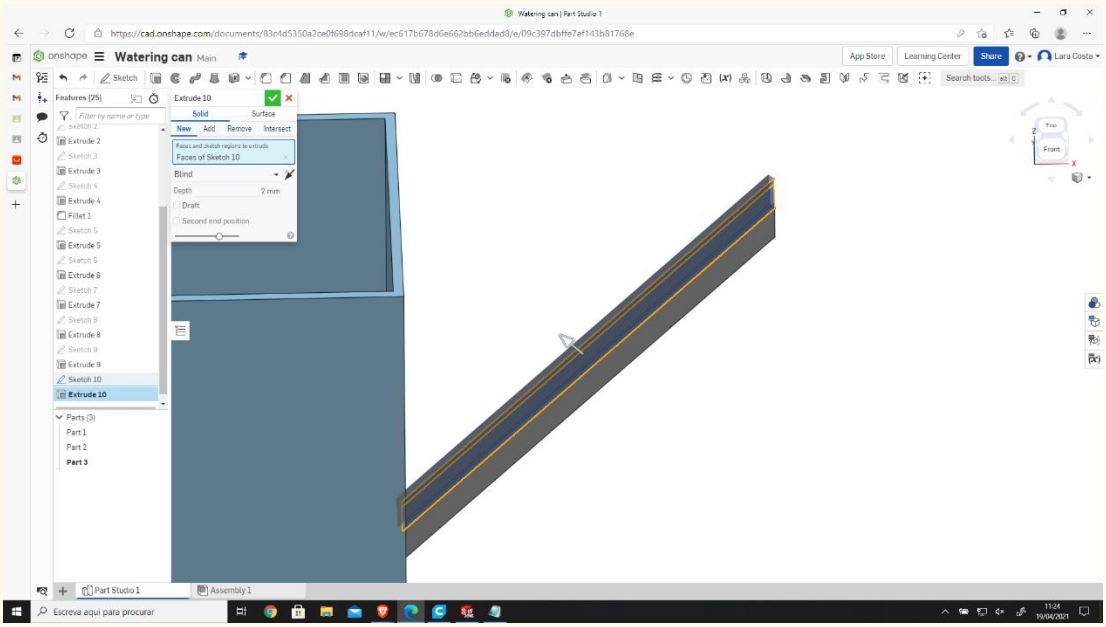
Step 45

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



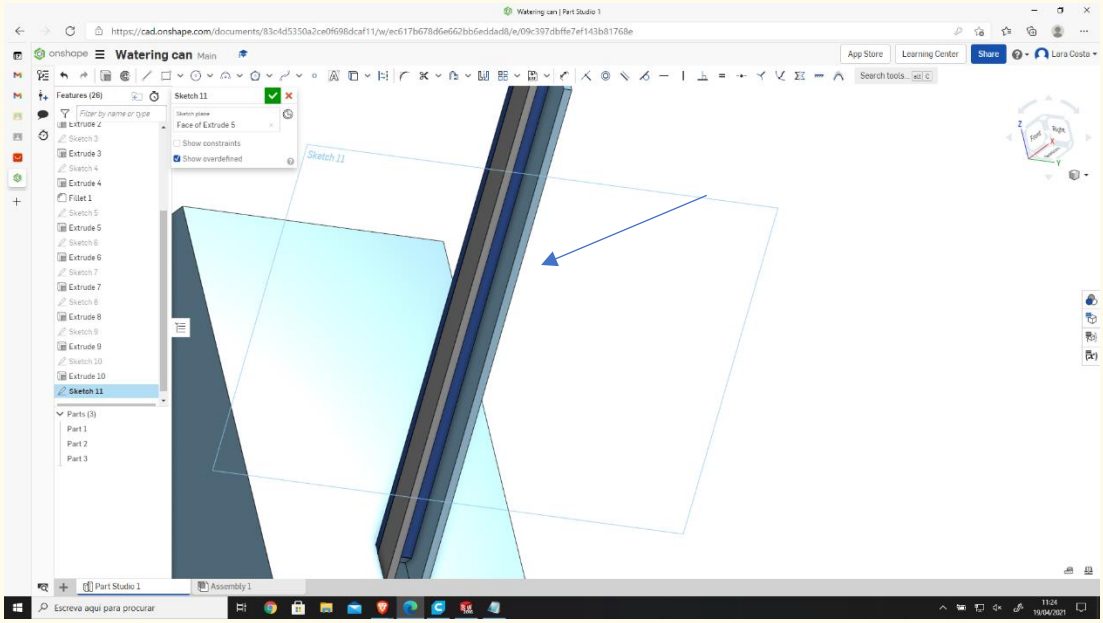
Step 46

Select Extrude, then put the Depth to 2mm.



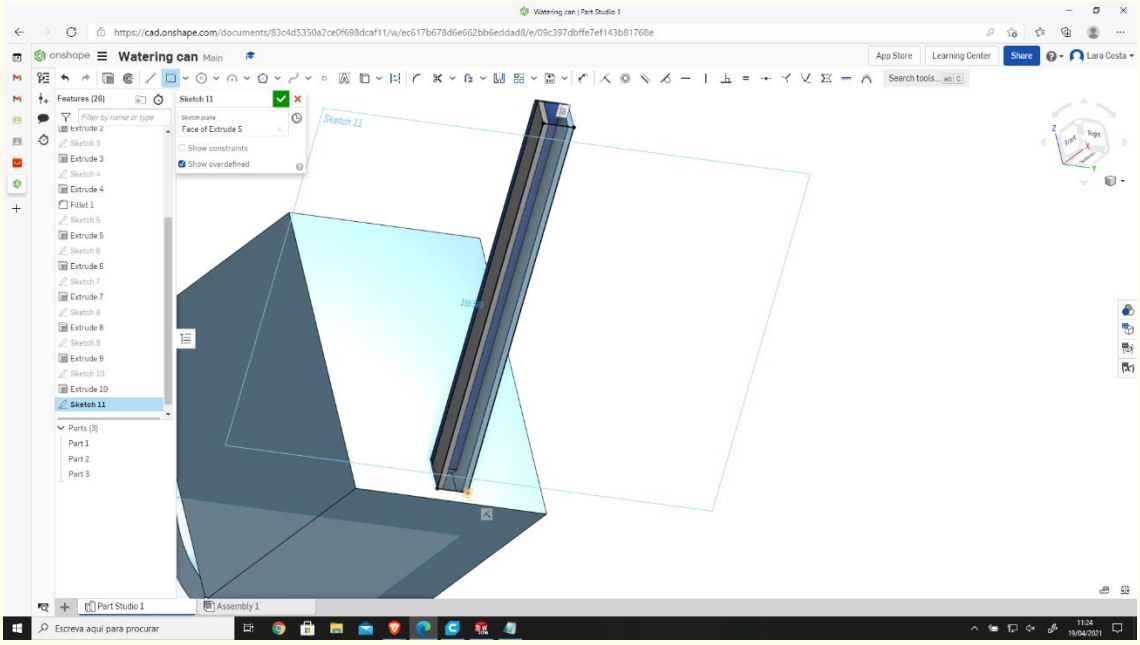
Step 47

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



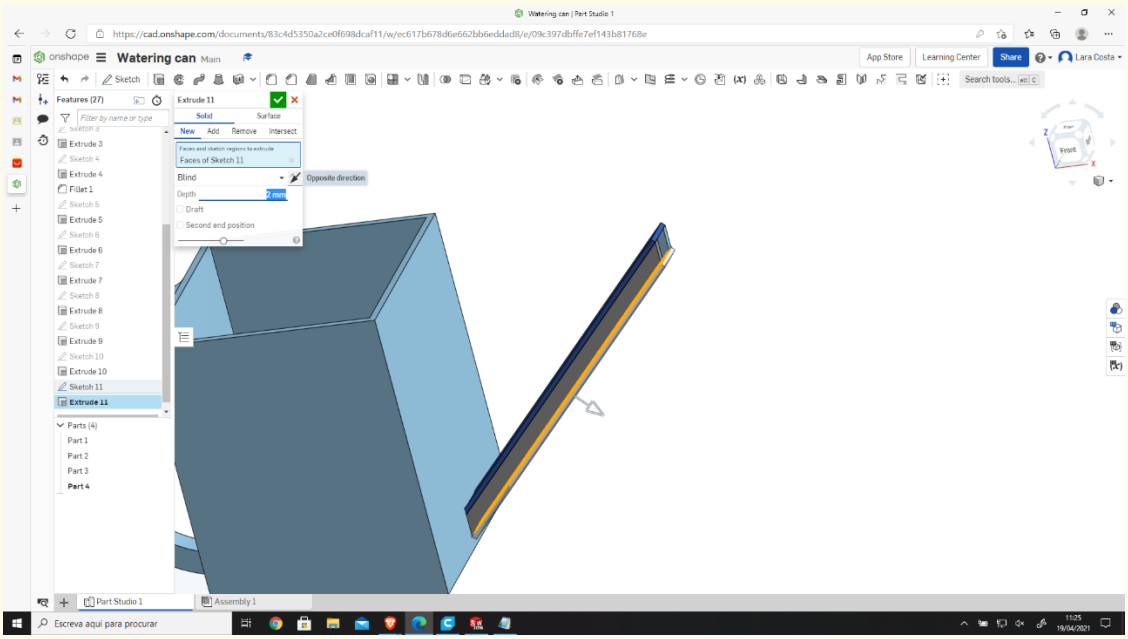
Step 48

Draw a rectangle.



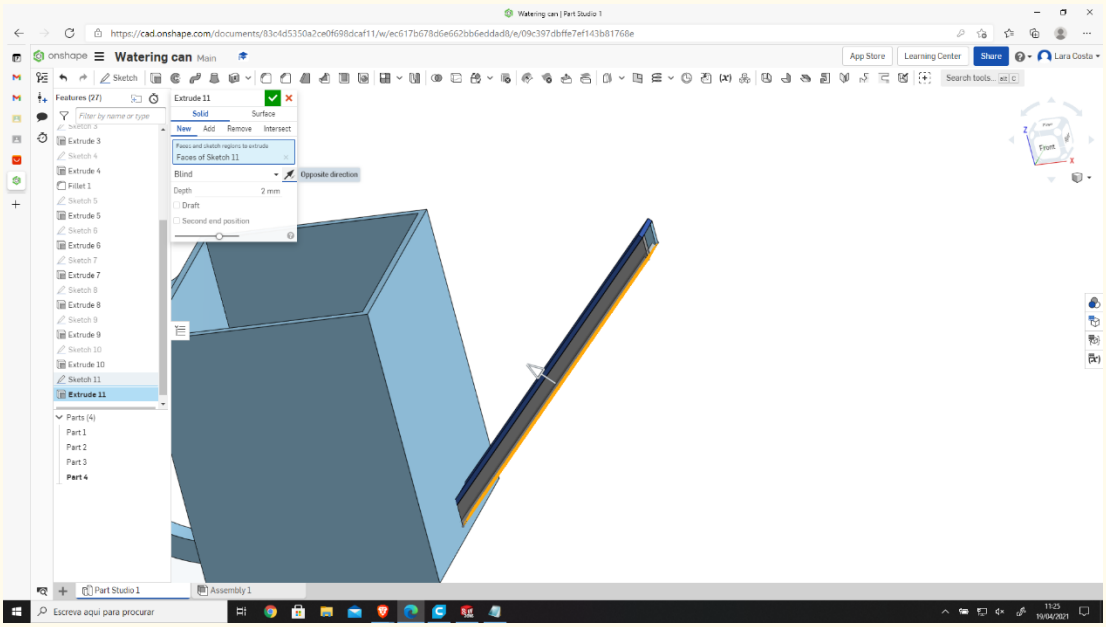
Step 49

Select extrude and give the measure 2mm.

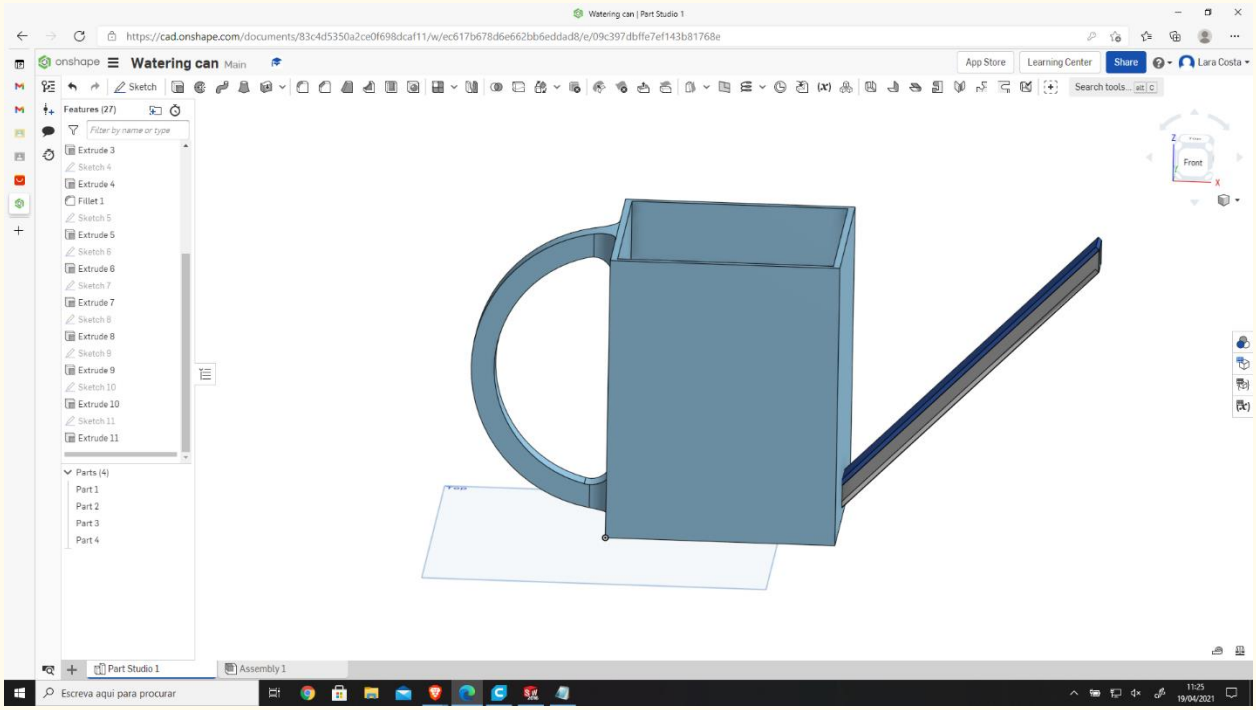


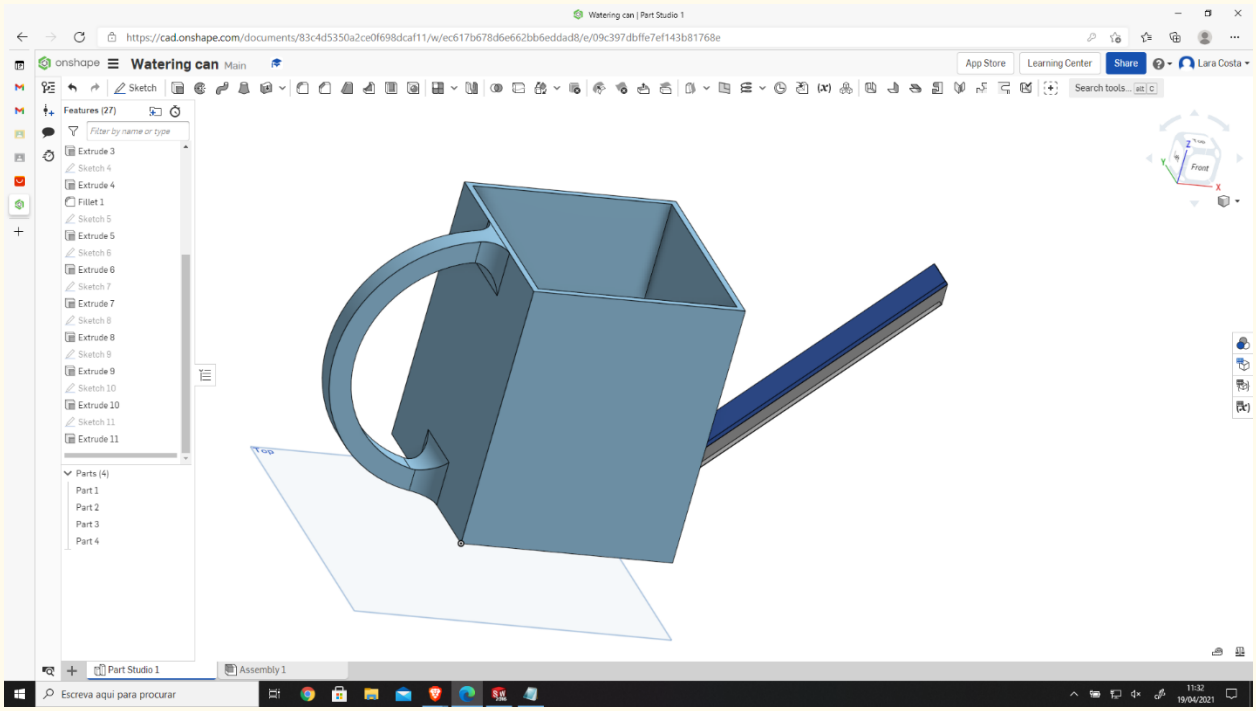
Step 50

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.





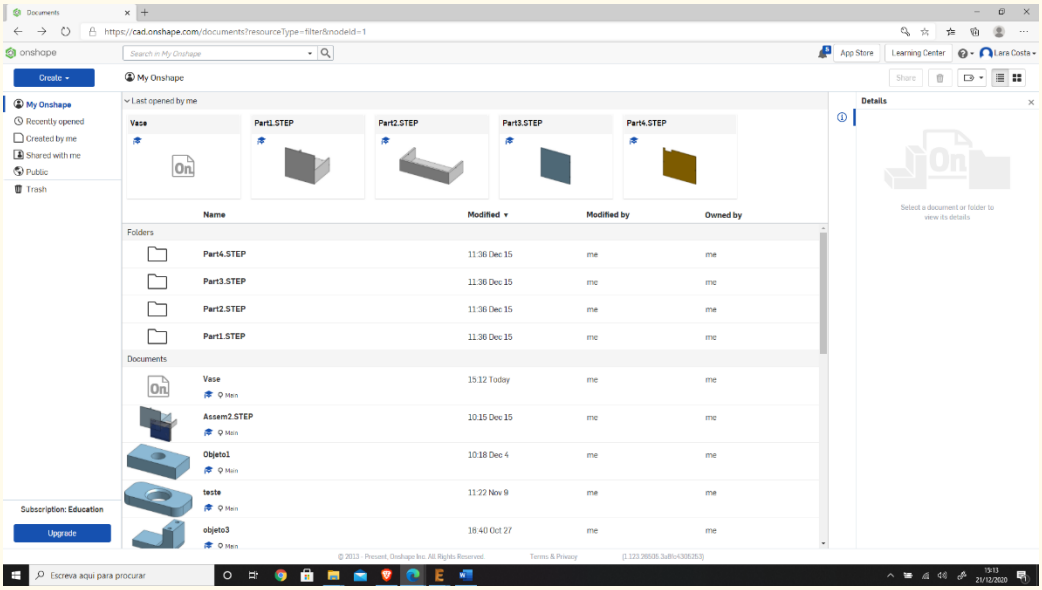
PROJECT: 3D DRAWING OF A GARDEN RAKE

- **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 competencies:**
    - 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:

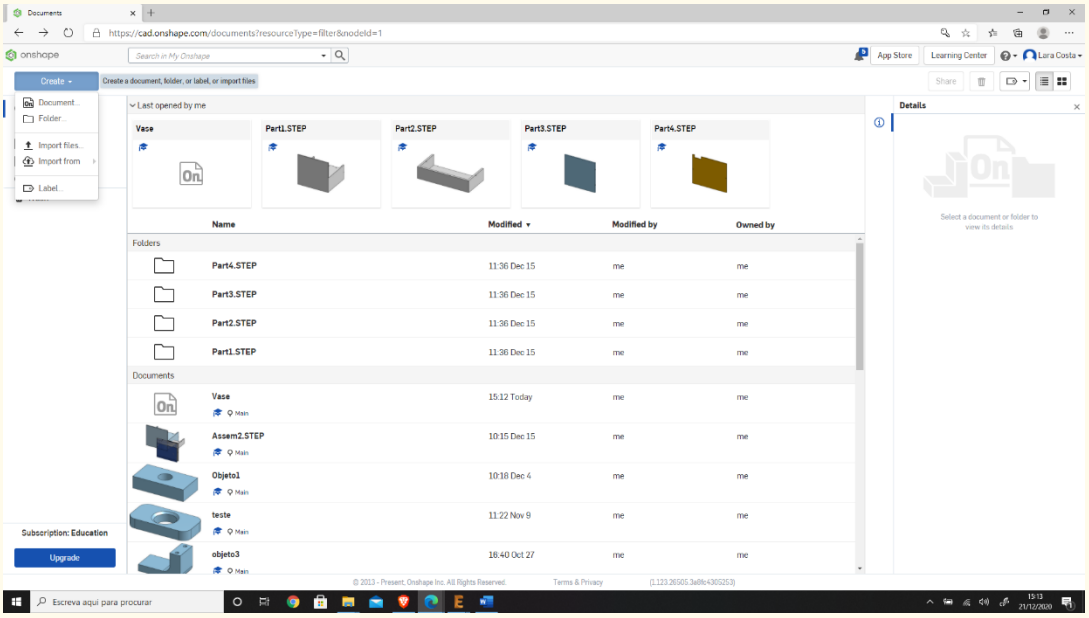
Step 1

Open Onshape.



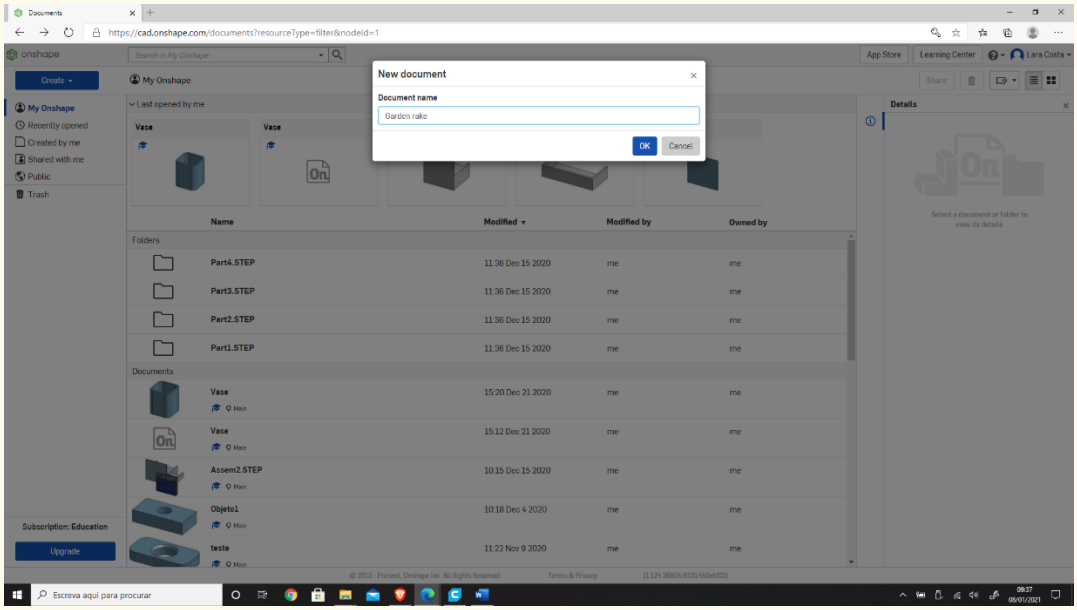
Step 2

Create a document.



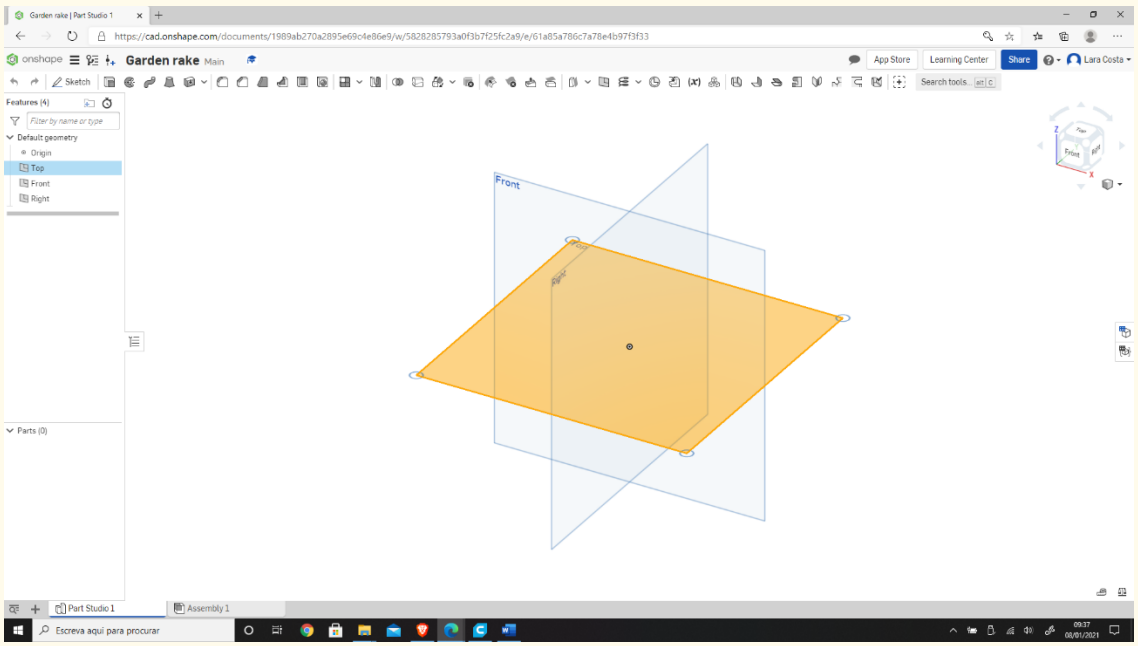
Step 3

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



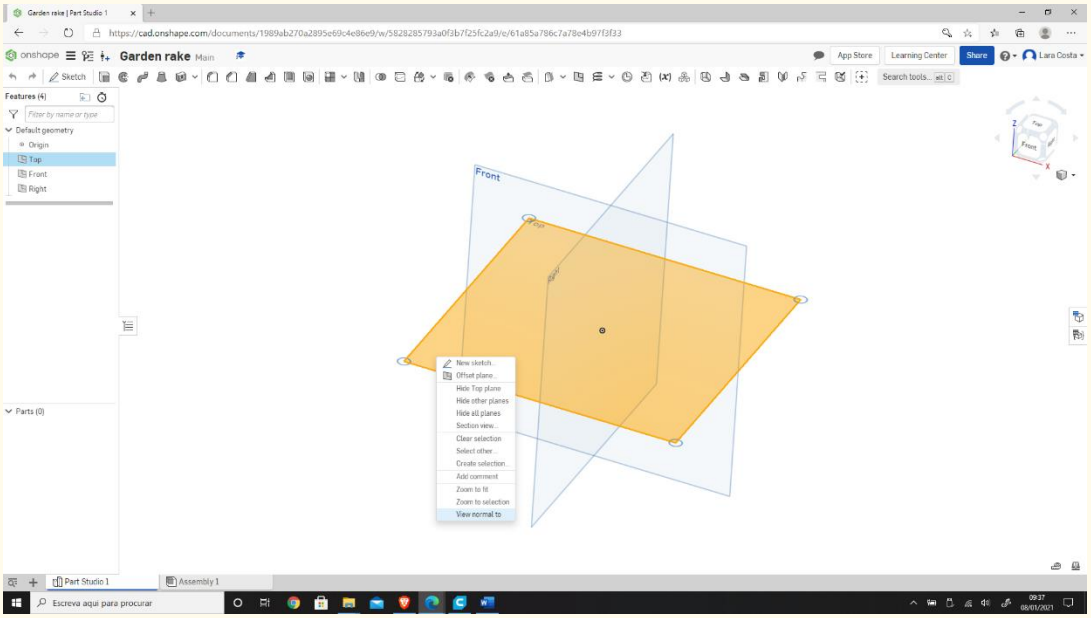
Step 4

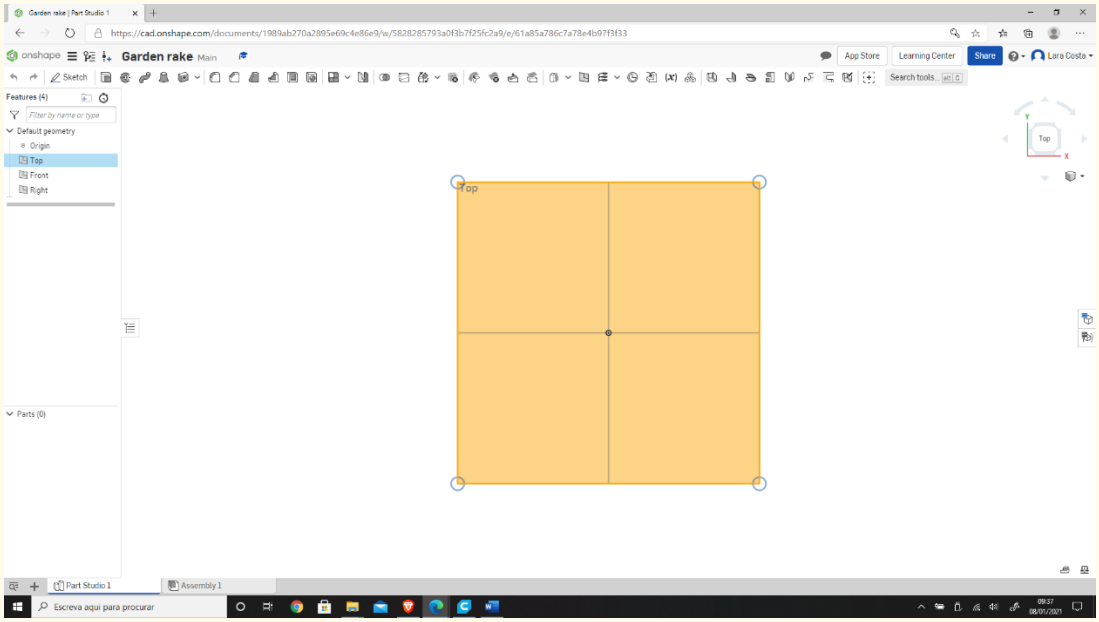
Select the plane (top) to start drawing.



Step 5

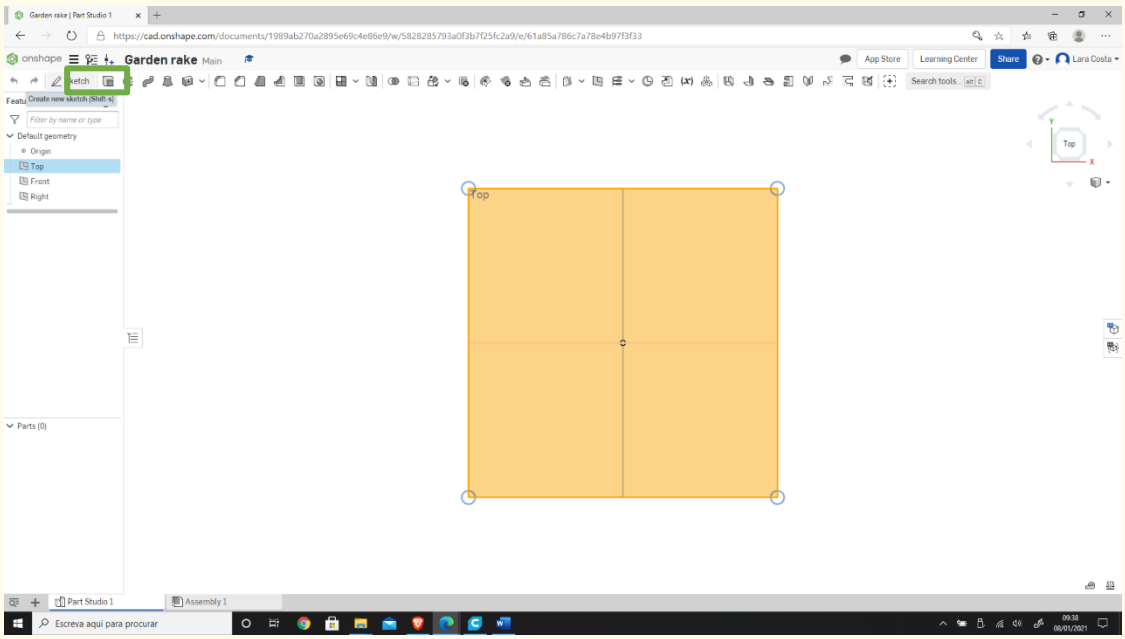
Right-click and select normal view.
The plan should look like the 2nd image.





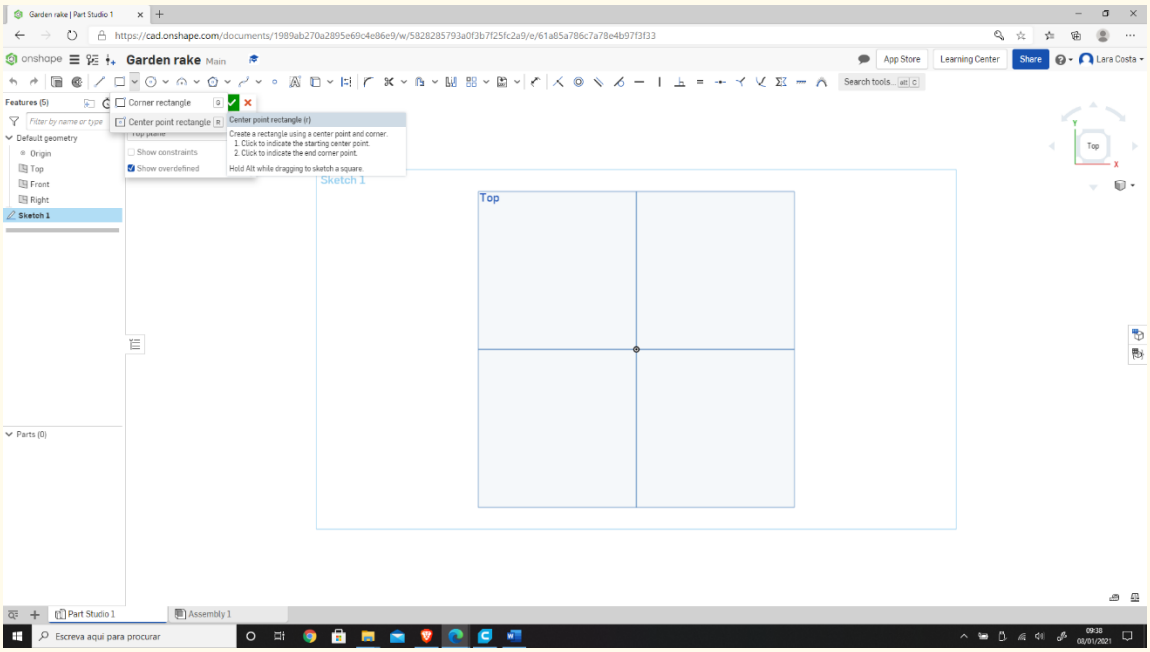
Step 6

Click Sketch.



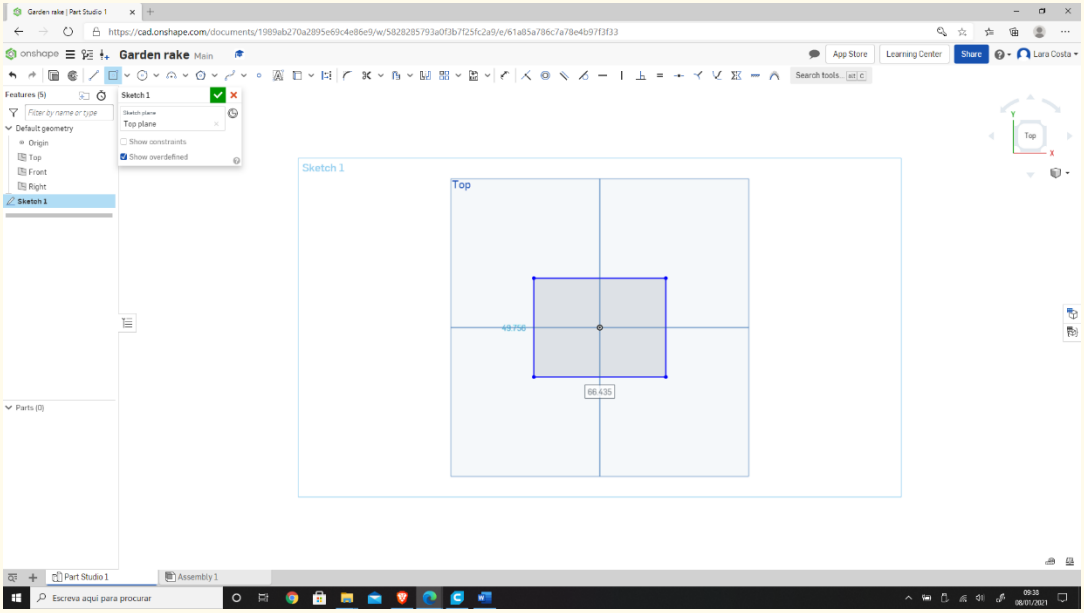
Step 7

Select the center point of the rectangle to draw.



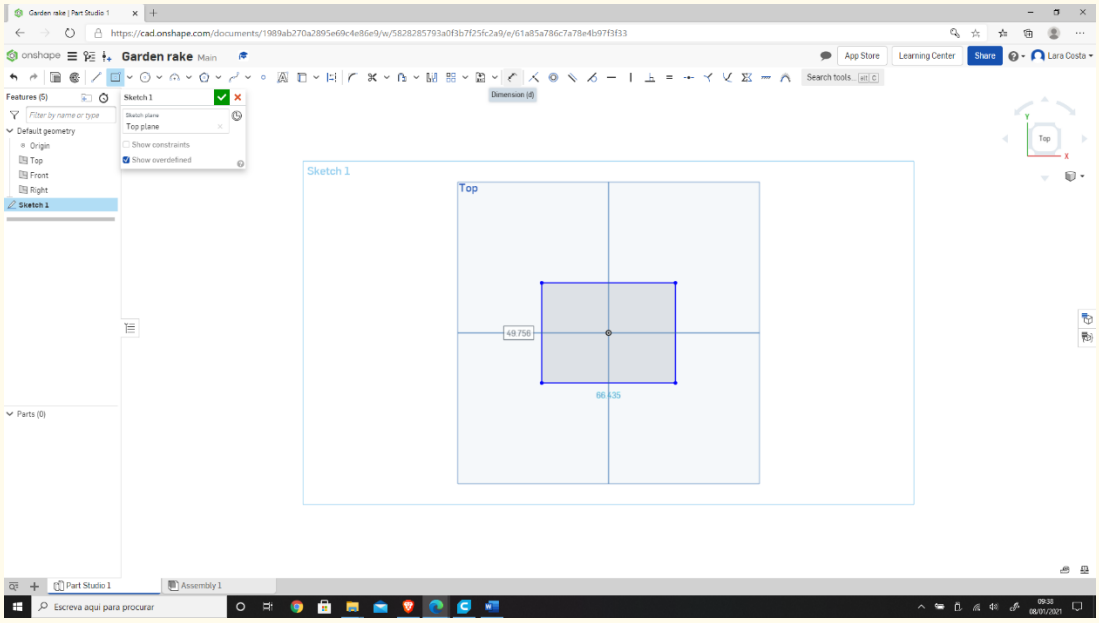
Step 8

Draw the rectangle.



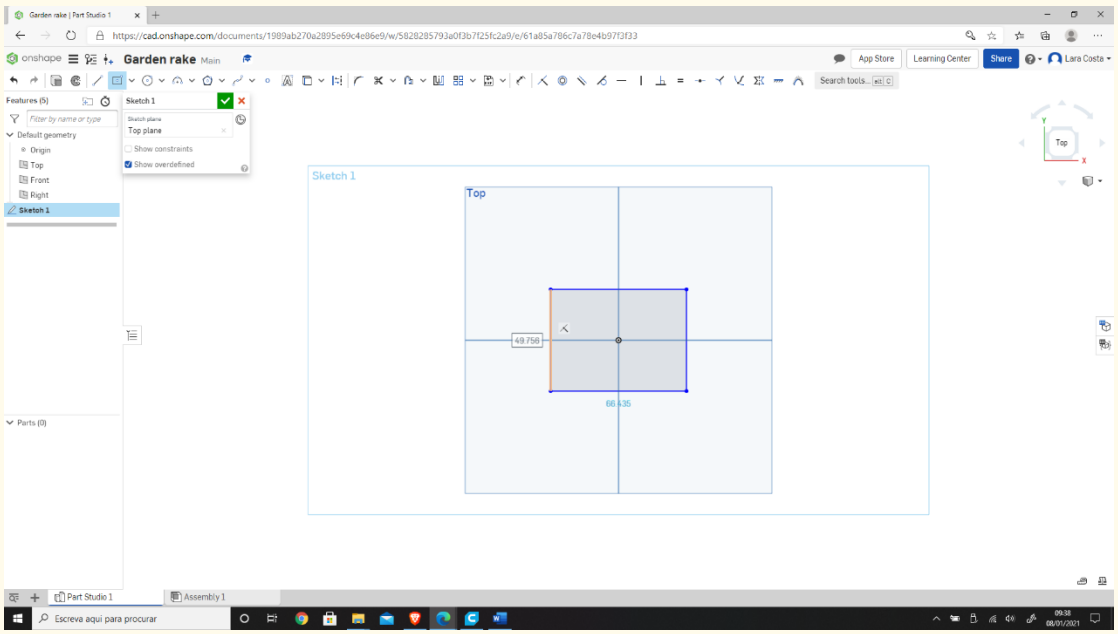
Step 9

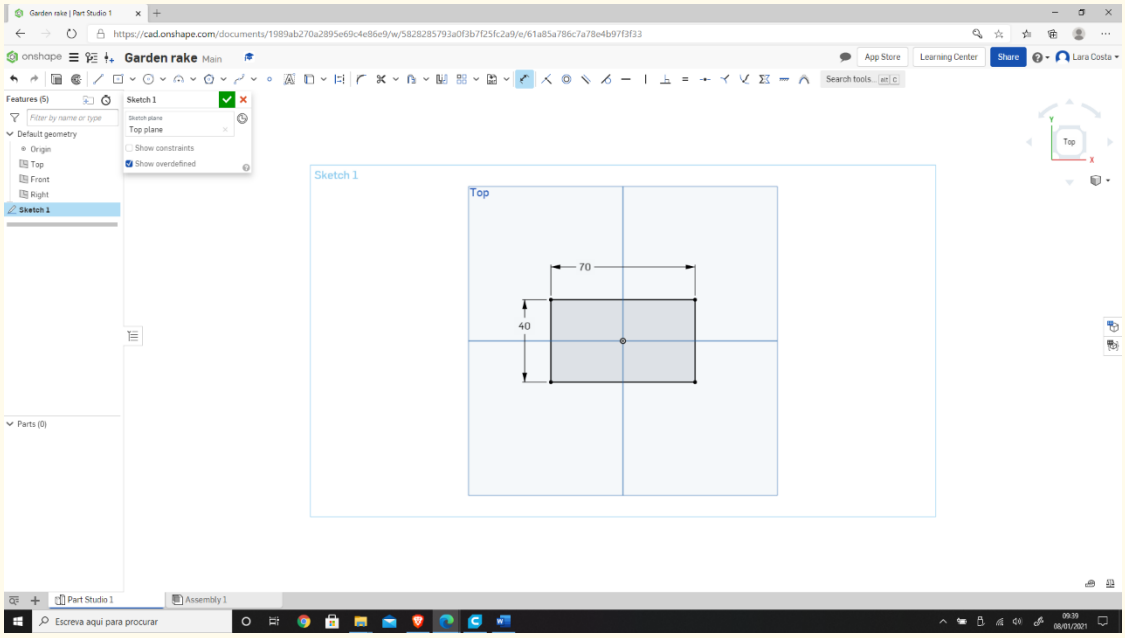
Select “Dimension” to define the measures.



Step 10

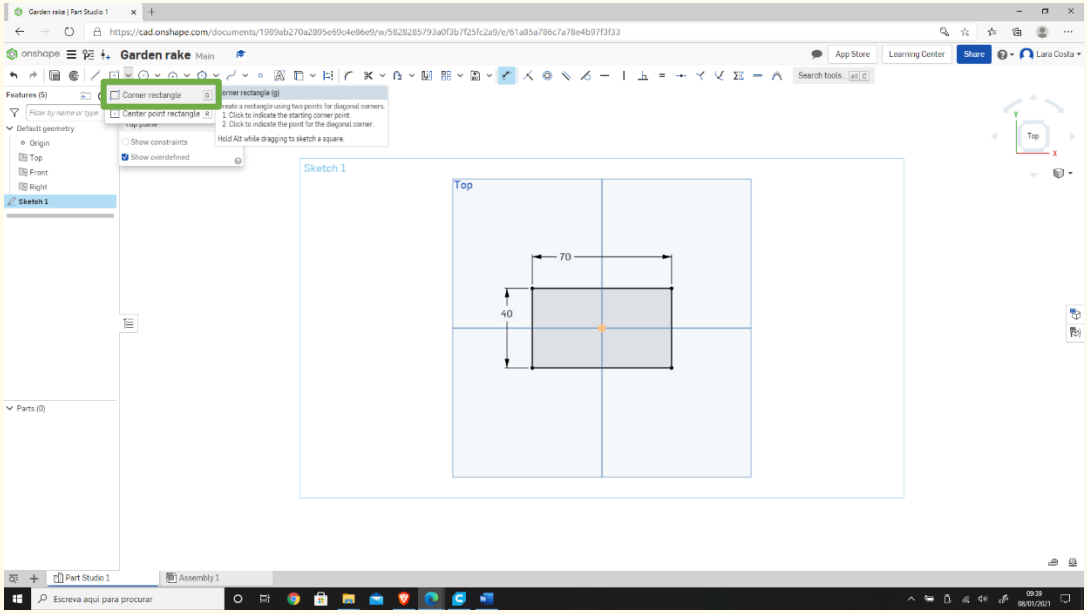
Select the line (first left then top) and give the measure 2nd figure.





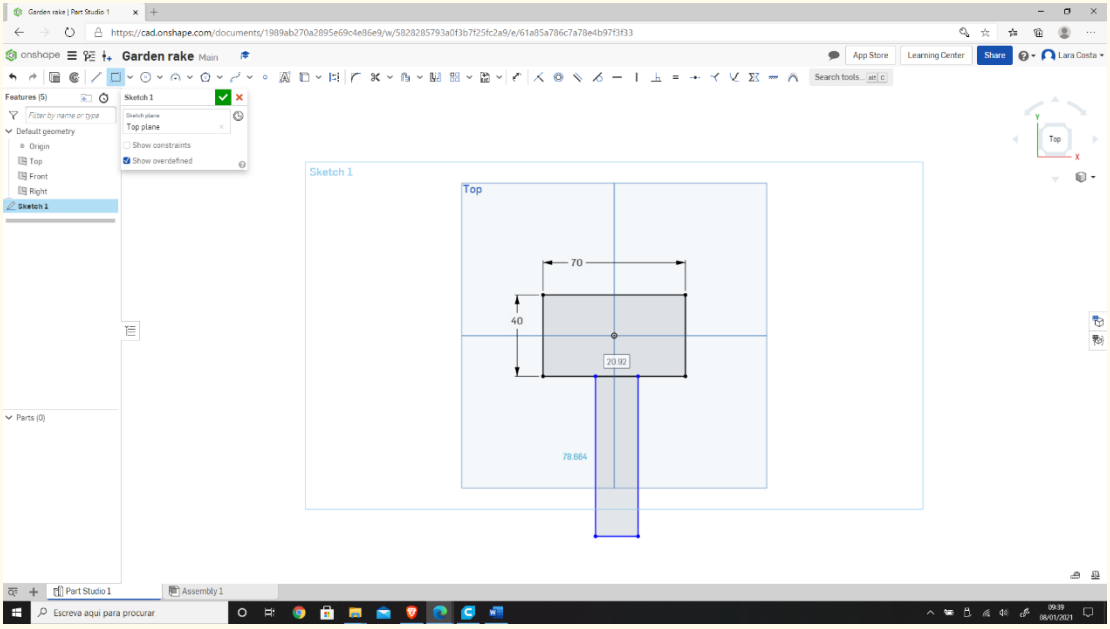
Step 11

Select corner rectangle.



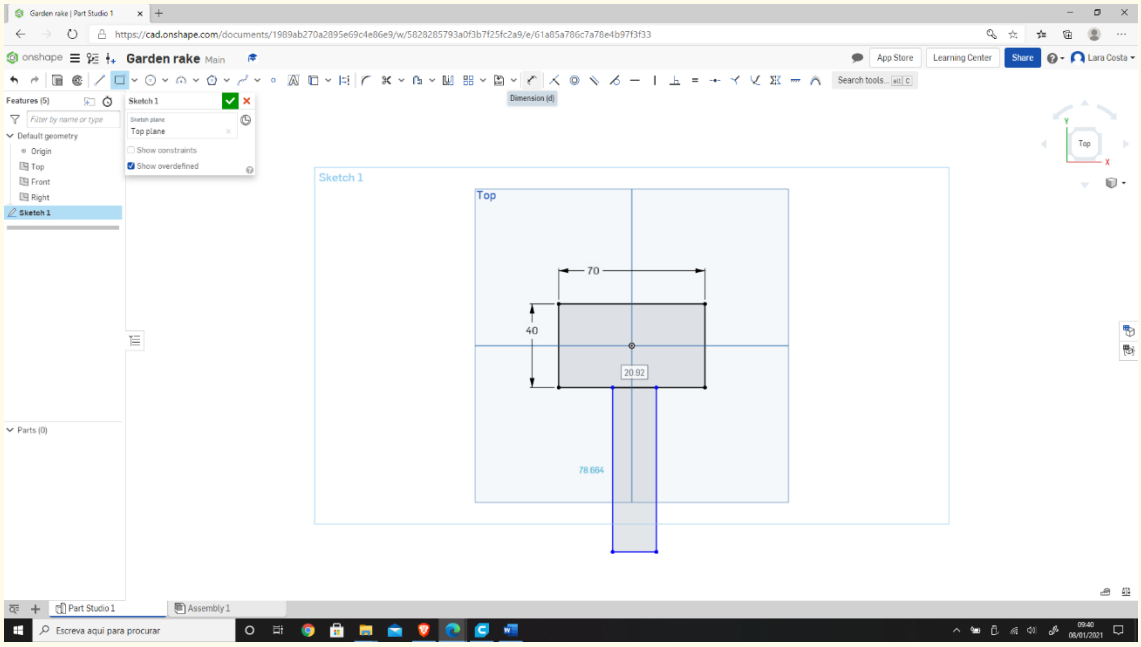
Step 12

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



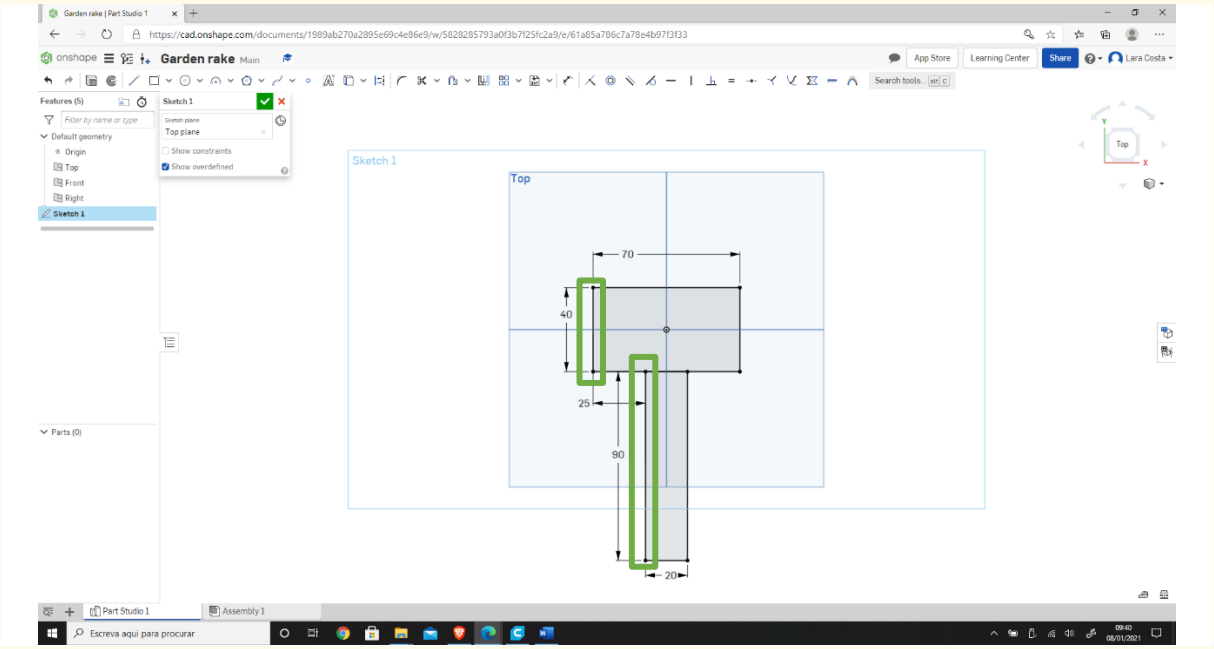
Step 13

Select Dimension to give the measures.



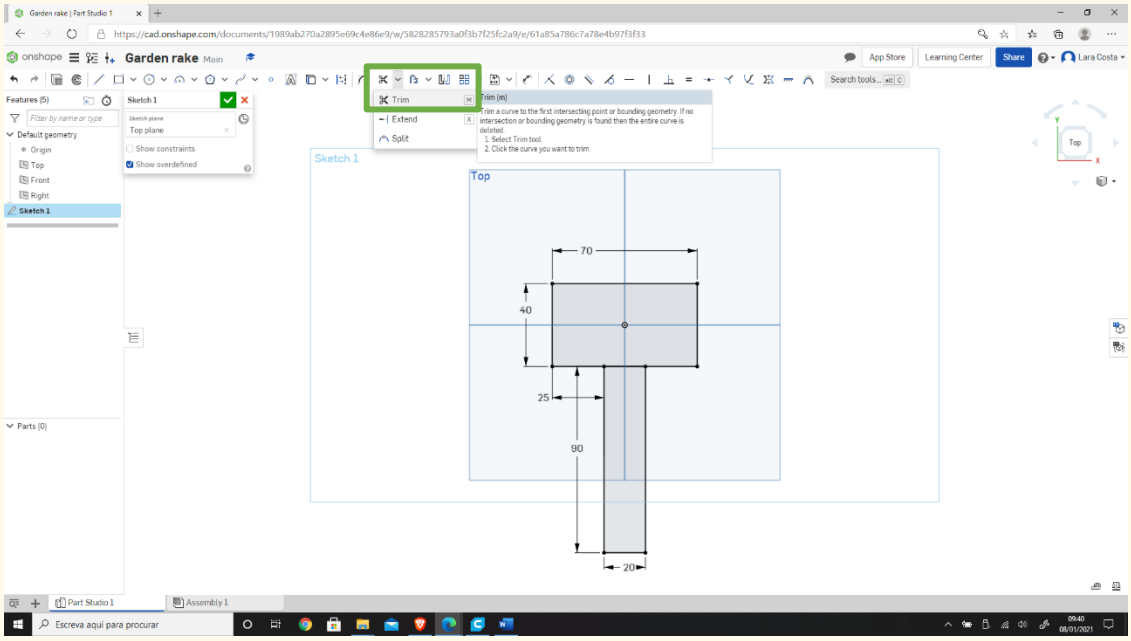
Step 14

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).



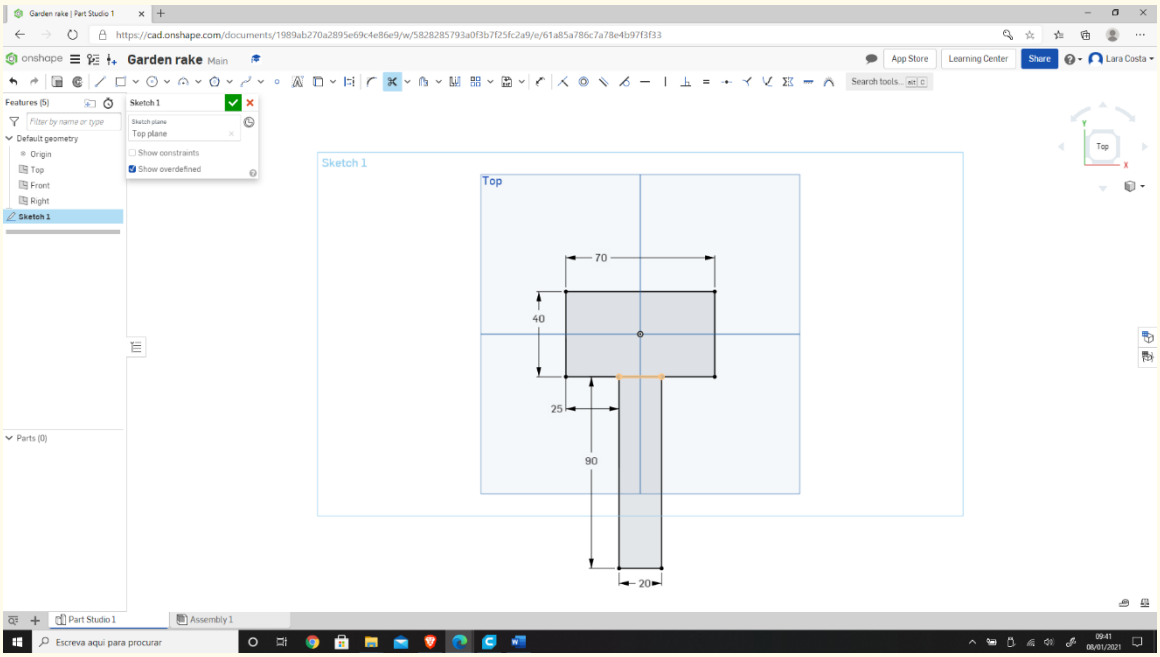
Step 15

Select Trim.



Step 16

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



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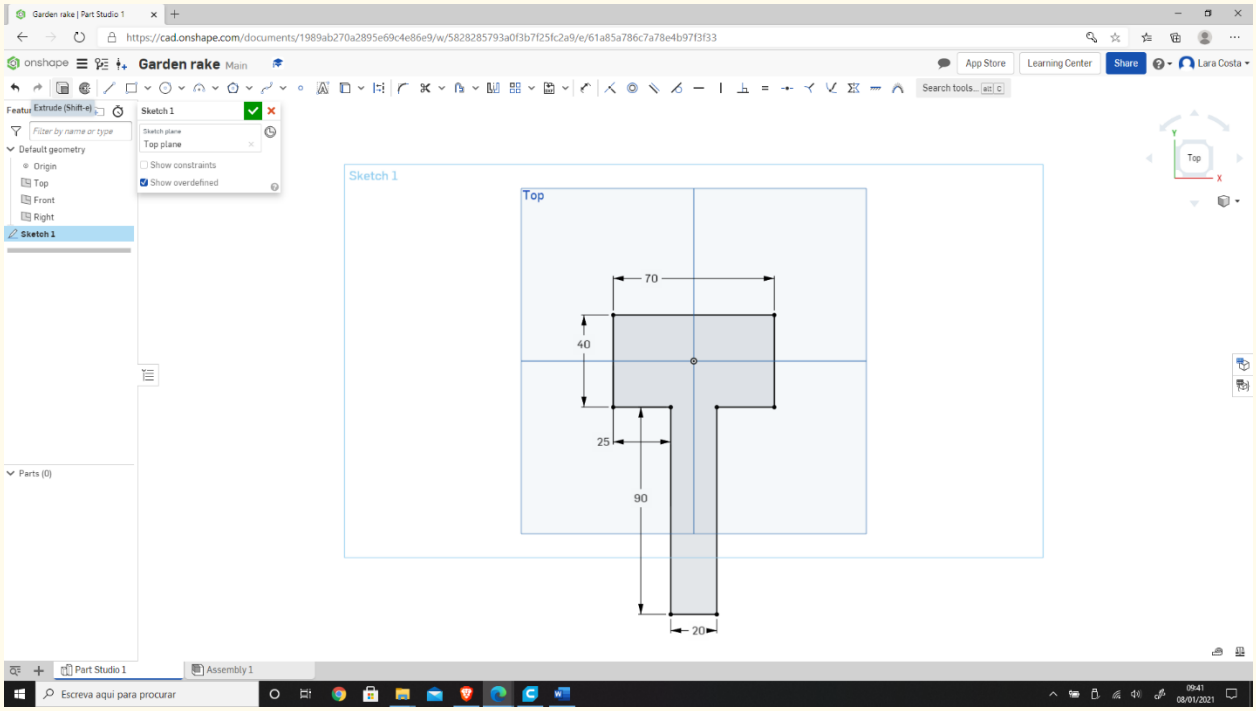
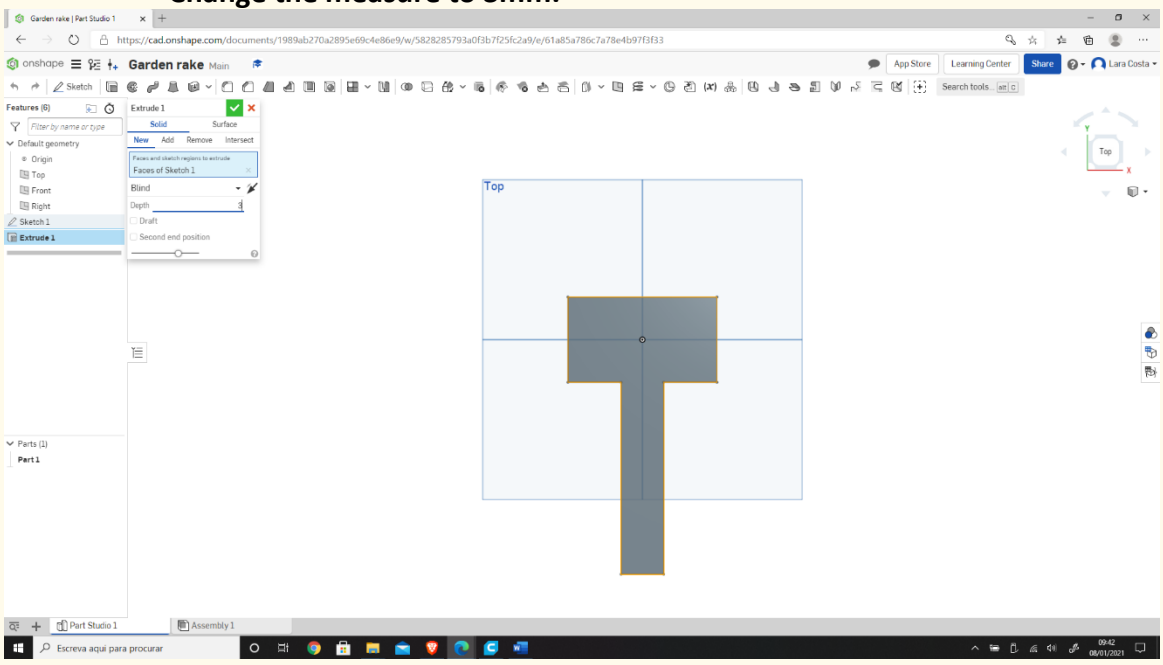
Step 17

Select extrude.



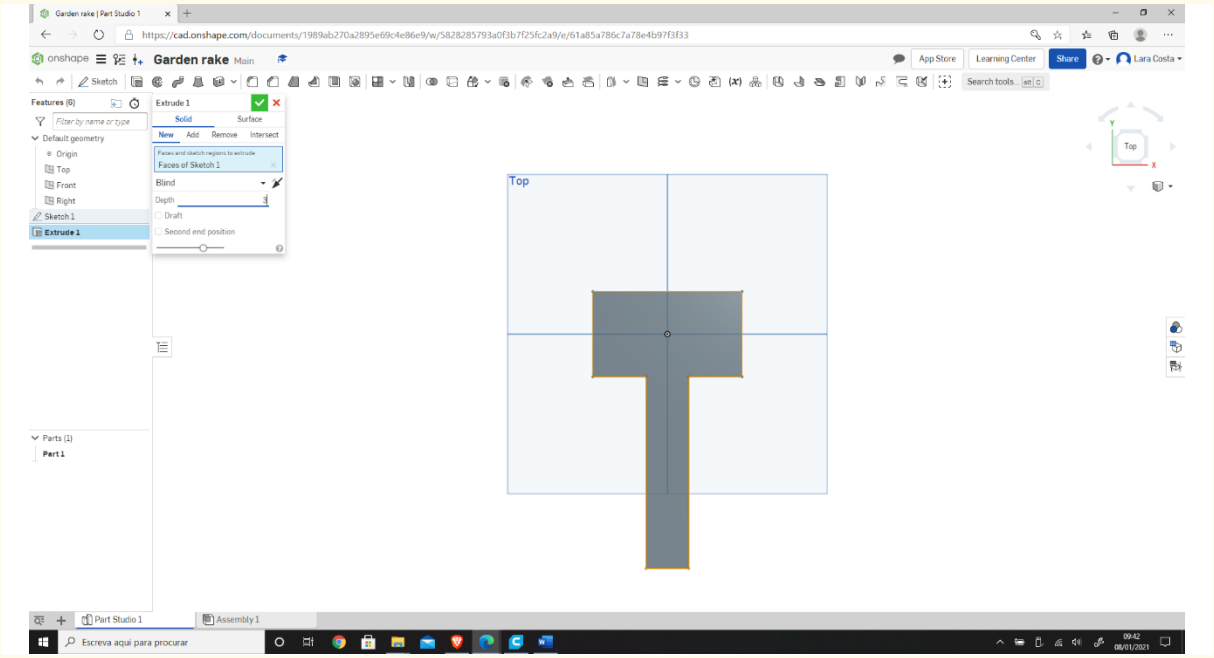
Step 18

Change the measure to 3mm.



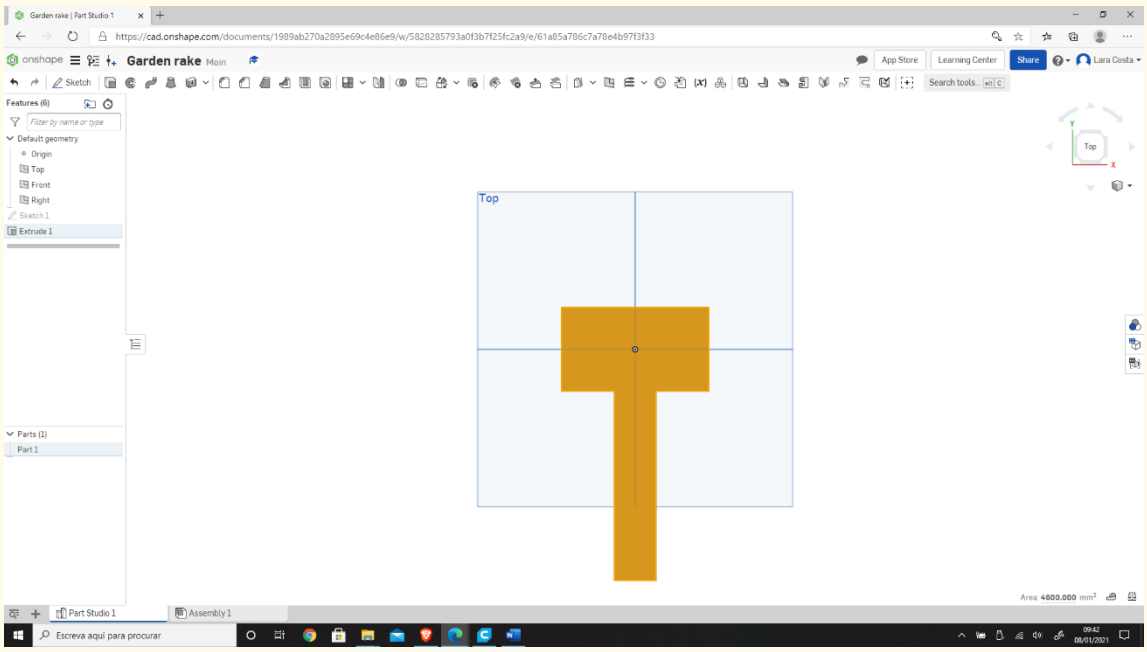
Step 19

Click on the arrow ✅, and this is the result.



Step 20

Select the front.



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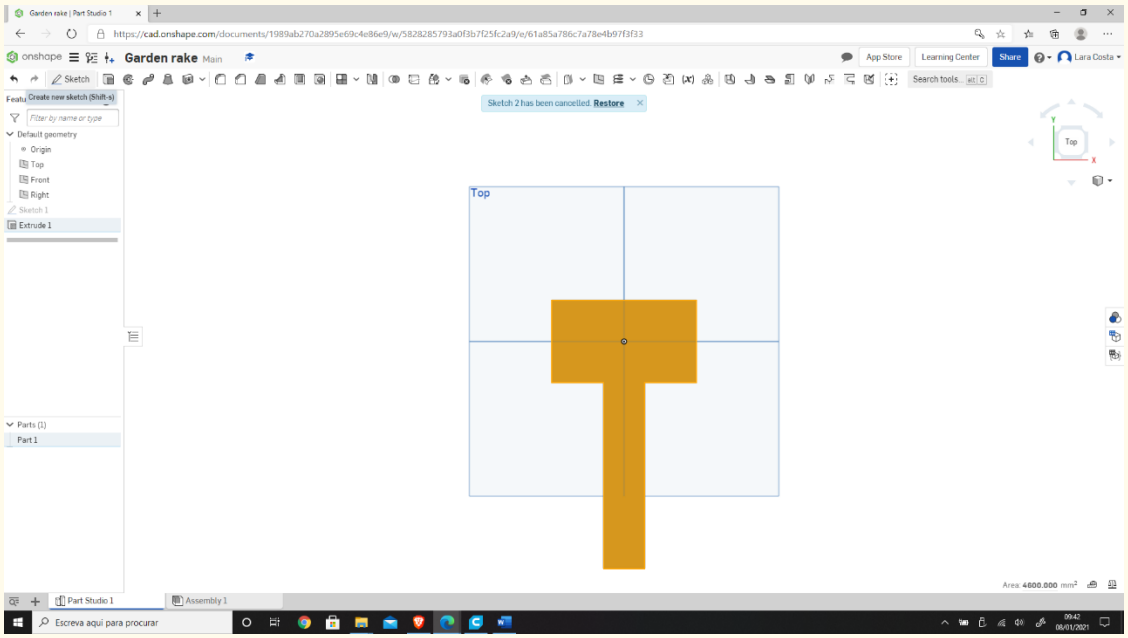
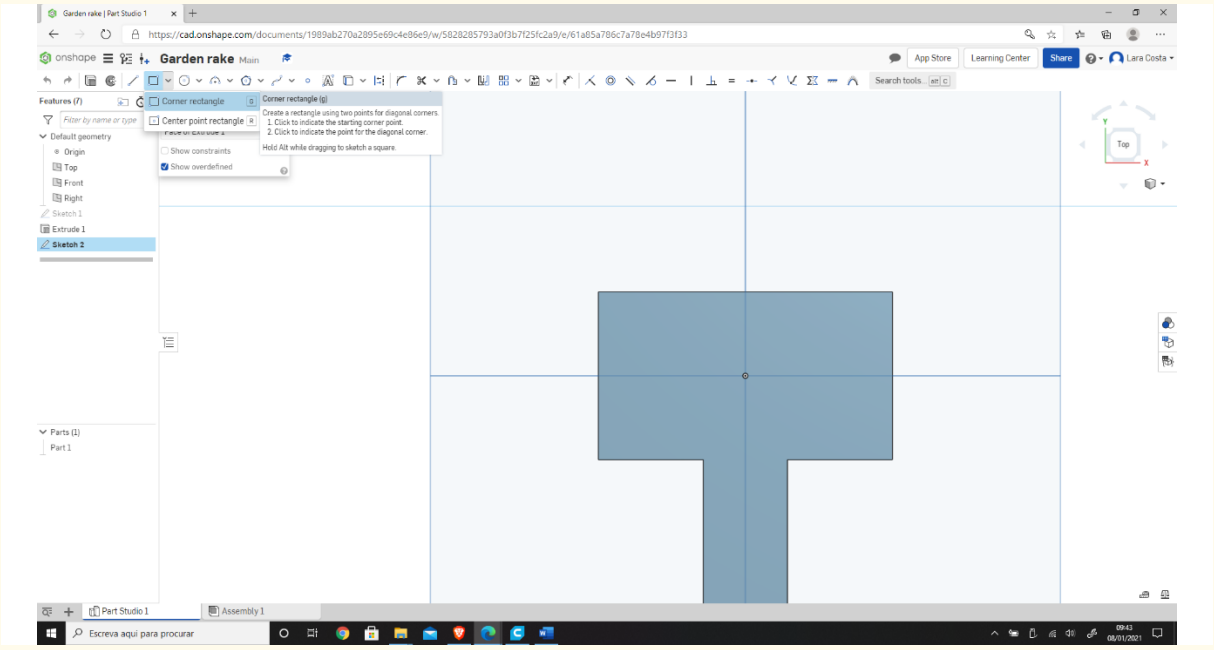
Step 21

Click Sketch.



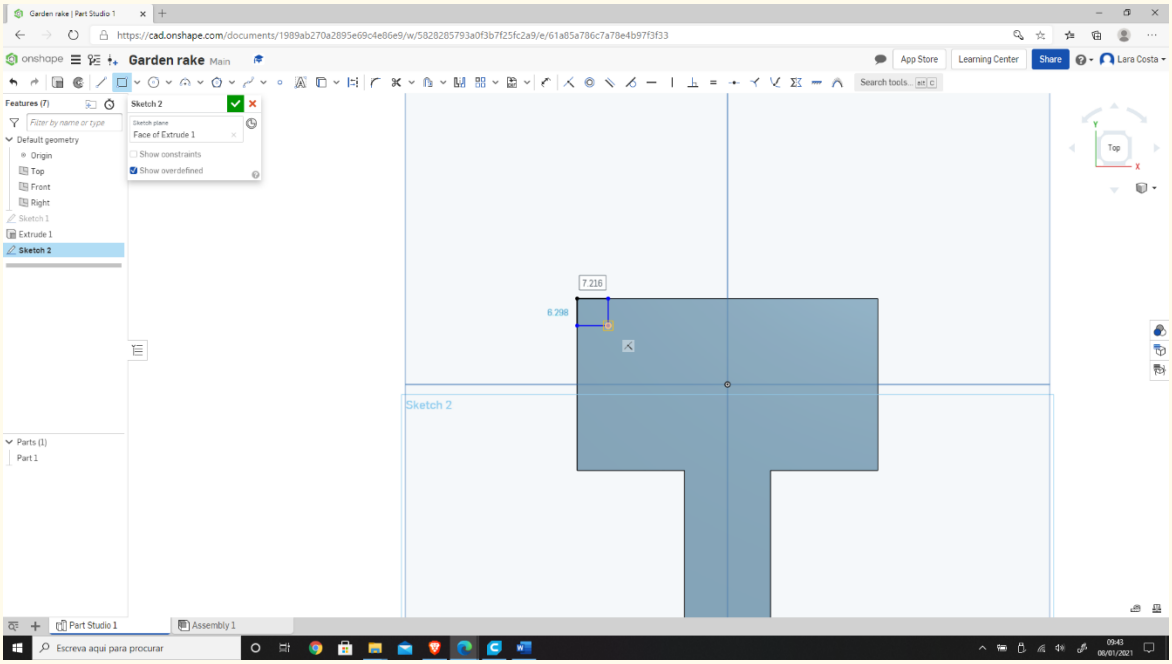
Step 22

Select Corner rectangle.



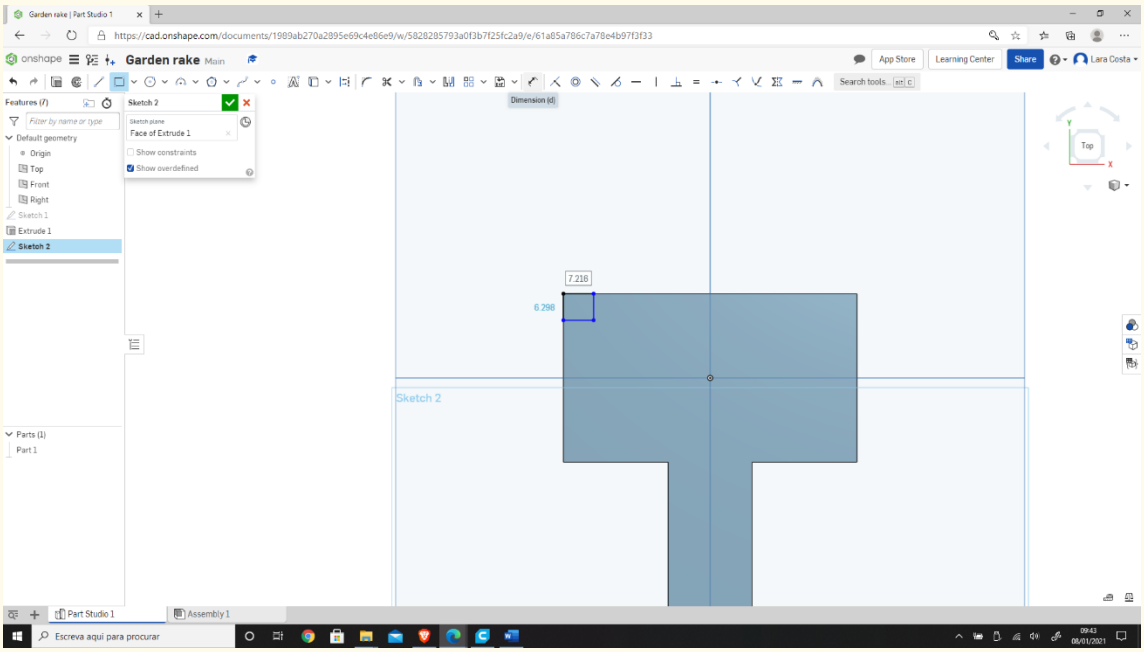
Step 23

Draw a similar rectangle as shown.



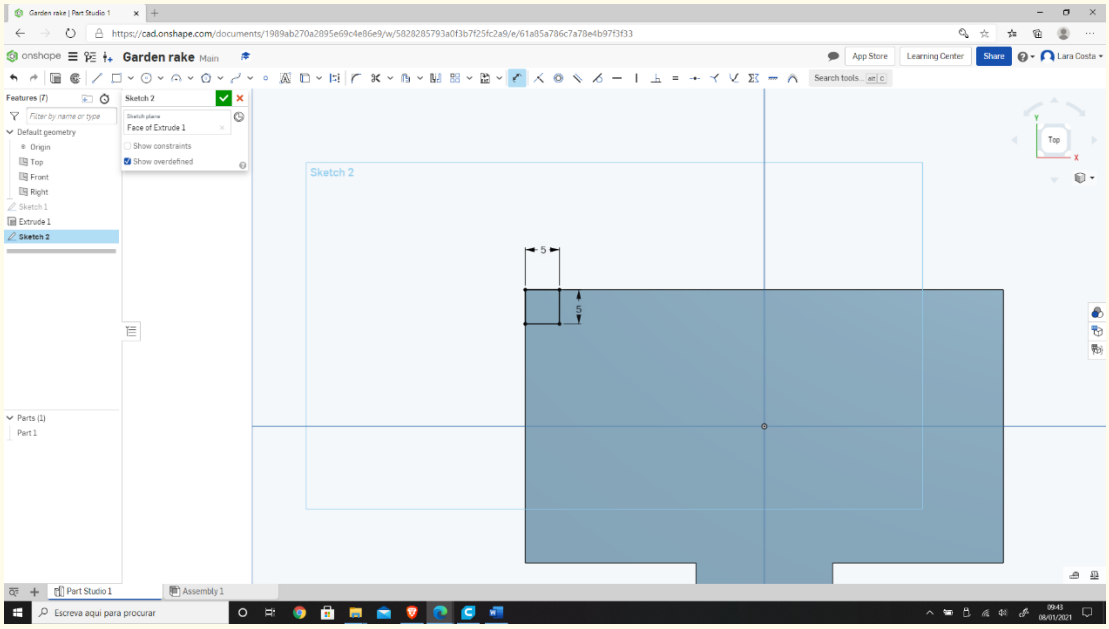
Step 24

Select Dimension to give the measures.



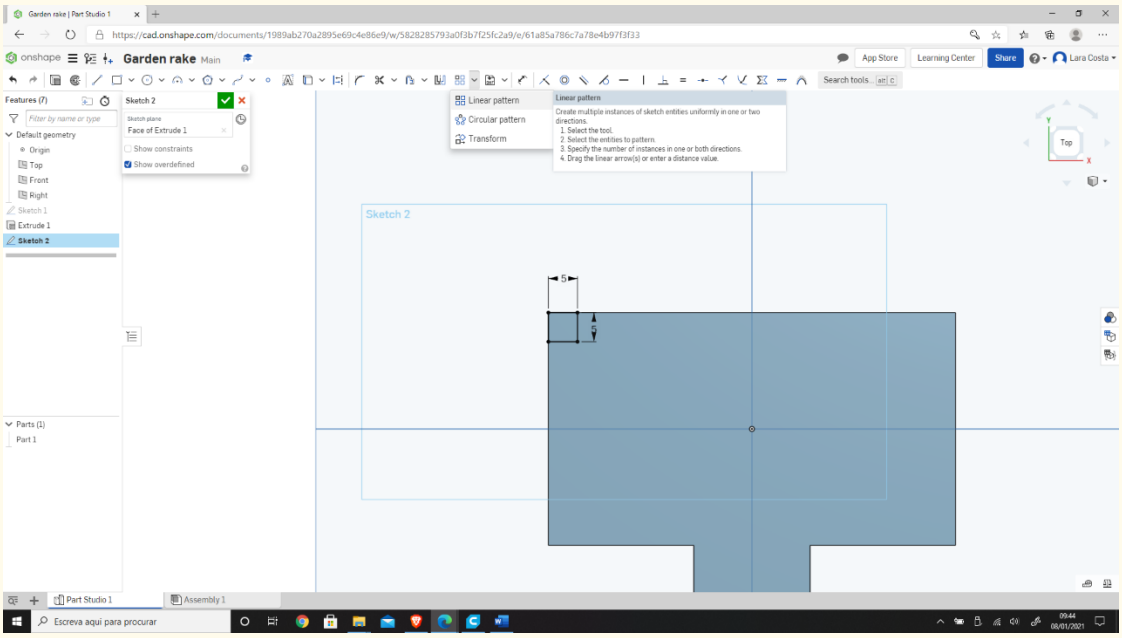
Step 25

Enter the measures as shown in the image below.



Step 26

Then select a Linear pattern.



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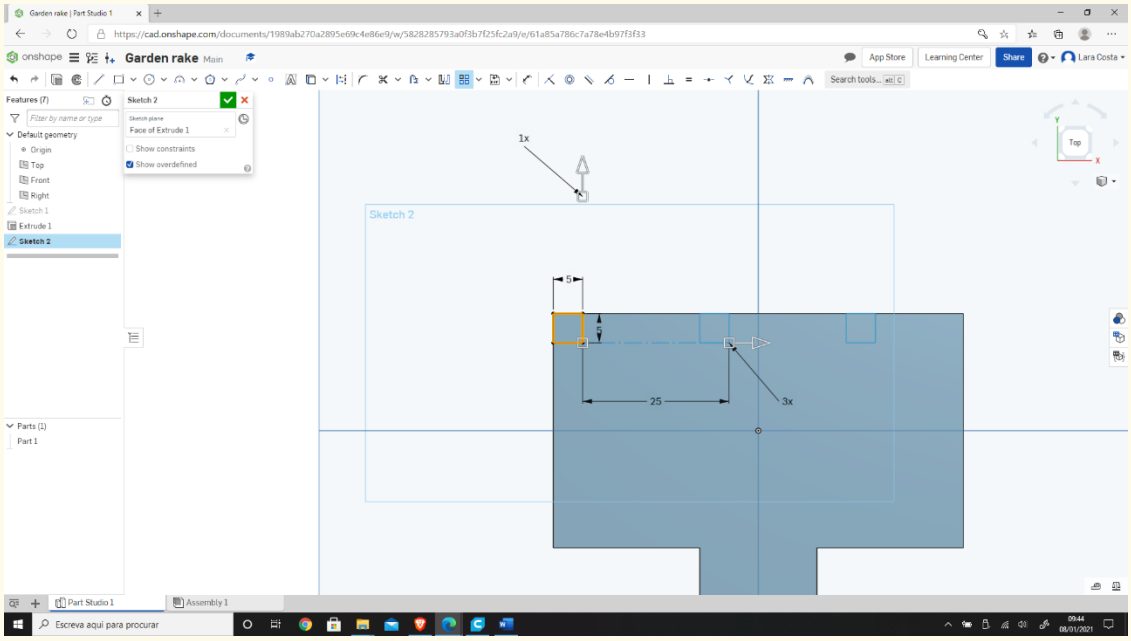
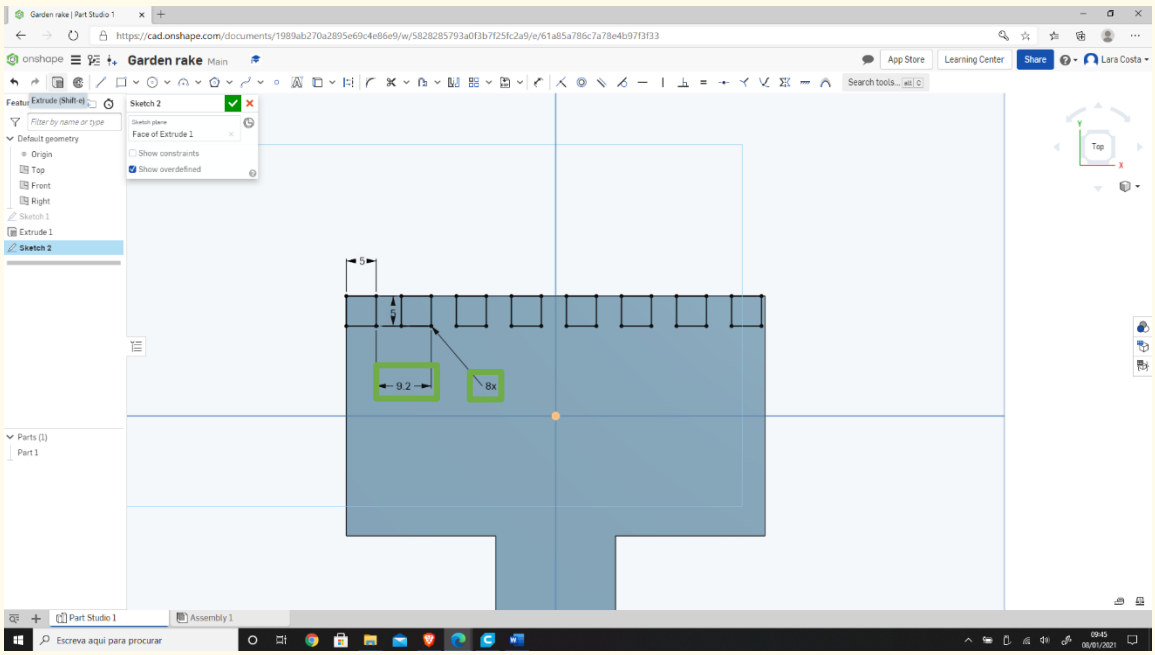
Step 27

Select all the lines (starting on the right).



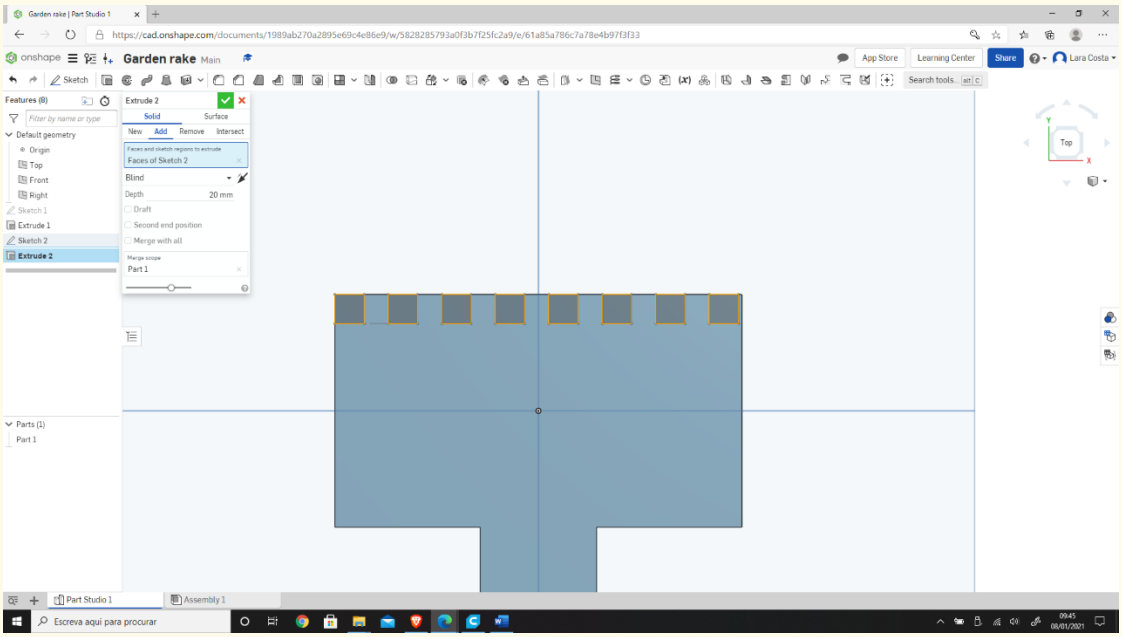
Step 28

Enter the parameters as shown in the image below (inside the green rectangles).



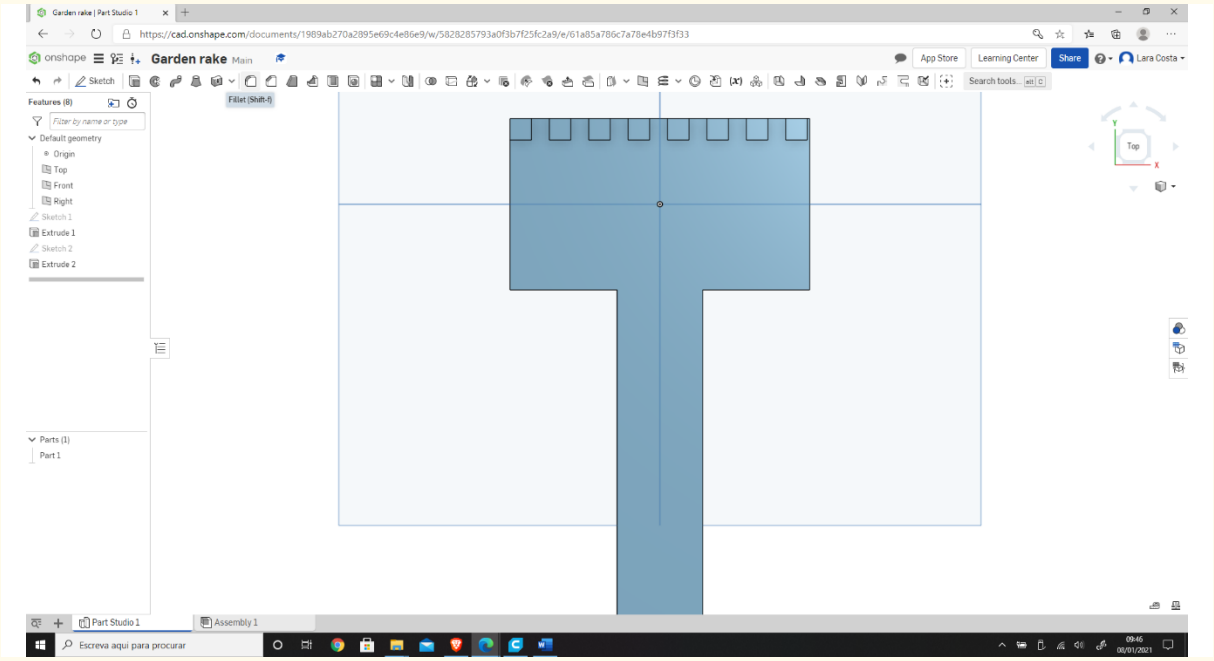
Step 29

Select Extrude, and the measure is 20mm.



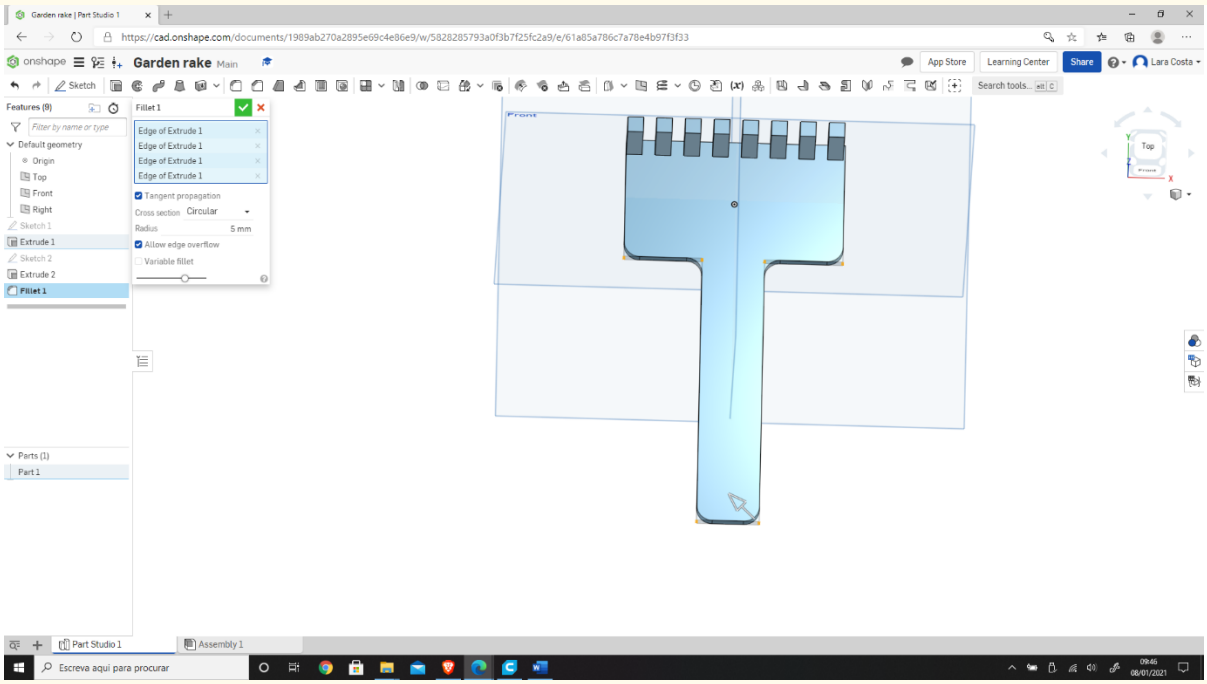
Step 30

Select Fillet.



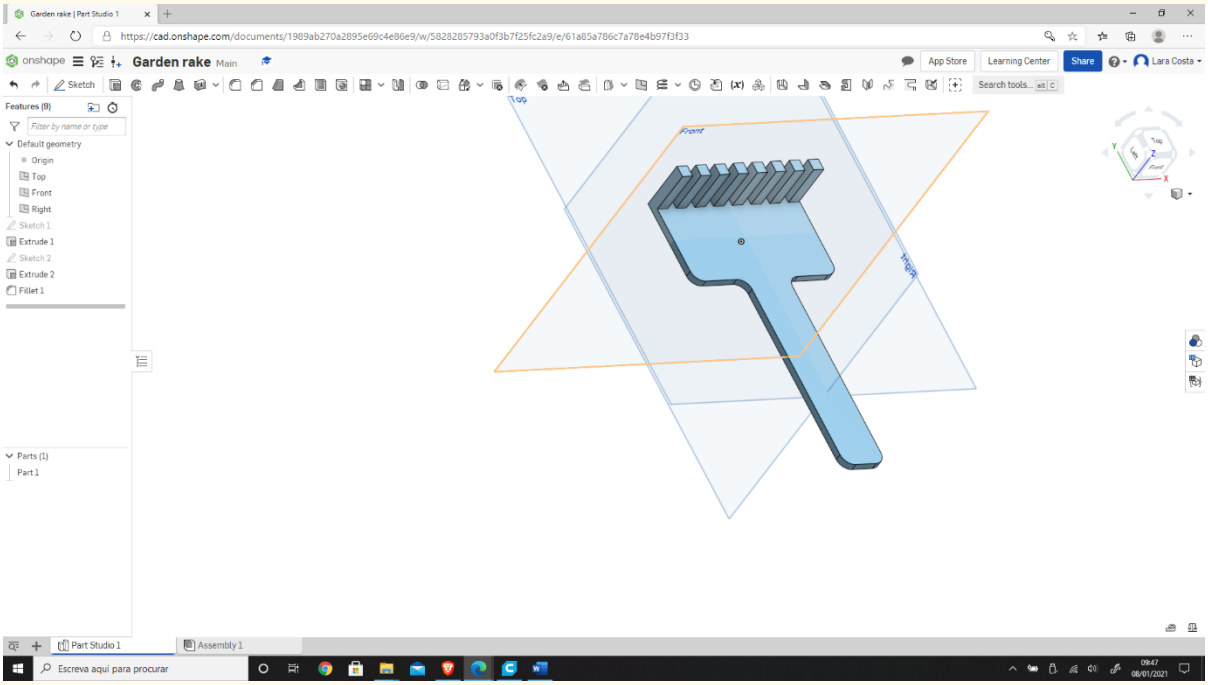
Step 31

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



Step 32

This should be your result.



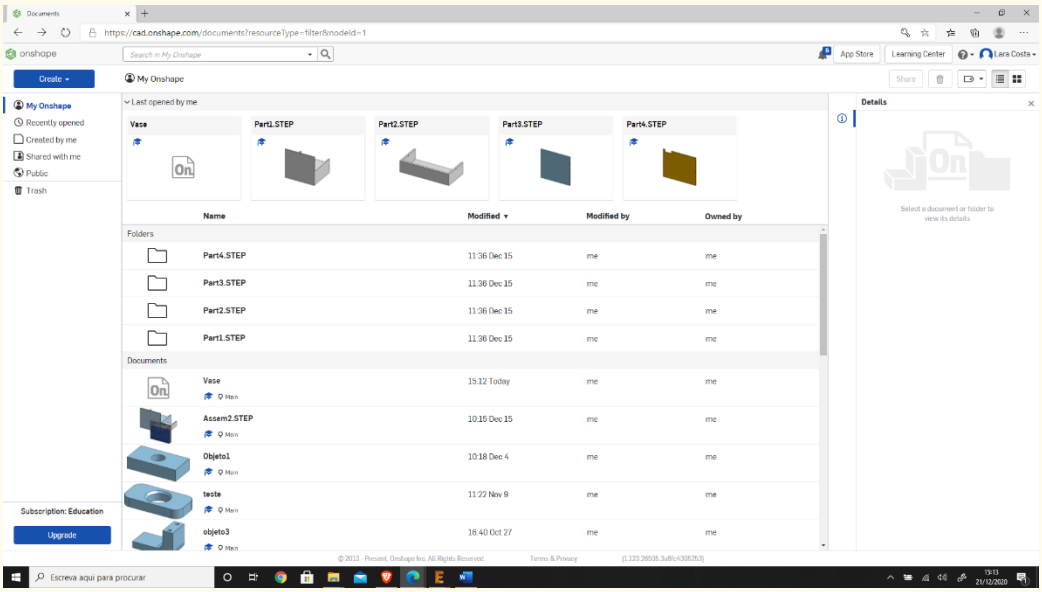
PROJECT: 3D DRAWING OF A VASE

- **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 competencies:**
    - 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:

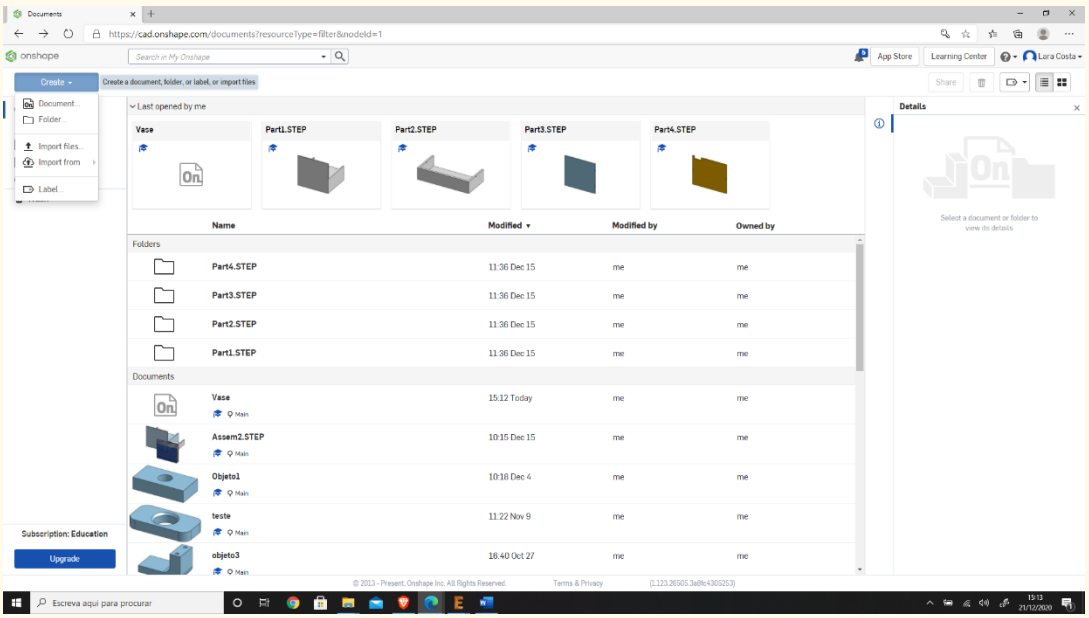
Step 1

Open Onshape.



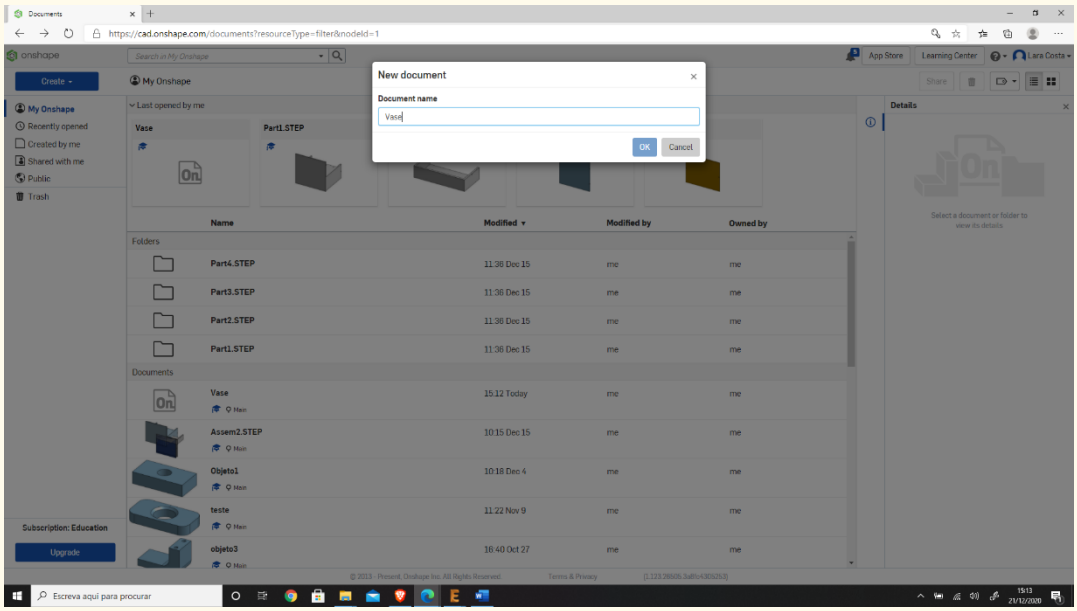
Step 2

Create a document.



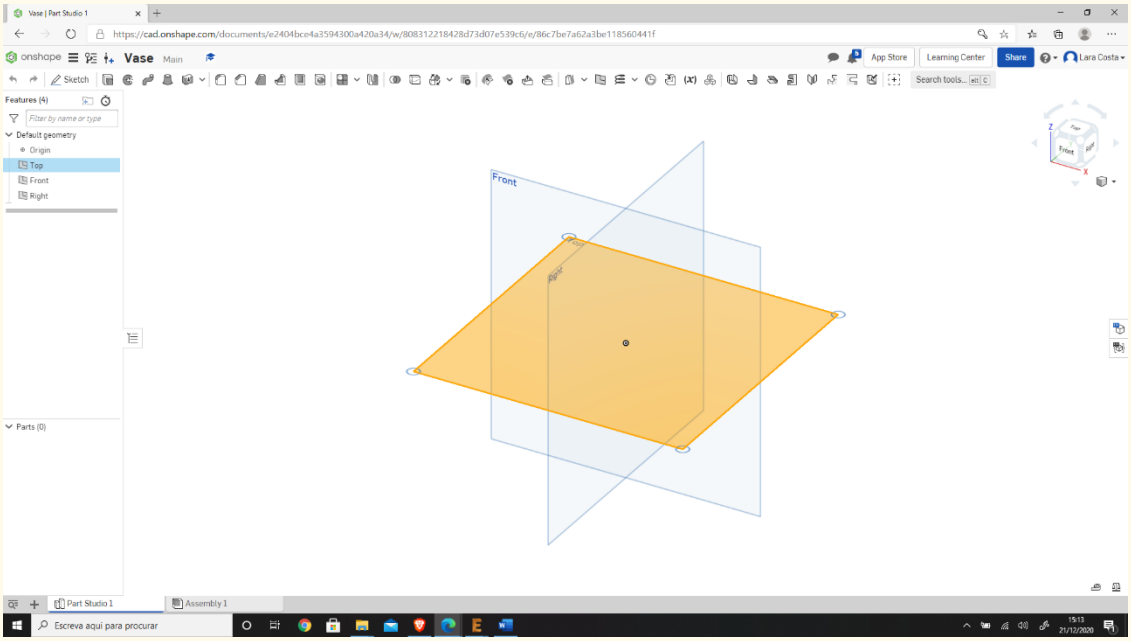
Step 3

Give a name to your document such as “Vase.”



Step 4

Select the plane (top) to start drawing.



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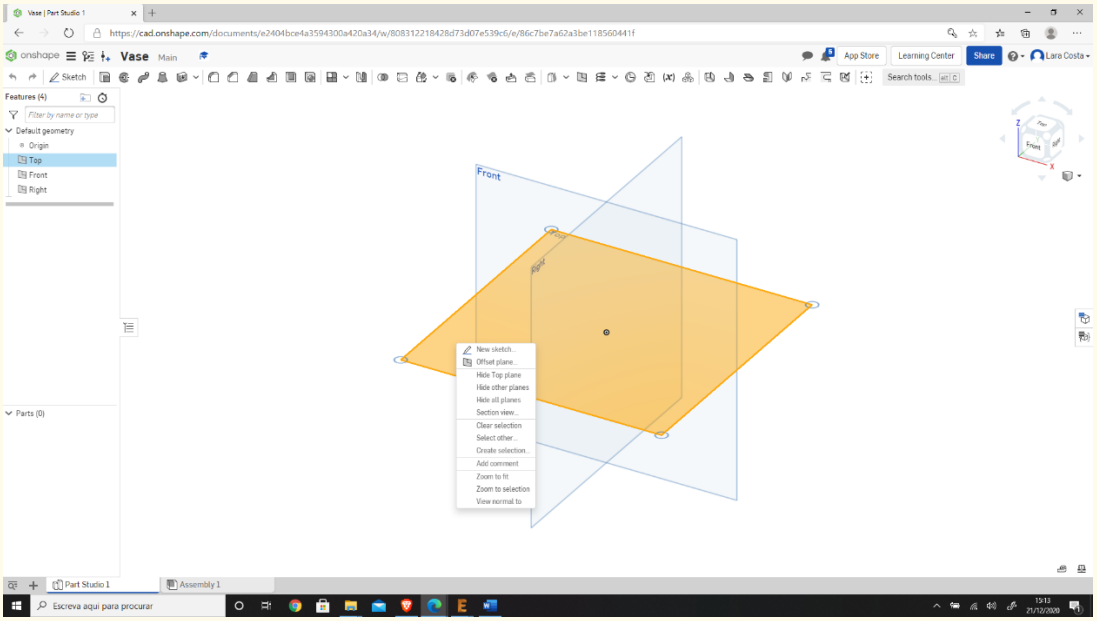
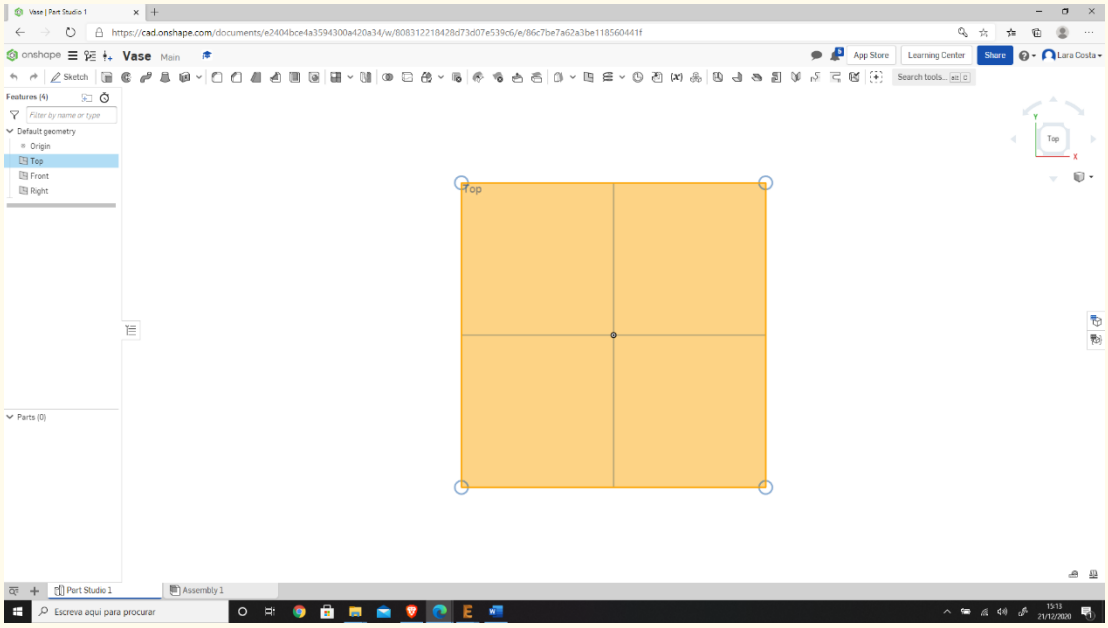
Step 5

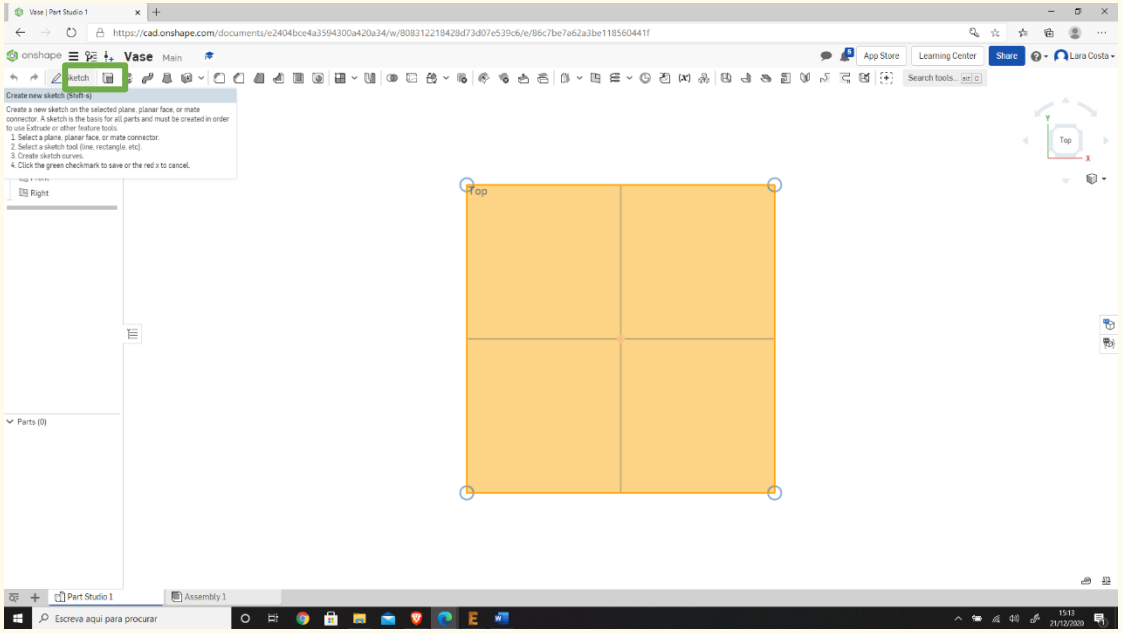
Right-click and select “normal view to”.
The plan should look like the 2nd image.





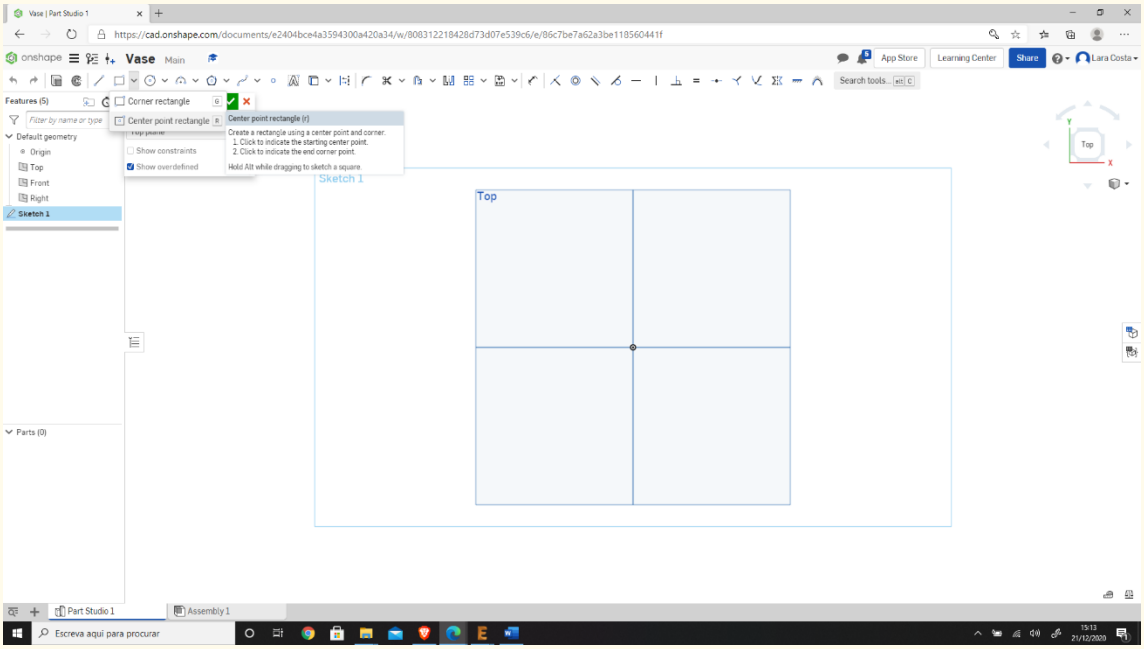
Step 6

Click Sketch.



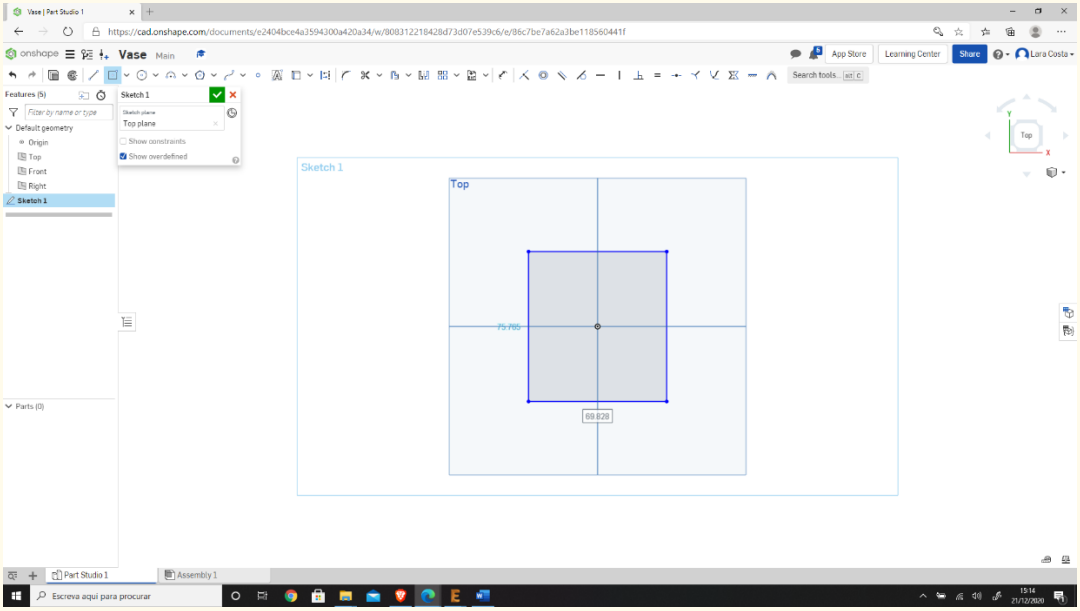
Step 7

Select center point rectangle to draw.



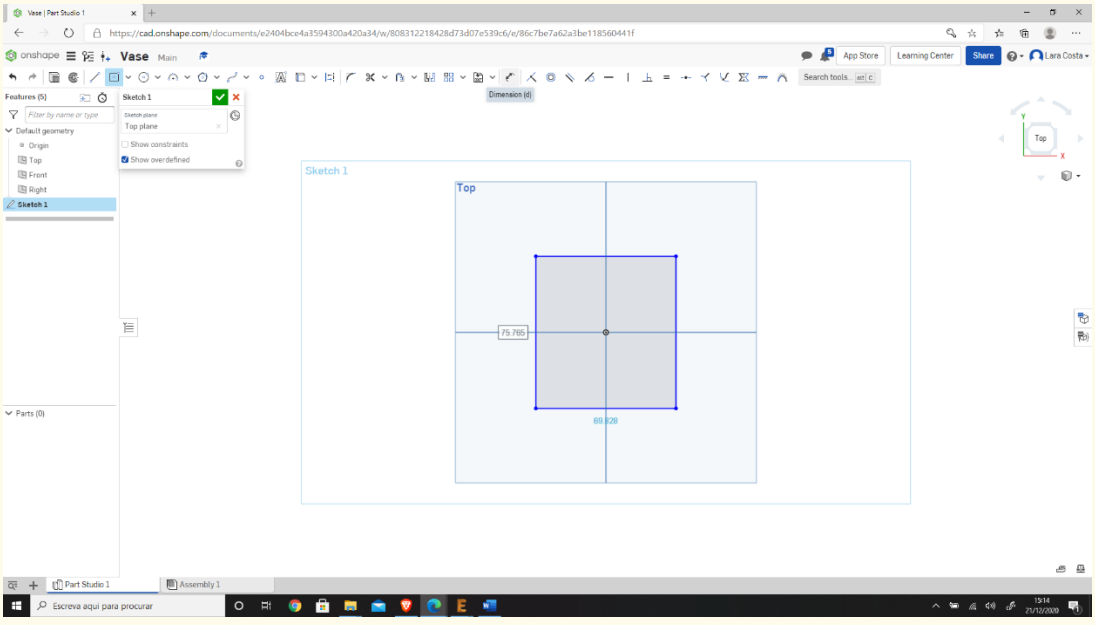
Step 8

Draw the rectangle.

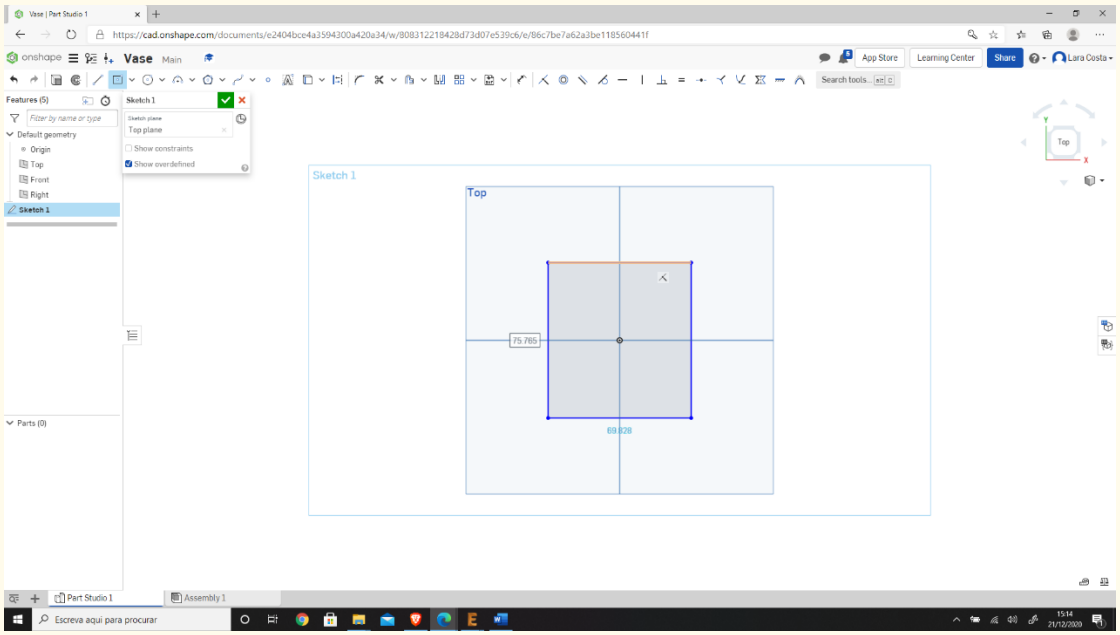


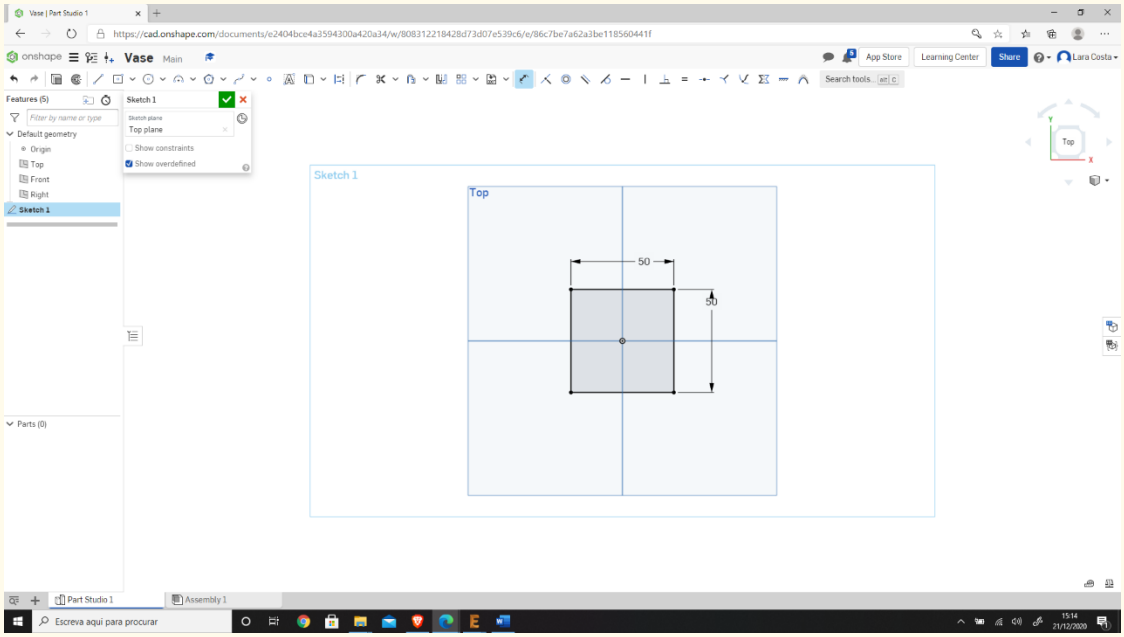
Step 9

Select Dimension to give the measures.



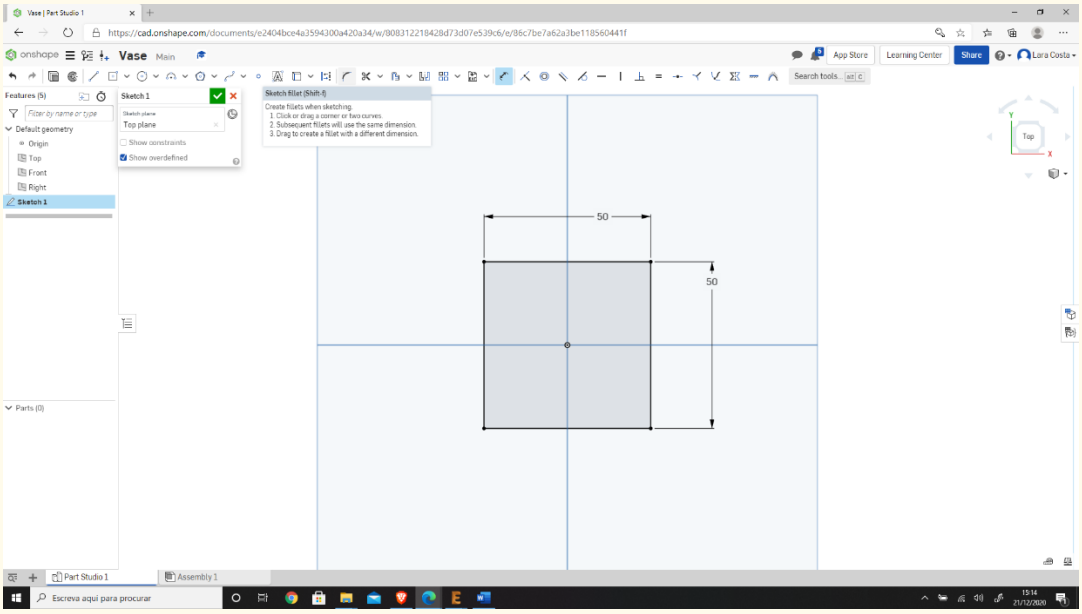
Step 10 **Select the line (first top then right) and give the measure 2nd figure (50mm).**





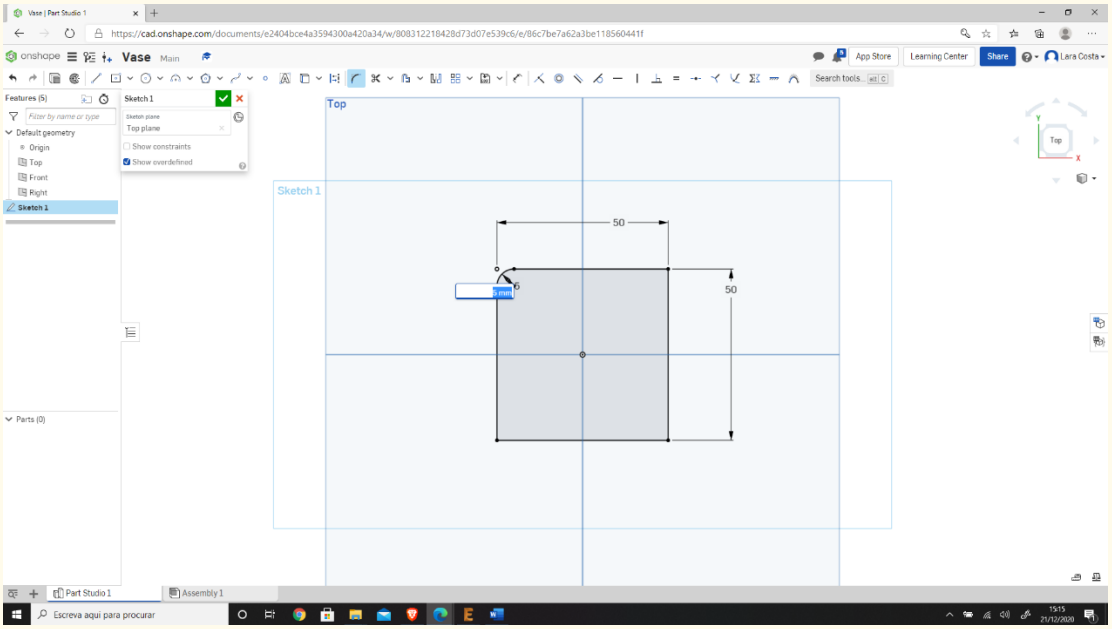
Step 11

Select sketch fillet.



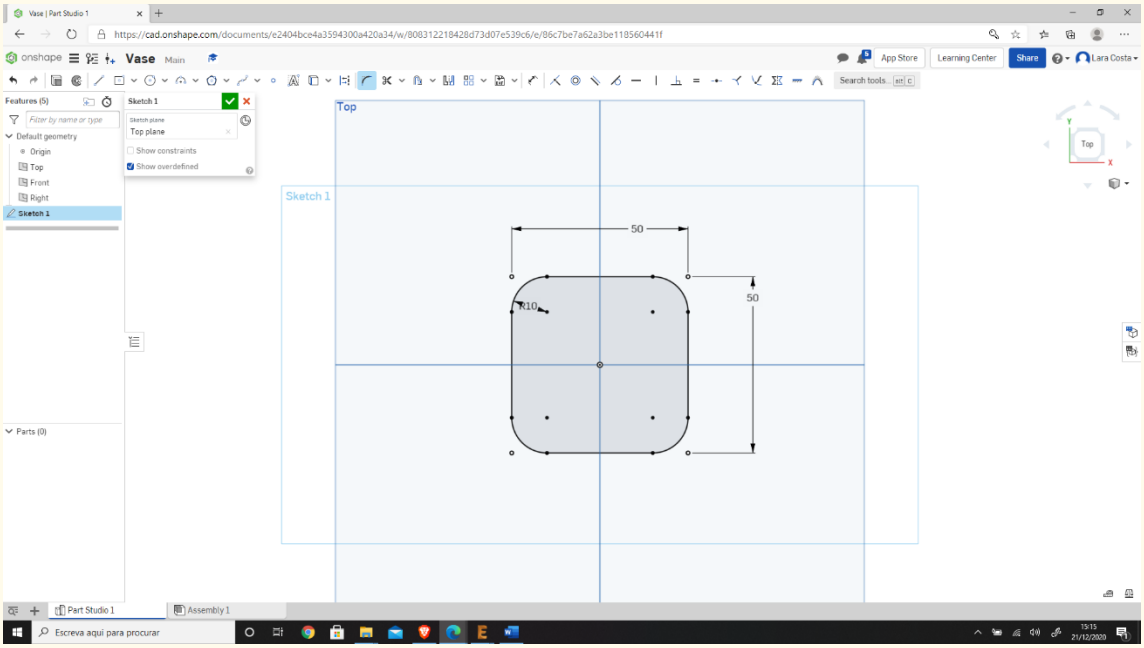
Step 12

Select two lines and click enter.



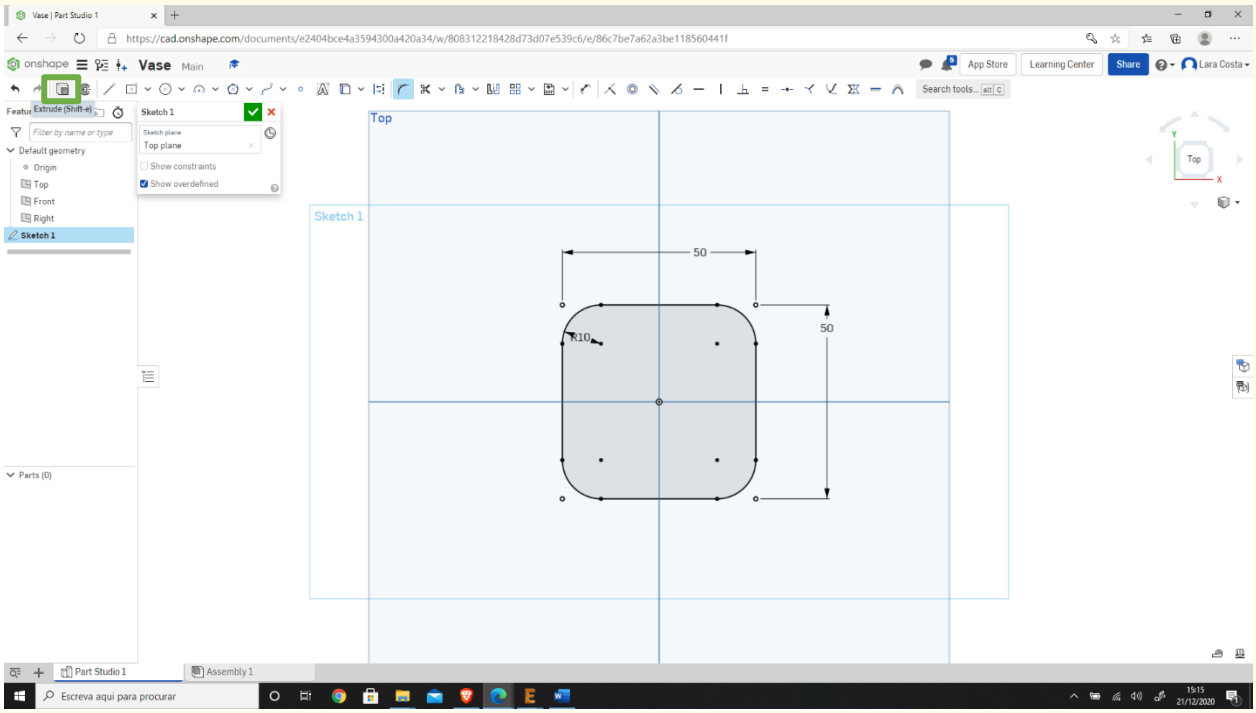
Step 13

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



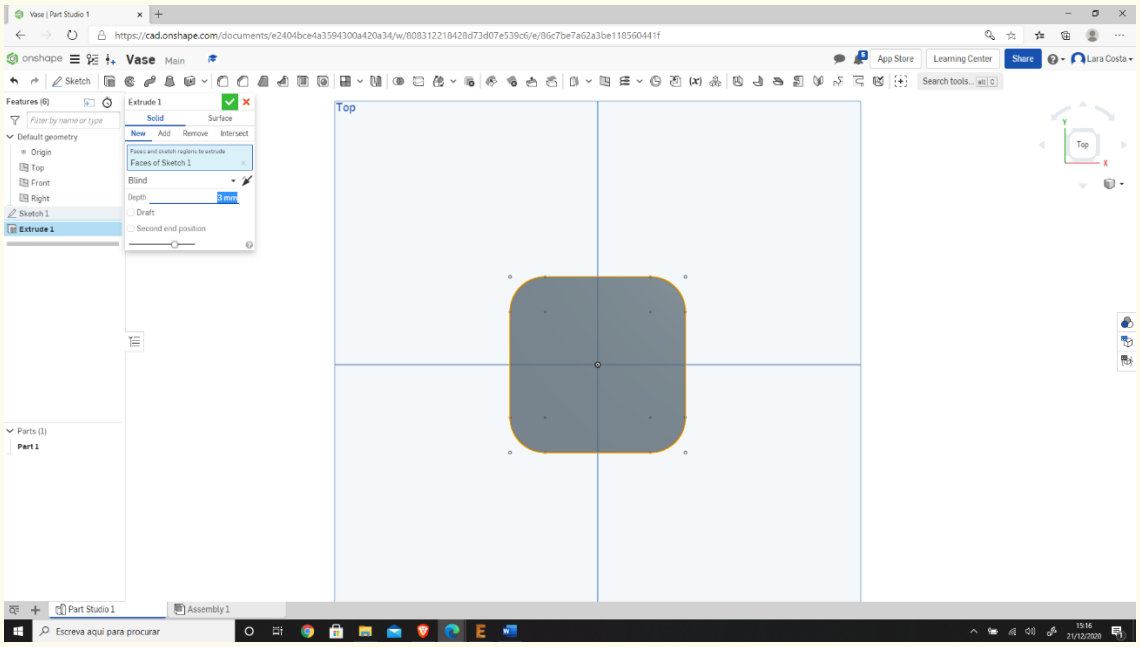
Step 14

Select Extrude.



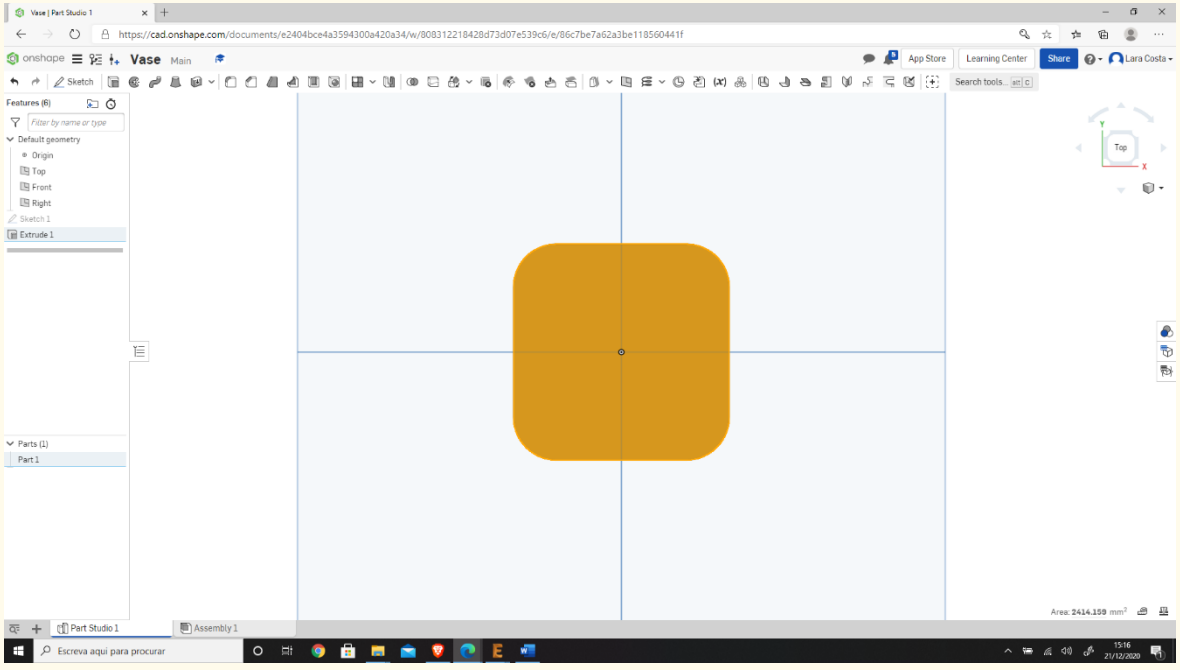
Step 15

Put Depth equal 3mm and click ✓.



Step 16

Select the front face.



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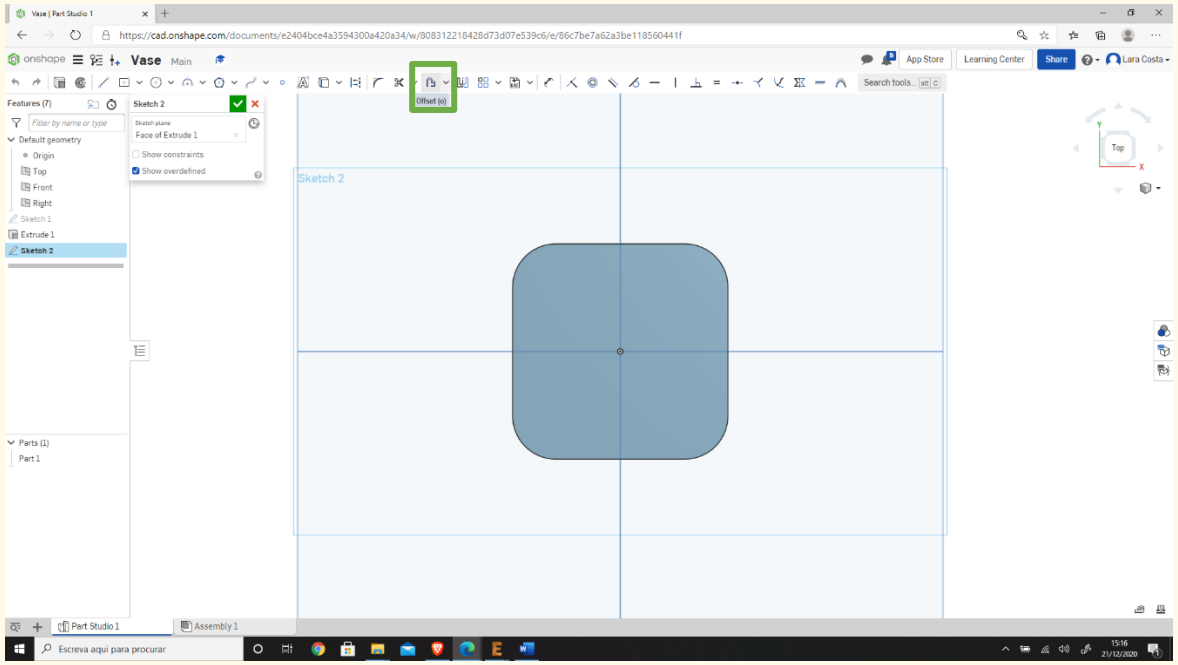
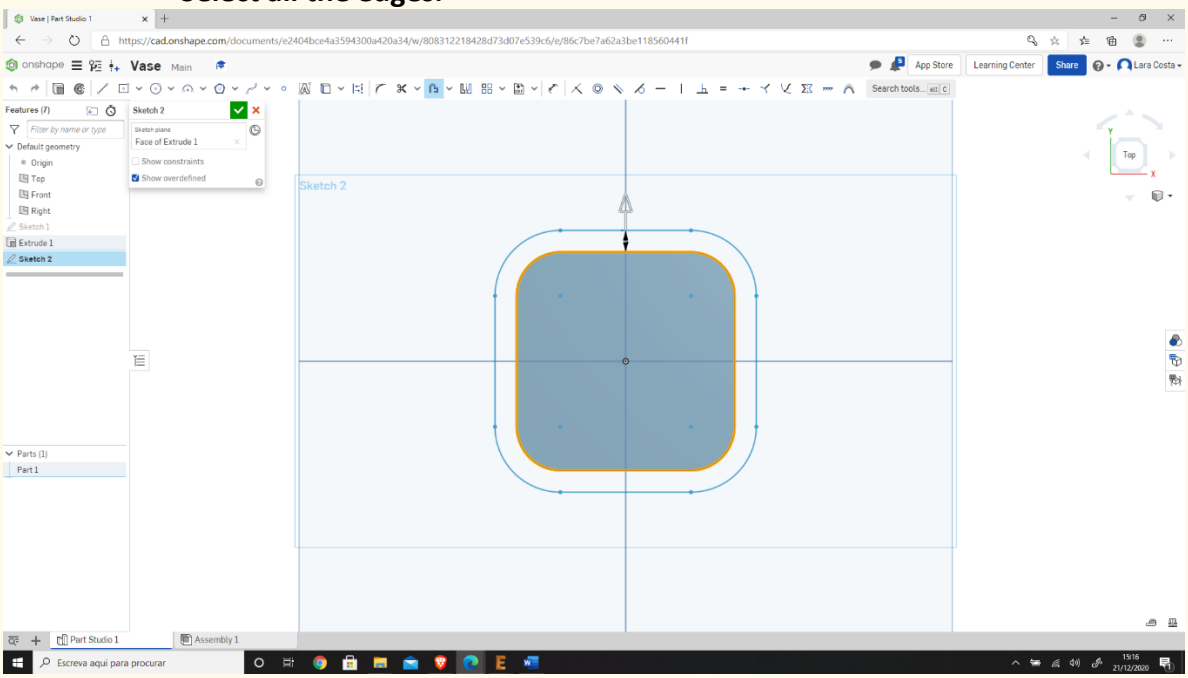
Step 17

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



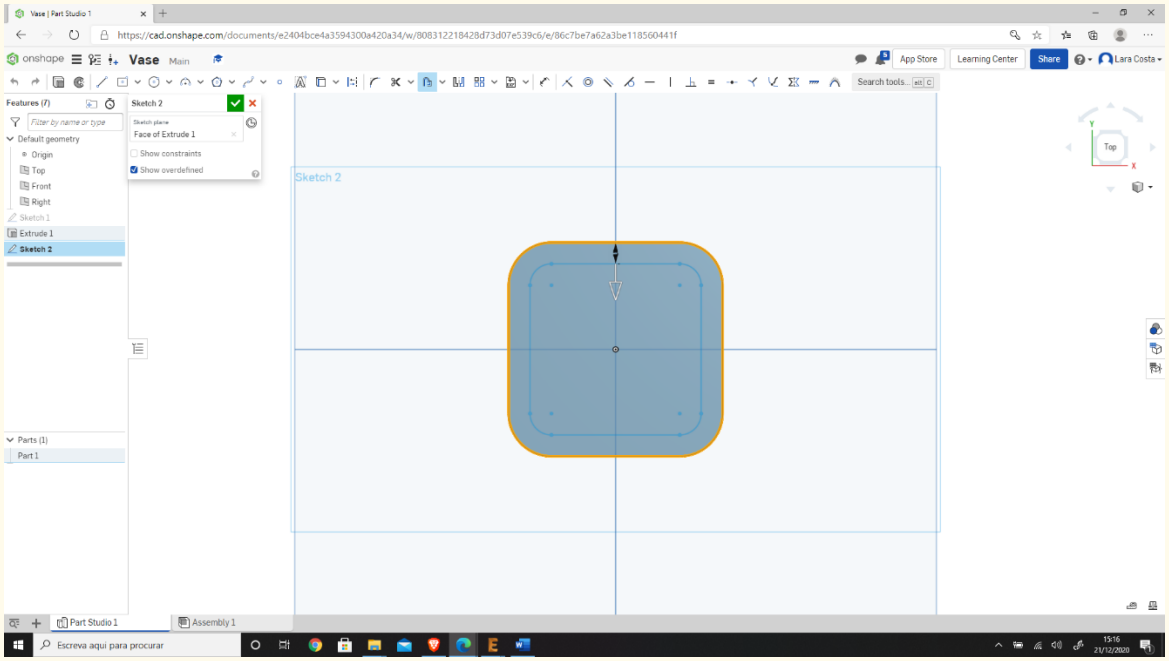
Step 18

Select all the edges.



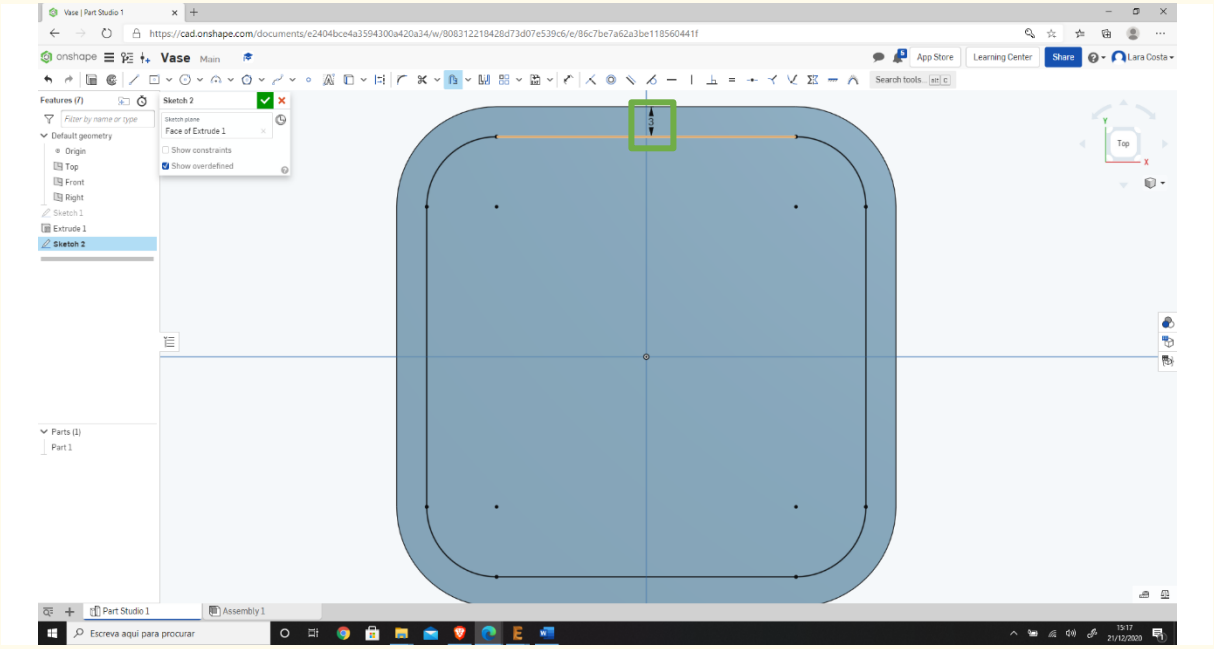
Step 19

Click on the arrow if necessary to have this result.



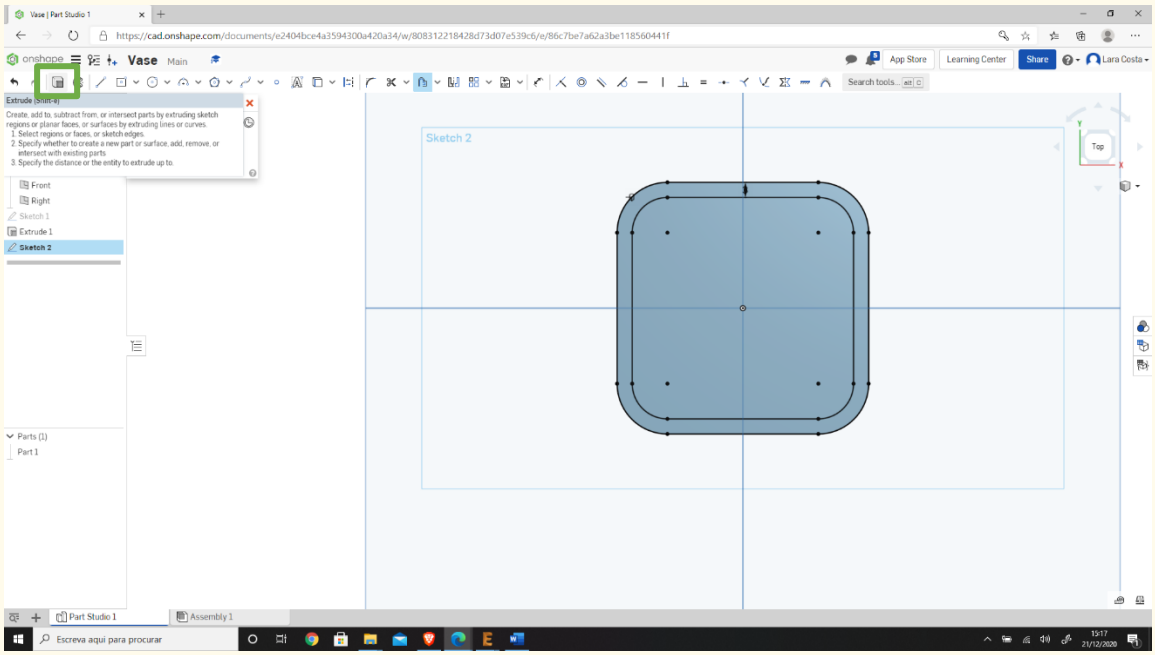
Step 20

Change the measure to 3mm.



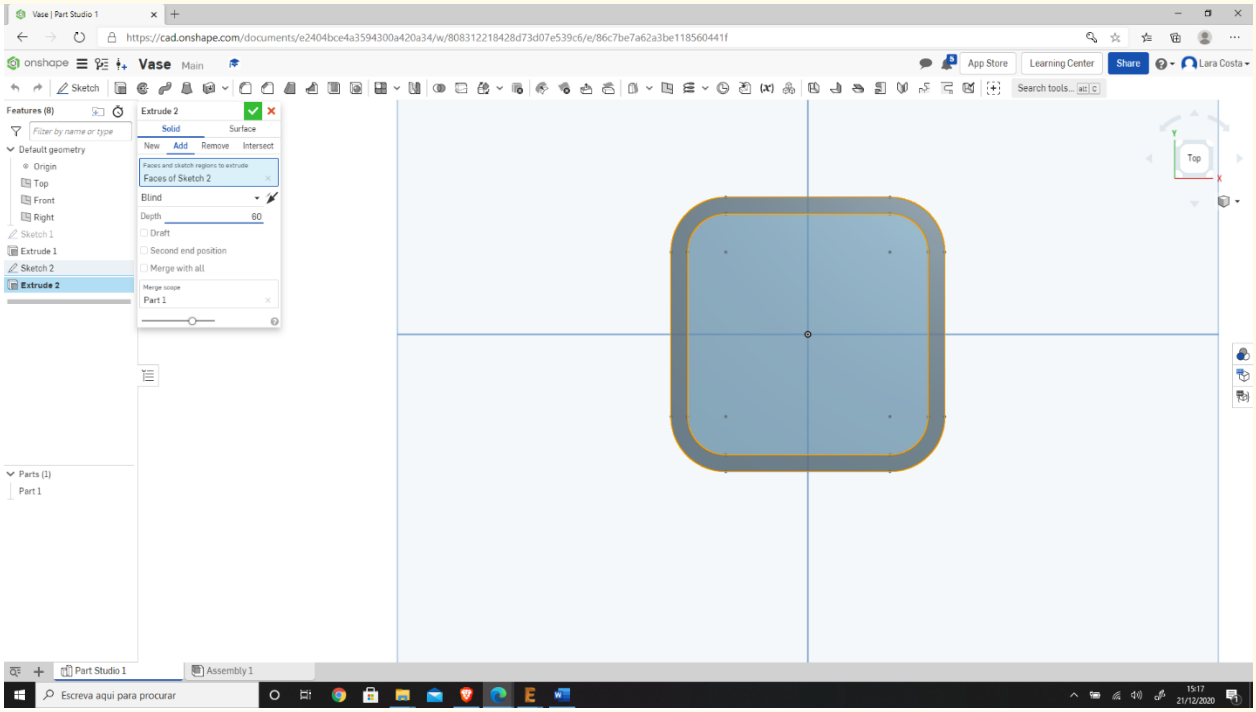
Step 21

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



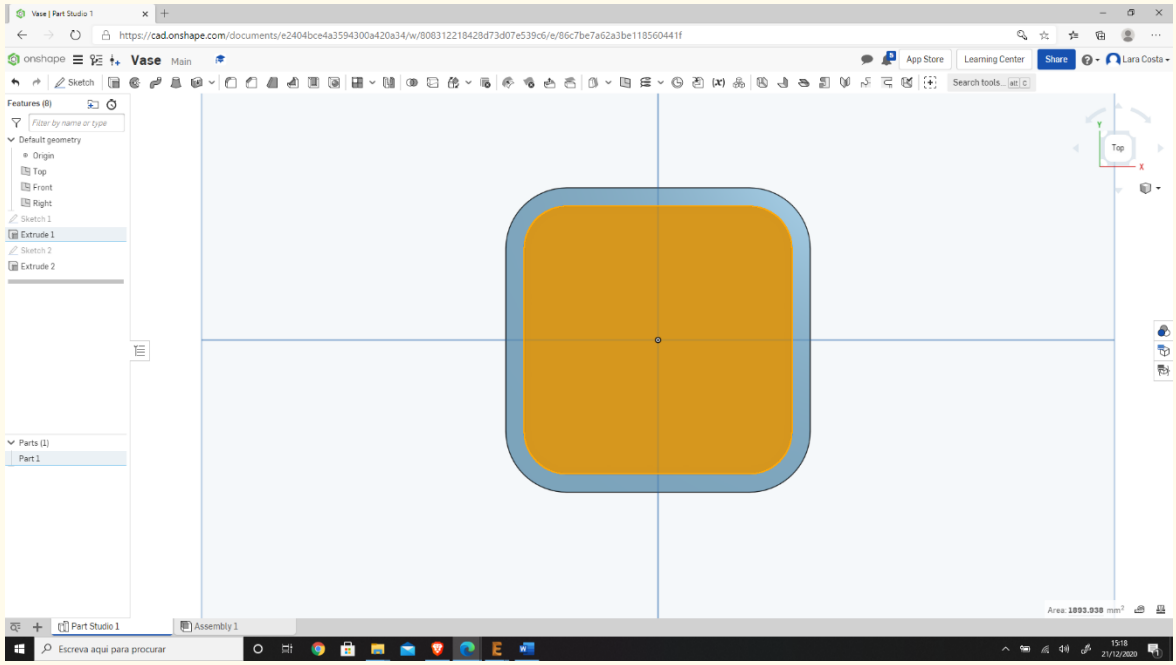
Step 22

Change the Depth to 60mm and click ✅.



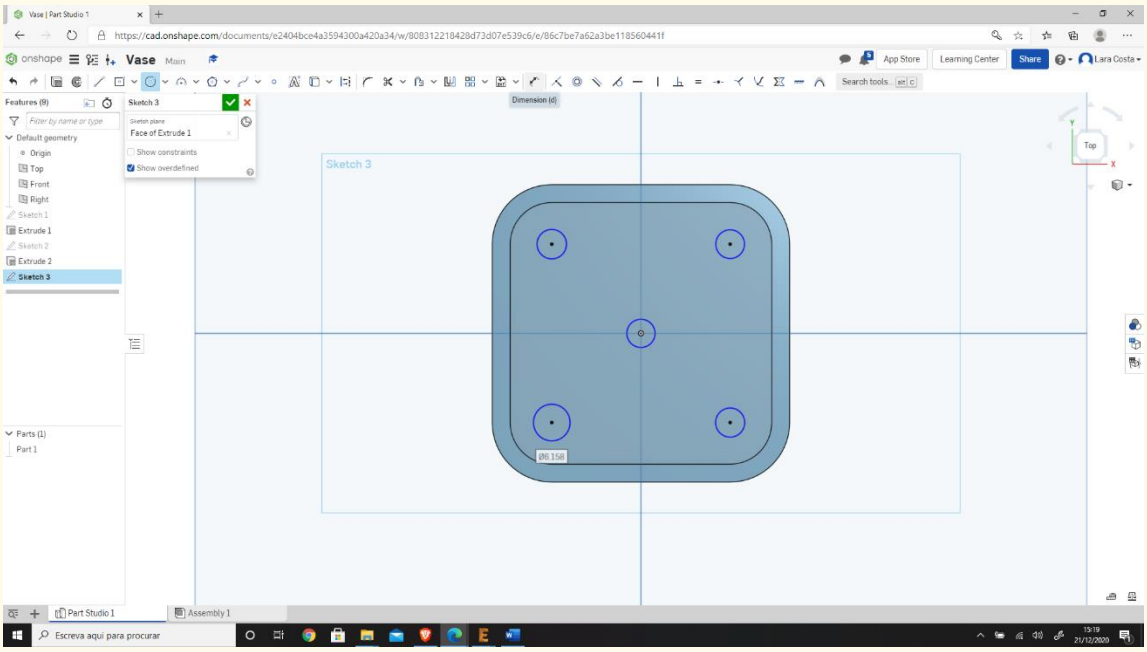
Step 23

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



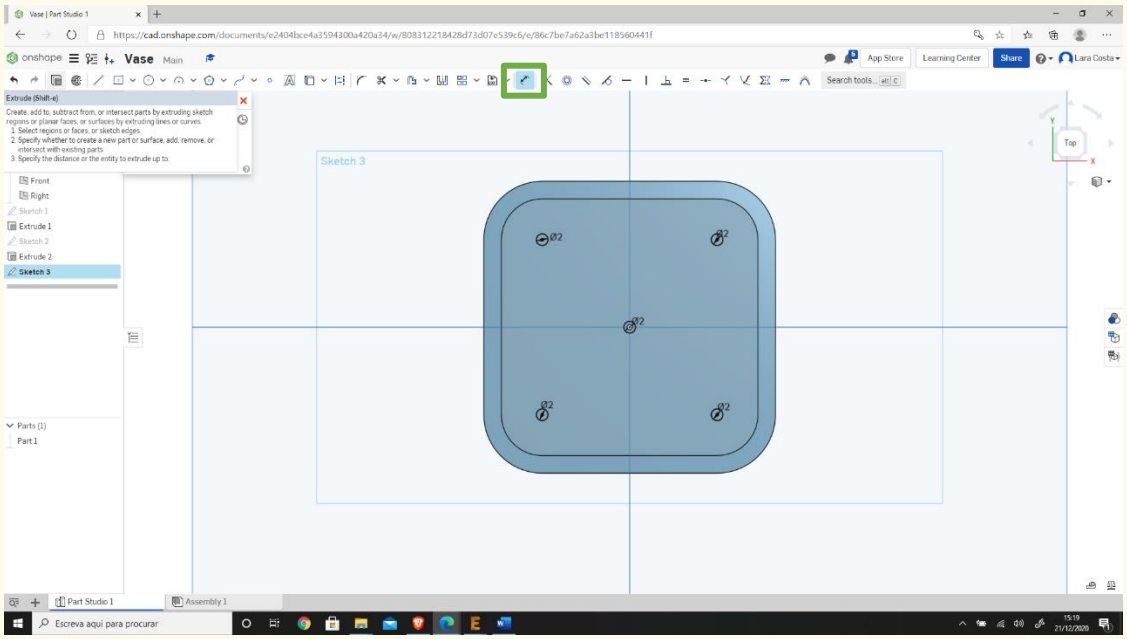
Step 24

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



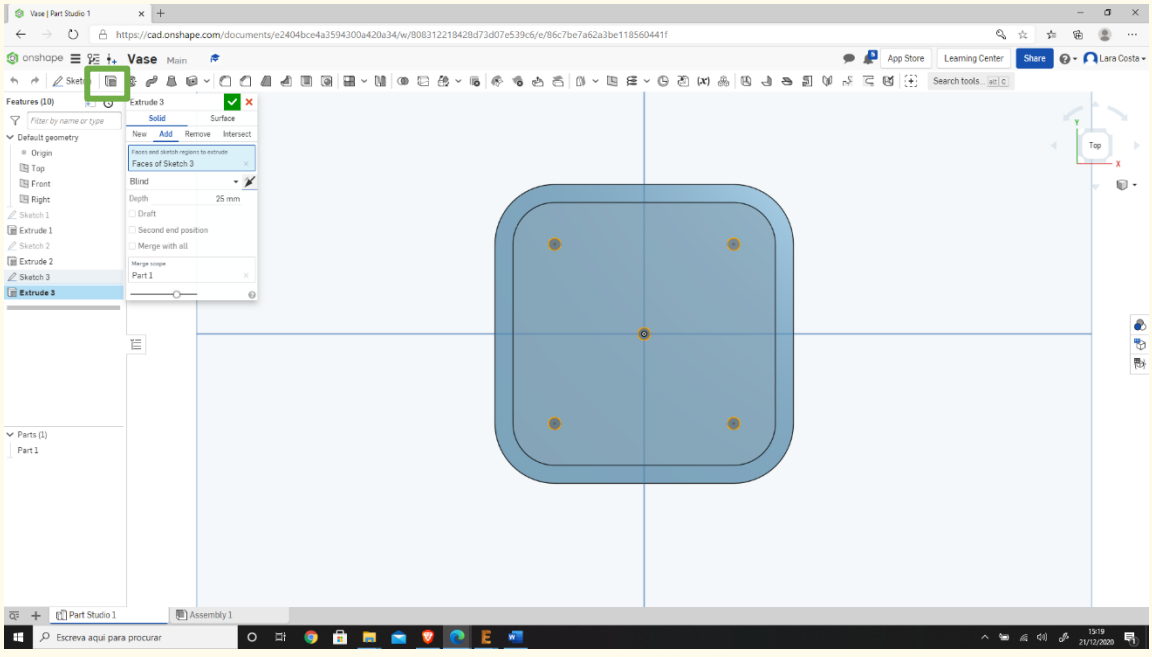
Step 25

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



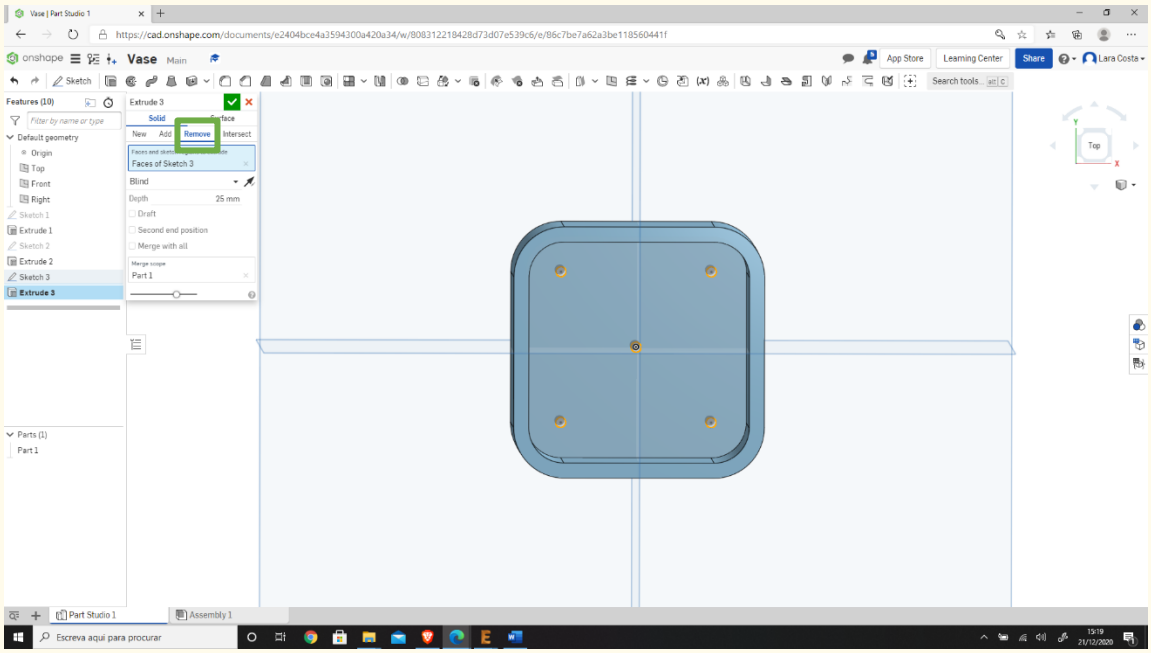
Step 26

Then select extrude.



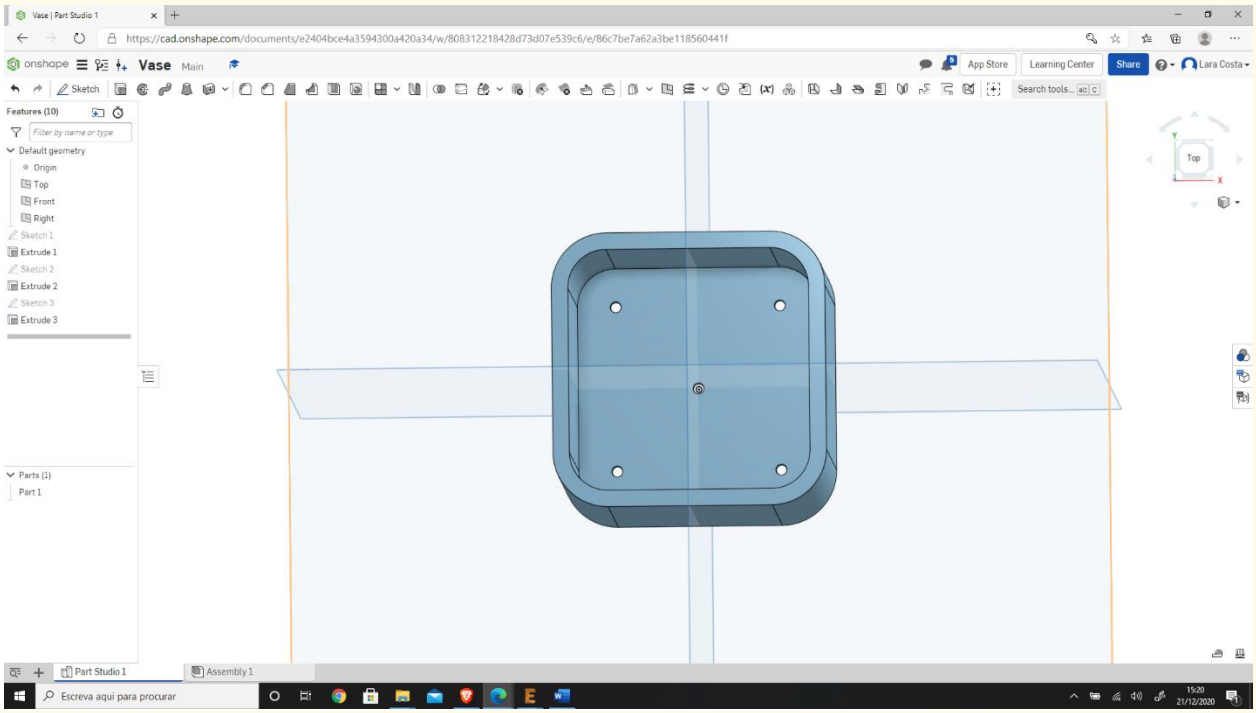
Step 27

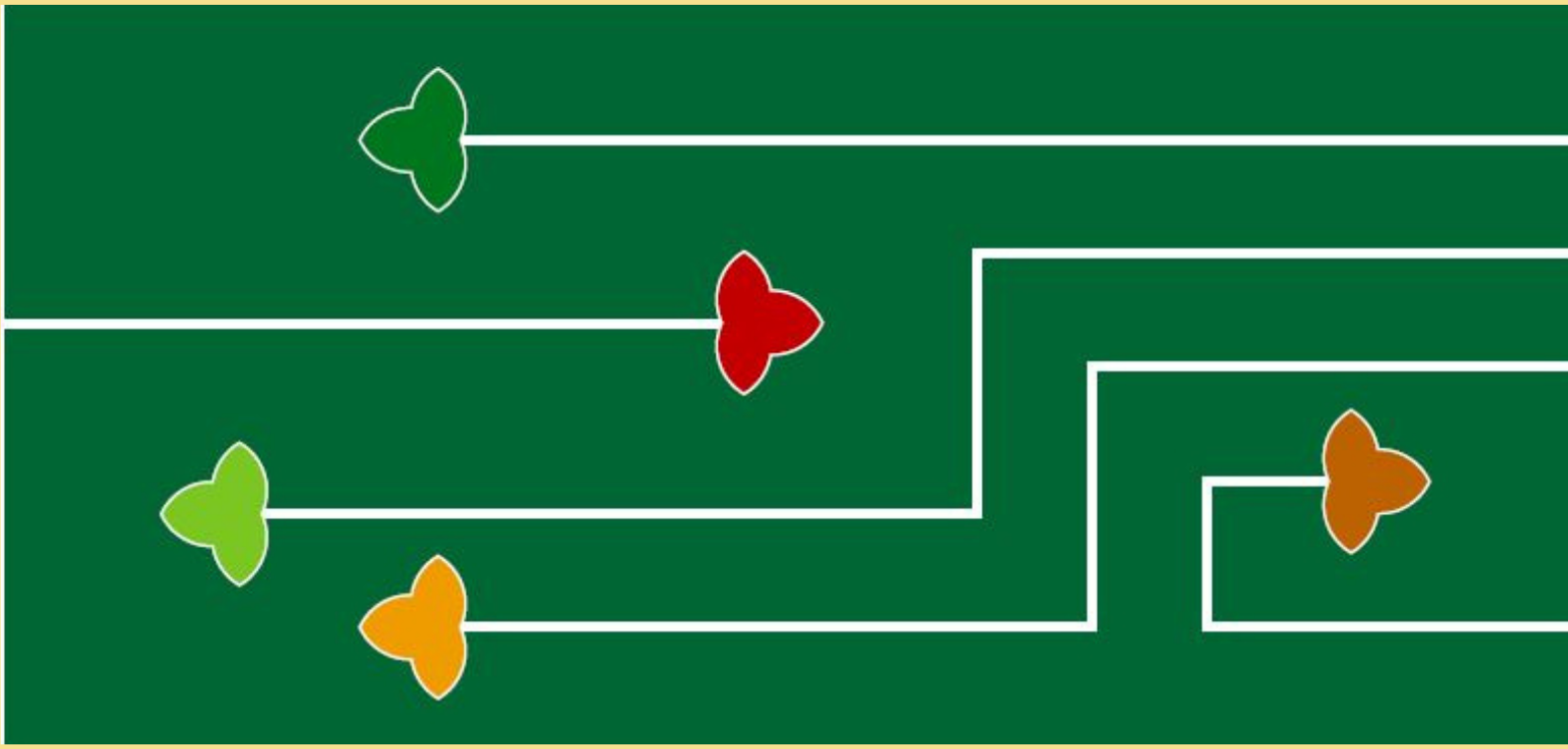
Click Remove ✅.



Step 28

This is how your project should look like.





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