Module 2

Solid modeling

Roadmap

1. **Solid modeling preliminaries and introduction to Onshape**

Solid modeling – Preliminaries

- There is no wrong or right way when doing solid modeling for CFD. The only rule you should keep in mind is that by the end of the day you should get a smooth, clean, and watertight geometry.
- The quality of the mesh and hence of the solution, greatly depends on the geometry. So always do your best when creating the geometry.
- During the solid modeling sessions, we are going to show you how to get started with the geometry generation tools. The rest is on you.
- The best way to learn how to use these tools is by doing.
- Think about a strategy to create your design, we call this design intent.
    - Should I extrude or revolve the sketch?
    - How should I set the dimensions?
    - Should I use multiple parts?
    - Do I need to parametrize my design?
- We are going to work with design intent during the hands-on sessions.

Solid modeling – Preliminaries

A few open-source/free CAD/solid modeling applications

- **Onshape:** it is a full cloud-based 3D CAD system. It runs in a web browser and on any device with a working web browser. It has the same capabilities of commercial CAD systems. You can create a free account and start to use it immediately. **This is the tool we will use during this training.** https://www.onshape.com/

- **Salome:** it is a history-based CAD tool (but not a 100% parametric). It has quite extensive capabilities for creation and manipulation of solid geometries. Its capabilities mirror those of commercial CAD systems. http://www.salome-platform.org/

- **Autodesk 360 fusion:** it is a full desktop 3D CAD system. It runs in windows and Mac. It is free for student, enthusiasts, hobbyists, and startups. http://www.autodesk.com/products/fusion-360/overview

- **Free-CAD:** it is a history-based CAD tool (parametric design). Light CAD software, good for not very complicated mechanical designs. http://sourceforge.net/apps/mediawiki/free-cad/

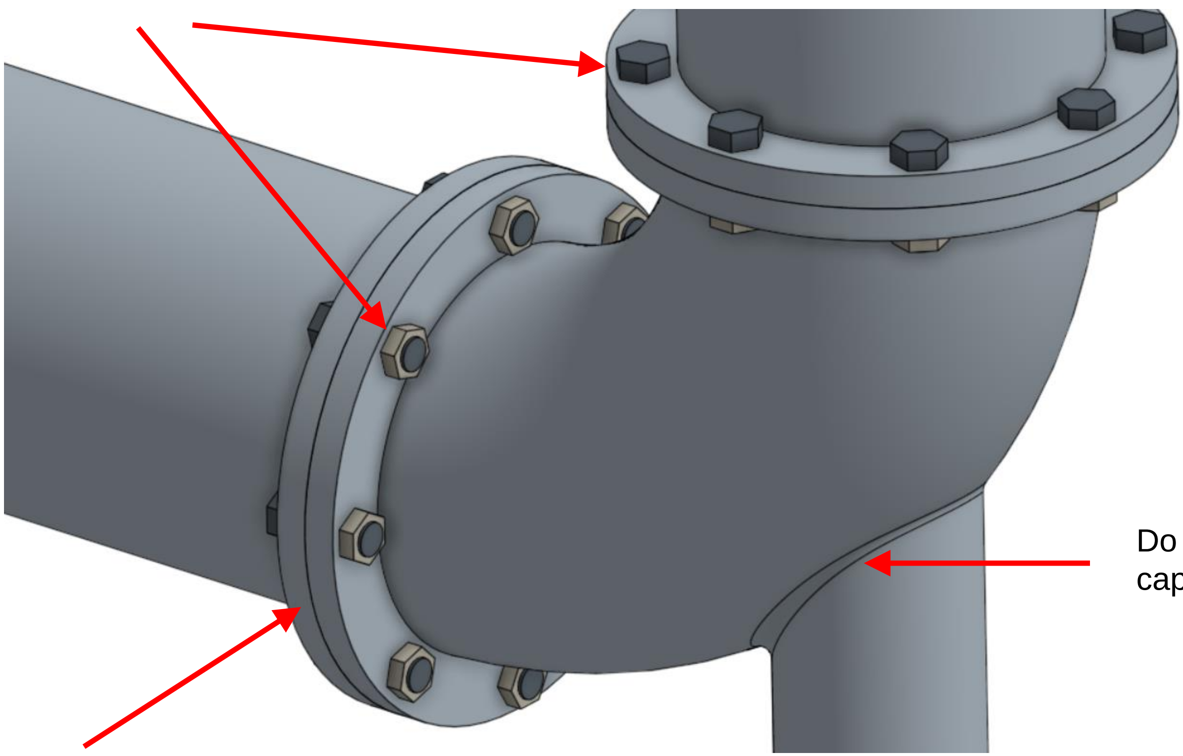
- **OpenVSP:** it is a parametric aircraft geometry tool. It allows users to create a 3D model of an aircraft defined by common engineering parameters. http://www.openvsp.org/

- **Openscad:** it is a 3D programming modeling tool. It reads in a script file that describes the object and renders the 3D model from this script file. http://www.openscad.org/

Geometry defeaturing

- Many times, it is not necessary to model all the details of the geometry. In these cases, you should consider simplifying the geometry (geometry defeaturing).
- **Geometry defeaturing** can save you a lot of time when generating the mesh. So be smart, and use it whenever is possible.

Are the nuts and bolts necessary in my simulation?



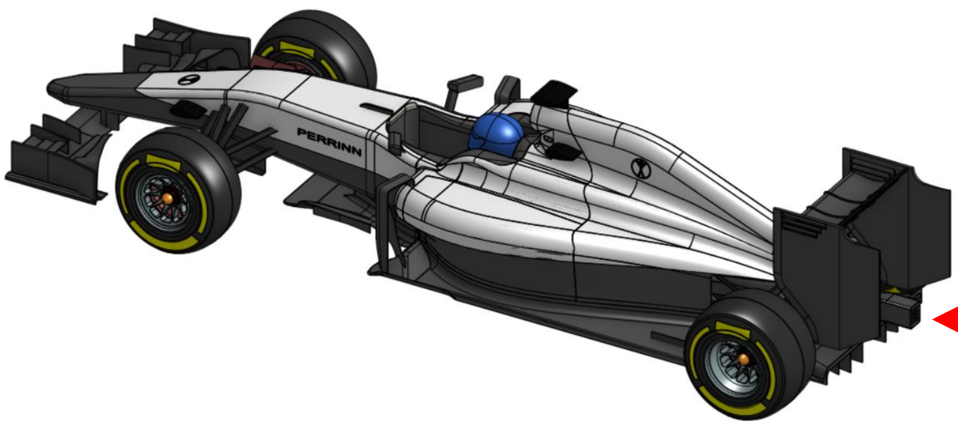
Do we really need to capture the fillet details?

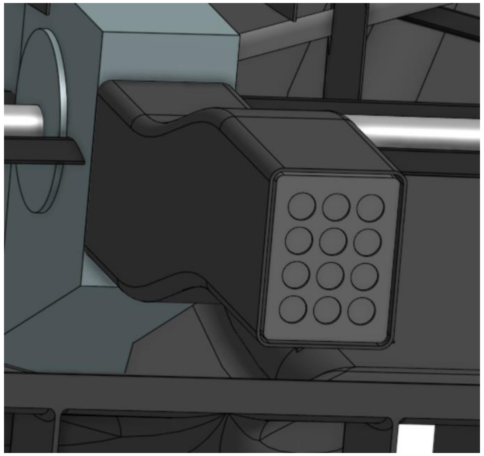
Do we need to model the flange?

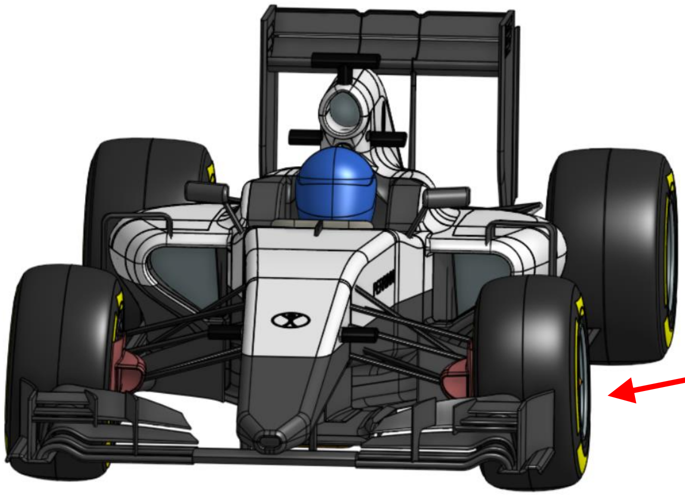
Solid modeling – Preliminaries

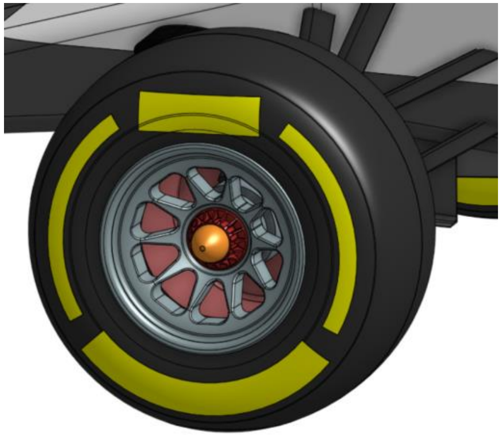
Geometry defeaturing

- Would you use all these geometry details for a CFD simulations?







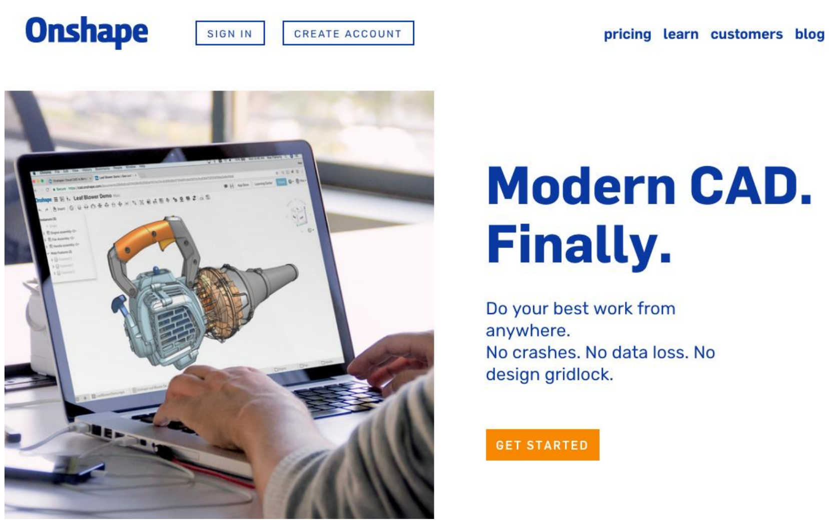


Solid modeling using Onshape

Onshape preliminaries

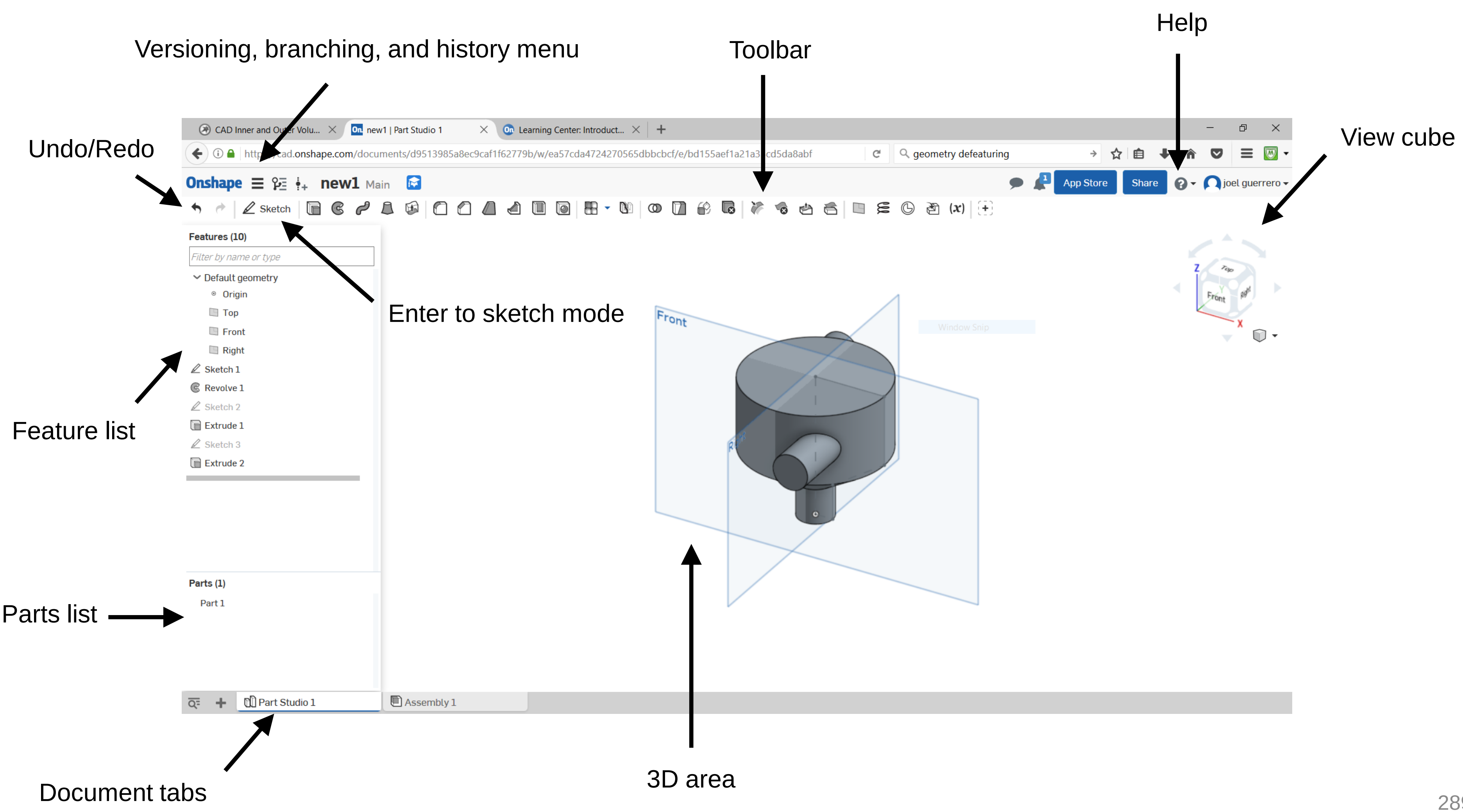
Solid modeling using Onshape

- 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 and personal use.
- To start using Onshape register at: https://www.onshape.com/



Solid modeling using Onshape

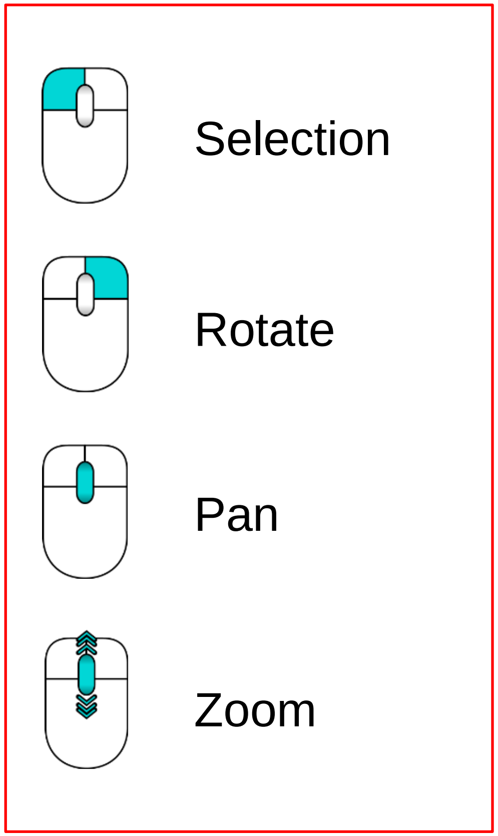
- Even if you have not used a CAD software before, you will find the GUI easy to use.
- You will notice that there is no save button because everything you do is automatically saved.

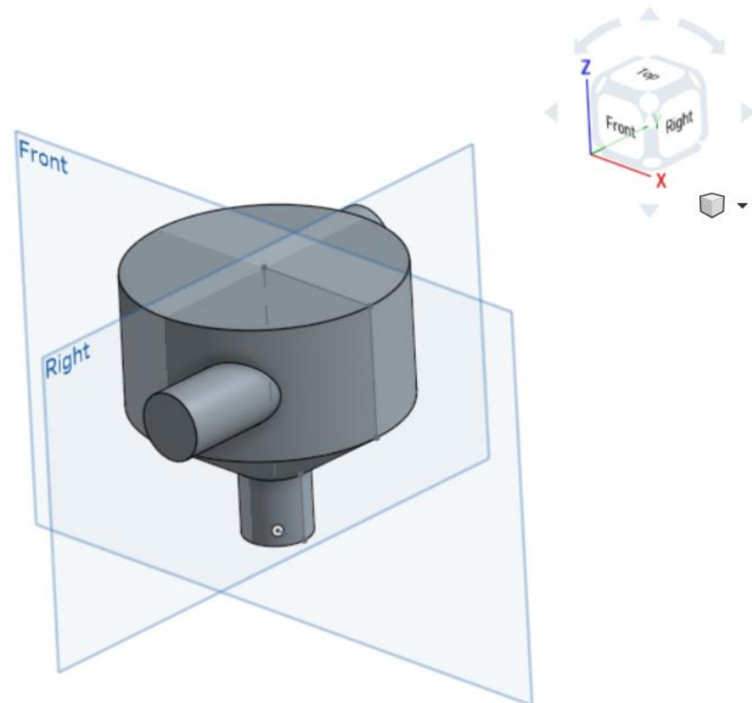


Solid modeling using Onshape

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

Mouse interaction in the
3D viewer



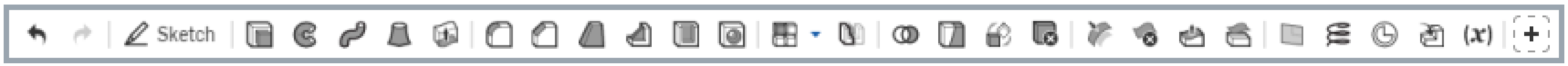


- To deselect click in an empty region in the 3D area

Solid modeling using Onshape

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

Feature toolbar:



Sketch toolbar:

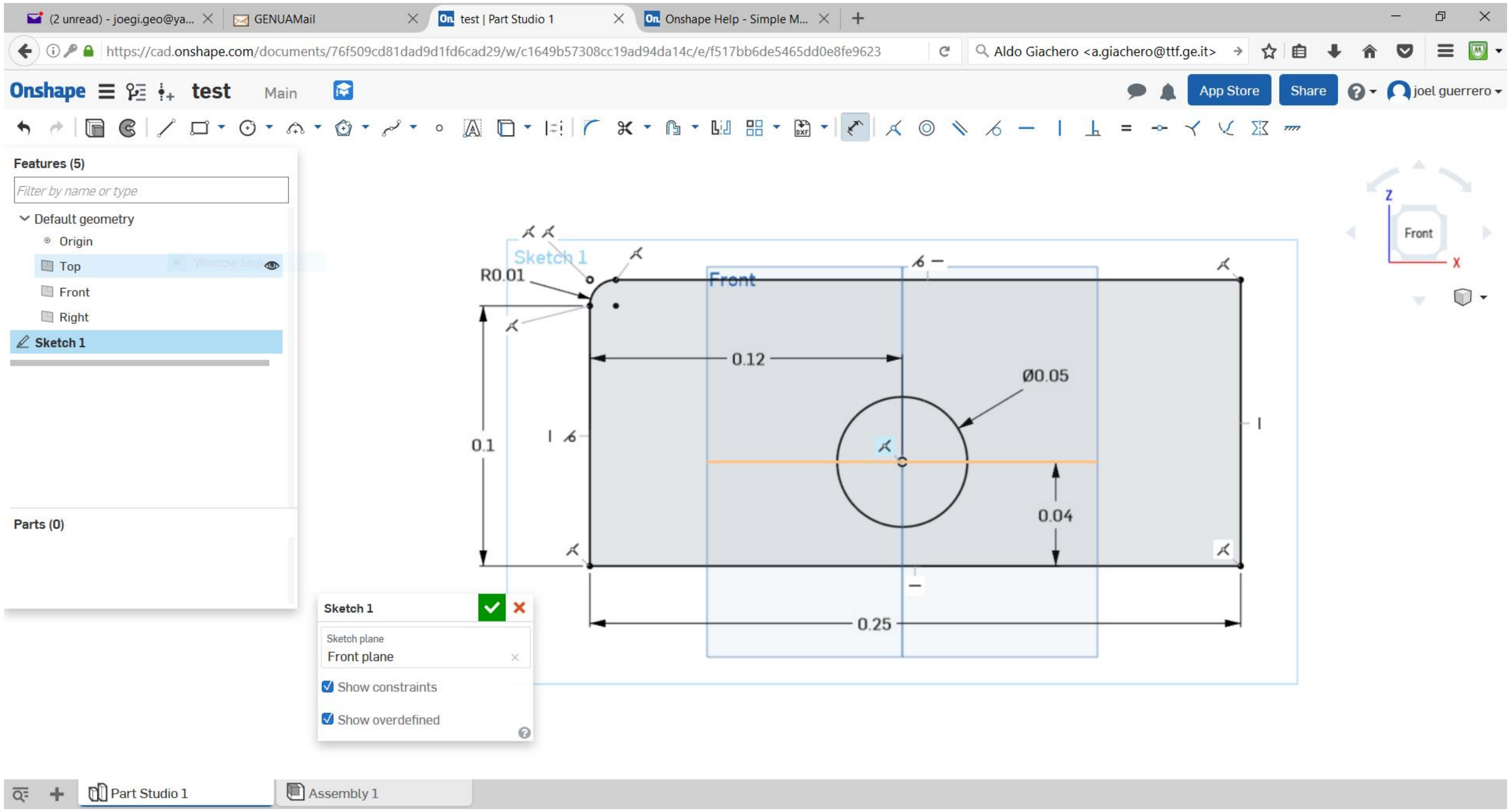
Assemblies toolbar:

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.

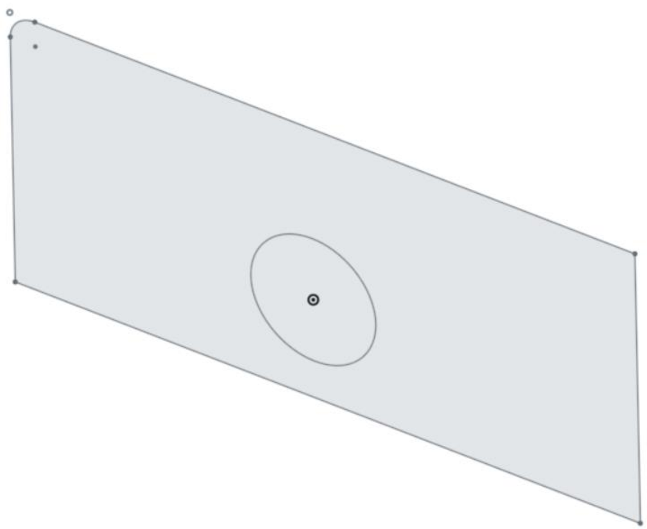
Solid modeling using Onshape

- 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.

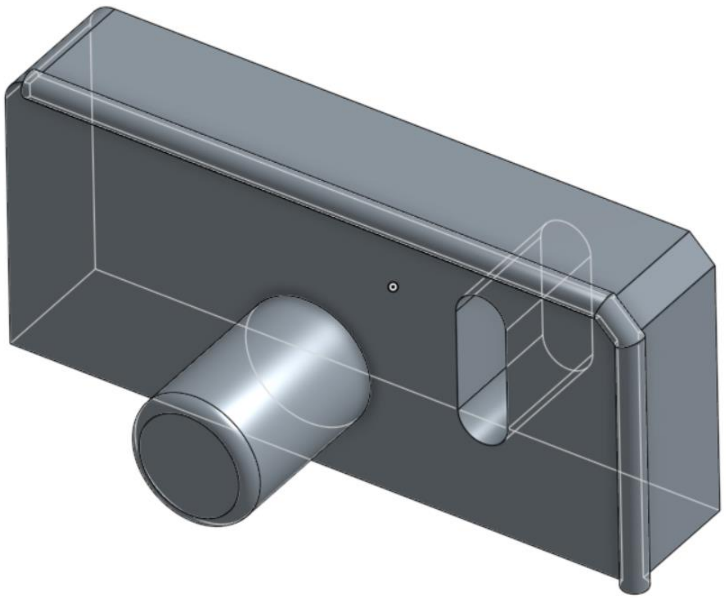


Solid modeling using Onshape

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



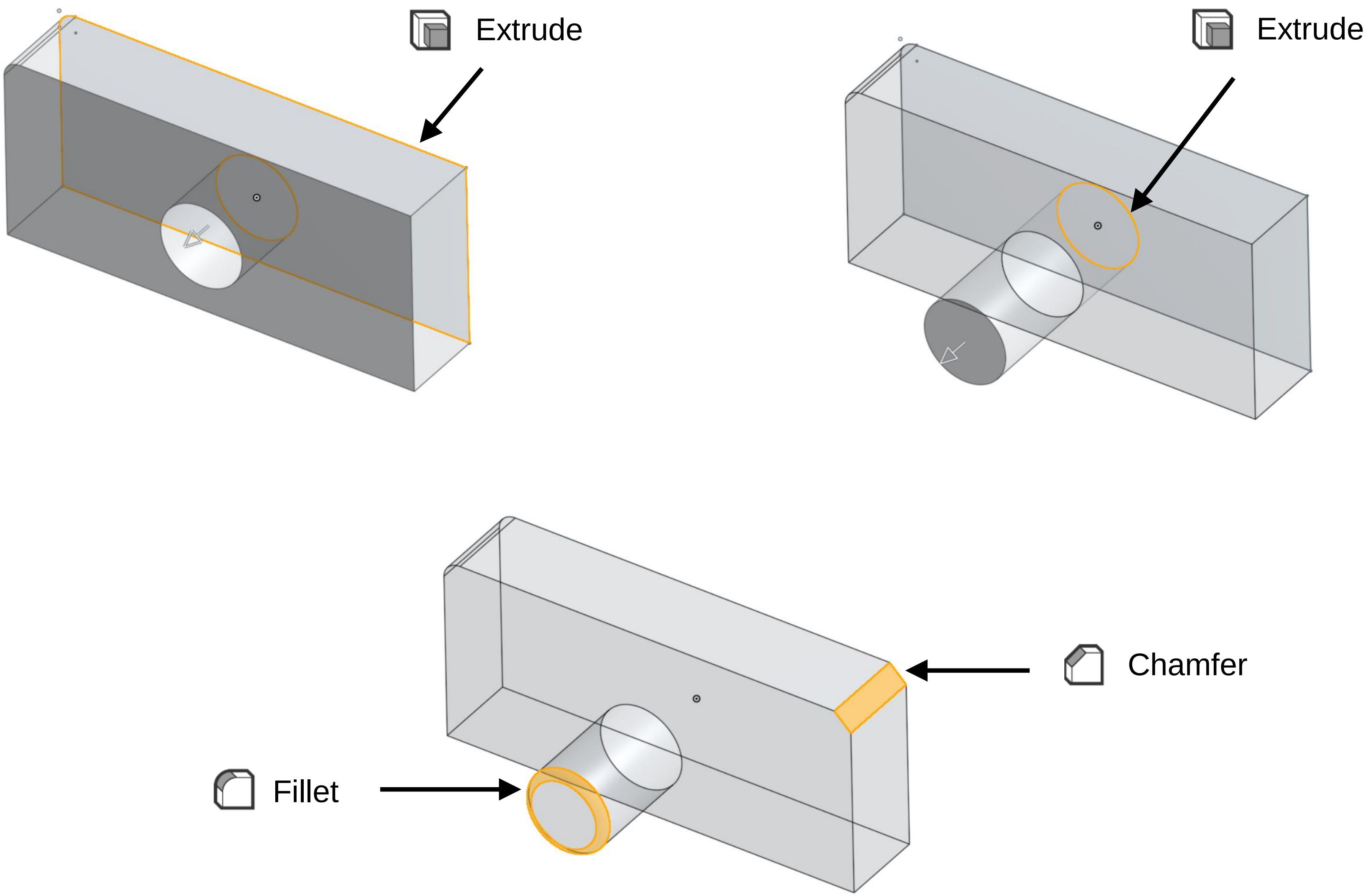
Starting sketch



Final solid model

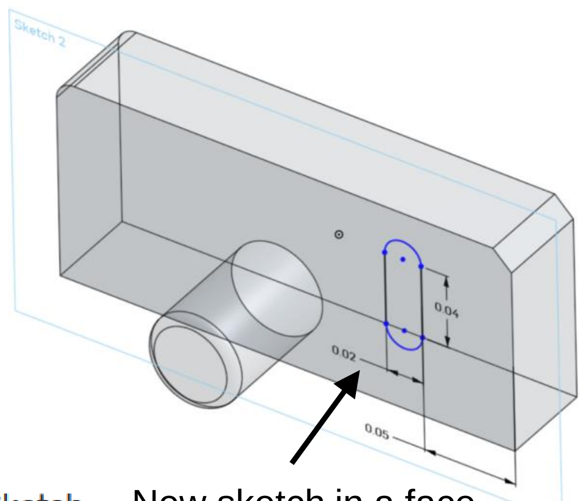
Solid modeling using Onshape

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



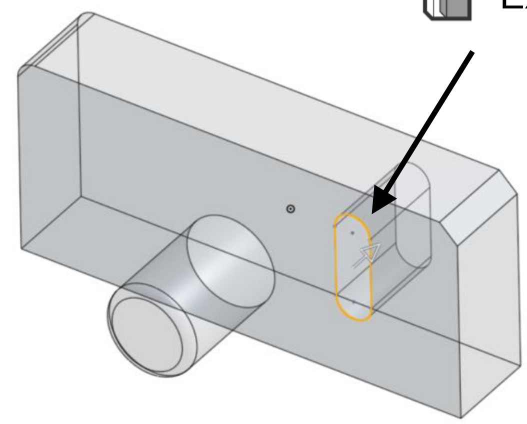
Solid modeling using Onshape

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

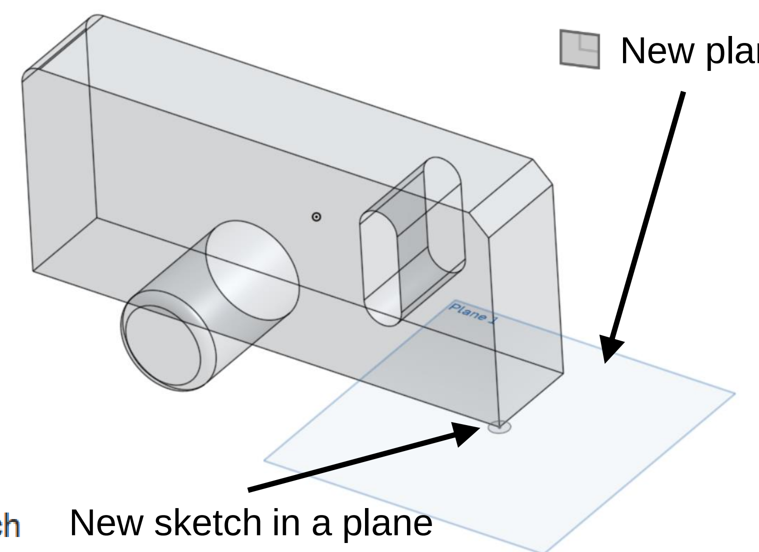


Sketch New sketch in a face

Extrude

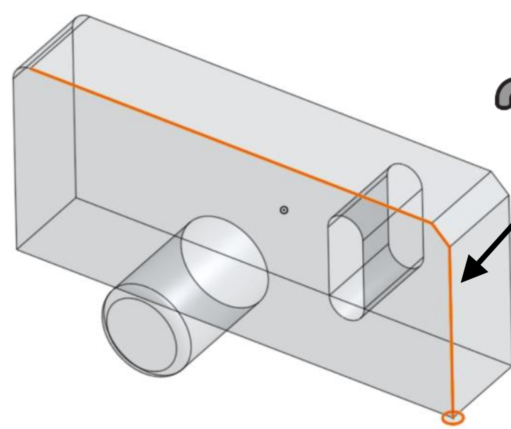


New plane



Sketch New sketch in a plane

Sweep



Solid modeling using Onshape

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

Keyboard shortcuts

General

Keys	Action
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

Keys	Action
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

Keys	Action
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

Keys	Action
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

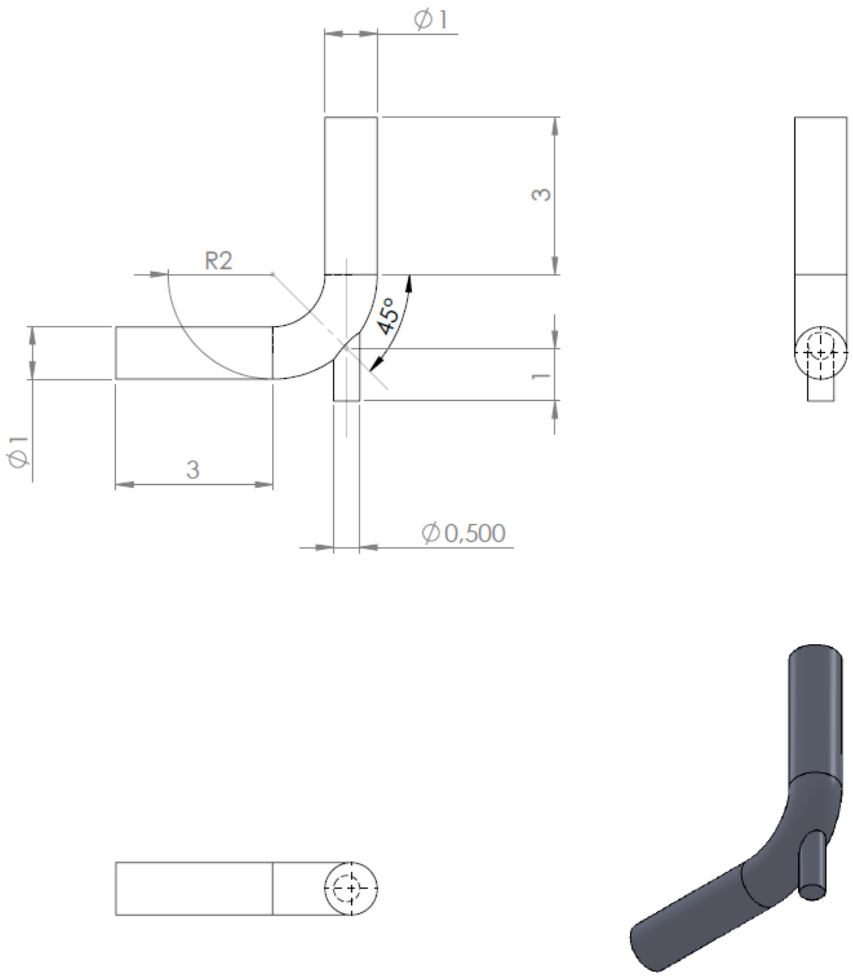
Keys	Action
shift	Suppress inferencing
l	Line
g	Corner rectangle
r	Center point rectangle
c	Center point circle
a	3 point arc
shift f	Fillet
m	Trim
x	Extend
o	Offset
u	Use
d	Dimension
i	Coincident
b	Parallel
t	Tangent
h	Horizontal
v	Vertical
e	Equal
q	Toggle construction

Drawings

Keys	Action
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

Solid modeling using Onshape

- Let us create this solid using the dimensions illustrated.



Note: all the dimensions are in meters

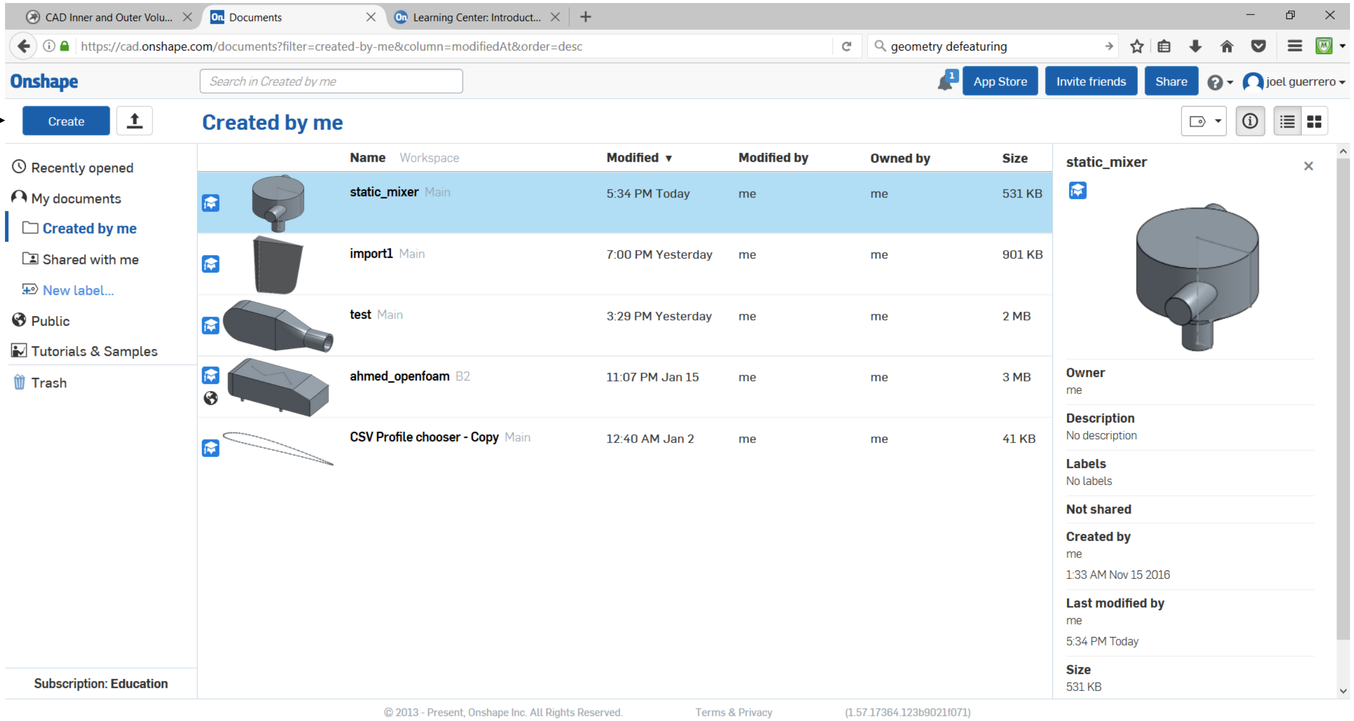
Solid modeling using Onshape

- 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 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 design intent.
    - Choose one feature over other.
    - Dimensioning strategy.
    - Order of the operations.
    - Parametrization.
    - Single or multiple parts.
    - Top-bottom or bottom-up modeling technique.

Solid modeling using Onshape

- Enter the document page and create a new design

Create new document

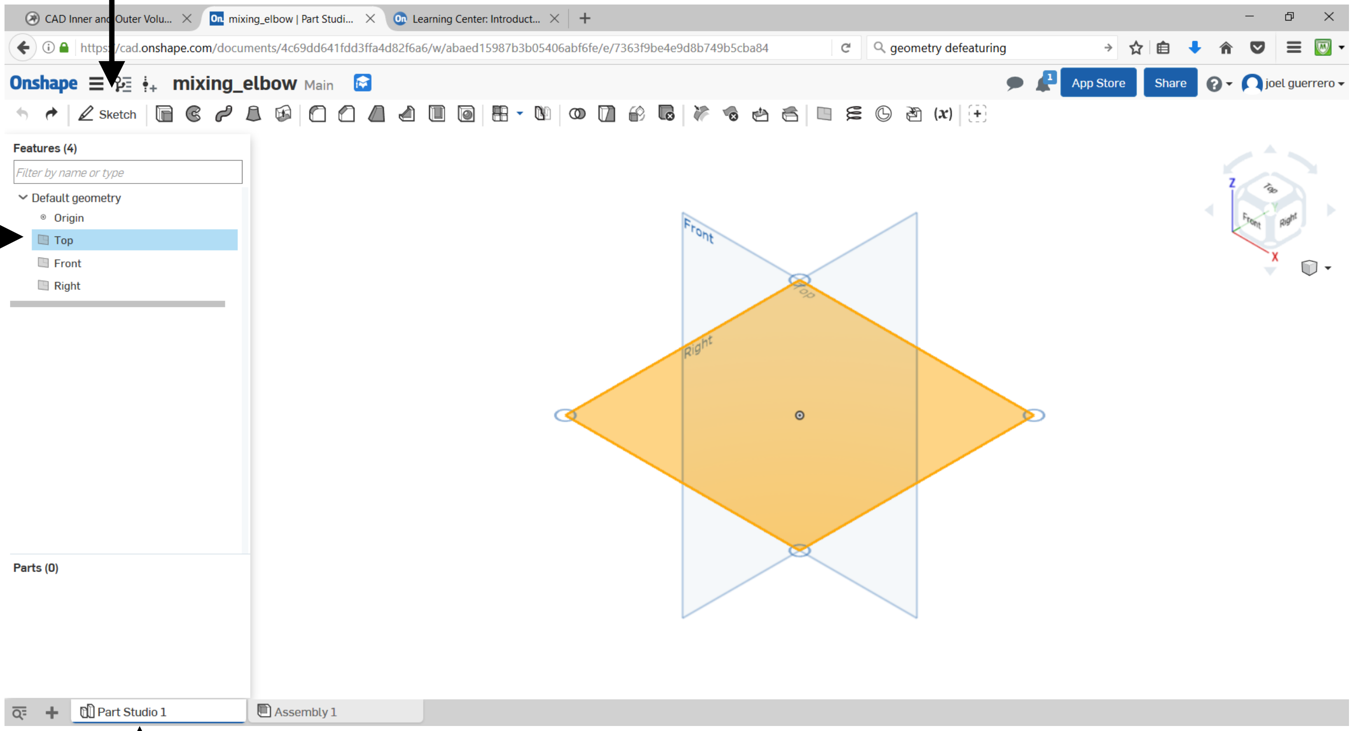


Solid modeling using Onshape

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

2. Start a new sketch

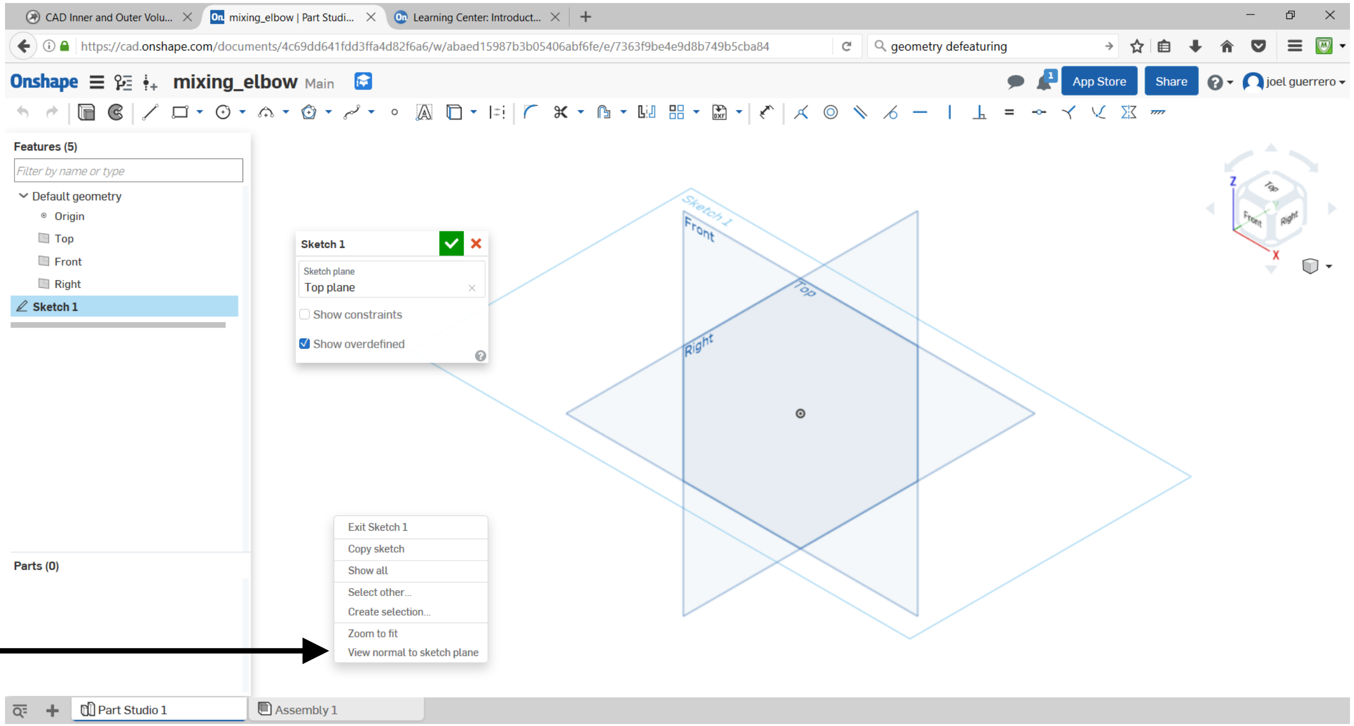
1. Select this plane



Part studio page

Solid modeling using Onshape

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

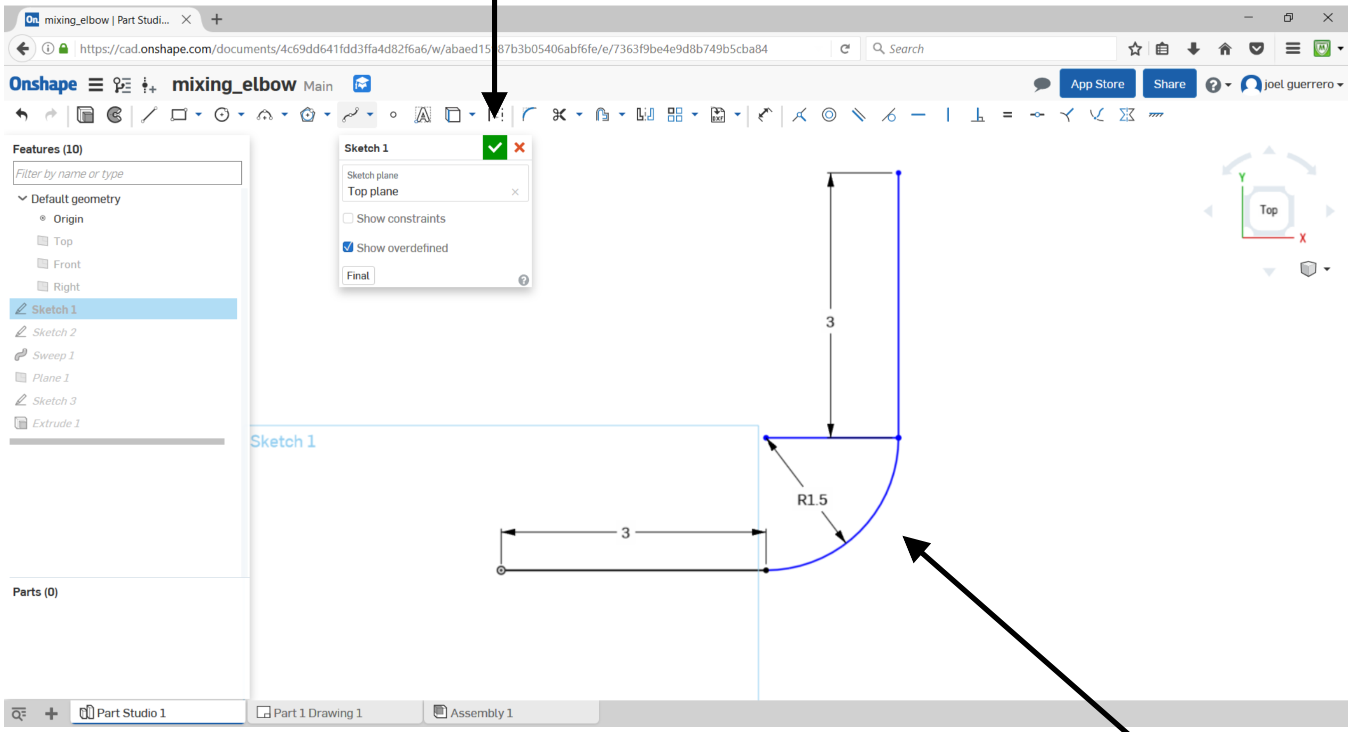


Right click on the 3D area and select view normal to sketch plane

Solid modeling using Onshape

- Using the sketching features, draw the following line.

When you are done sketching press the checkmark



Use the dimensions illustrated to draw this polyline

In sketch mode:

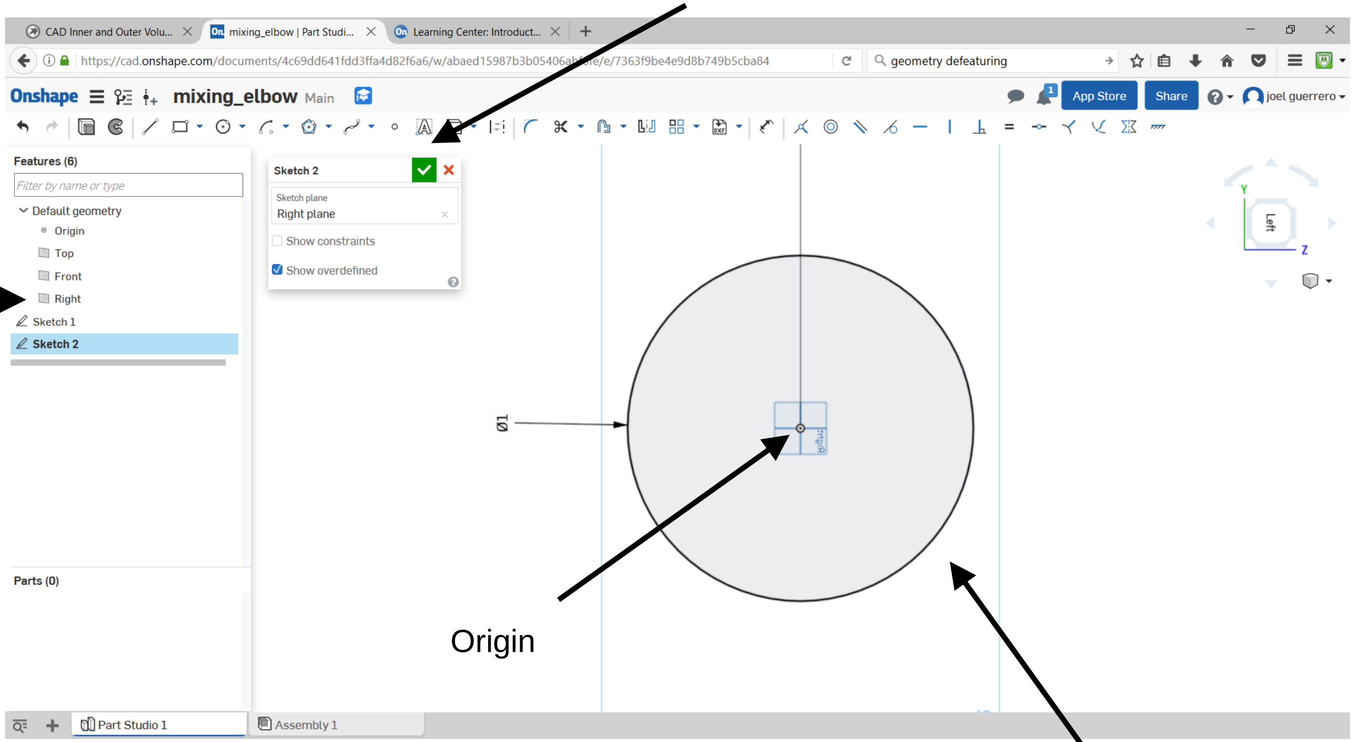
- Blue geometry is free to move.
- Black geometry is fully defined.
- Red geometry is over-constrained.

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 and start a new sketch



Origin

Use the dimensions illustrated to draw the circle

Solid modeling using Onshape

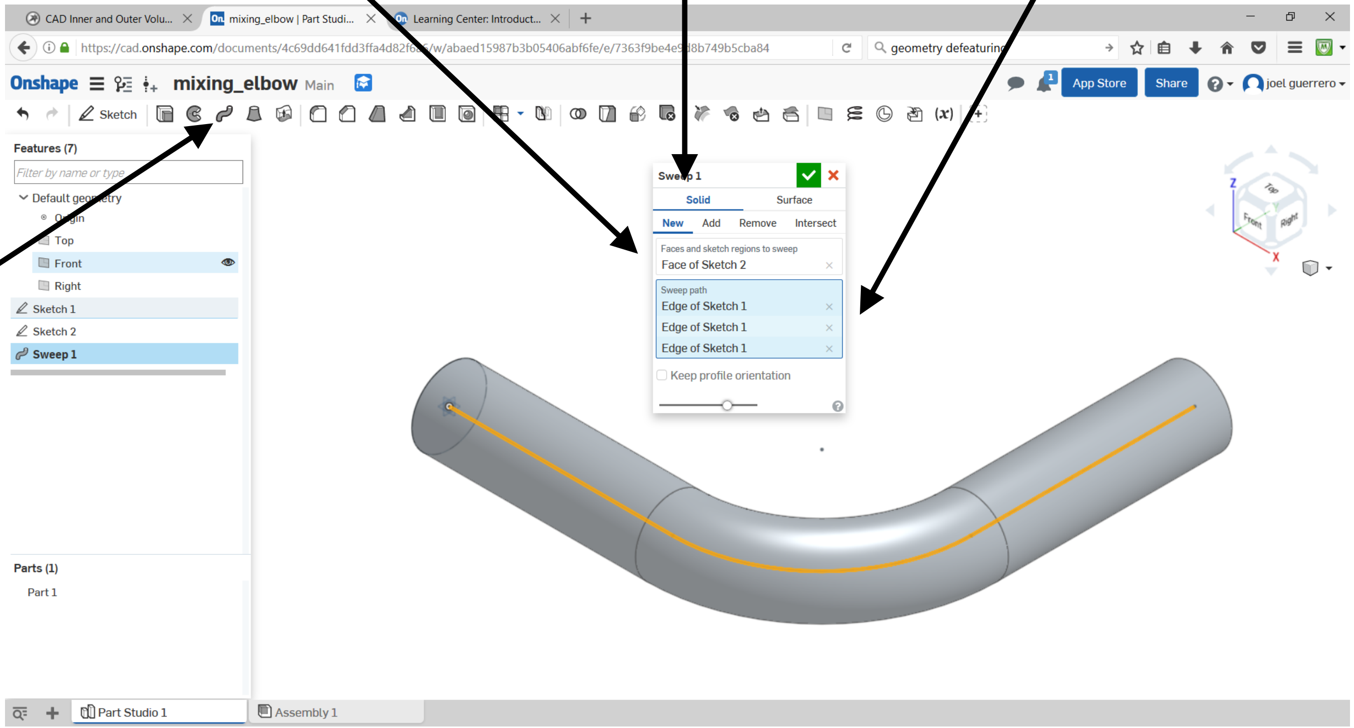
- Use the sweep feature to create a new solid.

Select the circle as the face to sweep

Select new solid

Select the lines as the sweep patch

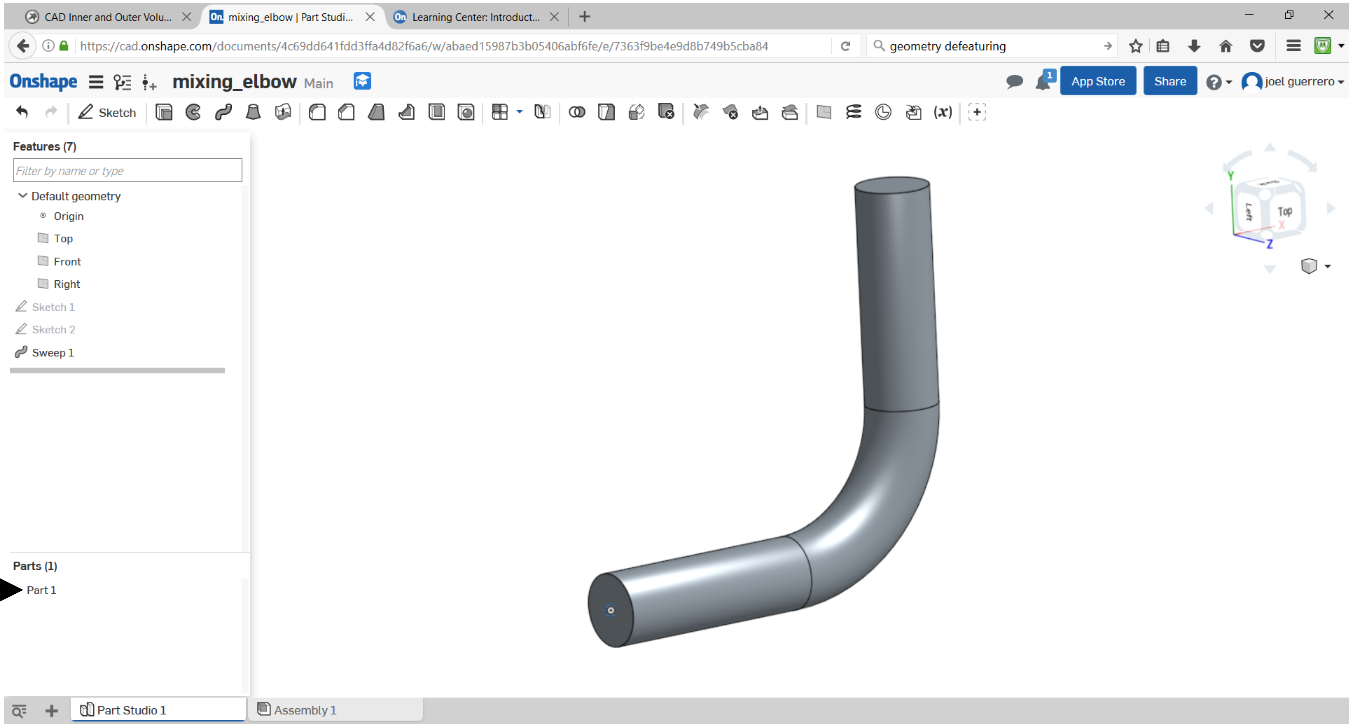
Sweep feature



Solid modeling using Onshape

- At this point, you should have this solid.

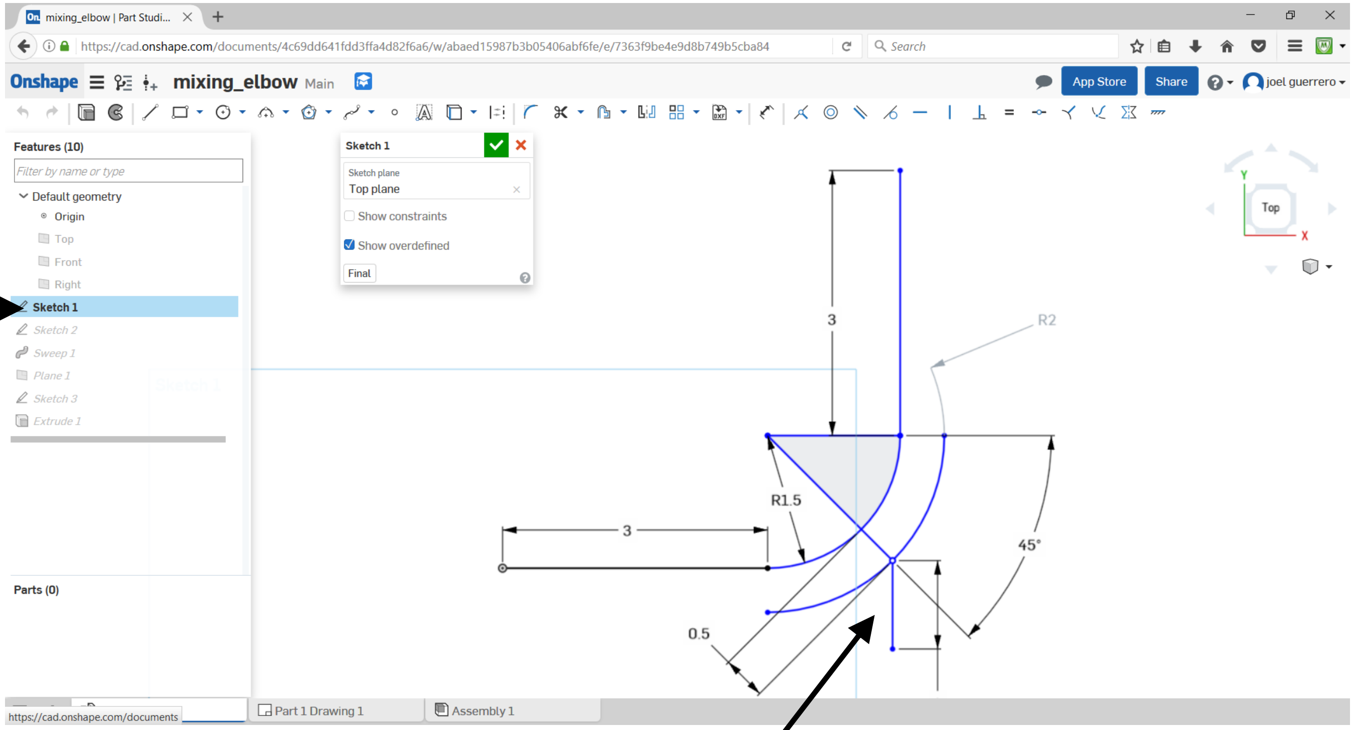
Solid name.
Right click to rename
or view the properties



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

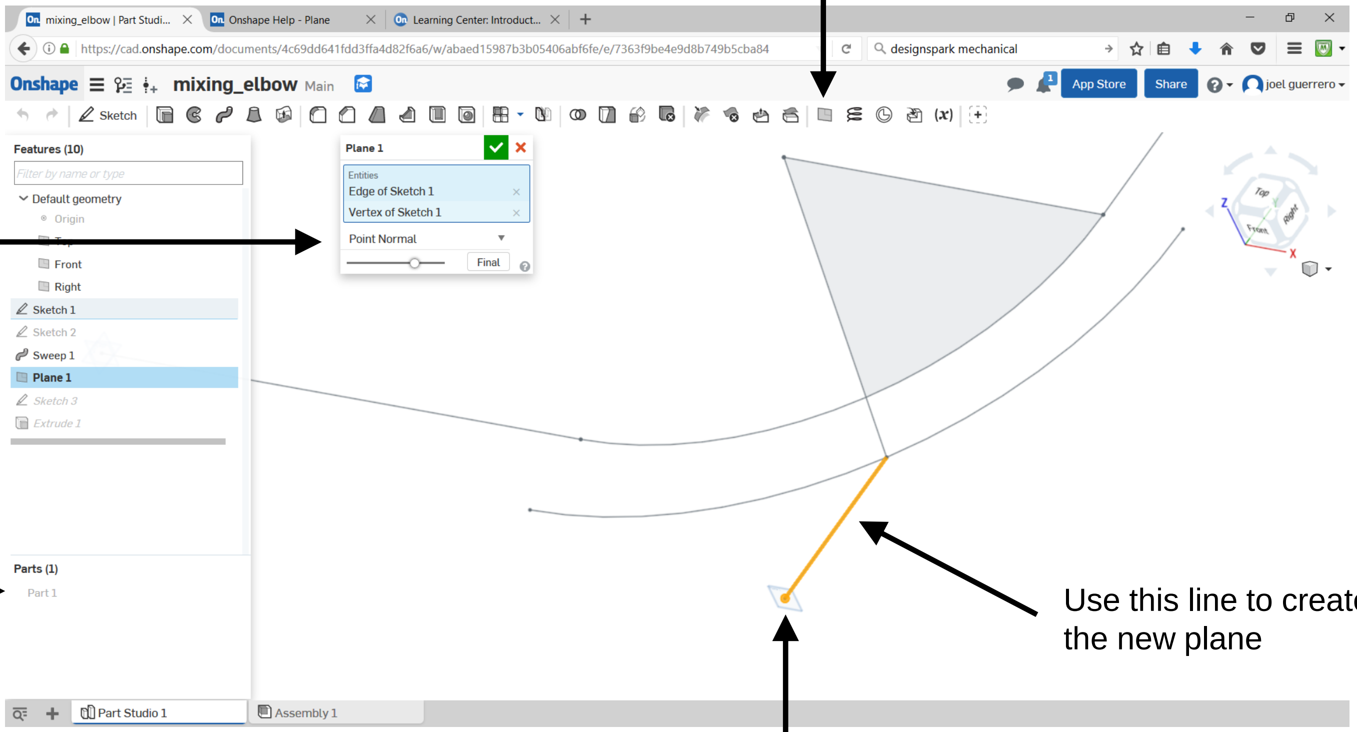
Solid modeling using Onshape

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

1. Create new plane

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

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

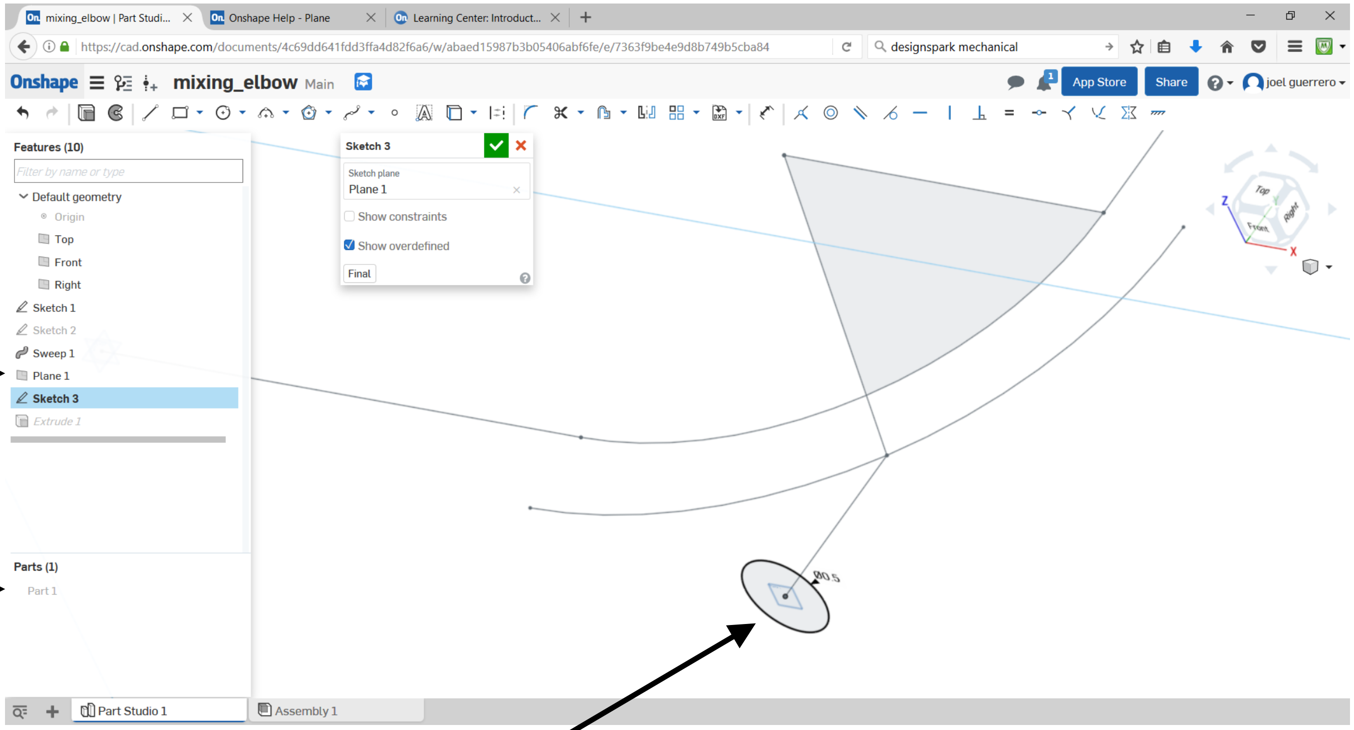


Use this line to create the new plane

Use this point to create the new plane

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

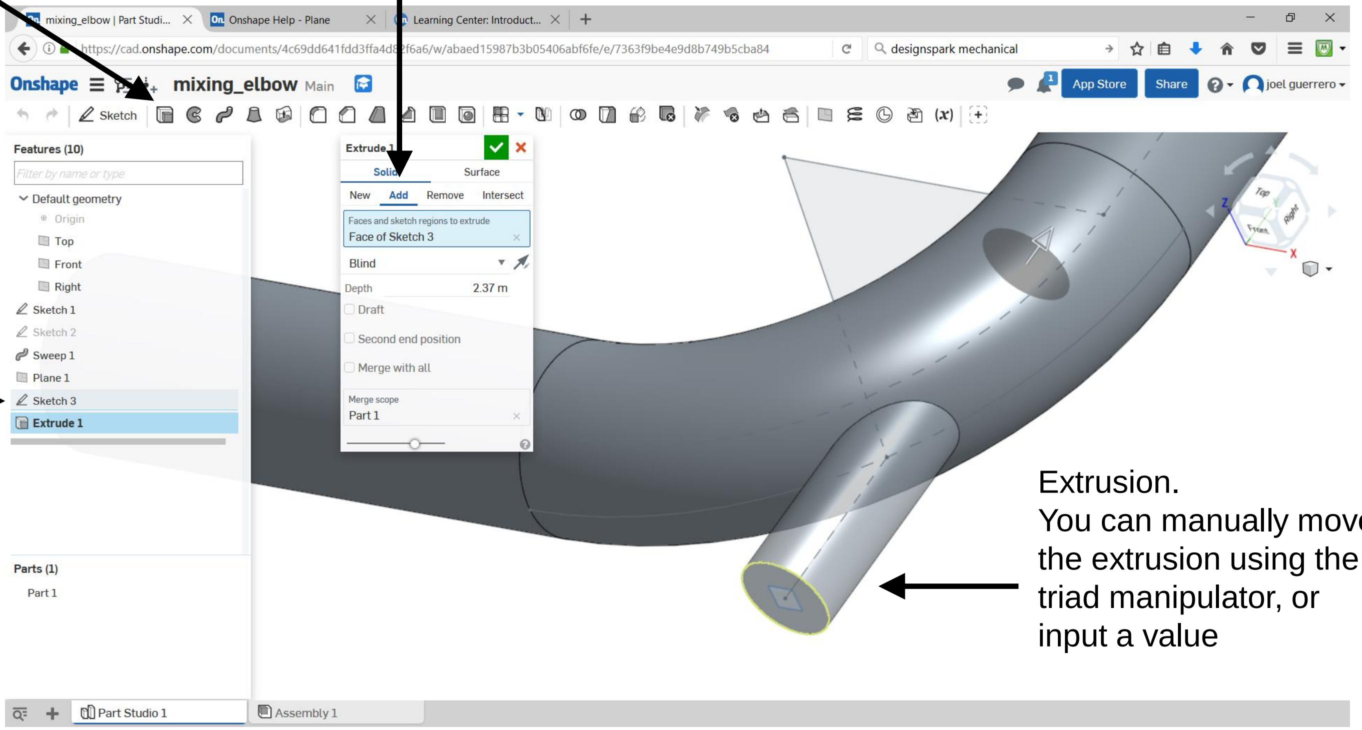
Solid modeling using Onshape

- Using the feature extrude to create a new solid using the previous sketch.
- Extrude the circle until in intercept the solid.

Add the new solid to the previous part (boolean operation)

Feature extrusion

Use this sketch as the base for the extrusion

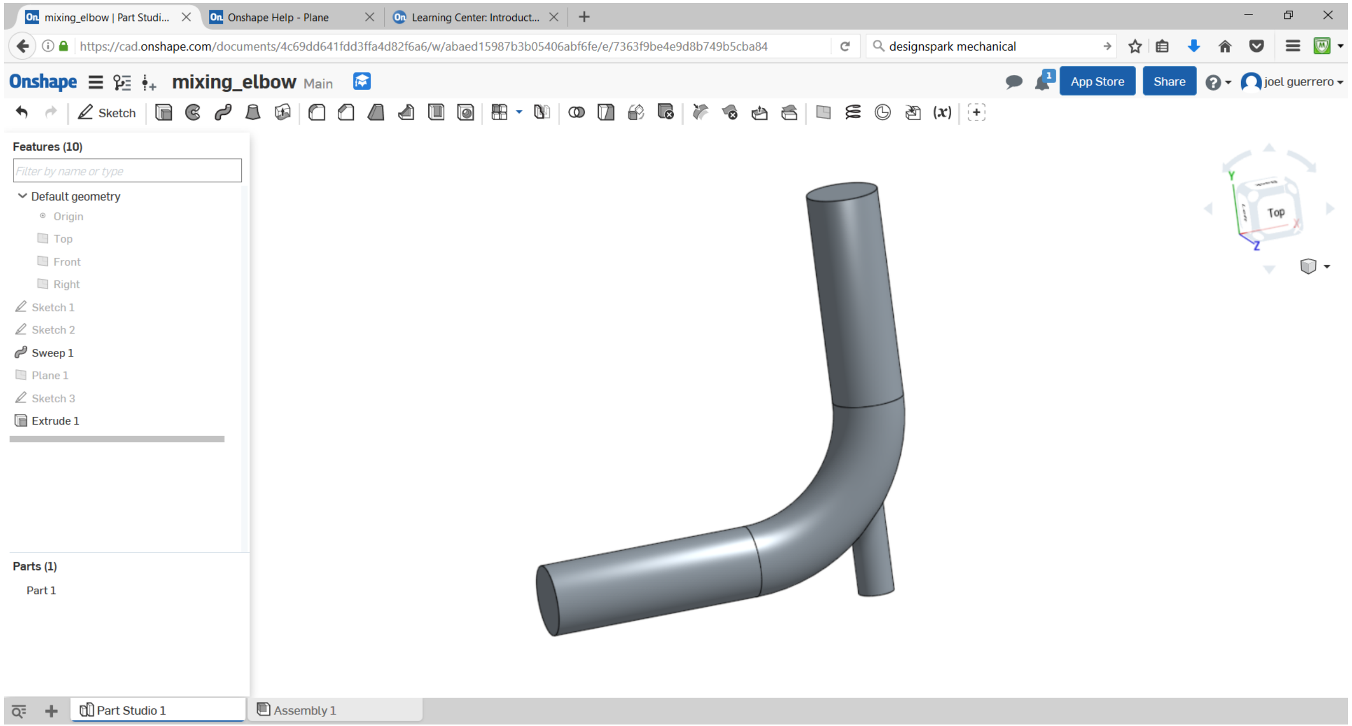


Extrusion.
You can manually move the extrusion using the triad manipulator, or input a value

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

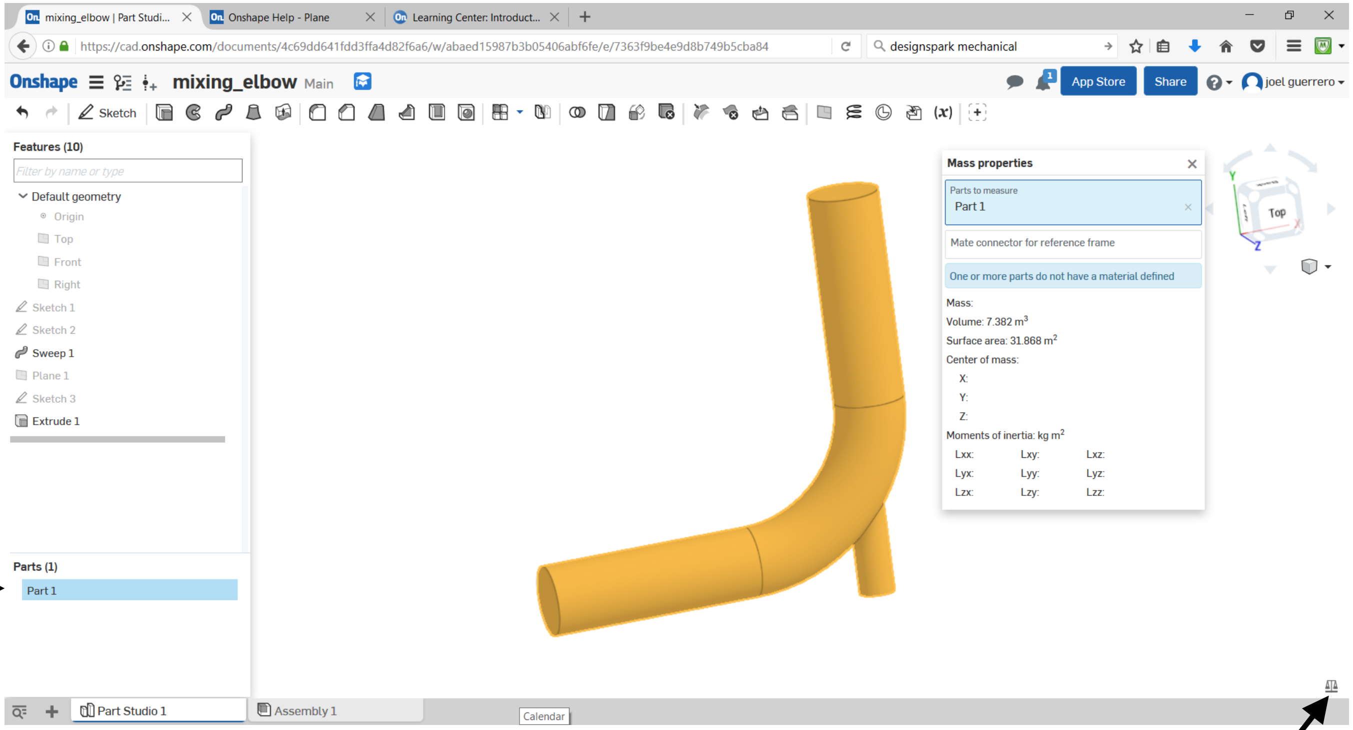
Solid modeling using Onshape

- At this point you should have the following solid.



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.

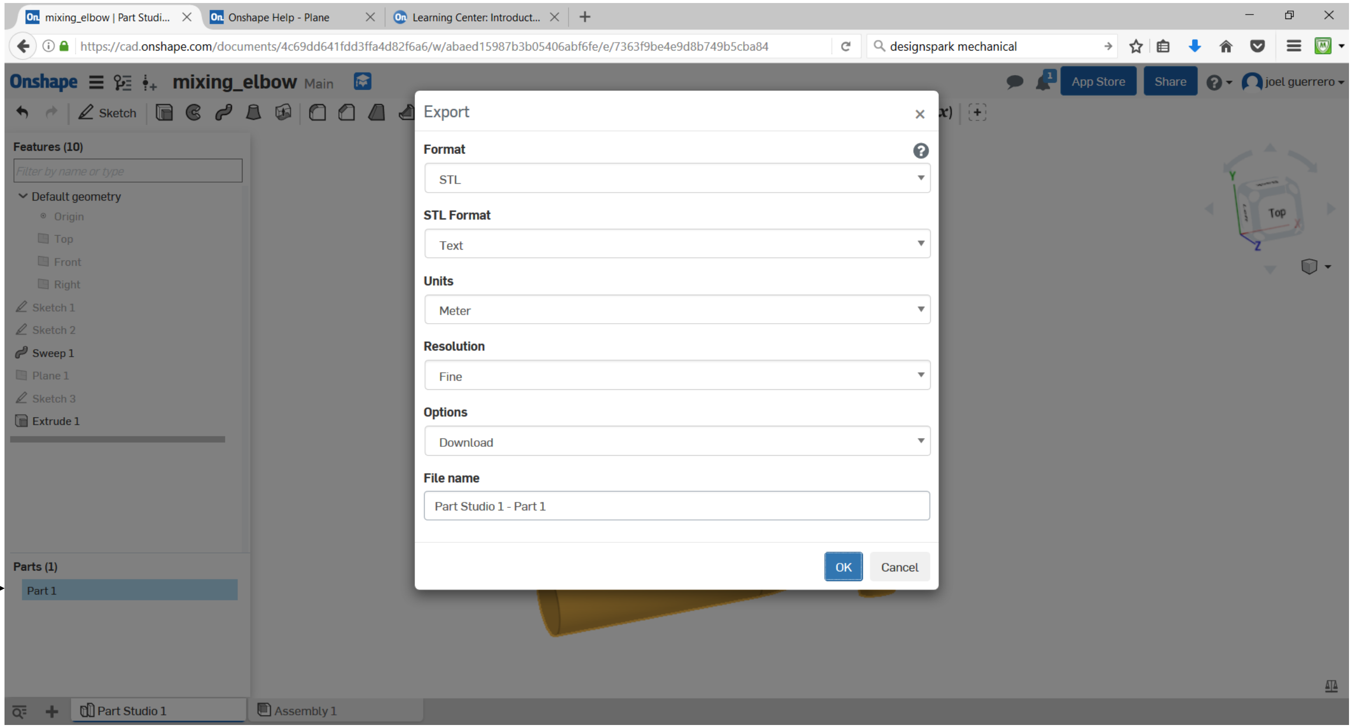


Select the part.
Right click and select assign material.

Mass properties icon

Solid modeling using Onshape

- To export the solid model, 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.

Solid modeling using Onshape

- Parametric modeling and feature-based modeling are two of the most powerful tools available in any CAD/solid modeling applications.
- 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.

- To learn more about Onshape, visit their learning center:

  https://learn.onshape.com/

- Finally, feel free to visit our youtube channel where you will find a few solid modeling videos in the context of CFD and OpenFOAM® :

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

Solid modeling using Onshape

- At the following links, you can find a few geometries that you can use to setup cases from scratch:

  - Cylinder:
https://cad.onshape.com/documents/b2db4af4637f36280b2625ea/w/dd2c83d792b42e03a0147ae5/e/c5d67f5471676a7aaaf83cbb

  - Mixing elbow:
https://cad.onshape.com/documents/1cc919d8e75c2e47e8c1d50e/w/0efa002648eb2fb80ec4bec4/e/a742bf4113c626735e1d8f1a

  - Static mixer:
https://cad.onshape.com/documents/58f7930861743e1074559ea6/w/96672317c9167265f9d10181/e/e4b6b1baffa90ca207afe974

  - Mixing tank:
https://cad.onshape.com/documents/e00307c191ce168d1d8c2e05/w/fc5d69b18559ec3893a1a80a/e/ba4b5ca5b34ad7335a2915a3

  - NACA 0012 airfoil
https://cad.onshape.com/documents/23a8d70e384ccf508c9e406e/w/f71671d54aaab7b6d55e413e/e/bed29662f1ff50b614b02636

  - Three element airfoil:
https://cad.onshape.com/documents/590a4195a6145a1089cfb96f/w/838a3095da90d5dd66a3150e/e/5d65878bed975a1a94102846

  - Onera M6 wing:
https://cad.onshape.com/documents/e176caaa70bfd4719cafe3d7/w/9d8b5771d7382000b0762e65/e/ec4669f8c28761e60e35b32f

  - Sailing boat:
https://cad.onshape.com/documents/ad885ed6298e6d95e372f573/w/1cfda457fe3ad410332aad9c/e/8dac4fcaf6e34a43e9676fbc

  - Ahmed body:
https://cad.onshape.com/documents/e1ecbacd95be9ed0962aa410/w/f0295899197e2f3d851000fd/e/aa40f8f5d26b7117dd0a5111