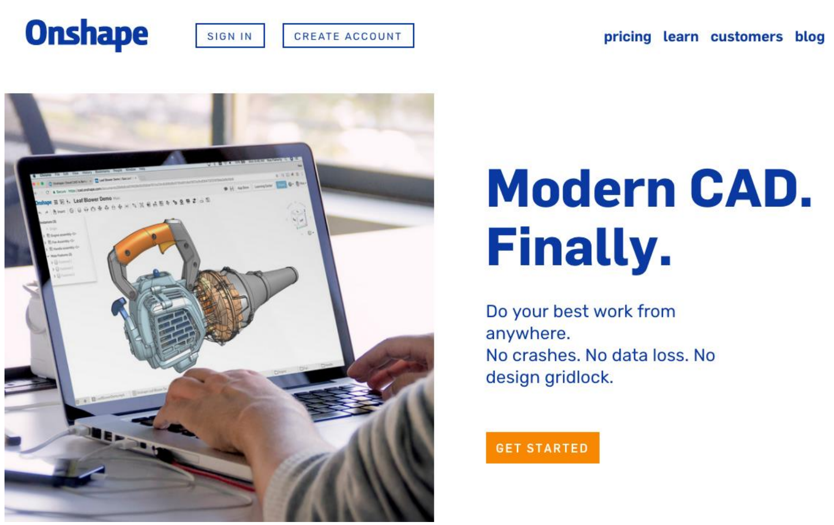
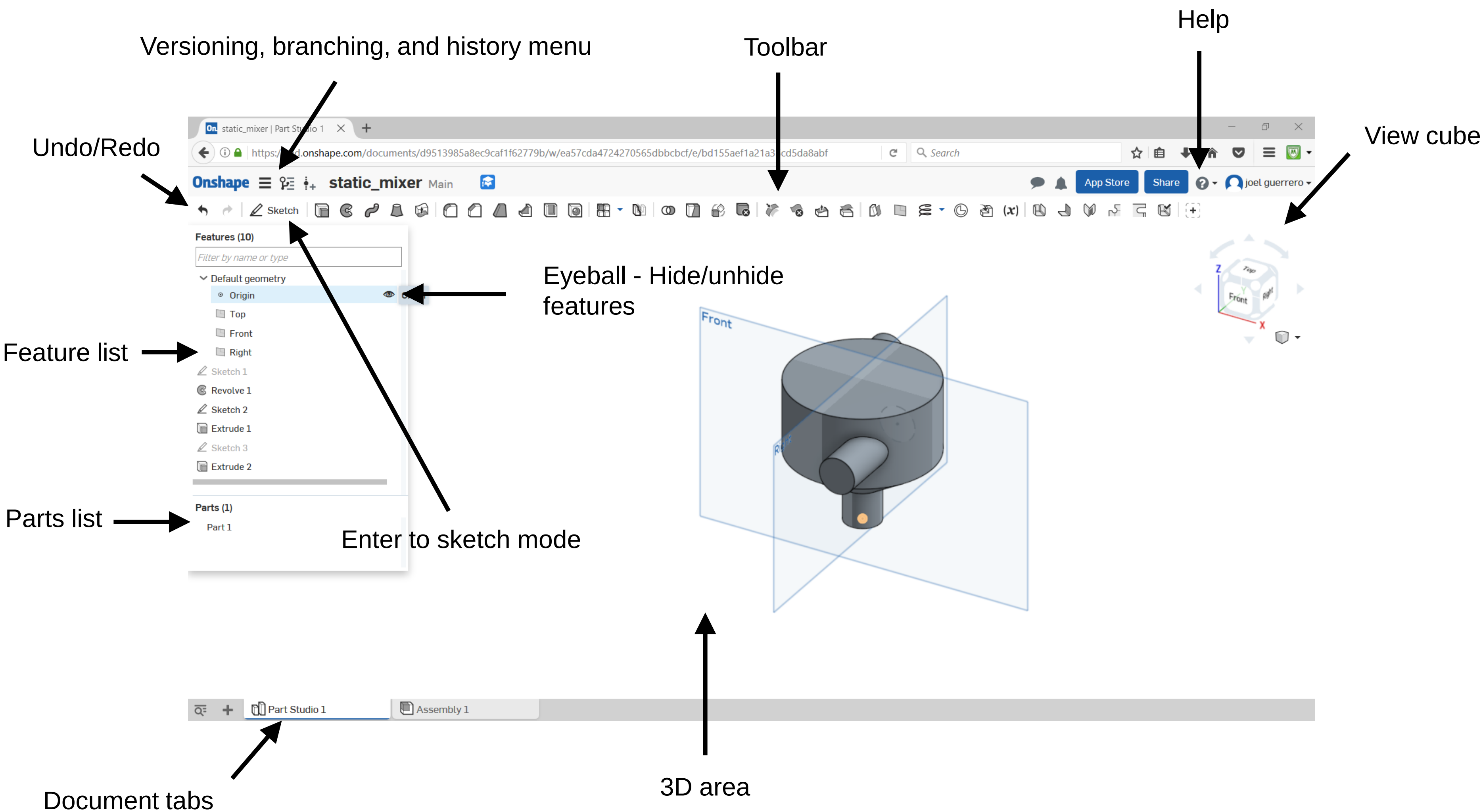
Introduction to solid modeling using Onshape

- Onshape is a CAD/solid modeling application.
- It provides powerful parametric and direct modeling capabilities.
- It is cloud based therefore you do not need to install any software.
- Documents are shareable.
- Multiple users can work in the same document at the same time (simultaneous editing).
- It runs in any device with a working web browser.
- It is freely available for Educational use.
- To start using Onshape register at: https://cad.onshape.com/



Introduction to solid modeling using Onshape

- Even if you have not used a CAD software before, you will find the GUI easy to use.
- You will notice that there is no save button because everything you do is automatically saved.



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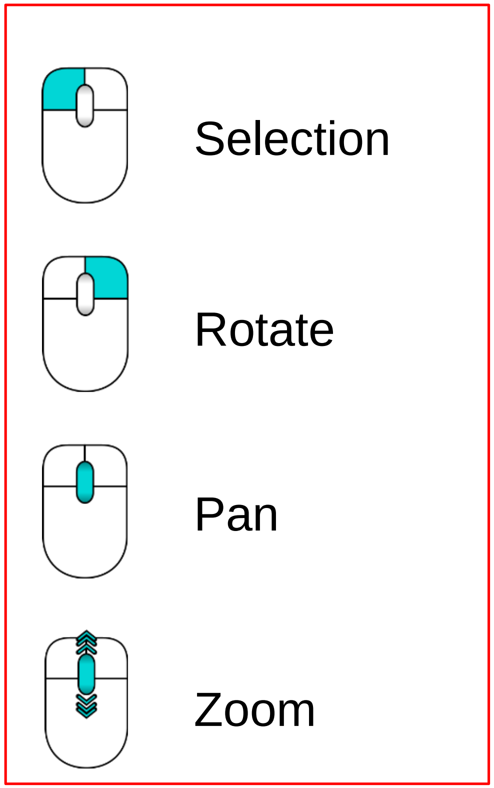
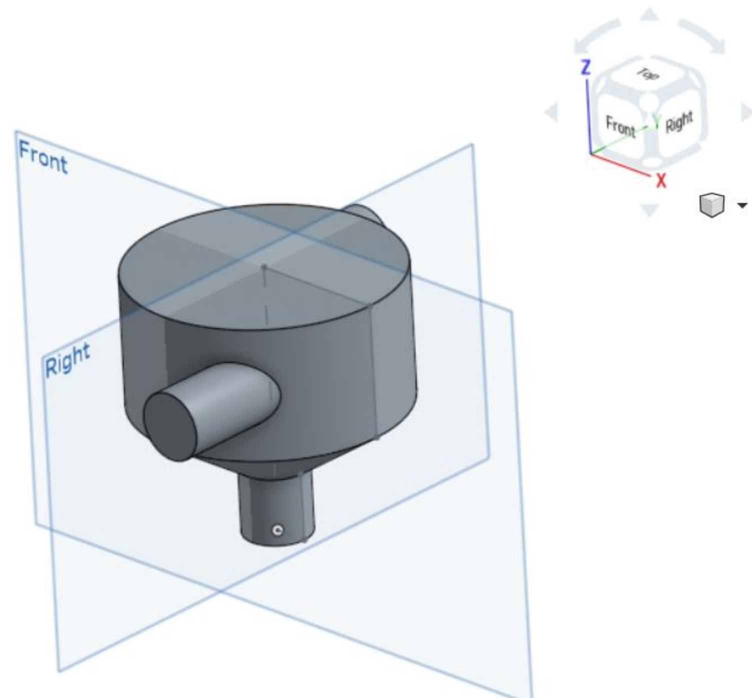
- Mouse interaction in the 3D area (it can be configure in the preference area).

Mouse interaction in the 3D viewer





- To deselect click in an empty region in the 3D area or press space-bar.

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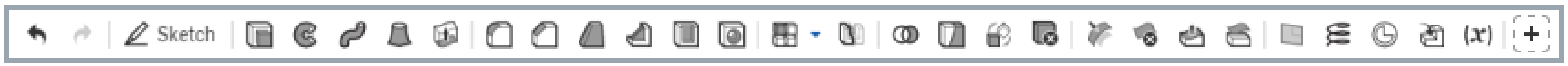
- When dealing with parts, assemblies and drawing in Onshape, you will find the following toolbars:

Feature toolbar:



Sketch toolbar:

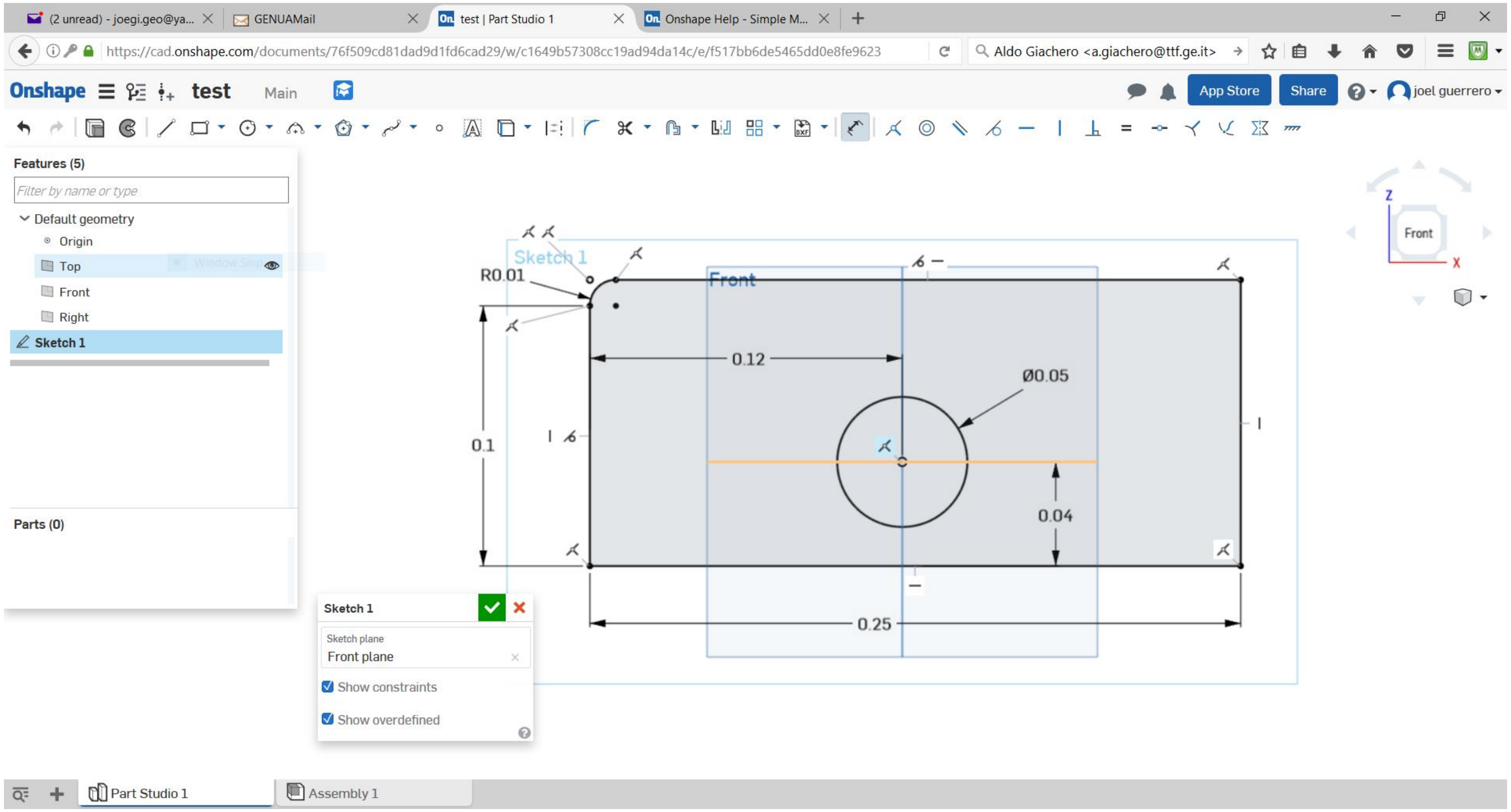
Assemblies toolbar:

Drawings toolbar:

- Each icon in the toolbar corresponds to a different feature.
- If you mouse over the toolbar icons you will get a pop-up window with the instructions of how to use the feature.
- If you need more information about each feature, use the help.

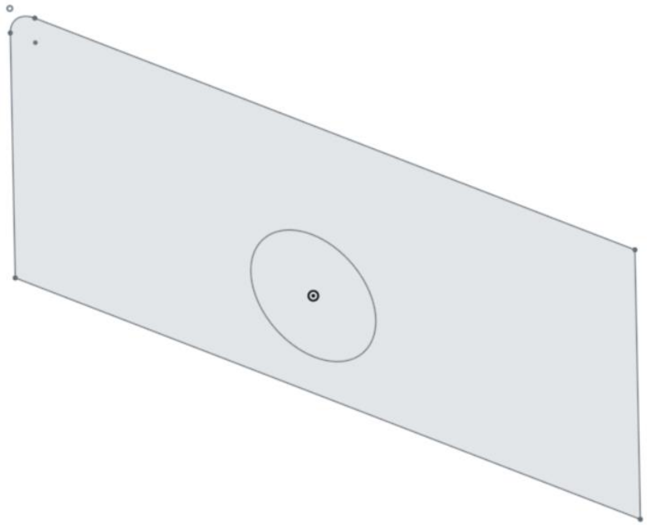
Introduction to solid modeling using Onshape

- Parametric modeling and feature based modeling are crucial components in the design experience.
- Onshape is parametric and feature based, with a relative fast learning curve.
- Sketches are the core of good 3D designs and parametrization.
- And dimensions and constrains are the glue that keep sketches together.

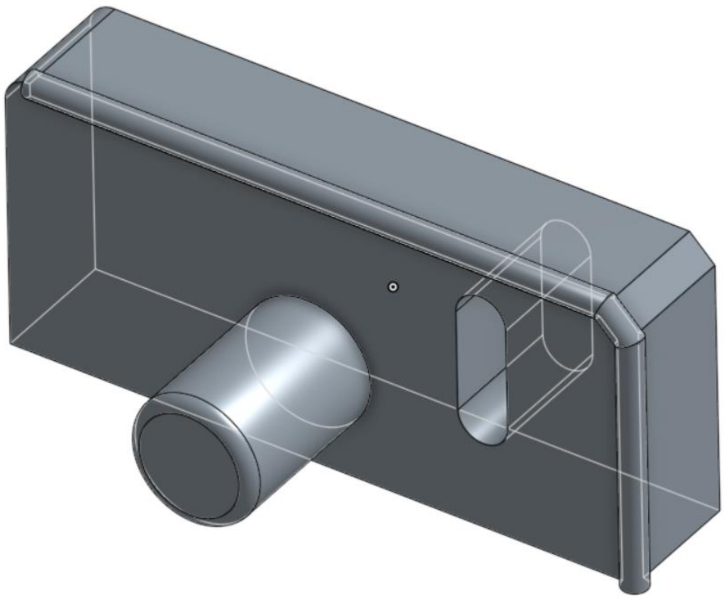


Introduction to solid modeling using Onshape

- A simple sketch, can be used to do many things using the parametric modeling and feature based modeling options available in Onshape.



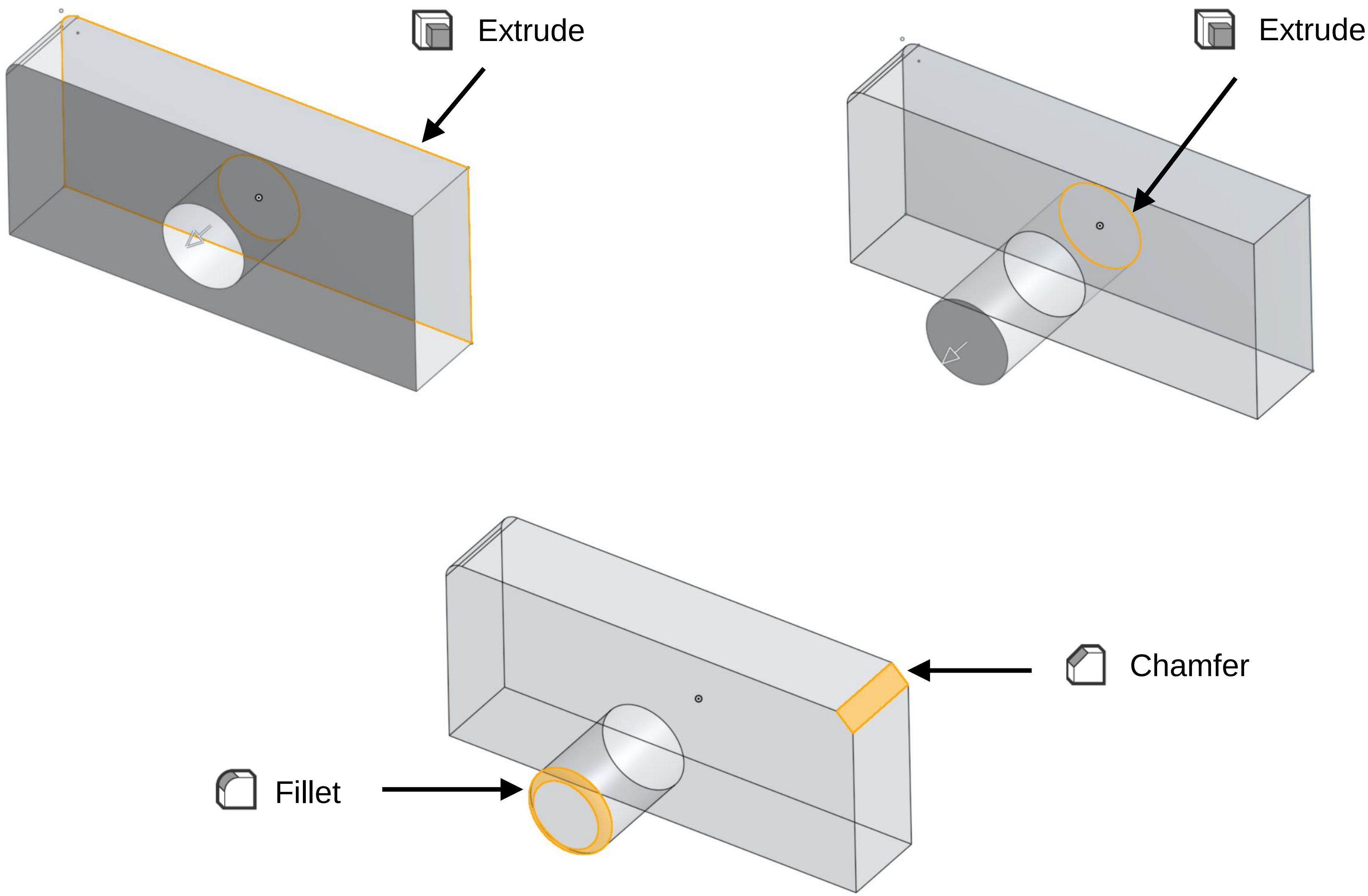
Starting sketch



Final solid model

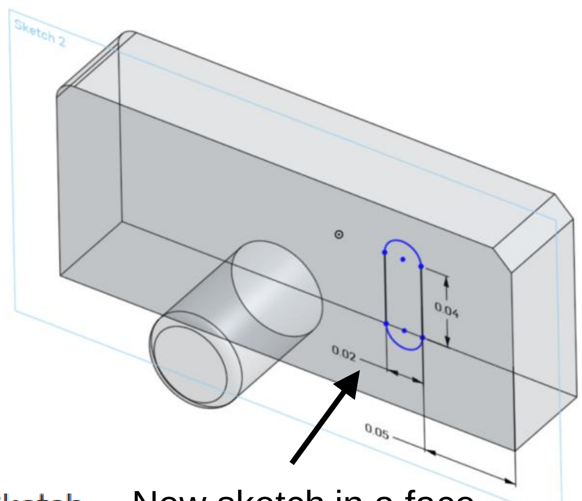
Introduction to solid modeling using Onshape

- A simple sketch, can be used to do many things using the parametric modeling and feature based modeling options available in Onshape.



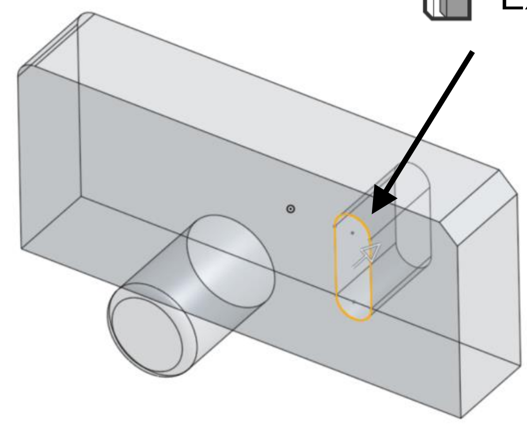
Introduction to solid modeling using Onshape

- A simple sketch, can be used to do many things using the parametric modeling and feature based modeling options available in Onshape.

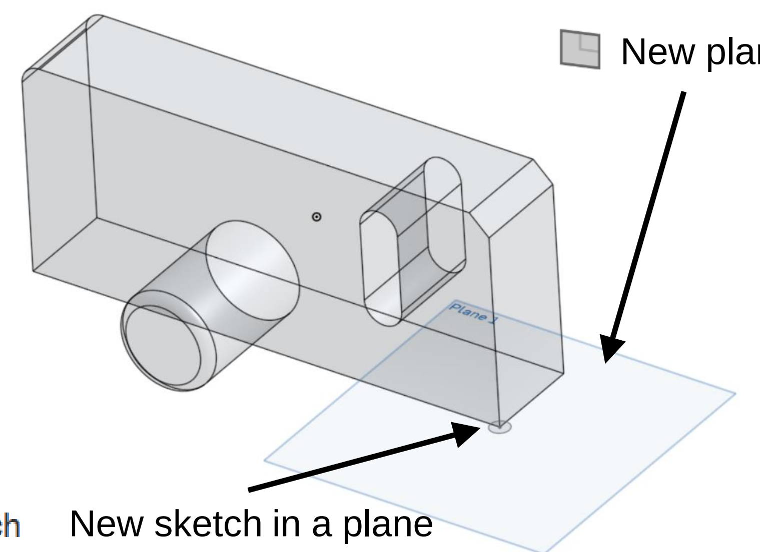


Sketch New sketch in a face

Extrude

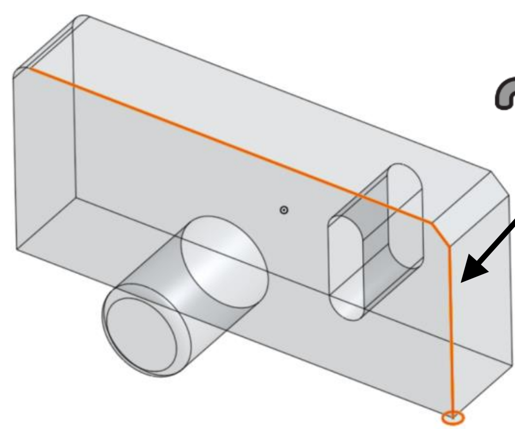


New plane



Sketch New sketch in a plane

Sweep



Introduction to solid modeling using Onshape

- And finally, some keyboard shortcuts that may turn out useful for your work:

Keyboard shortcuts

General

shift ?	Keyboard shortcuts
ctrl / ⌘ z	Undo
ctrl / ⌘ y	Redo
delete	Delete selection
space bar	Clear selection
shift n	Rename selection
esc	Cancel command
enter	Accept command
shift enter	Accept & repeat command
shift click	Open in new window
ctrl / ⌘ click	Open in new tab
ctrl u	Feedback
alt t	Search tabs

Assembly

shift	Lock mate inference
ctrl / ⌘ c	Copy
ctrl / ⌘ v	Paste
m	Mate
ctrl m	Mate connector
i	Insert dialog
shift s	Snap mode
a	Flip primary axis
q	Reorient secondary axis
j	Hide/show mates
k	Hide/show mate connectors

Part Studio

shift s	Sketch
shift e	Extrude
shift f	Fillet
ctrl m	Mate connector
shift	Lock mate inference
a	Flip primary axis
q	Reorient secondary axis
k	Hide/show mate connectors

3D view

shift z	Zoom in
z	Zoom out
f	Zoom to fit
w	Zoom to window
←→↑↓	Rotate
shift ←→↑↓	Pan
shift 1	Front view
shift 2	Back view
shift 3	Left view
shift 4	Right view
shift 5	Top view
shift 6	Bottom view
shift 7	Isometric view
shift 8	Section view
n	View normal to
p	Hide/show planes
y	Hide selected part
shift y	Show selected part

Sketch

shift	Suppress inferencing
l	Line
g	Corner rectangle
r	Center point rectangle
c	Center point circle
a	3 point arc
shift f	Fillet
m	Trim
x	Extend
o	Offset
u	Use
d	Dimension
i	Coincident
b	Parallel
t	Tangent
h	Horizontal
v	Vertical
e	Equal
q	Toggle construction

Drawings

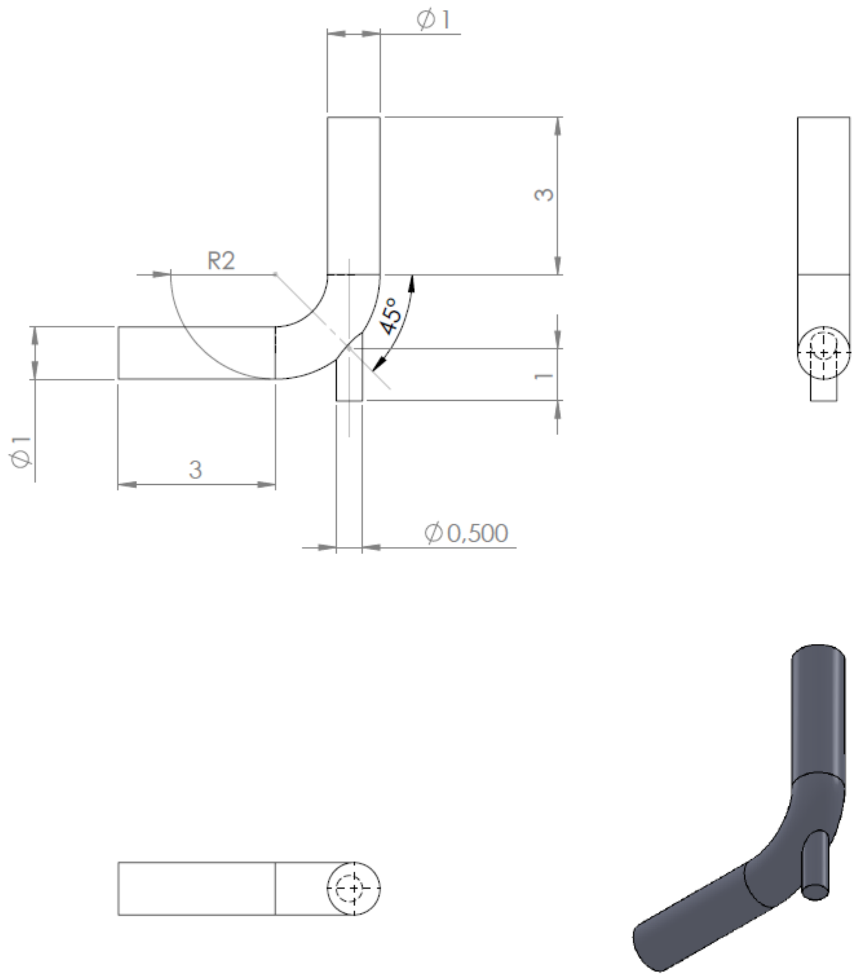
shift z	Zoom in
z	Zoom out
f	Zoom to fit
w	Zoom to window
p	Projected view
d	Linear dimension
shift r	Radial dimension
shift d	Diameter dimension
n	Note
ctrl q	Update drawing
l	Line
ctrl s	Display sheet menu
pg dn	Next sheet
pg up	Previous sheet
home	First sheet
end	Last sheet

Introduction to solid modeling using Onshape

- This is all we need to know about part modeling in Onshape.
- Let us work with a few simple geometries to understand how Onshape works.
- We also will show you a few clicks and picks you should be aware of.
- Remember, study and practice is the best way to build modeling skills.

Introduction to solid modeling using Onshape

- Let us create this solid model using the dimensions illustrated.



Note: all the dimensions are in meters

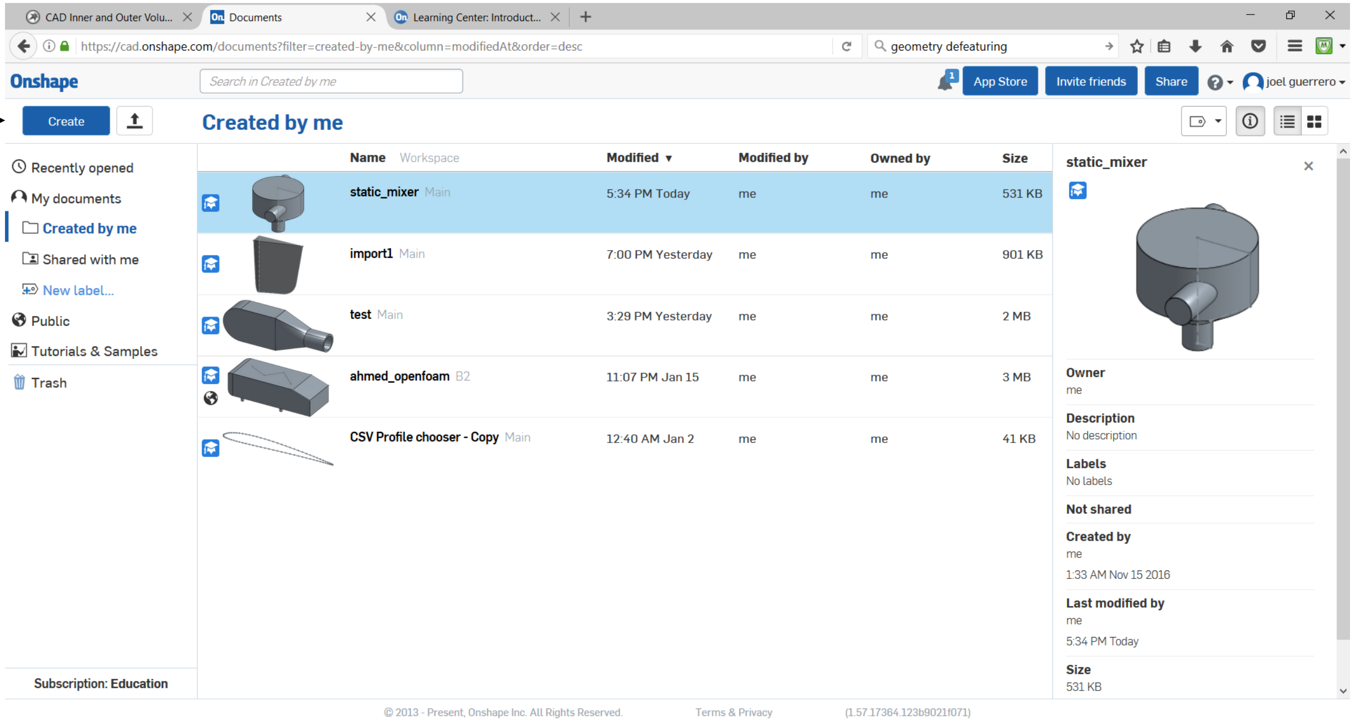
Introduction to solid modeling using Onshape

- Remember, there is no wrong or right way to make a model, but there are sometimes better ways.
- The fact that there are many ways to accomplish a task when creating a model, gives you the freedom to work in a way that is comfortable to you. Hereafter we are going to show you our way.
- If you have an idea how your design may need to change in the design process, then you should make it in a way to make those changes more efficient.
- Think about a strategy to use to create your design or <u>design intent</u>.
    - Choose one feature over other.
    - Dimensioning strategy.
    - Order of the operations.
    - Parametrization.
    - Single or multiple parts.
    - Top-bottom or bottom-up modeling technique.

Introduction to solid modeling using Onshape

- Enter the document page and create a new design

Create new document

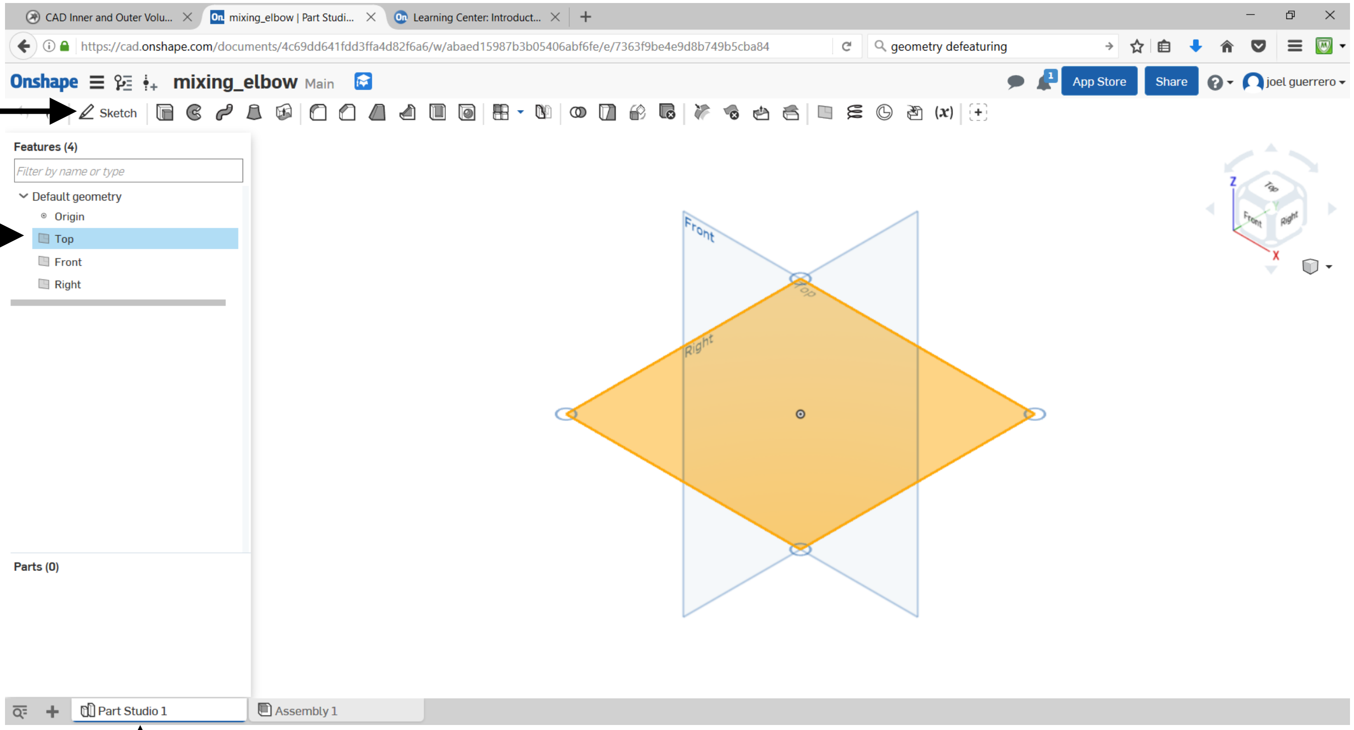


Introduction to solid modeling using Onshape

- In the part studio page, select the top plane and start a new sketch.
- If you mouse over the toolbar icons you will get a pop-up window with the instructions of how to use the feature.

Start a new sketch

Select this plane

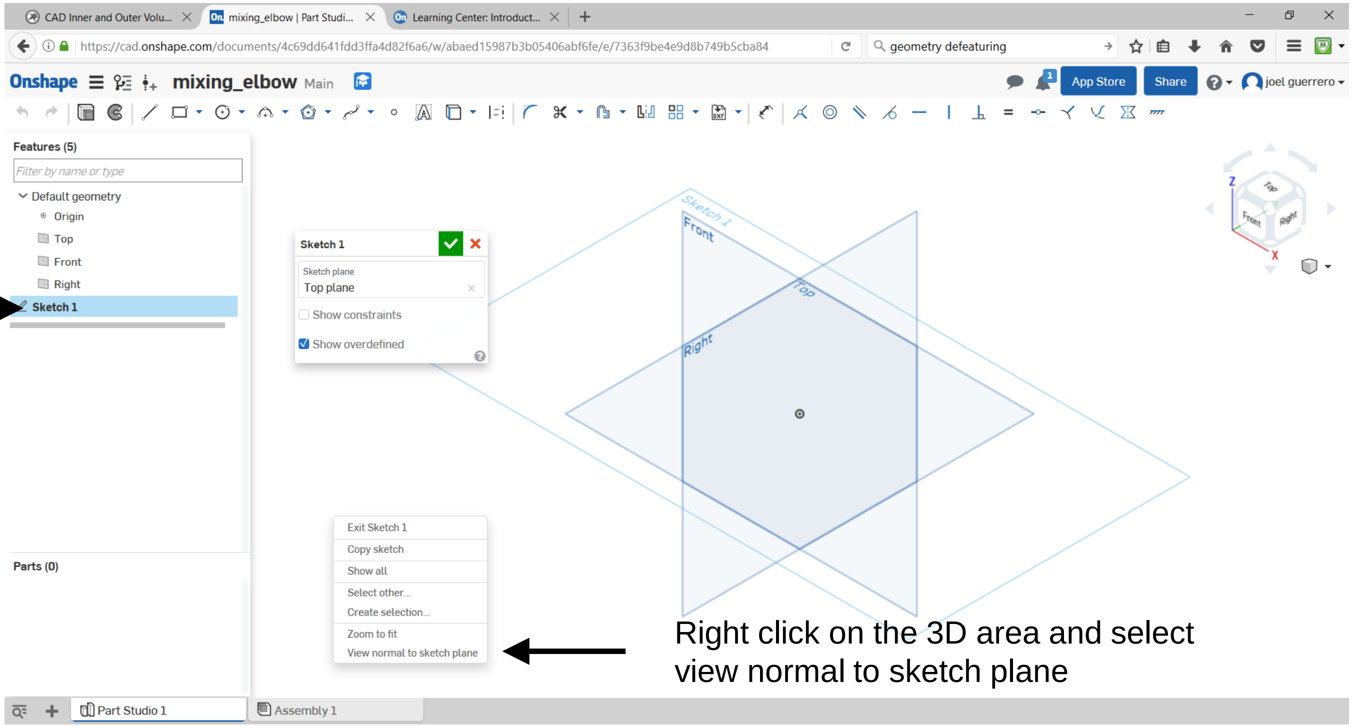


Part studio page

Introduction to solid modeling using Onshape

- In the part studio page, select the top plane and start a new sketch.

We are working in this sketch



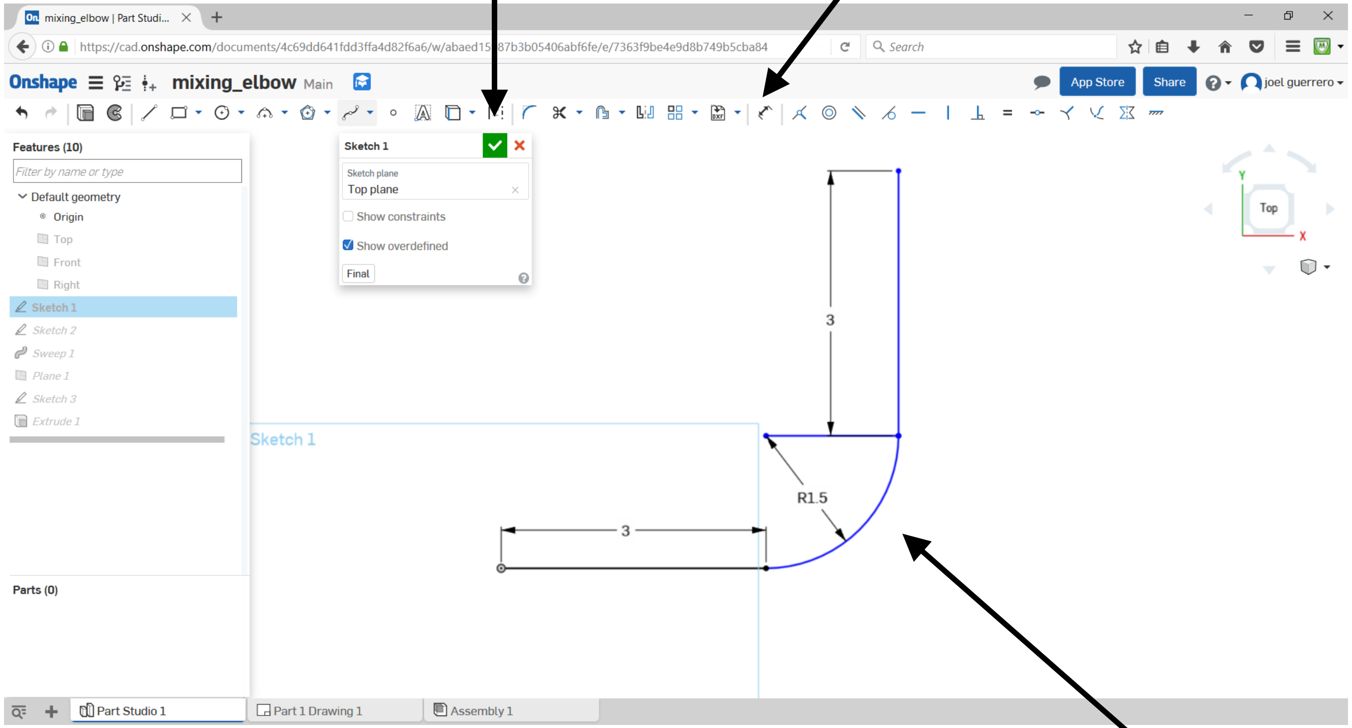
Right click on the 3D area and select view normal to sketch plane

Introduction to solid modeling using Onshape

- Using the sketching features, draw the following line.

When you are done sketching press the checkmark

To add dimensions



Use the dimensions illustrated to draw this line

In sketch mode:

- Blue geometry is free to move.
- Black geometry is fully defined.
- Red geometry is over-constrained.

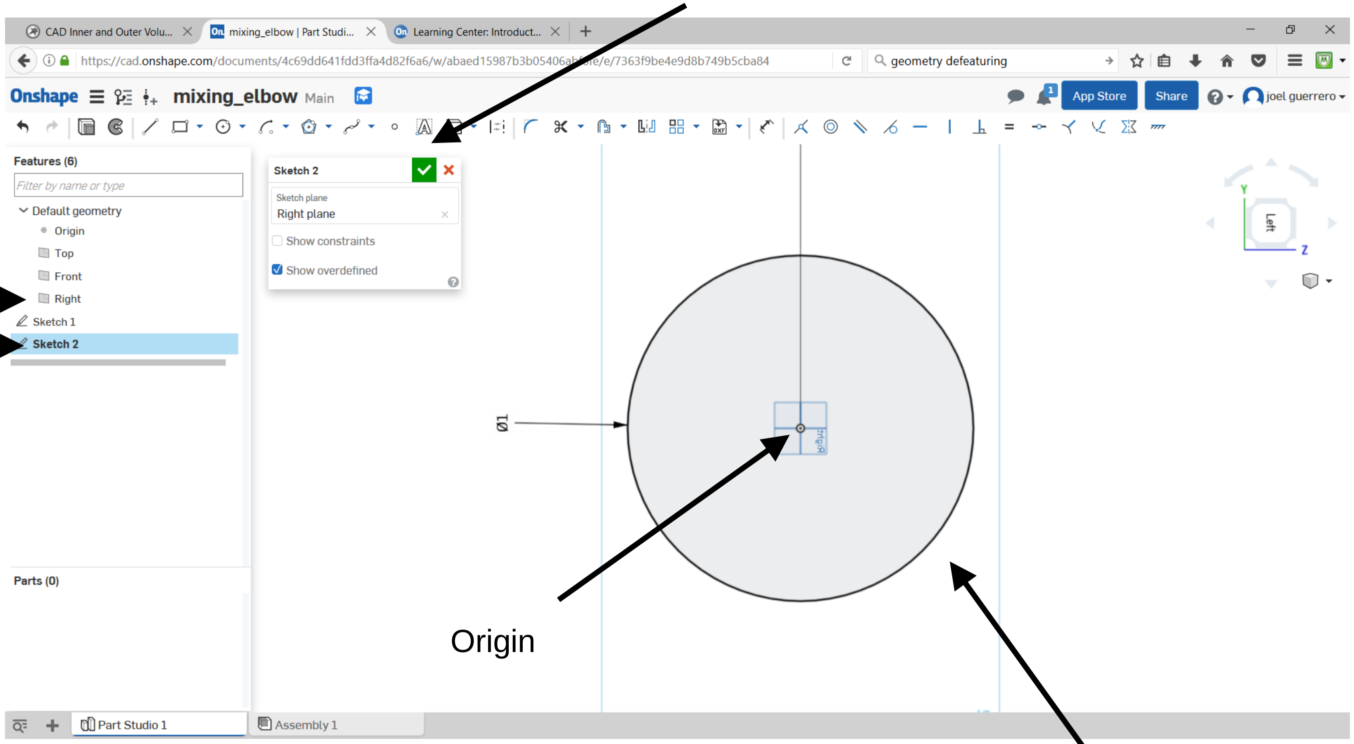
Introduction to solid modeling using Onshape

- Select the right plane and start a new sketch.
- Draw a circle with the center in the origin (the white point).

When you are done sketching
press the checkmark

Select this plane

We are working in this sketch



Origin

Use the dimensions illustrated to
draw the circle

Introduction to solid modeling using Onshape

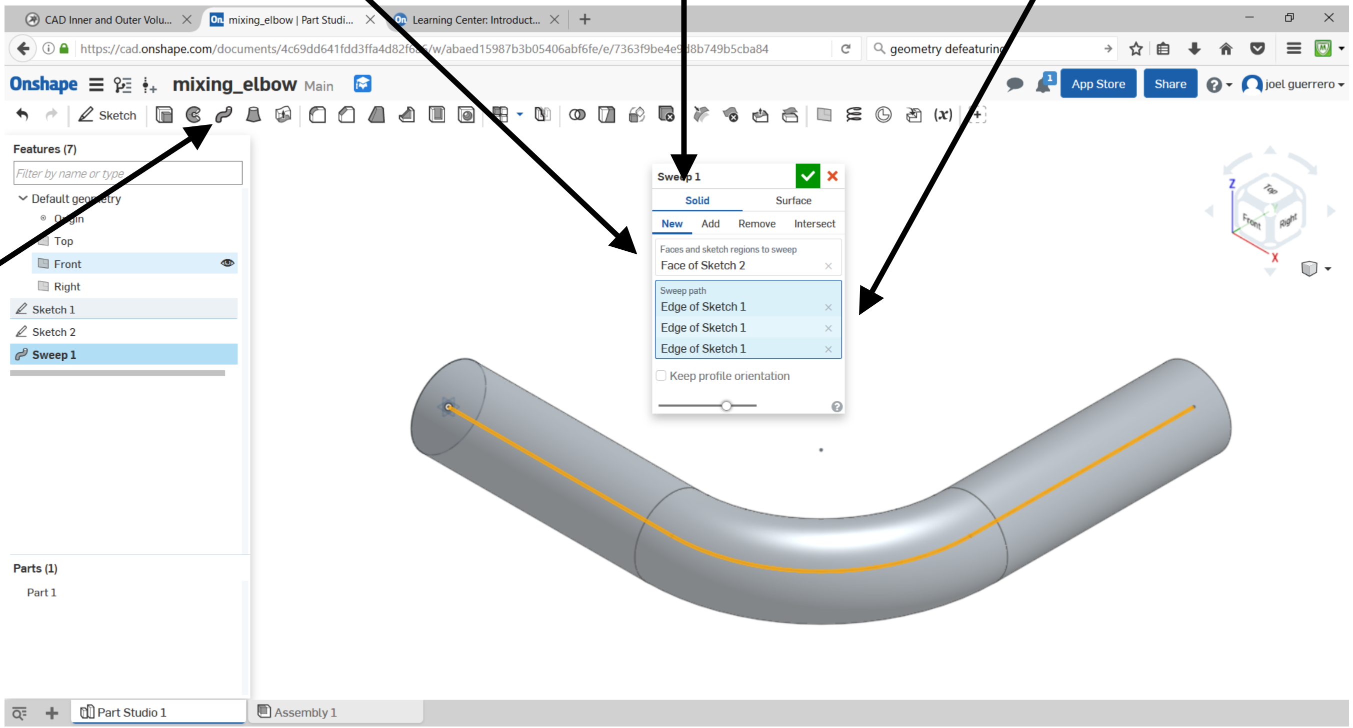
- Use the sweep feature to create a new solid.

Select the circle as the face to sweep

Select new solid

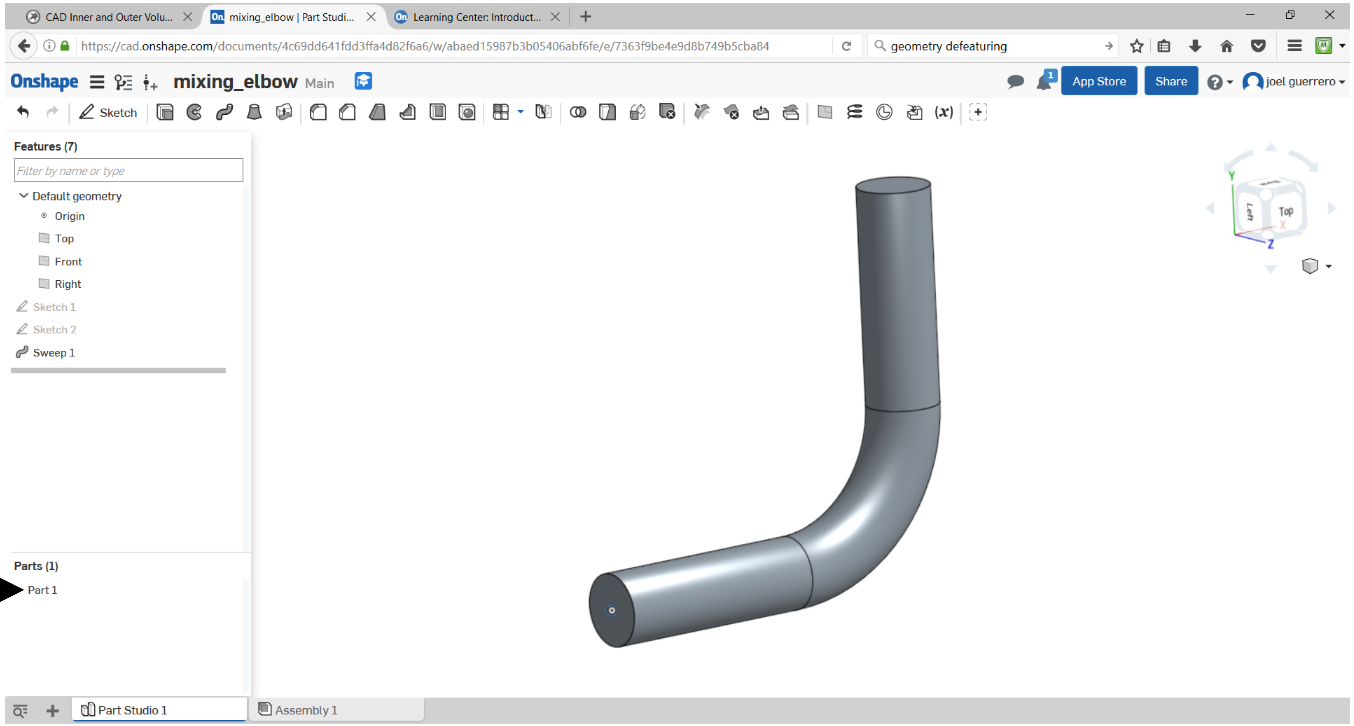
Select the lines as the sweep patch

Sweep feature



Introduction to solid modeling using Onshape

- At this point, you should have this solid.

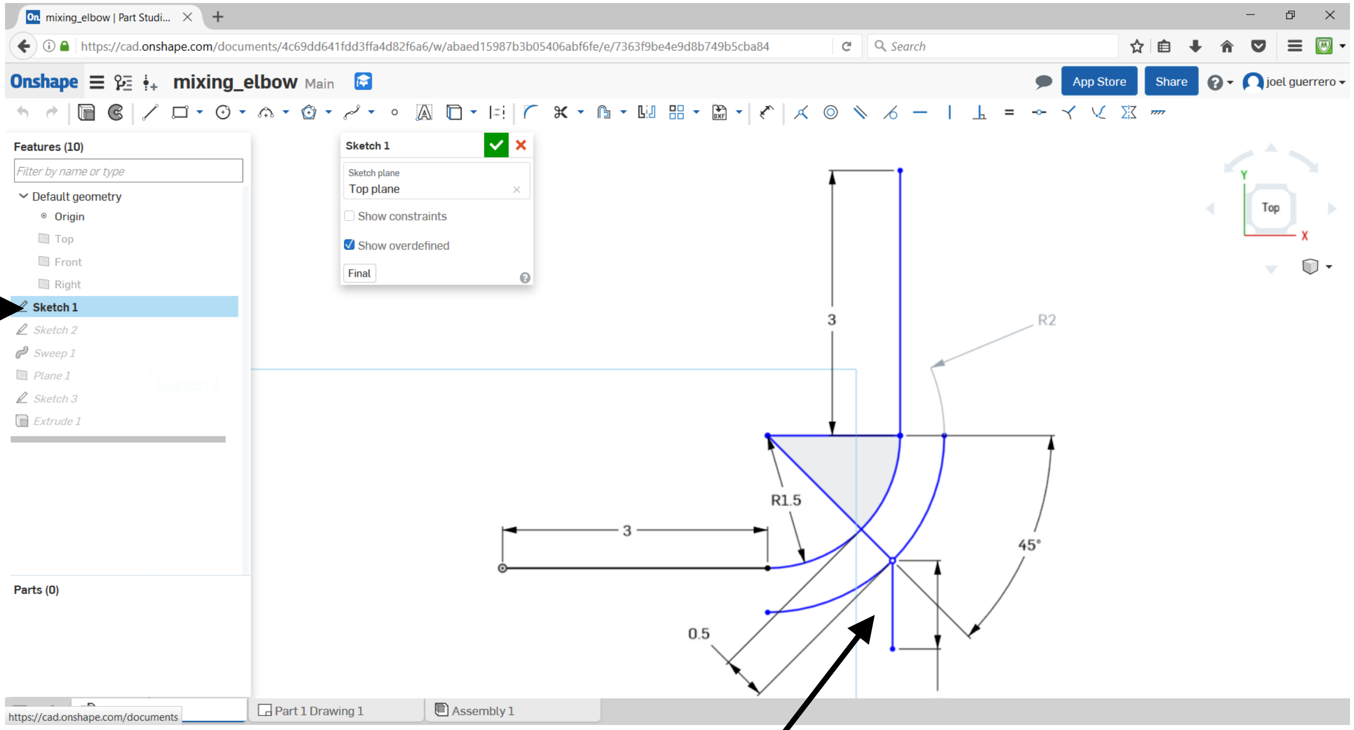


Solid name.
Right click to rename
or view the properties

Introduction to solid modeling using Onshape

- Let us add the new inlet to the pipe.
- Create a new sketch in the top plane or edit the initial sketch (hereafter we will edit the initial sketch).

Right click and choose the option edit



Sketch these lines using the dimensions illustrated. Pay attention to the angle and the offset distance.

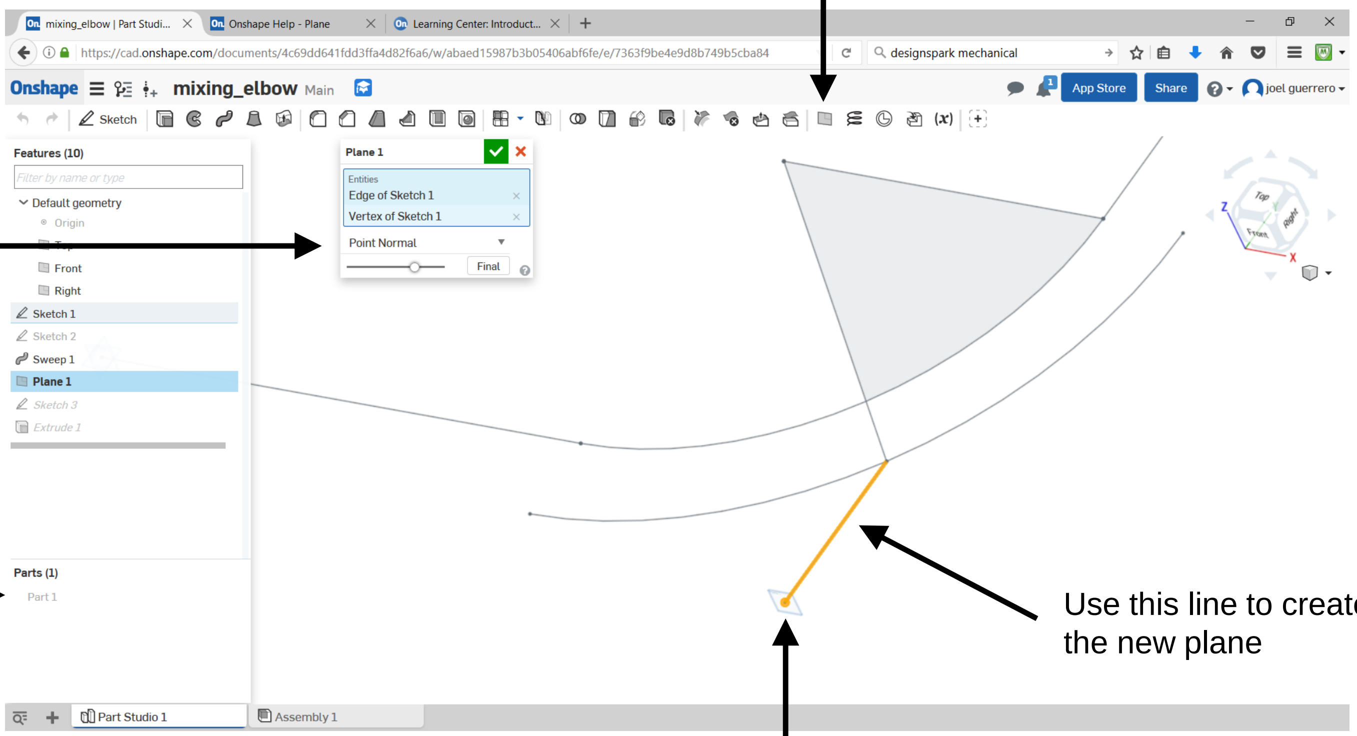
Introduction to solid modeling using Onshape

- Create a plane normal to a line and passing through a point

Create new plane

Select point normal, and select the line and point as illustrated

To get better visibility, you can hide the solid or adjust the transparency

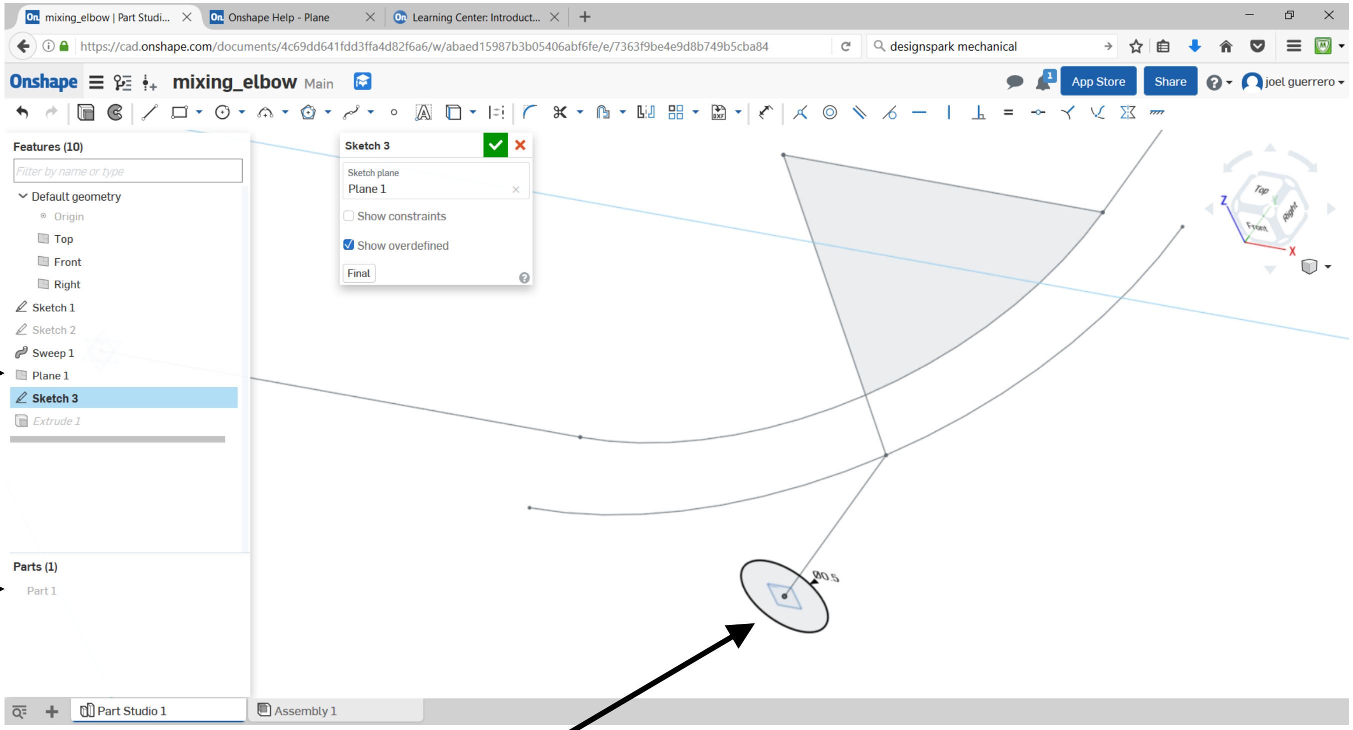


Use this line to create the new plane

Use this point to create the new plane

Introduction to solid modeling using Onshape

- Sketch a circle in the newly created plane.



New plane

To get better visibility, you can hide the solid or adjust the transparency

Sketch this circle in the newly created plane

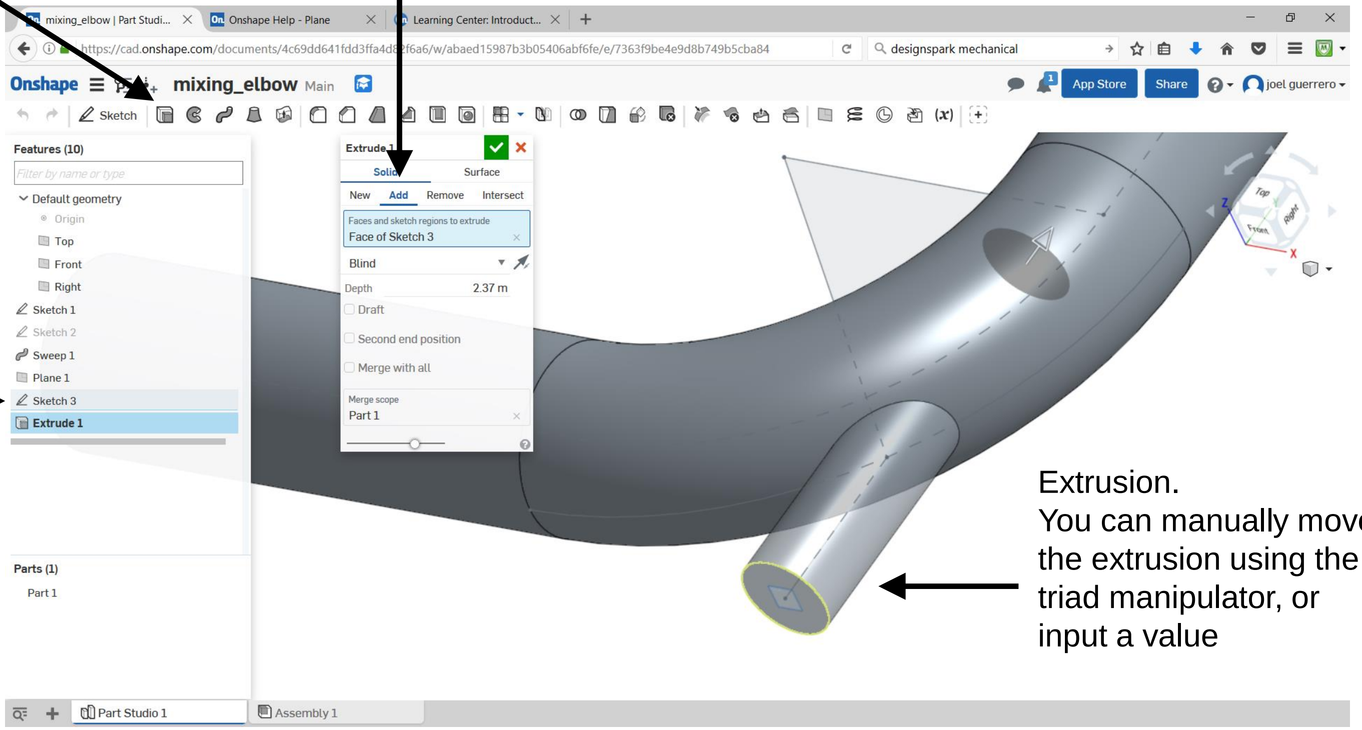
Introduction to solid modeling using Onshape

- Use the feature extrude to create a new solid using the previous sketch.
- Extrude the circle until it intercepts the solid.

Add the new solid to the previous part

Feature extrusion

Use this sketch as the base for the extrusion

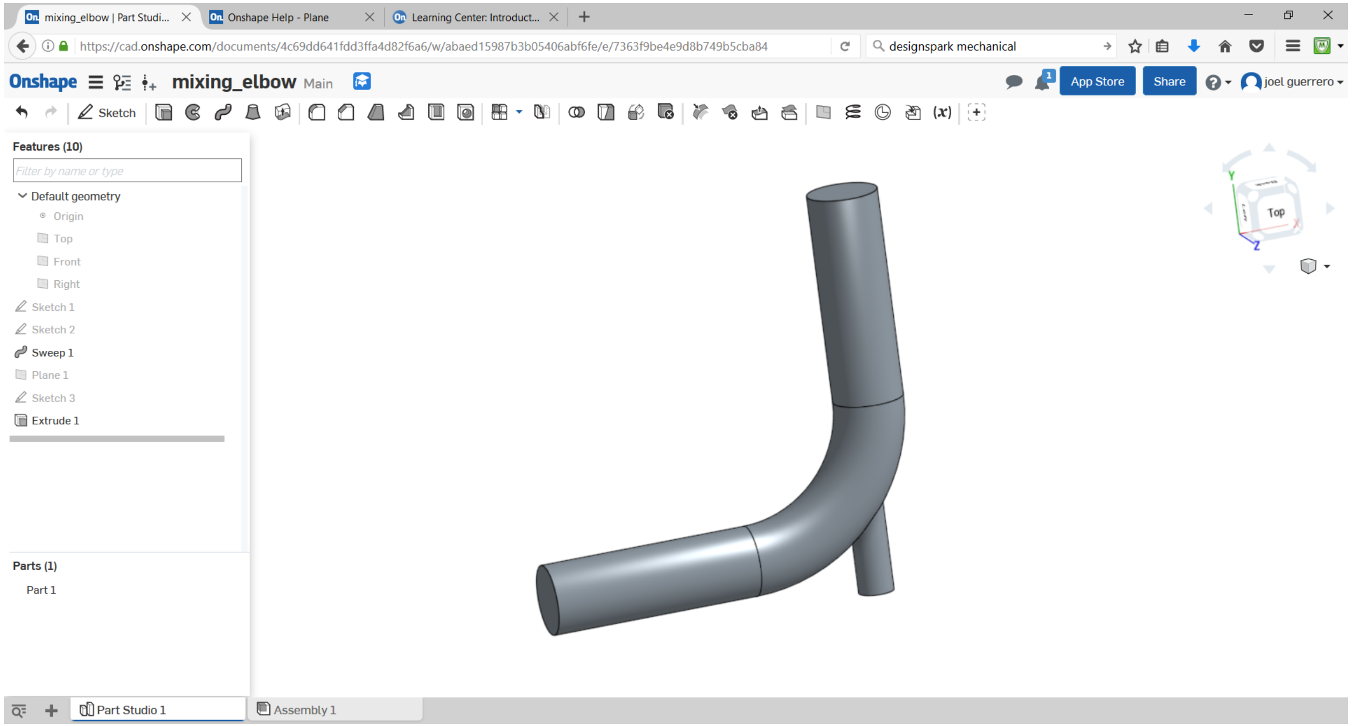


Extrusion.
You can manually move the extrusion using the triad manipulator, or input a value

Instead of the extrusion feature, you could use the sweep feature. You will need to create a longer sweep path.

Introduction to solid modeling using Onshape

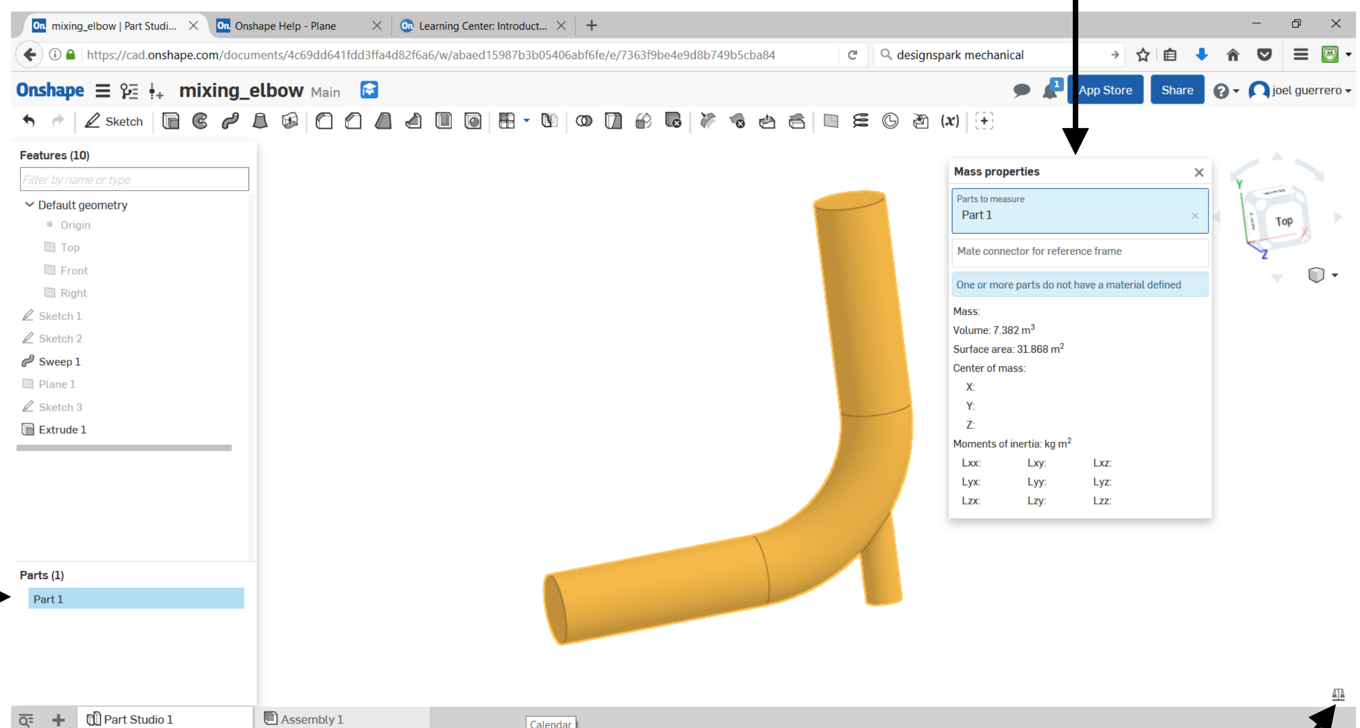
- At this point you should have the following solid.



Introduction to solid modeling using Onshape

- If you want to know the mass properties of the solid, select it, and then click on the mass properties icon.
- To get the inertia, you will need to assign a material.

Mass properties window



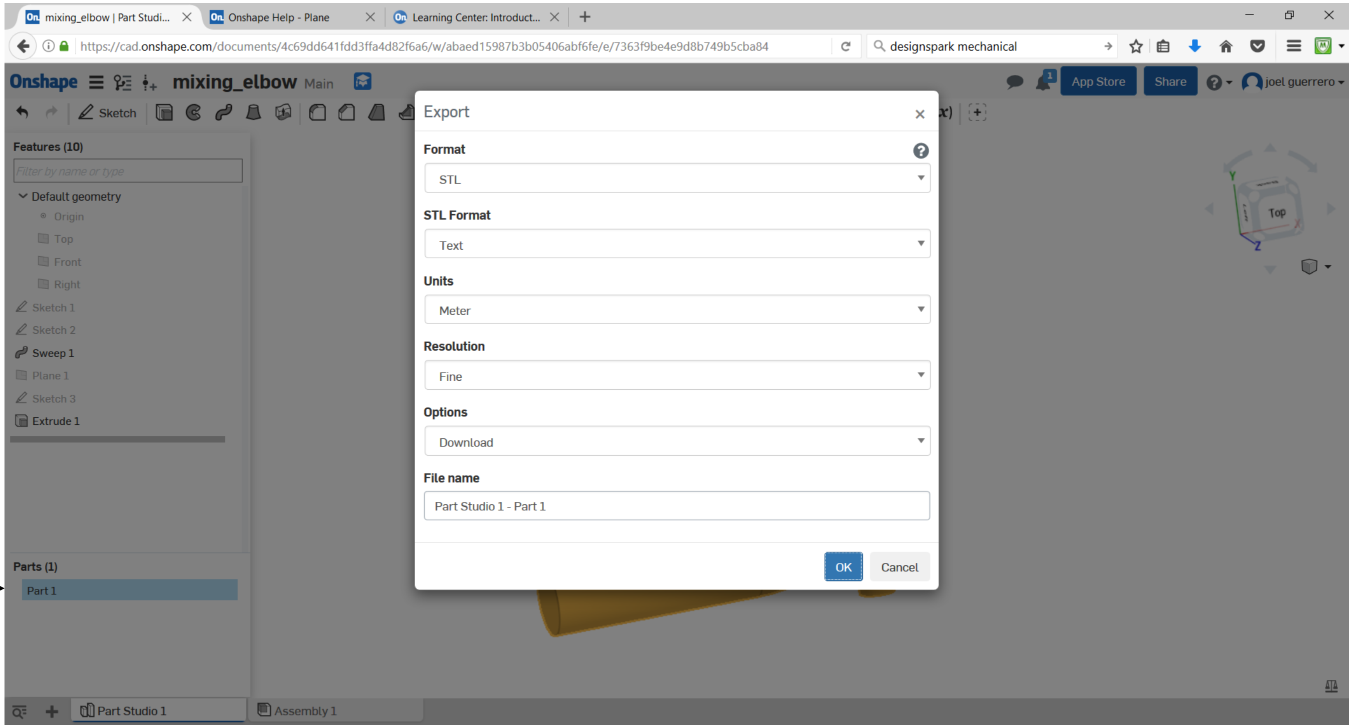
Select the part.
Right click and select assign material.

Mass properties icon

Introduction to solid modeling using Onshape

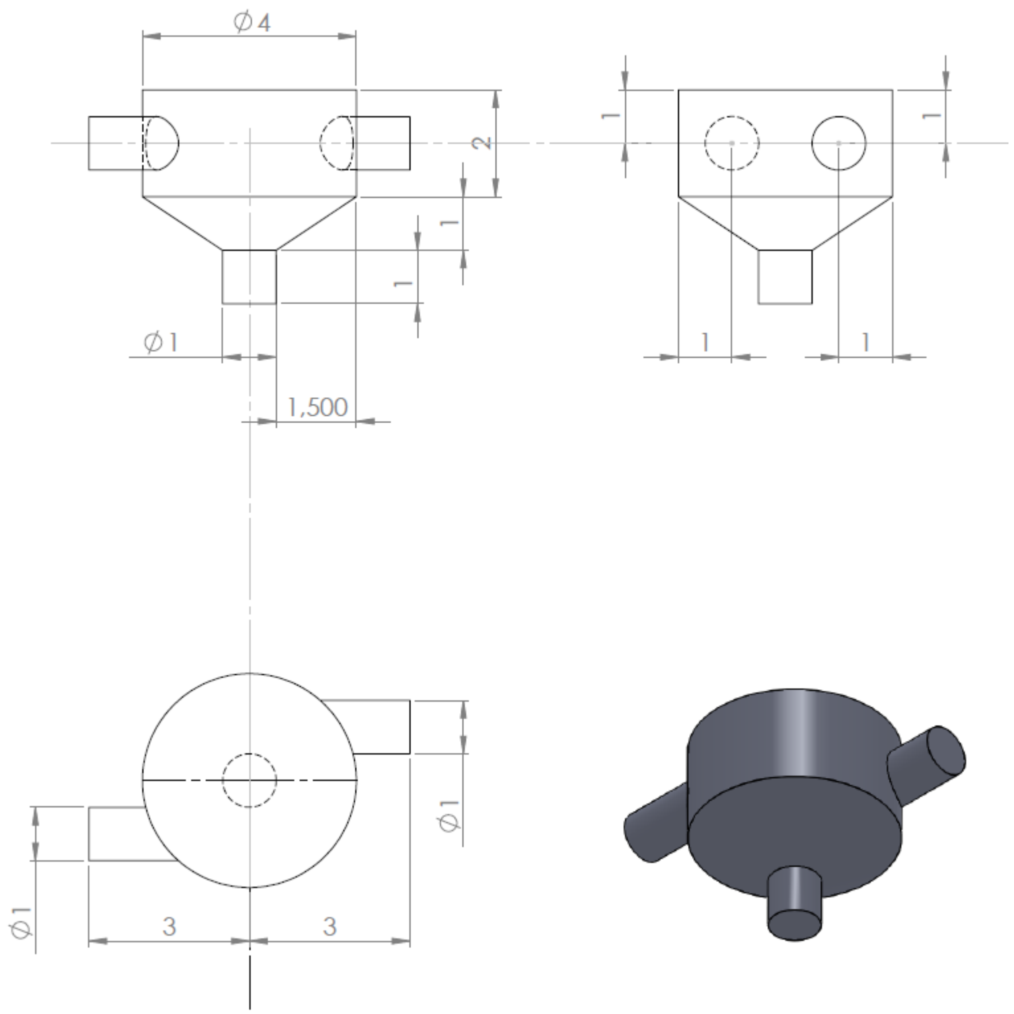
- To export the design, right click on the part name and select the option export.
- Choose the desired format. In this case choose STL.

Right click and select the option export.



Introduction to solid modeling using Onshape

- Let us create another solid model using the dimensions illustrated.



Note: all the dimensions are in meters

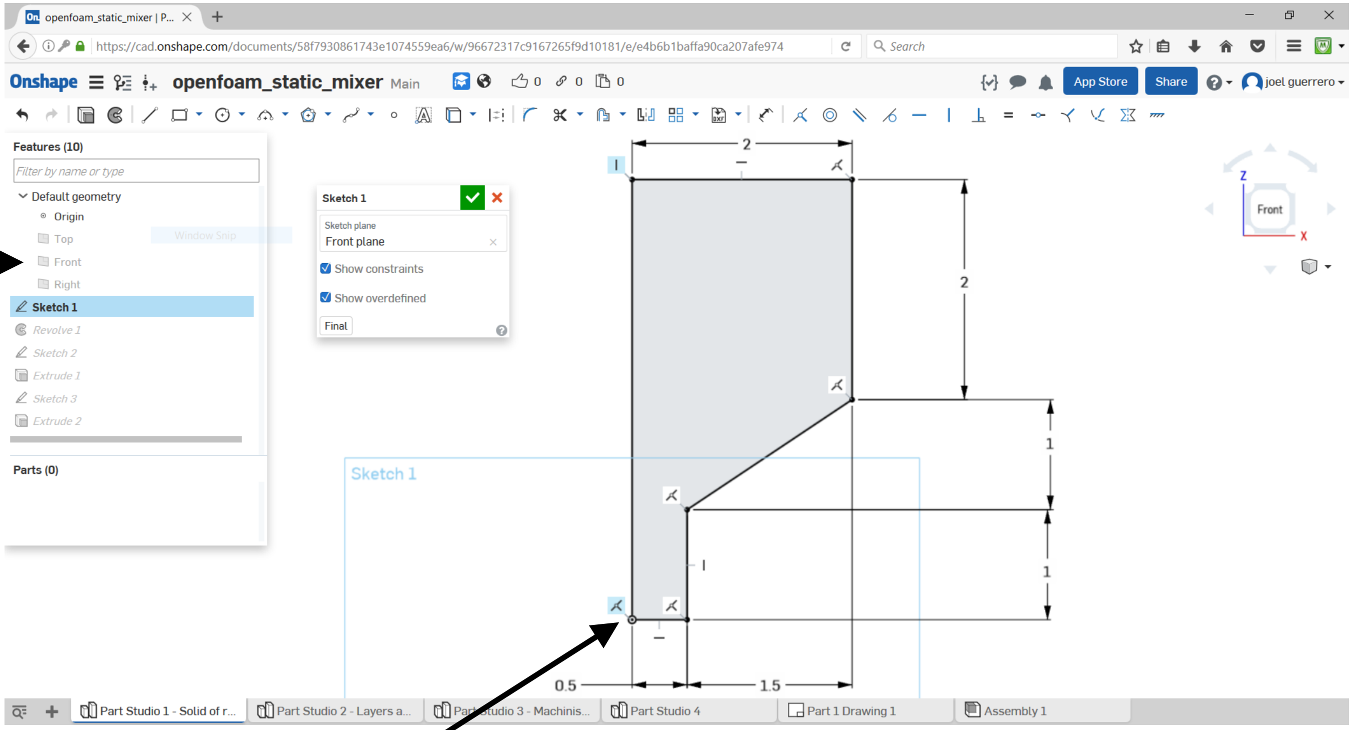
Introduction to solid modeling using Onshape

- We have mentioned that there are many ways to accomplish a task when creating a model.
- Hereafter we are going to generate the same solid model using three different approaches.
- And depending of our final goal, one approach may be better than the other one.
- This is design intent in action.
- We are going to try the following approaches:
    - One single sketch to generate the solid of revolution.
    - The layer approach to generate the main body.
    - The machinist approach. Here we start from a uniform solid and then we remove material.
- As we are already familiar with the interface and the design process, we are not going to explain all the steps.

Introduction to solid modeling using Onshape

- Let us draw one single sketch to generate the solid of revolution.
- Using the dimensions illustrated, create this sketch in the front plane.
- Pay attention to the constrains.

Select this plane to draw the sketch



Origin

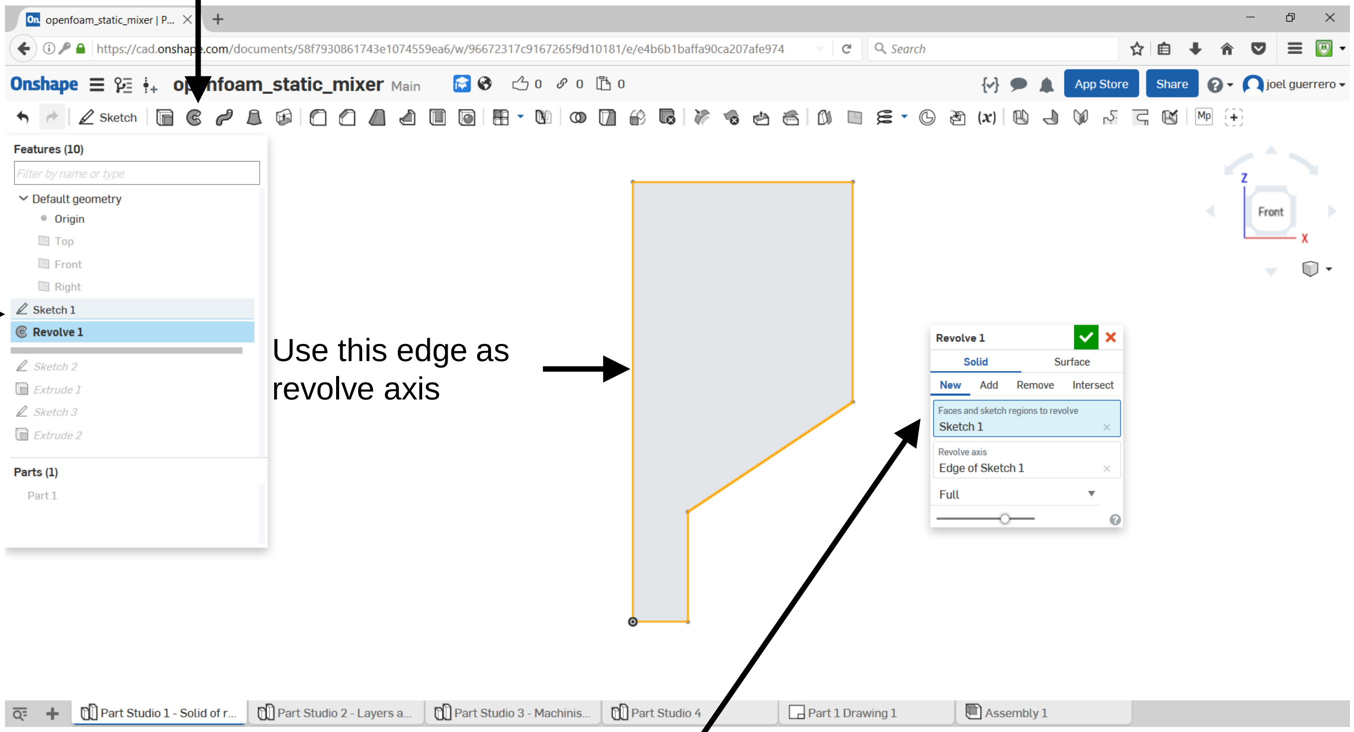
Introduction to solid modeling using Onshape

- Now using the previous sketch, generate a solid of revolution using the feature revolve.

Revolve feature

Sketch 1

Use this edge as revolve axis

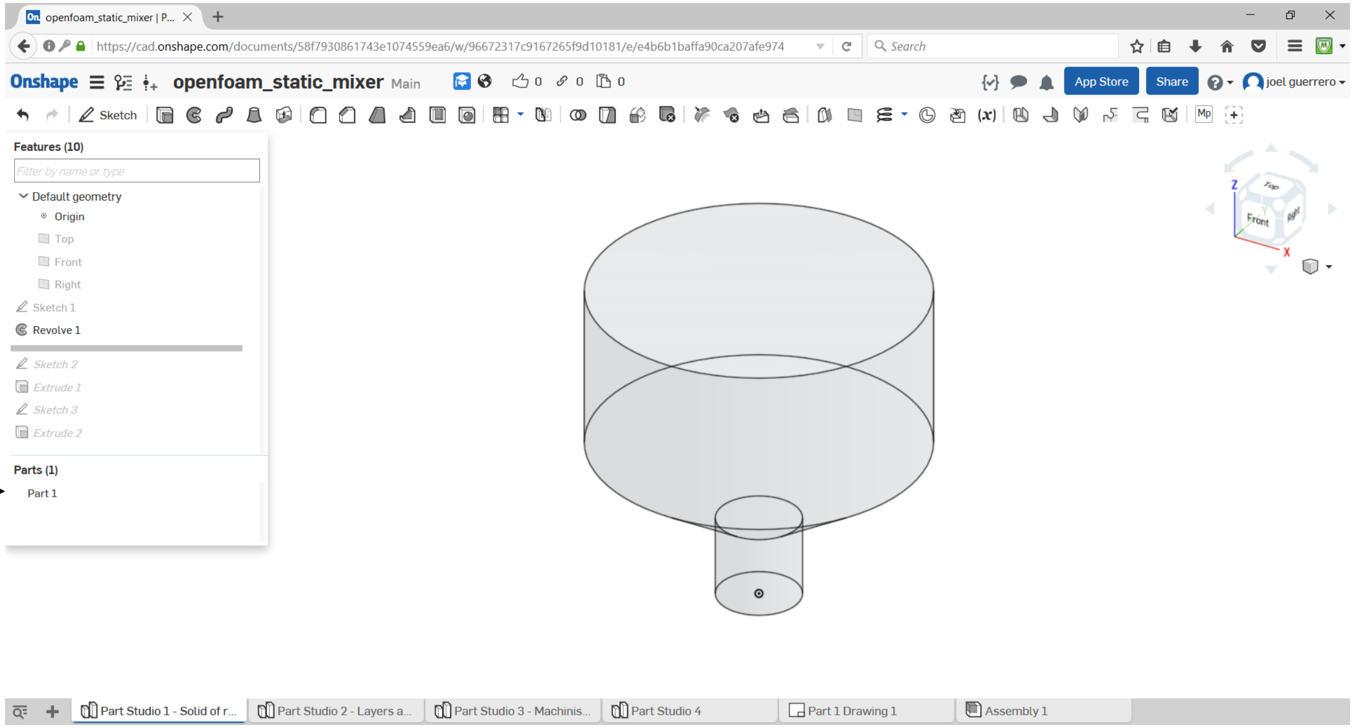


Select Sketch 1 as region to revolve

Introduction to solid modeling using Onshape

- At this point, you should have this solid.
- Notice that we changed the transparency.

Right click on Part 1 and select `Edit appearance` to change transparency and other properties

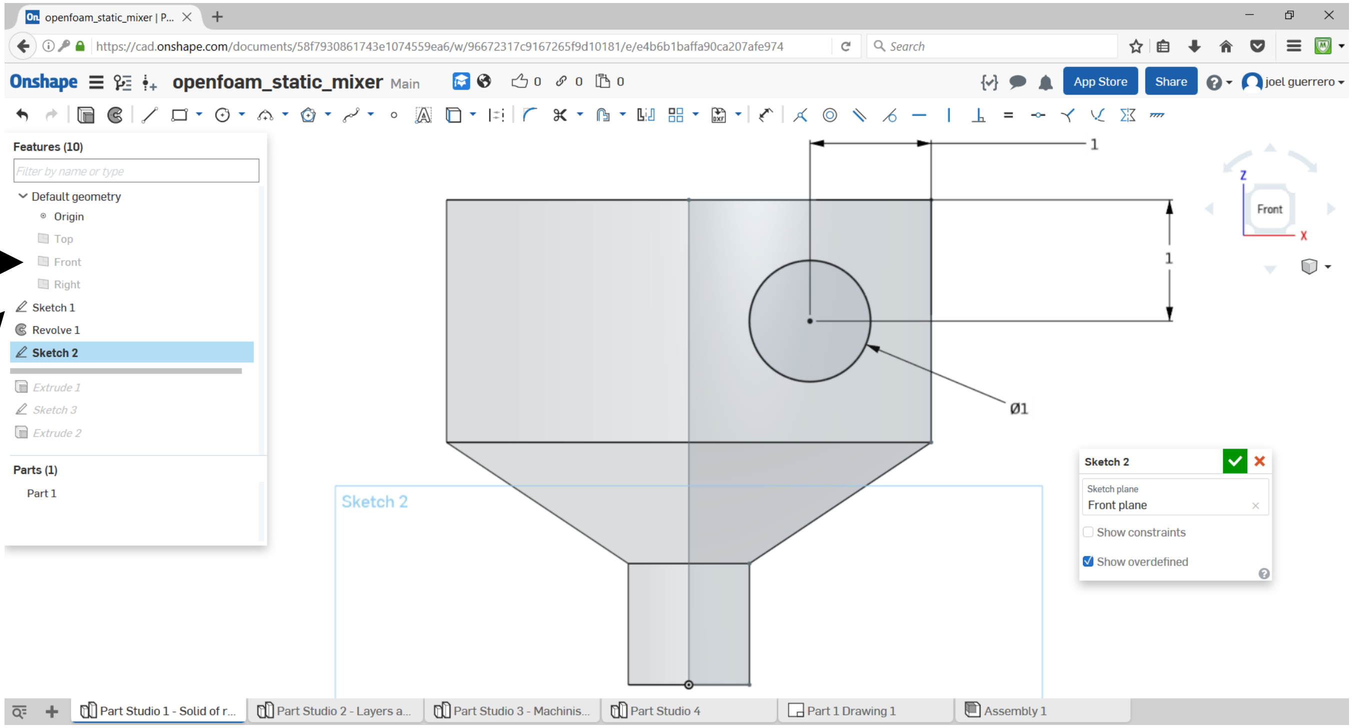


Introduction to solid modeling using Onshape

- Let us create the two extrusions, one at a time.
- Select the front plane and draw the following sketch using the dimensions illustrated.

Select the front plane to draw the sketch

Use sketch 1 as reference to set the dimensions

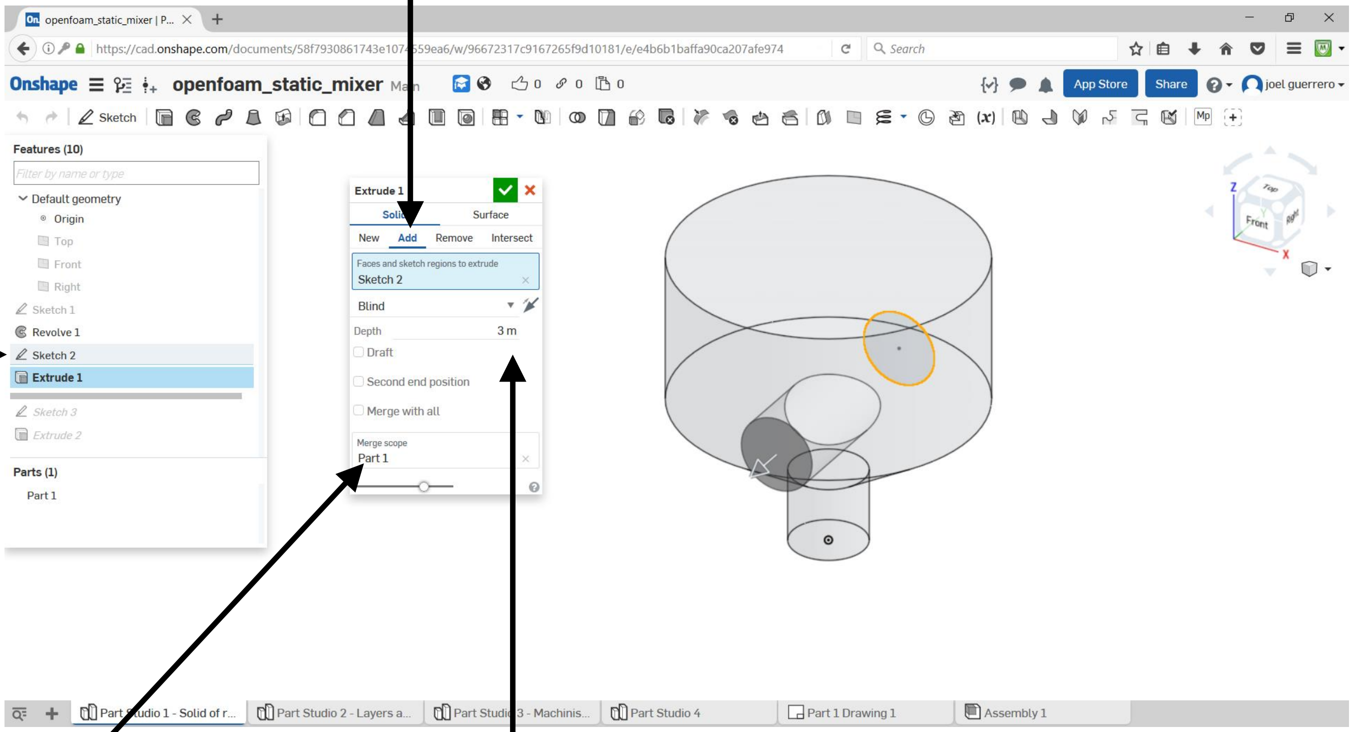


Introduction to solid modeling using Onshape

- Extrude the newly created sketch to obtain the following solid

Select Add to fuse the new solid with the previous one

Select this sketch as region to extrude



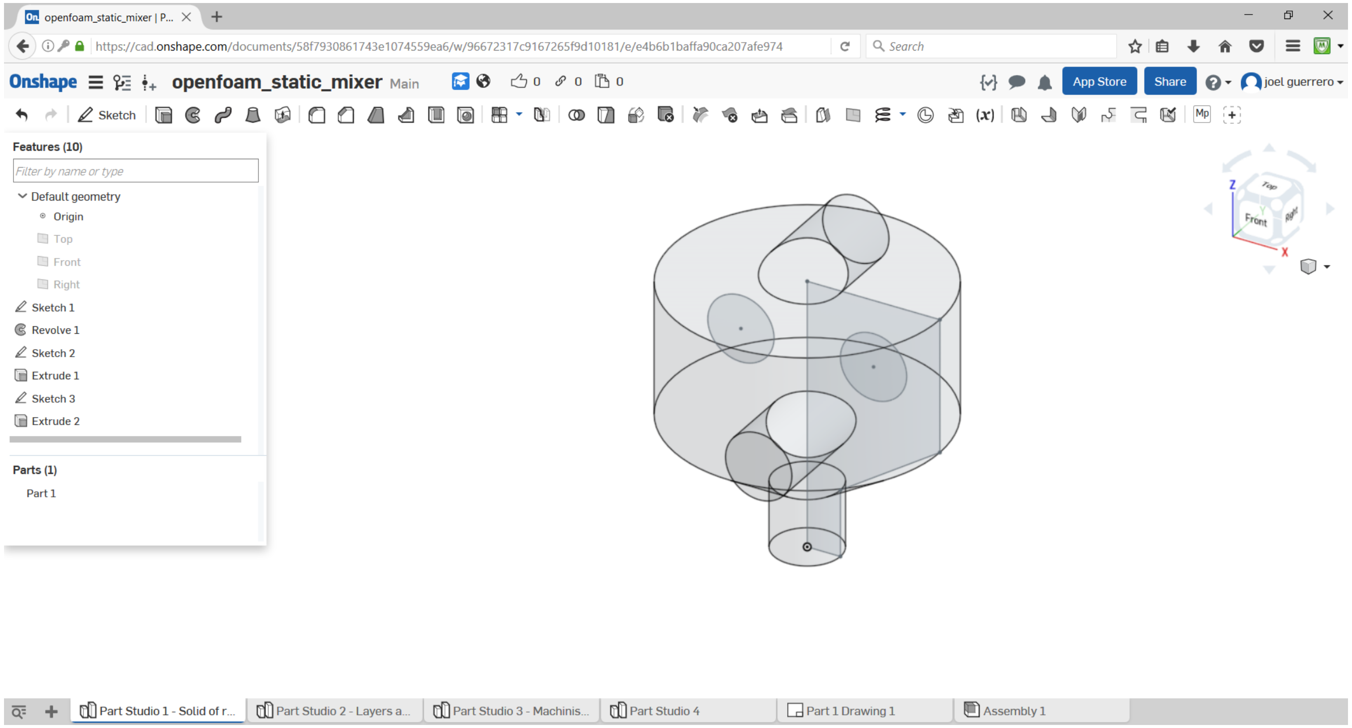
Merge with this part

Extrude a distance of
3 meters

Introduction to solid modeling using Onshape

- Do the same for the other extrusion. Remember to reverse the extrusion direction.
- At this point you should have the following solid model.

In this sample, we created the two extrusions using two different sketches, maybe it would have been better to mirror one sketch instead.

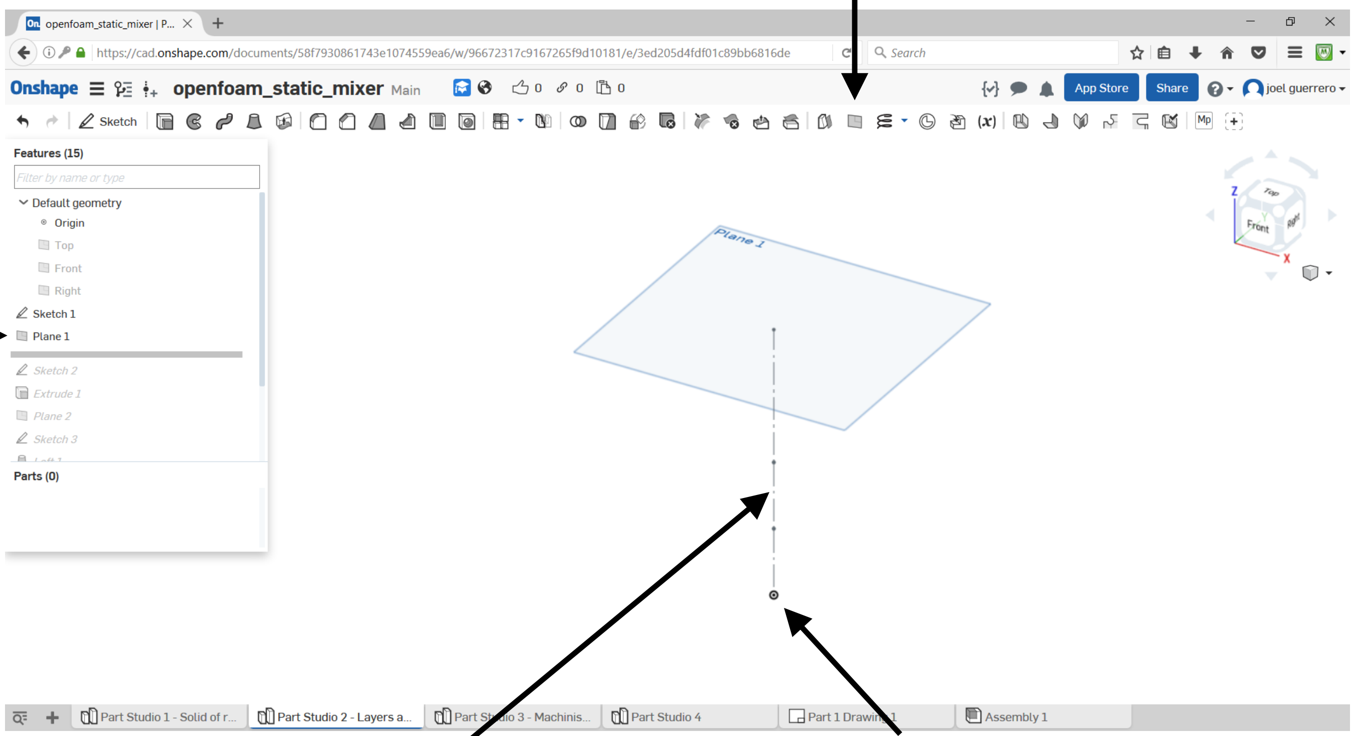


Introduction to solid modeling using Onshape

- Let us create the same solid but using layers instead.
- Create a new plane using the option point normal or offset.

Create new plane

New plane



If you use the offset option to generate the new plane, you will need to select the top plane and then offset it a distance of 4 meters

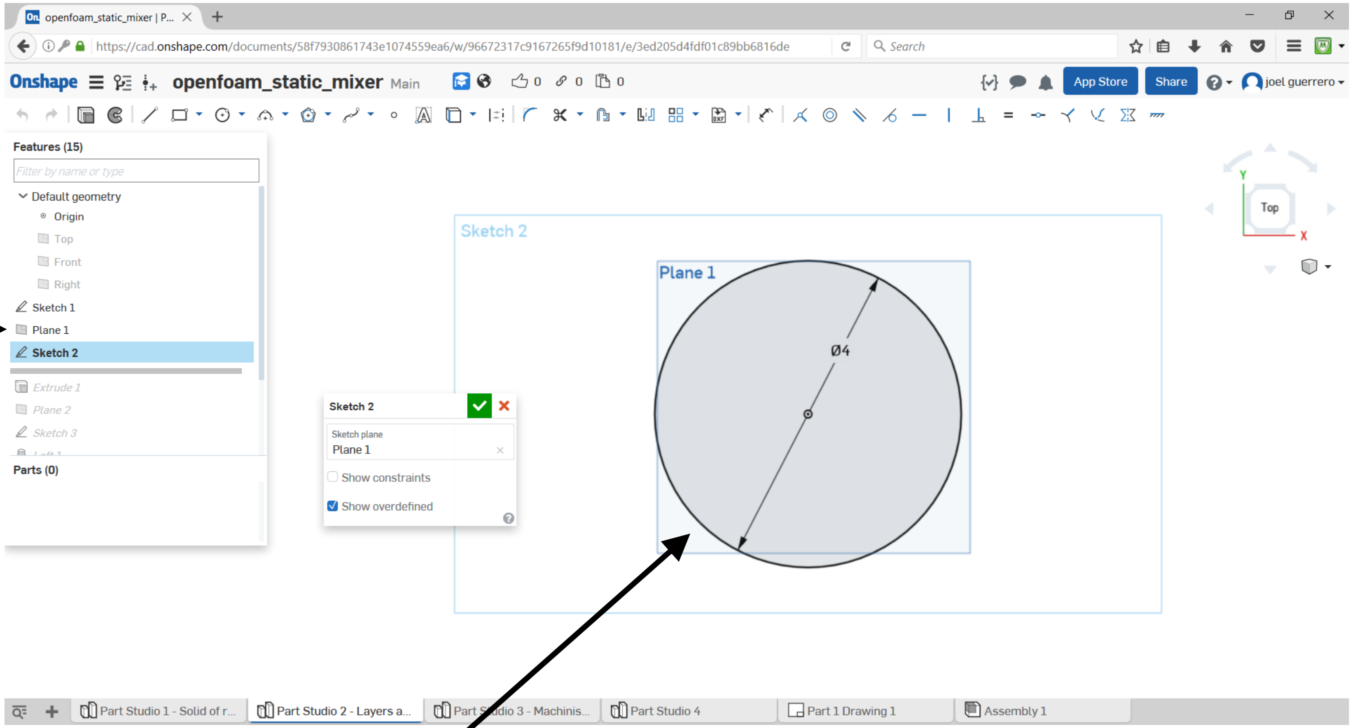
Origin

Construction axis sketched in the front plane (4 meters)

Introduction to solid modeling using Onshape

- Using the dimensions illustrated, create a circle in the new plane.

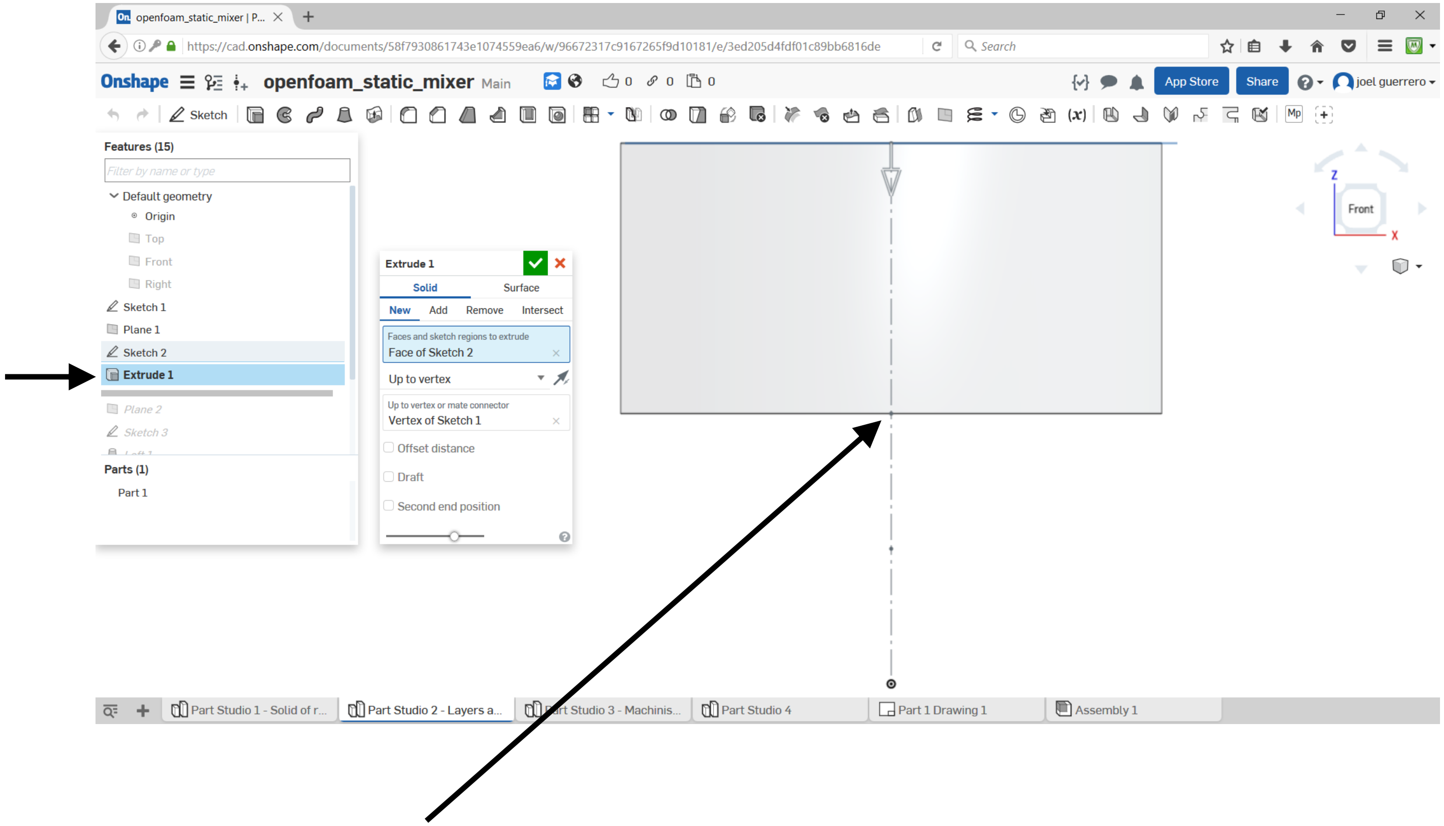
New plane



Sketch this circle

Introduction to solid modeling using Onshape

- Now extrude the circle to create the first layer.

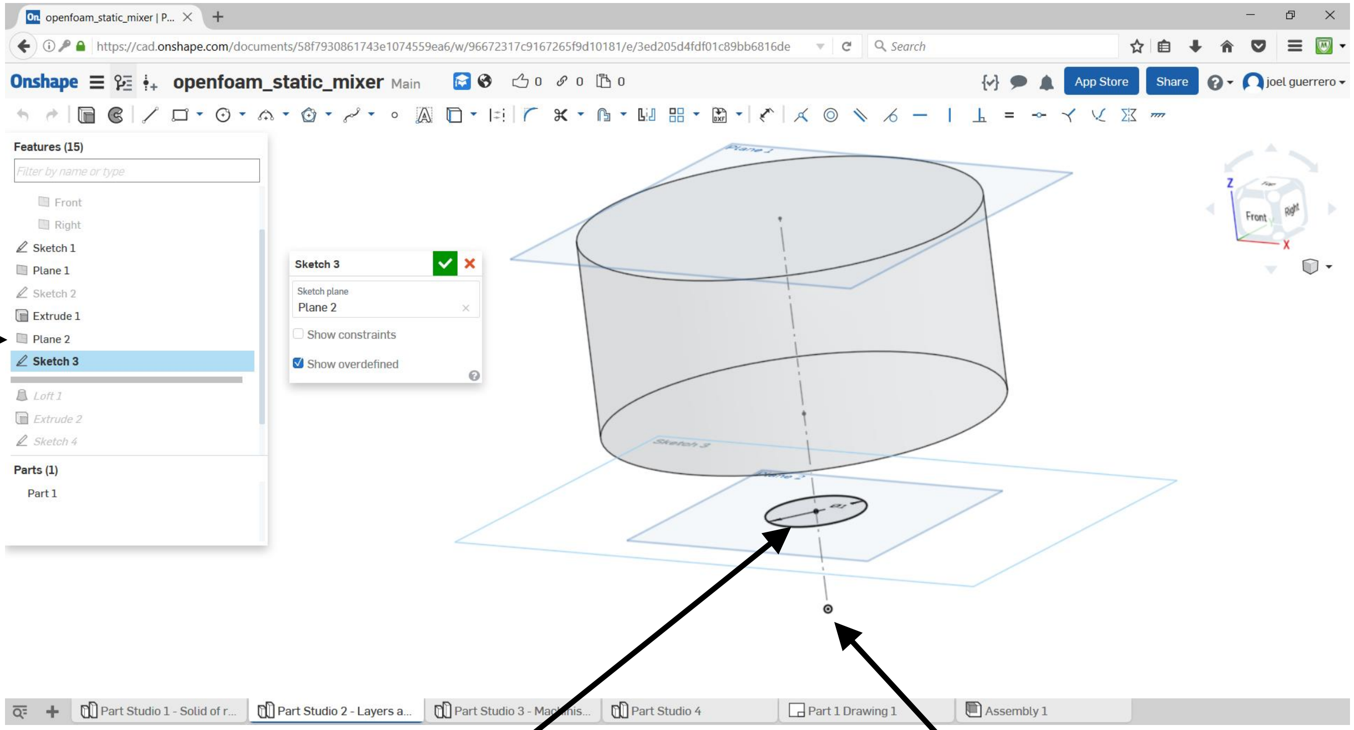


Generate the extrusion up to a reference vertex or a distance of 2 meters

Introduction to solid modeling using Onshape

- Now generate a new plane located one meter from the origin, as in the figure.
- Using the dimensions illustrated, draw a circle in the new plane.

New plane, you can create it using the option Point normal or offset.



Origin

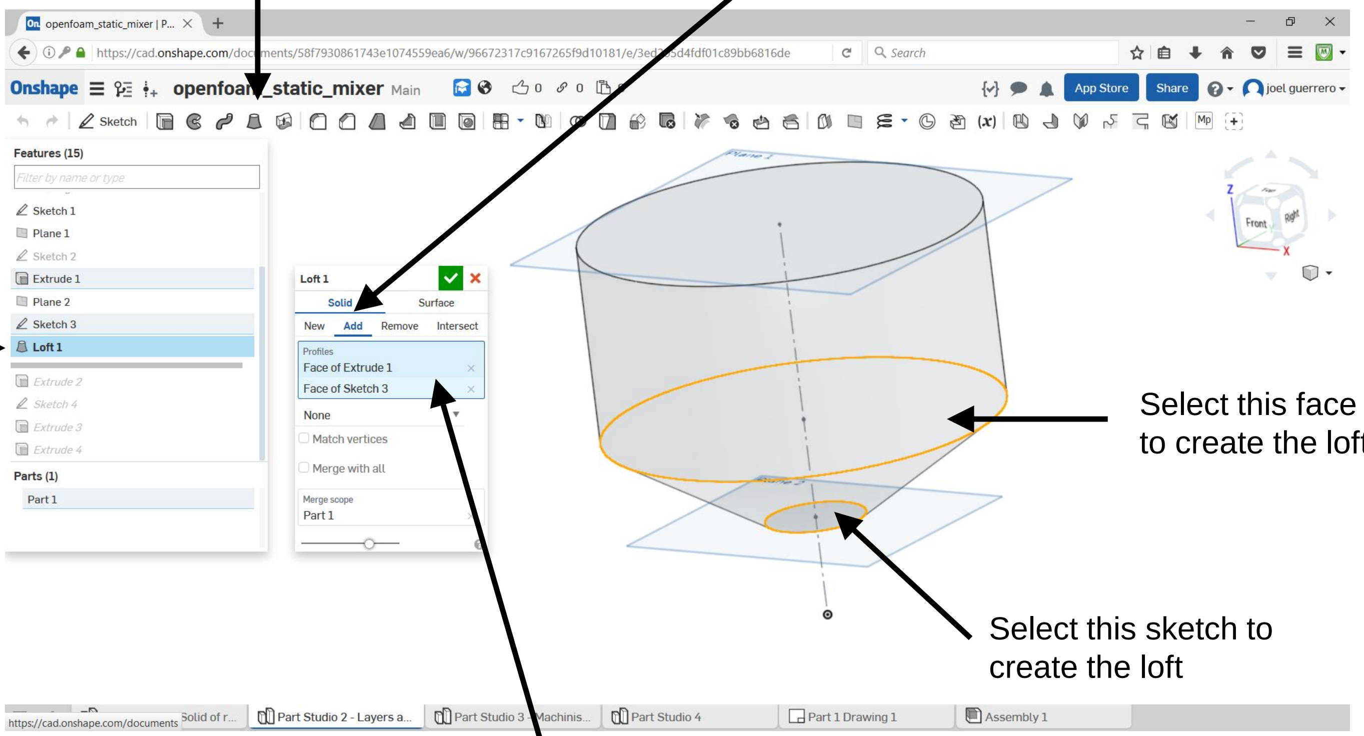
Sketch a circle with a diameter of 1 meter

Introduction to solid modeling using Onshape

- Let us create the second layer using a loft.
- The loft feature will create a solid between two (or more) profiles.

Loft

Select Add to fuse the new solid with the previous one



Select this face to create the loft

Select this sketch to create the loft

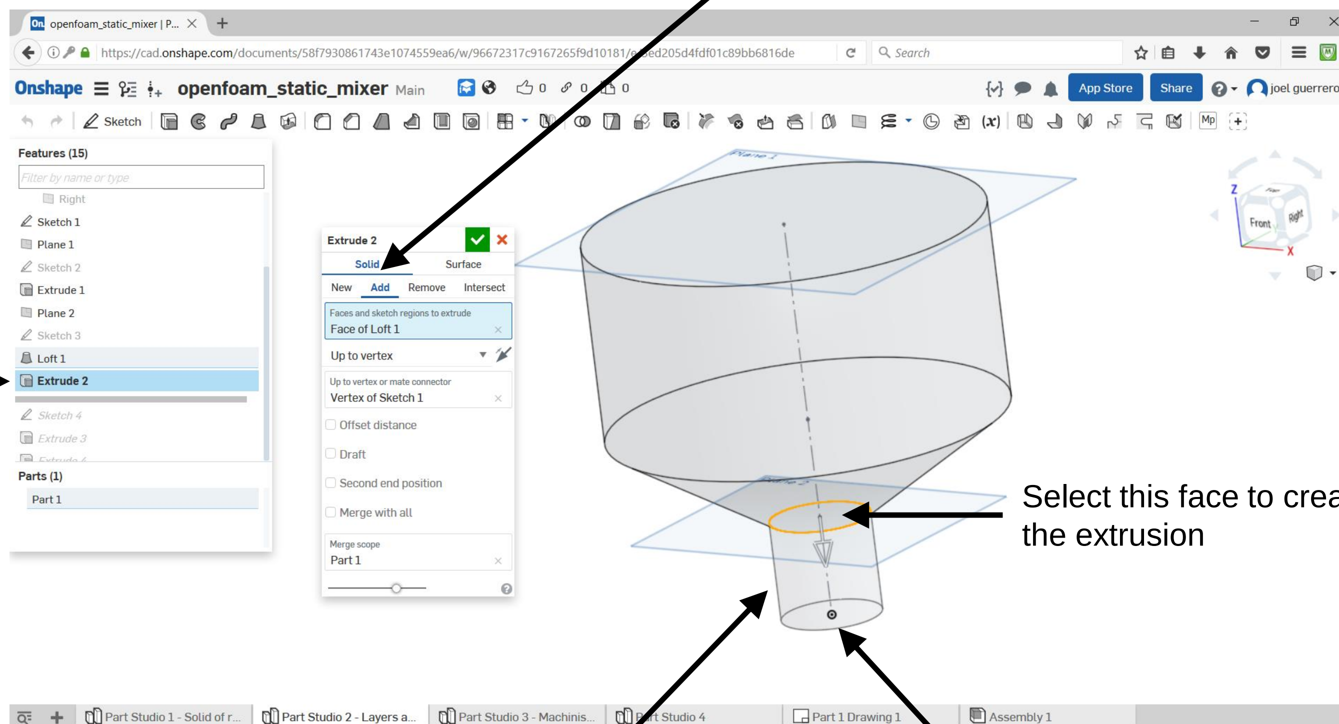
Selected faces

The faces need to be selected in order

Introduction to solid modeling using Onshape

- Let us create the final layer by extruding a face.

Select Add to fuse the new solid with the previous one



Select this face to create the extrusion

Origin

Create the extrusion up to the origin or a distance of 1 meter

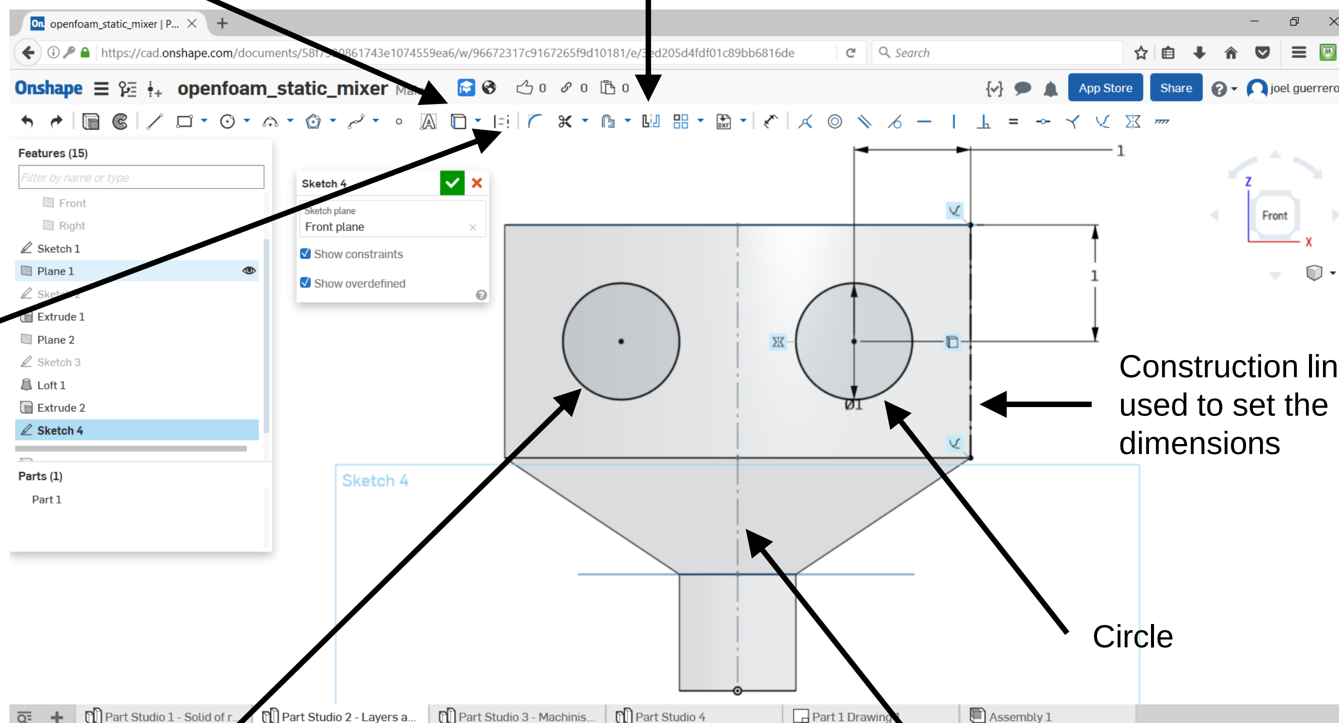
Introduction to solid modeling using Onshape

- Now that we have the main body, let us create the final two extrusions.
- Sketch the two circles as illustrated, notice that we are mirroring the right circle.

Use this feature to extract the reference line

Mirror feature

Convert to a construction line



Construction line used to set the dimensions

Circle

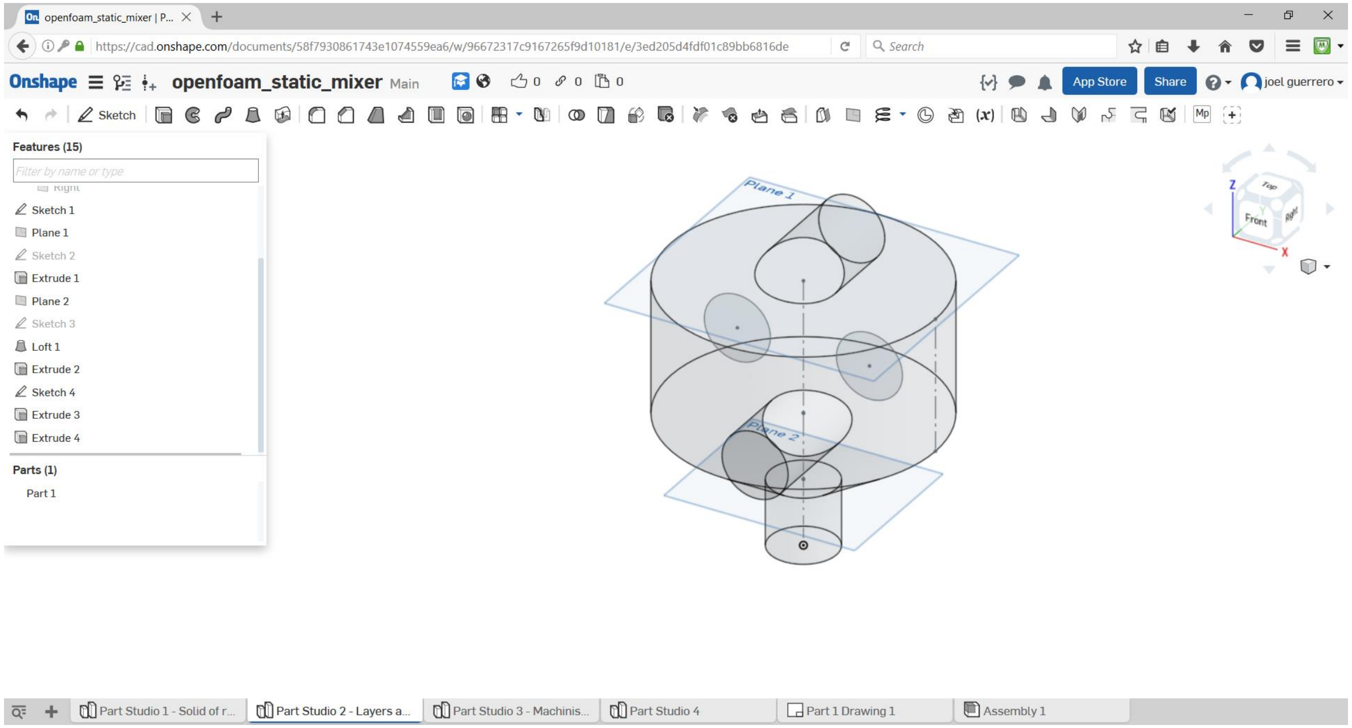
When extruding the two circles, remember to use the option add material to fuse the bodies. Extrude the circles a distance of 3 meters.

Mirrored circle

Construction axis to be used as a mirror axis

Introduction to solid modeling using Onshape

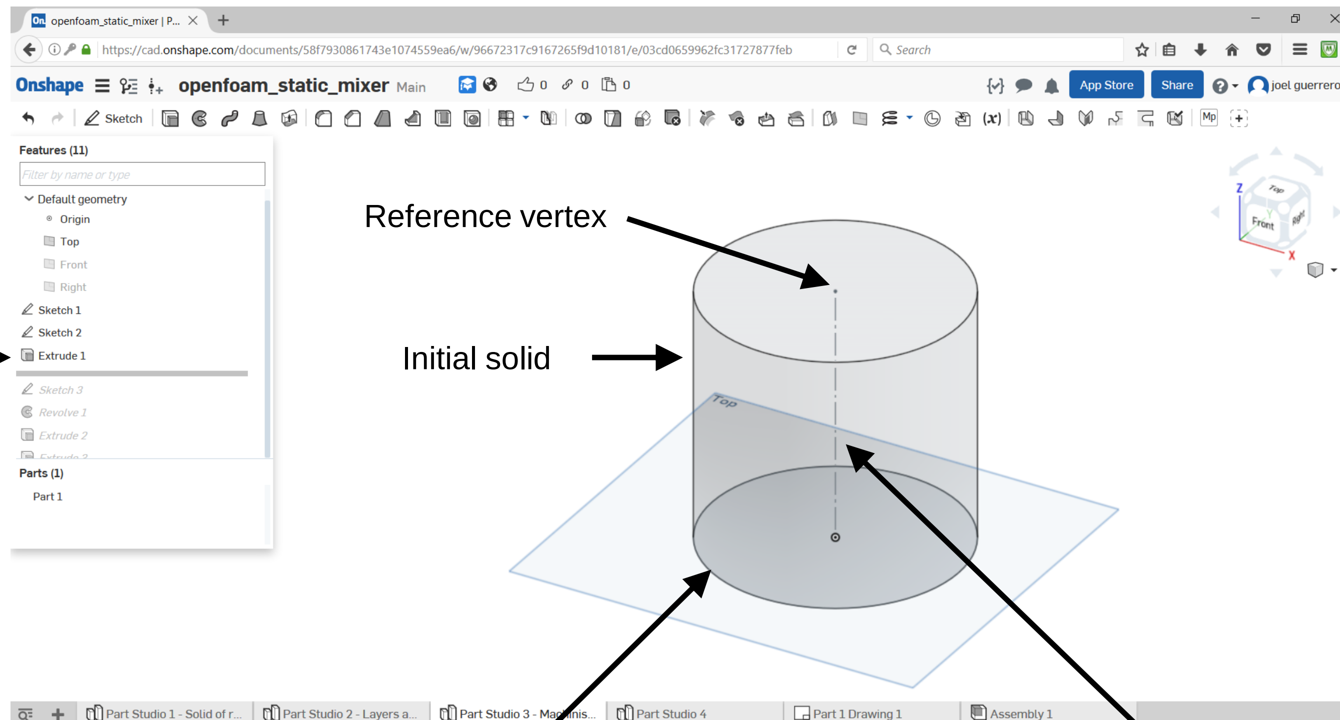
- At this point, you should have the following solid model.
- As you can see, is exactly the same result but we used more operations.



Introduction to solid modeling using Onshape

- Let use the machinist approach. This approach is useful is you are thinking on how you will manufacture this part.
- By starting from a uniform solid we start to remove material.

Extrude the circle a distance of 4 meters, or up to a reference vertex. You can also sweep the circle along the construction line

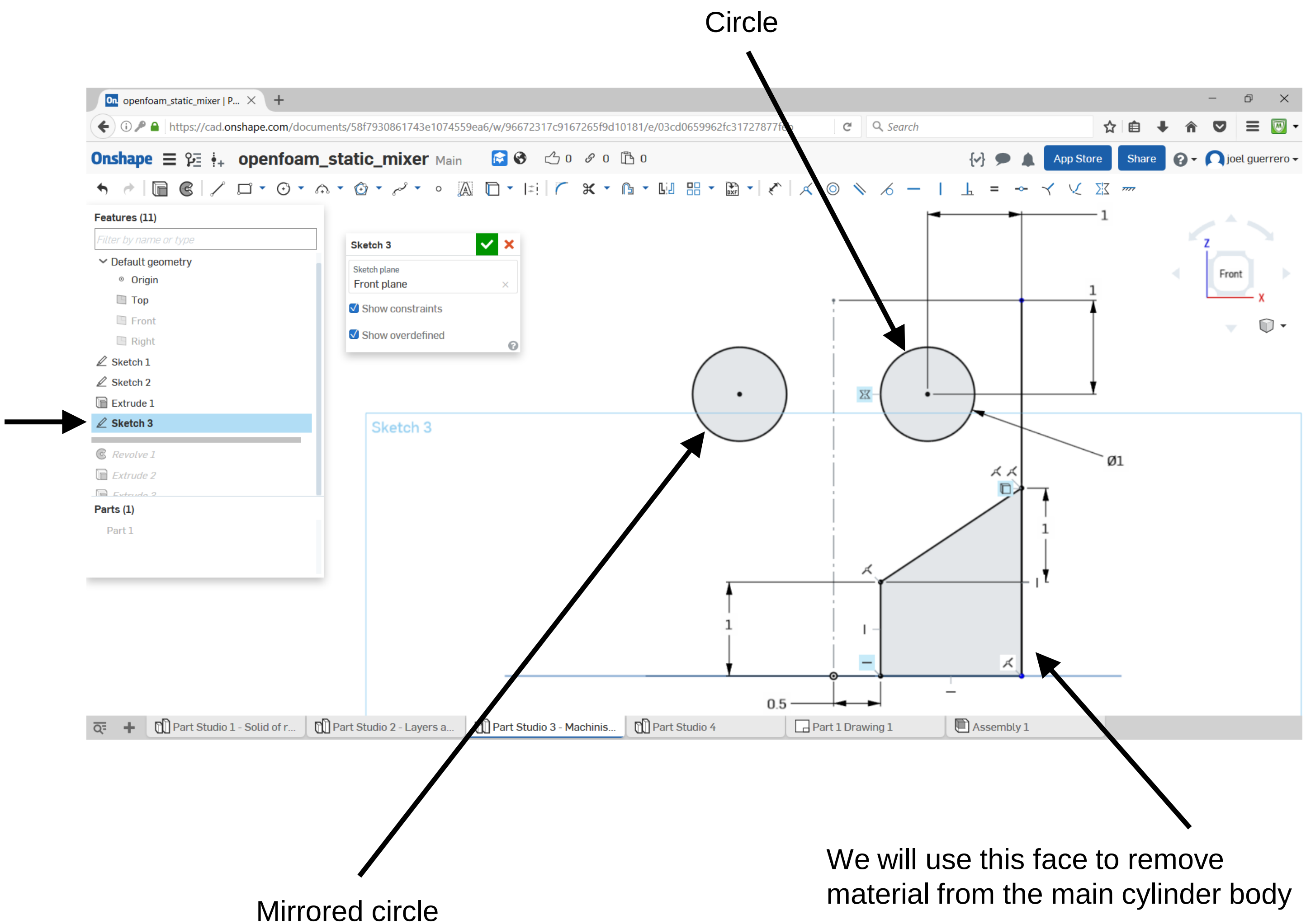


Sketch a circle with a diameter of 4 meters in the bottom plane (sketch 2 in the features list)

Sketch this construction line in the front plane (sketch 1 in the Features list)

Introduction to solid modeling using Onshape

- Let us draw the following sketches in the front plane.
- To set the dimensions, you will need to create reference lines/points.

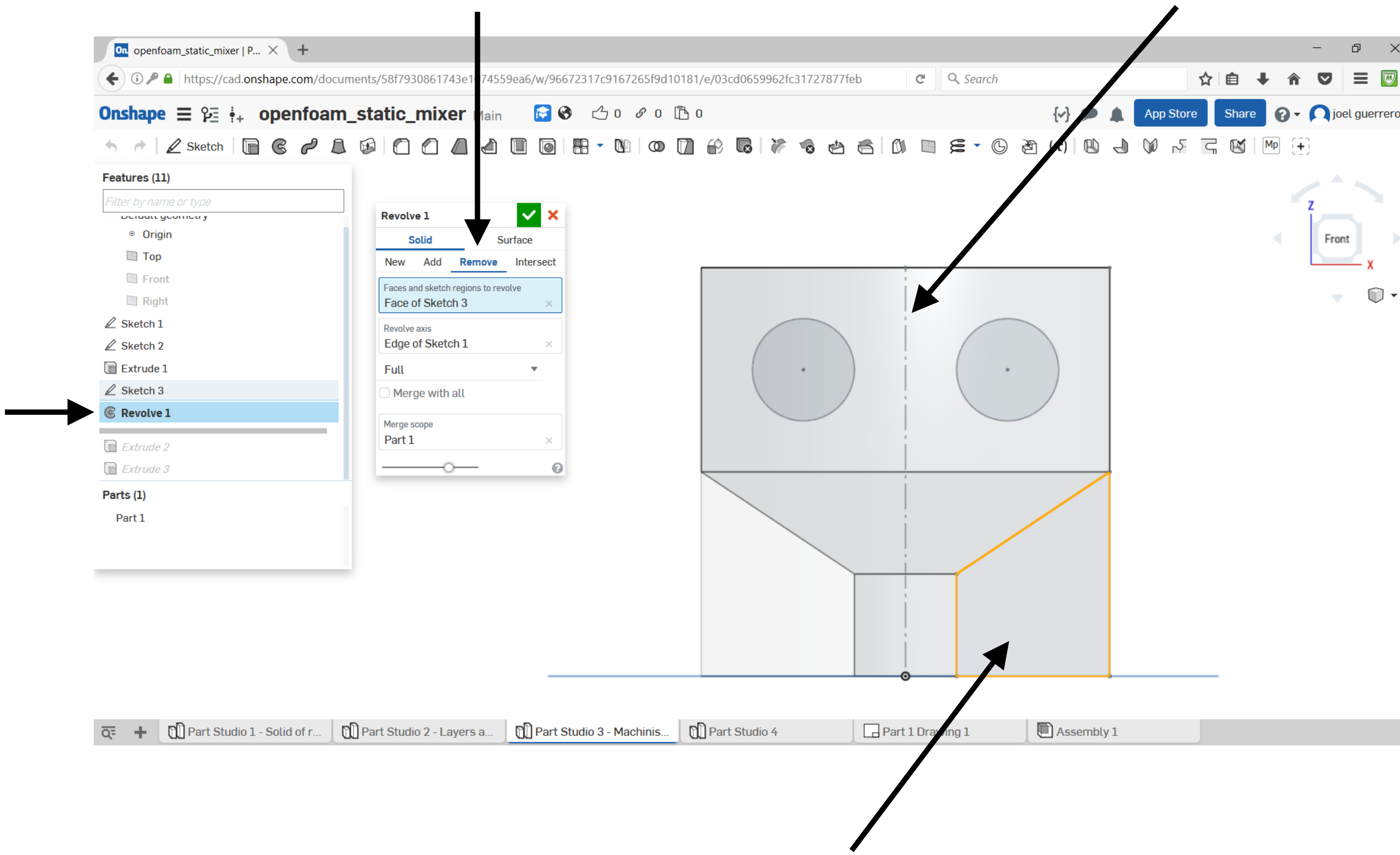


Introduction to solid modeling using Onshape

- Let us remove some material from the main cylinder part.
- We will use the revolve feature with the remove option.

Use the option remove in order to remove the material

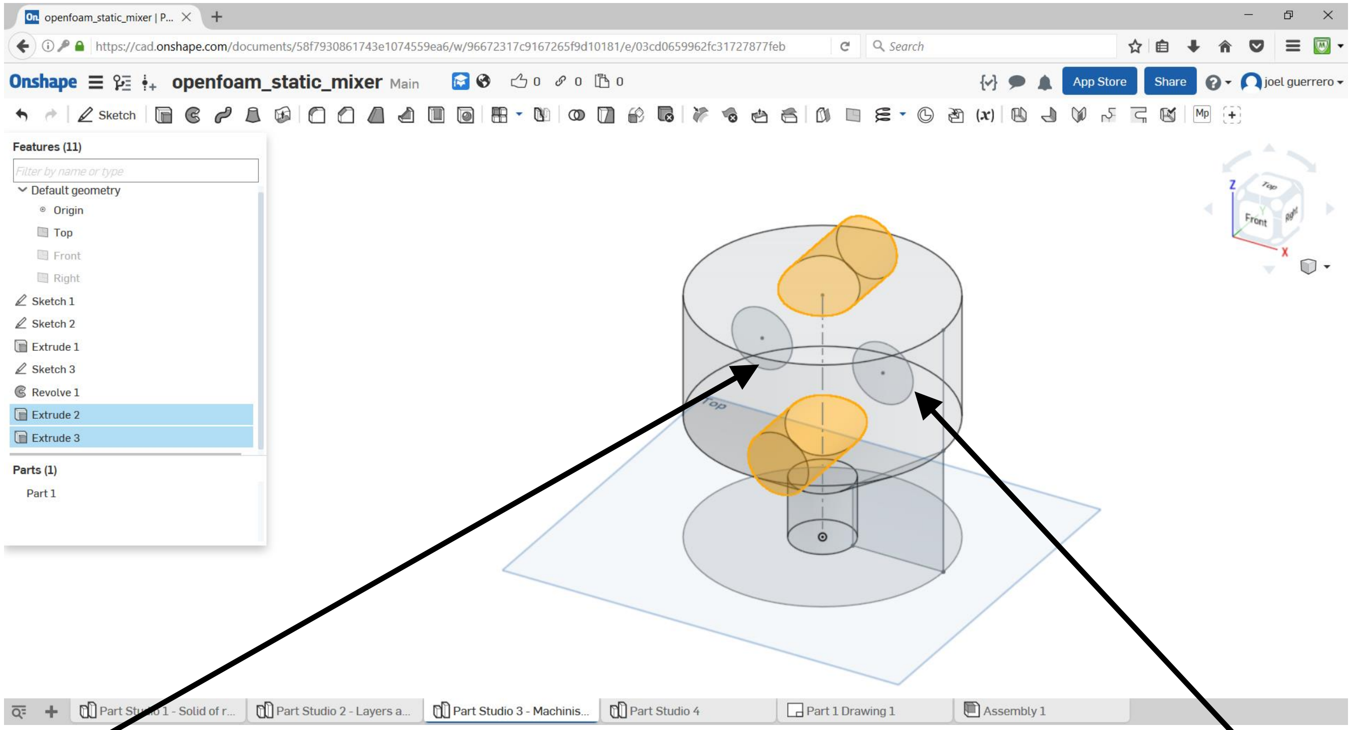
Revolve axis



Face to revolve.
Notice that this face belongs to sketch 3

Introduction to solid modeling using Onshape

- Finally, let us create the two extrusions.
- Remember to use the option add material two fuse the bodies.

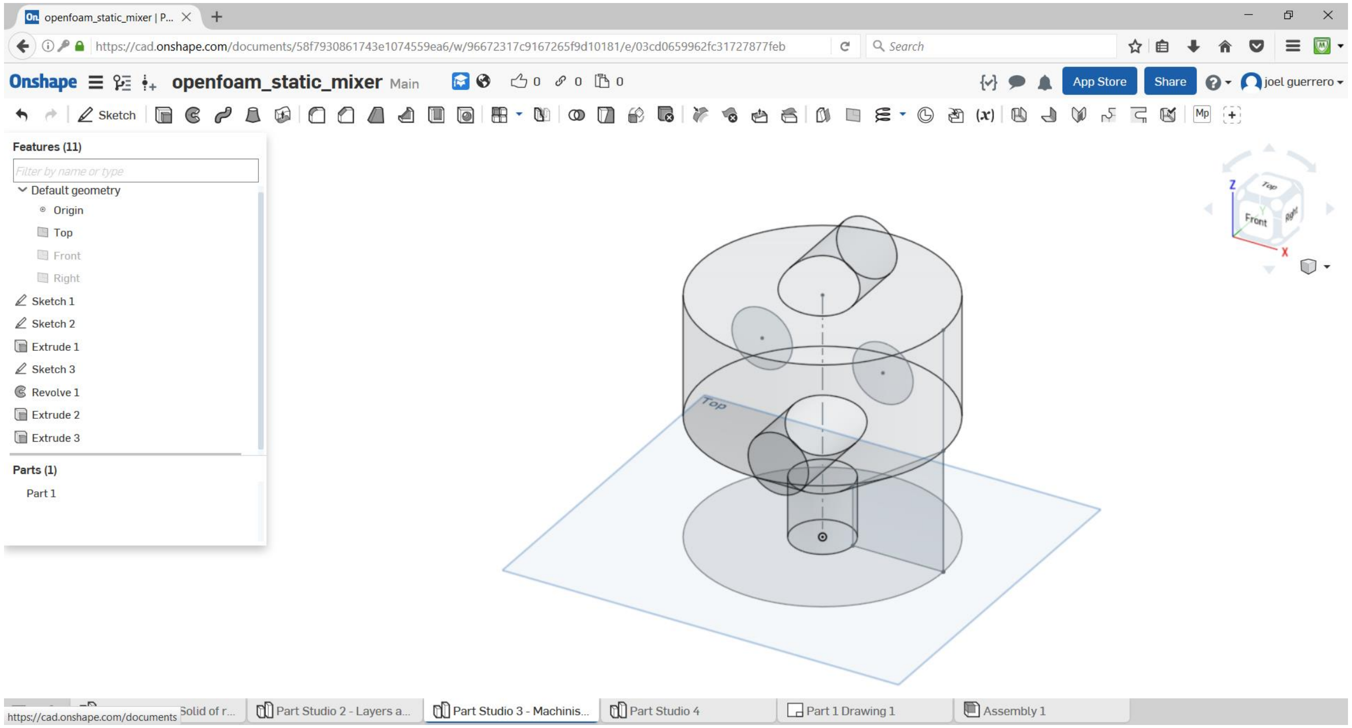


Select this face and extrude a distance of 3 meters

Select this face and extrude a distance of 3 meters

Introduction to solid modeling using Onshape

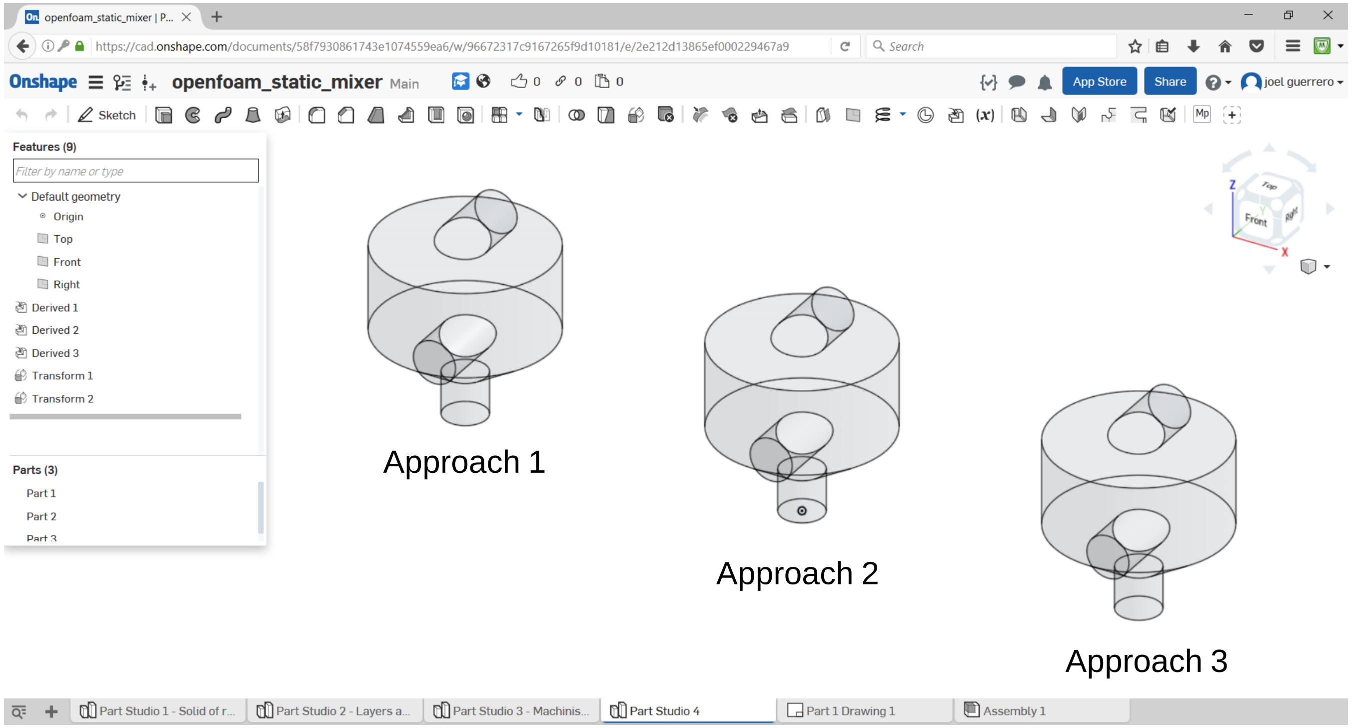
- At this point, you should have the following solid model.
- Again, we obtained the same result.



- If you are working with CNC lathes or additive manufacturing, this approach can be used to select the best way to remove (or add) material.

Introduction to solid modeling using Onshape

- We have just seen design intend in action.
- Design intent is just a strategy where your design is defined in such a way that changes produce desired, predictable results.
- As you can see, we can arrive to the same results in many ways. But it is better to work in such a way to get the results more efficient.



Introduction to solid modeling using Onshape

- Parametric modeling and feature-based modeling are two of the most powerful tools available in any CAD/solid modeling application.
- They are crucial components in the design experience, especially when dealing with design intent.
- Experimenting with dimension schemes is one of the best ways to improve your understanding of design intent.
- Design intent is incredibly powerful in a parametric CAD environment. But in case you are worried about fully understanding how design intent impacts your current models, we assure you that it comes naturally as you create designs.
- Just be sure to give some thoughts about an approach and strategy for creating your models, as well as the dimension schemes and relationships that will be applied.
- Once you have done that, your modeling skills will take off and you will find new confidence in how well you can model.
- Finally, feel free to visit our youtube channel where you will find a few solid modeling videos:

https://www.youtube.com/channel/UCNNBm3KxVS1rGeCVUU1p61g