CaYPT
Canadian Young Physicists' Tournament

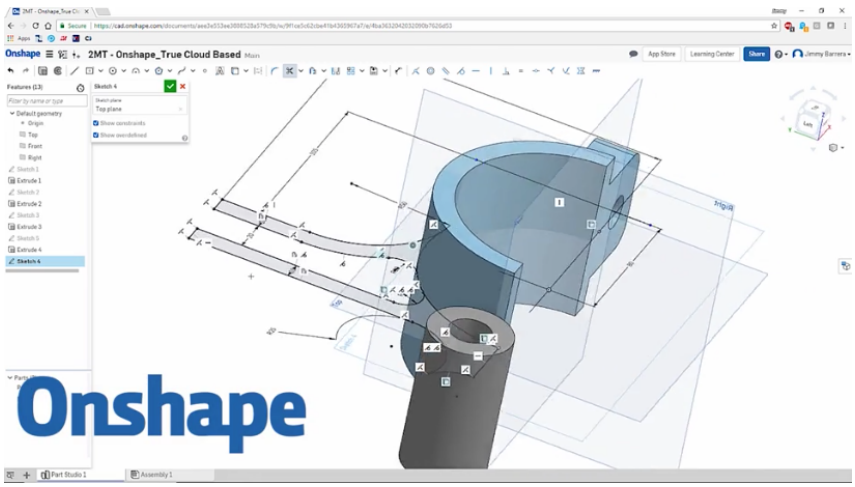
BASICS OF ONSHAPE CAD

Author: Jeffrey Li

Date Created: 05/19/22
Version 1.0

Abstract

In this SOP, the basic operations and tutorials of Onshape CAD are discussed. The ways to construct, modify, and calibrate models are included. Readers available in the lab are listed.



Contents

1 Principles of Operation of OnShape

Onshape is a cloud-based CAD and data management solution platform available through desktop and mobile apps.

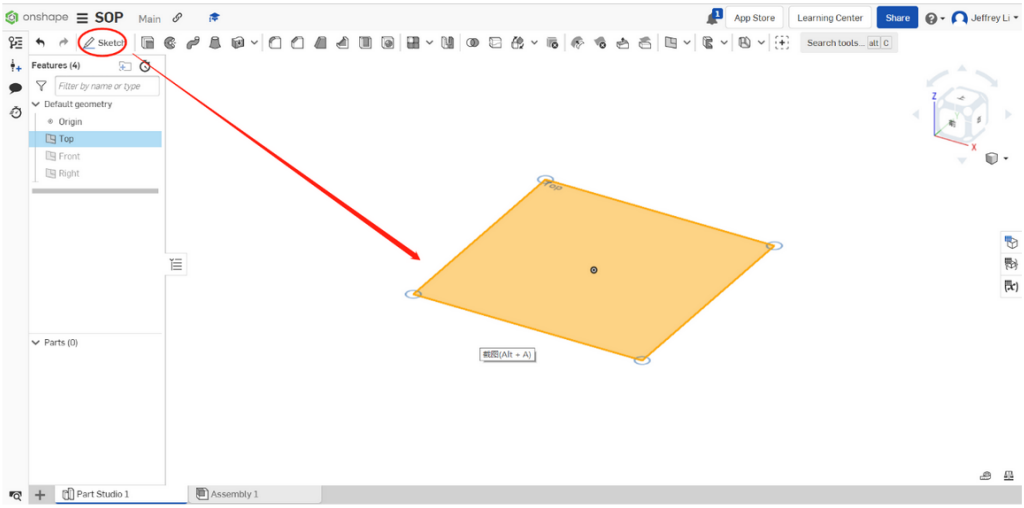
2 How to get access to Onshape for free

Onshape has two different versions: Professional and Educational. “Educational” is for gratis and the steps to access it are easy. Go to the Onshape website, click on “Create a student account” and you can access Onshape for free.

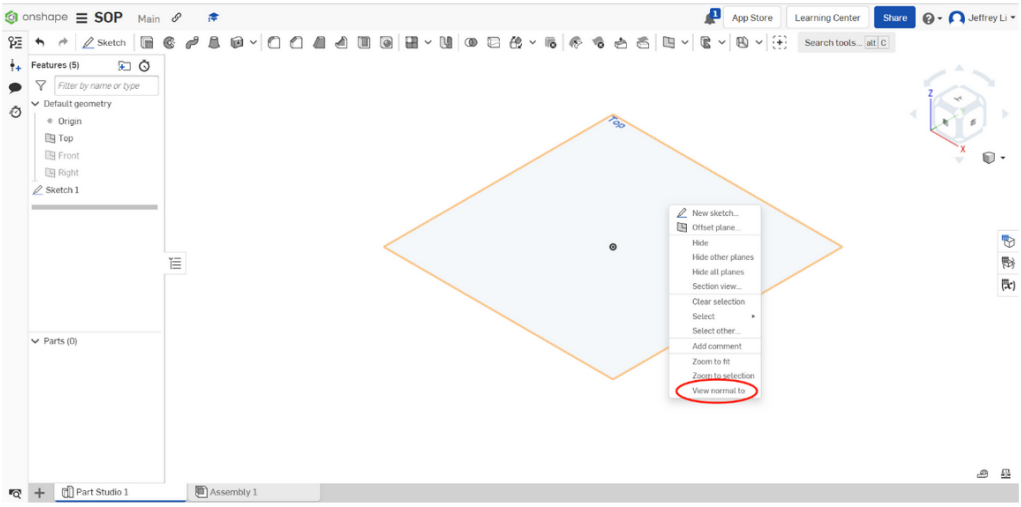
3 How to build a solid on Onshape:

Procedures:

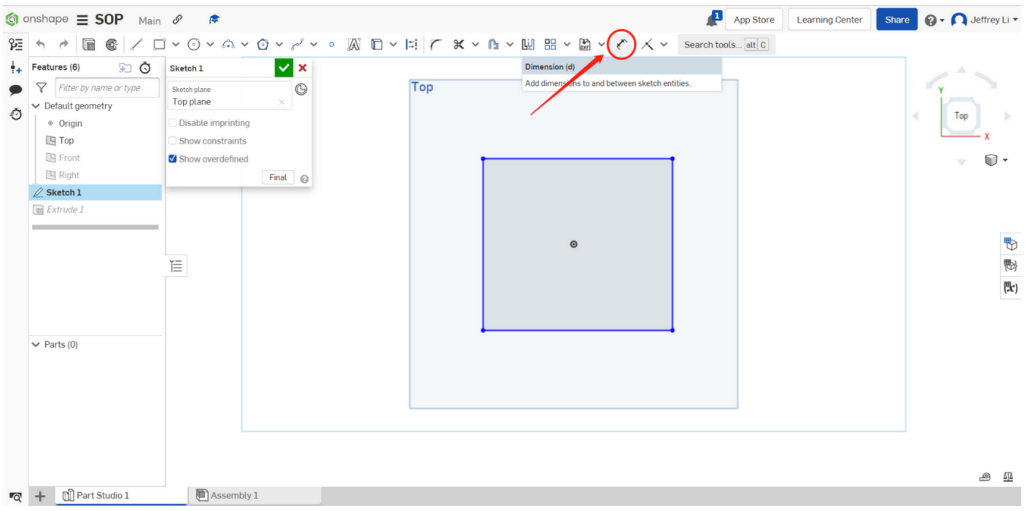
1. Click on the “Sketch” bottom and select a surface (Click on a surface and it will turn Orange)



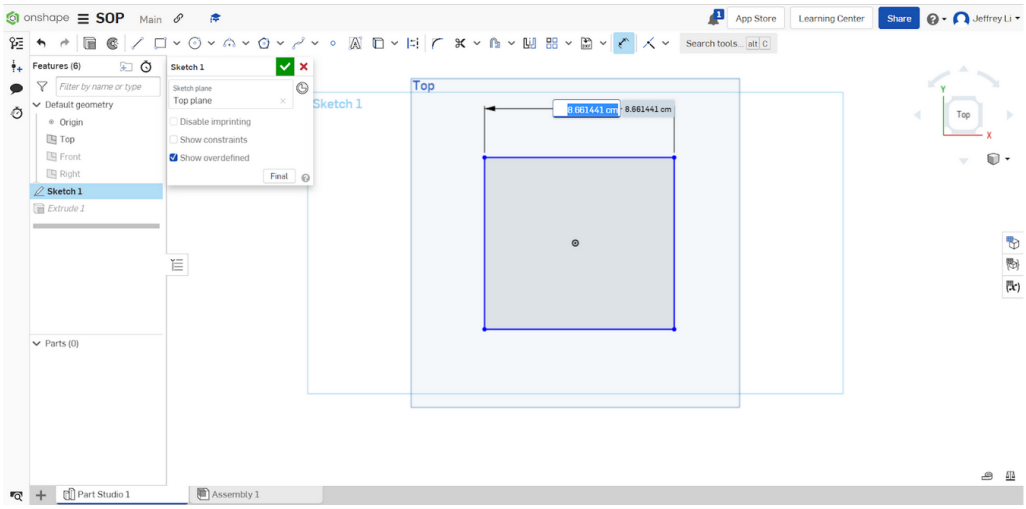
2. Adjust your view to the top of the surface by clicking “view normal to”



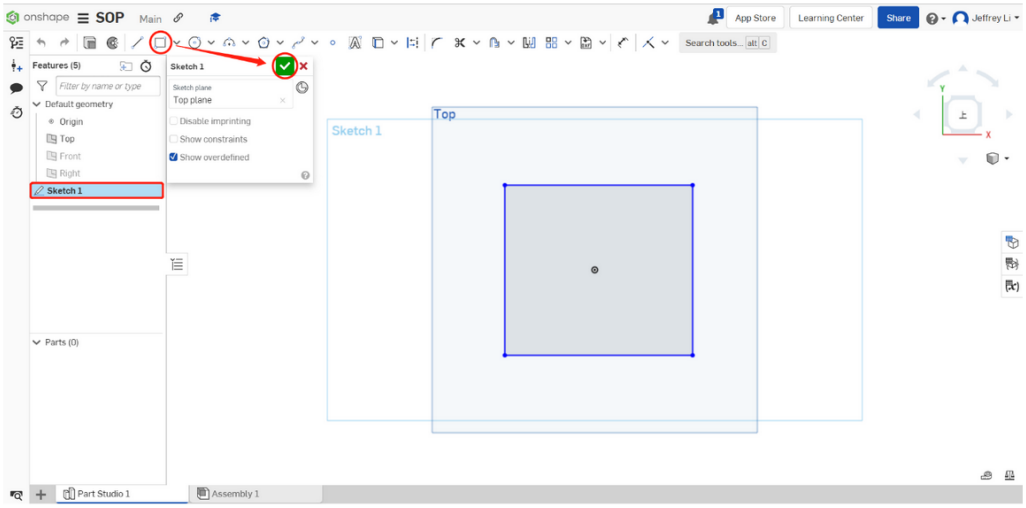
3. Draw a rectangle on the surface by clicking the rectangle sign, after drawing the dimensions of the rectangle, click on the screen again to finalize the dimensions of the rectangle. The dimensions of the rectangle can be adjusted by clicking on the “dimension”



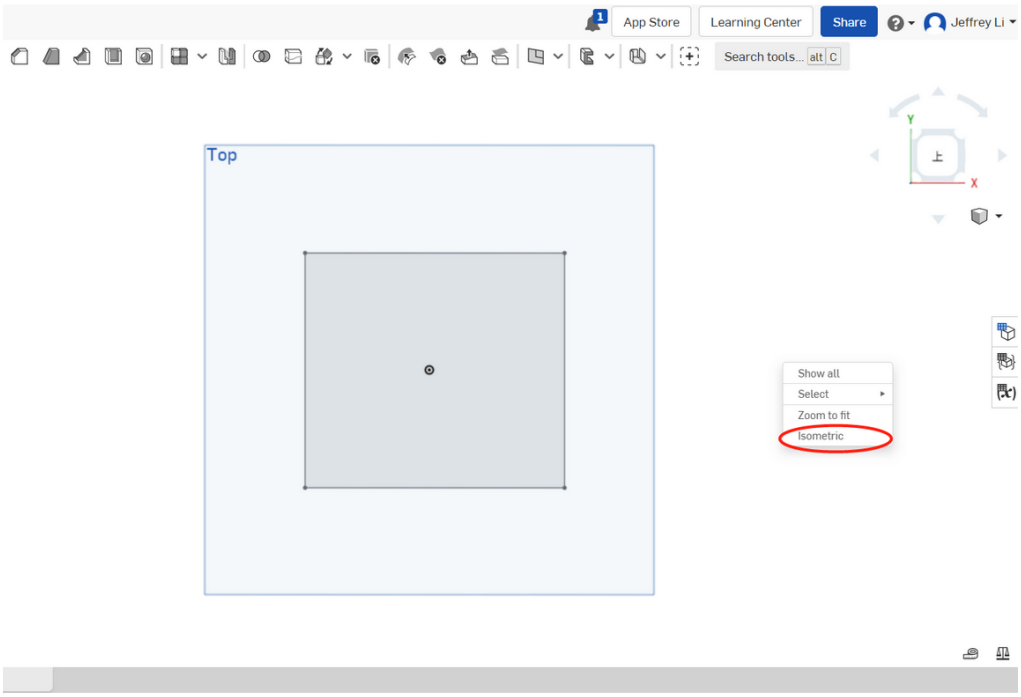
4. Click on the dimension then click on 1 side of the rectangle to measure its length and click on 2 sides to measure the distance between these 2 sides. The length of the side can be adjusted by changing the number shown on the dimension bar.



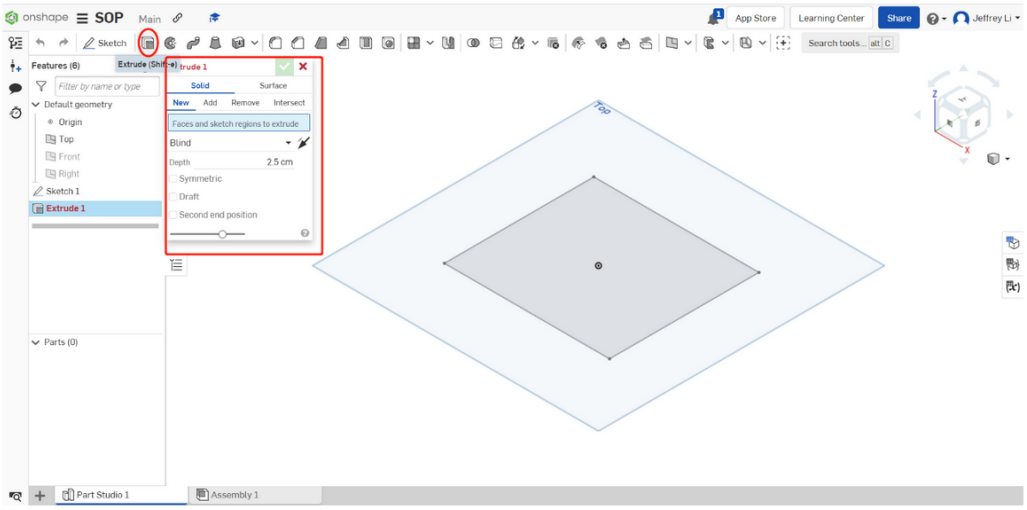
5. Then, click on the green check mark to finish "sketch." This sketch will be named "Sketch 1" in the "Feature" section.

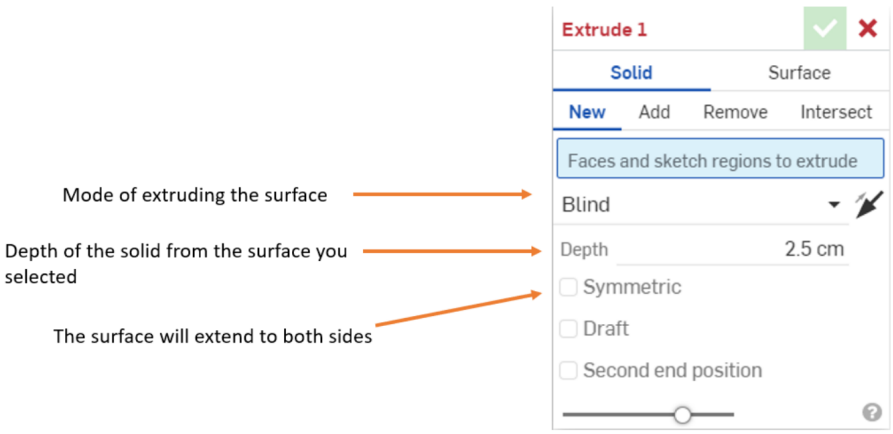


6. Right-click on the white region outside of the surface and select “Isometric.”

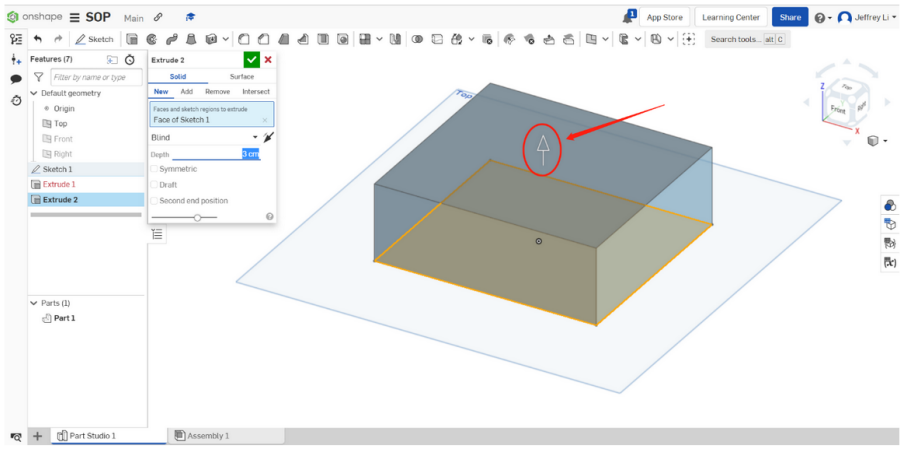


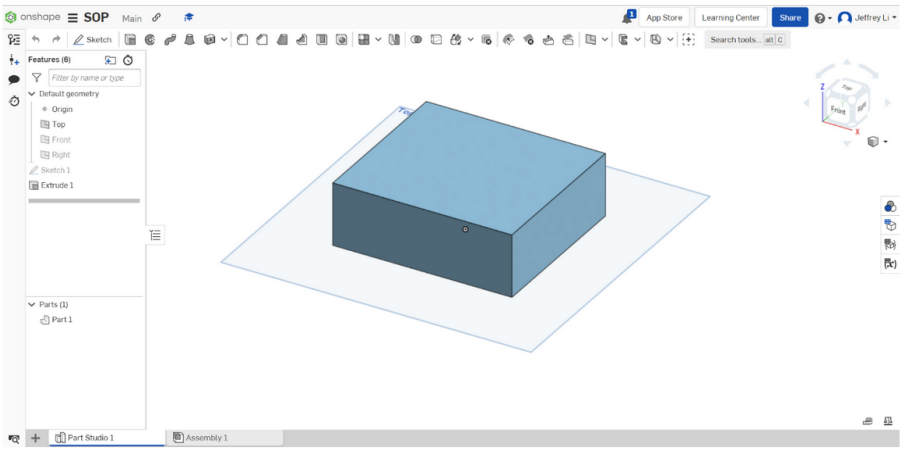
7. Click on the “Extrude” button, and the “Extrude” window will pop out.





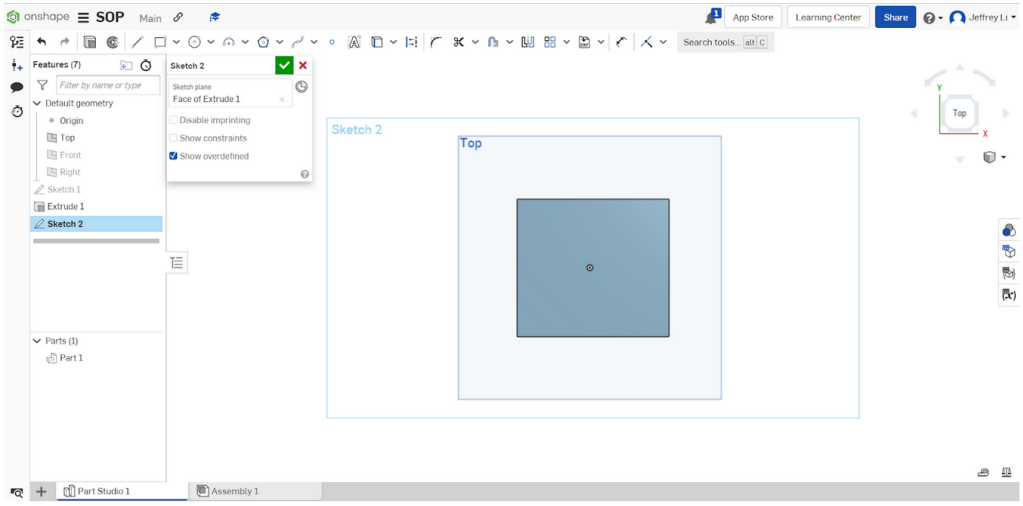
8. Select and extrude the surface for 3 cm, the arrow is to change the direction at which the surface expands by dragging the arrow. Then, click on the green check mark to create a solid.



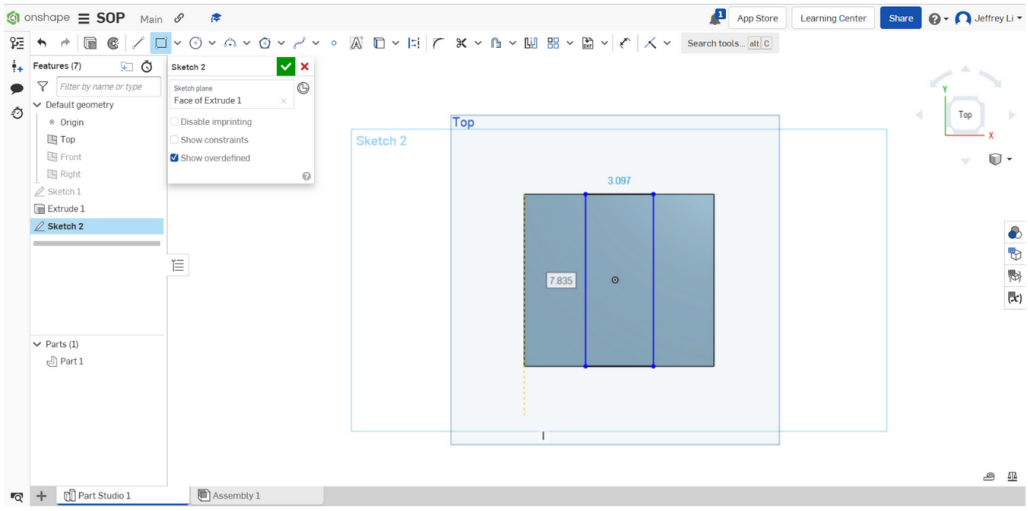


4 How to modify existing solid

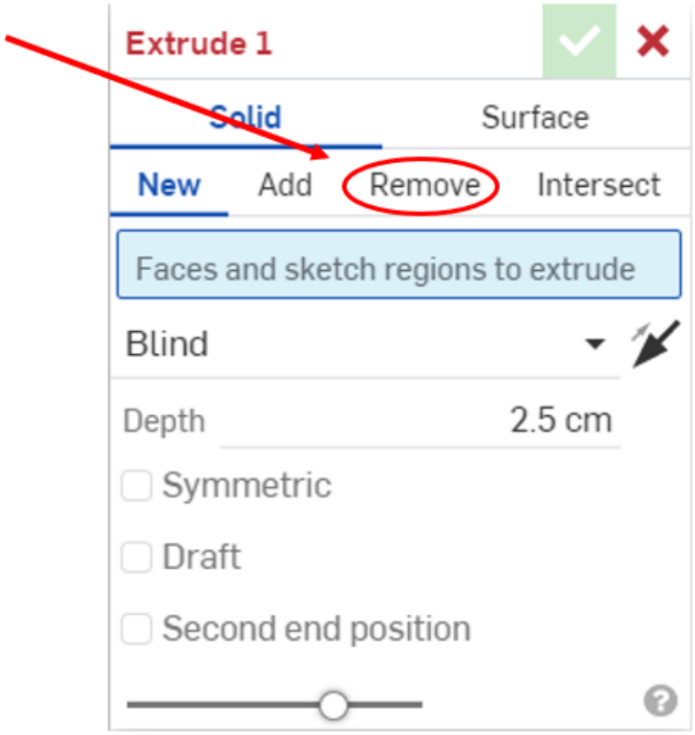
1. Select “Sketch” and chooses one surface of the solid as the sketching surface.



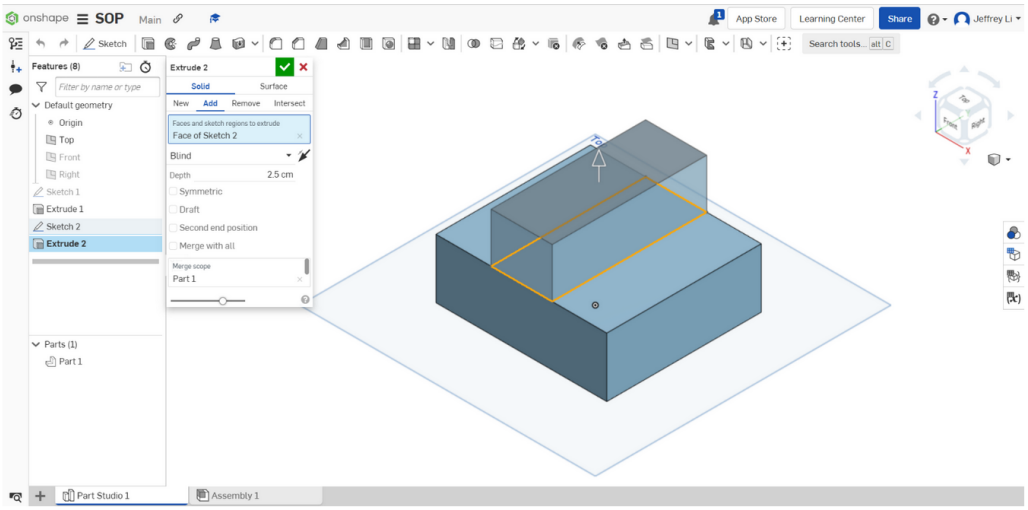
2. Draw the figure your want to modify on the surface.



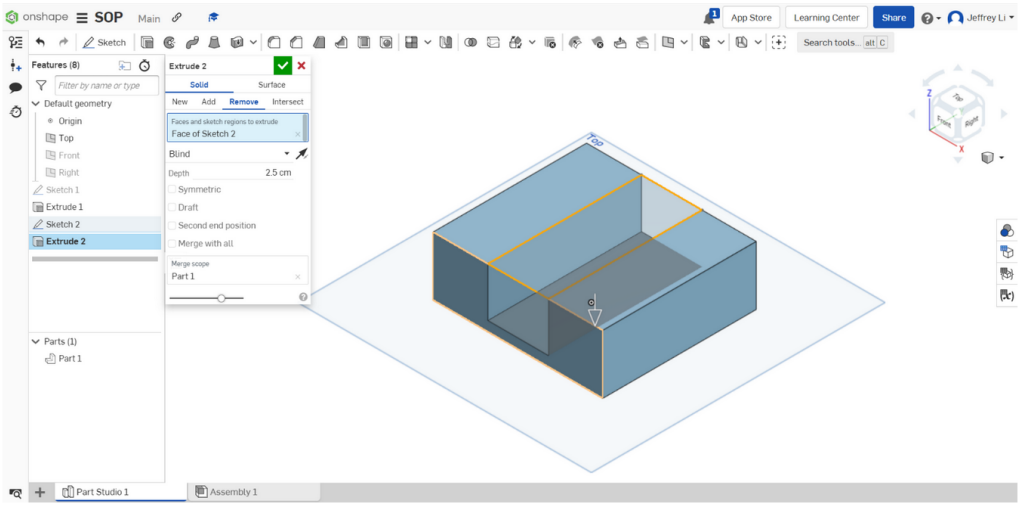
3. Finish sketching and select “Extrude.” On the “Extrude” window, use “Add” or “Remove” to either add a layer of solid on the figure drawn or remove the amount of solid in the figure drawn to a selected depth.


Extrude 1
Solid
Surface
New
Add
Remove
Intersect
Faces and sketch regions to extrude
Blind
Depth
2.5 cm
Symmetric
Draft
Second end position

4. **Add:**


onshape
SOP
Main
App Store
Learning Center
Share
Jeffrey Li
Sketch
Search tools...
Features (8)
Extrude 2
Solid
Surface
New
Add
Remove
Intersect
Faces and sketch regions to extrude
Face of Sketch 2
Blind
Depth
2.5 cm
Symmetric
Draft
Second end position
Merge with all
Merge scope
Part 1
Default geometry
Origin
Top
Front
Right
Sketch 1
Extrude 1
Sketch 2
Extrude 2
Parts (1)
Part 1
Part Studio 1
Assembly 1

5. **Remove:**



5 General Procedures of Operation

1. Draw a sketch on the selected surface and modify the dimensions to desired values.
2. Extrude the sketch to desired thickness.
3. Modify the solid by selecting one of the solid's surfaces as the sketch surface and draw figures that are going to be removed.
4. Select "Extrude" and remove the drawn figure to a certain depth.

6 Related SOP

[3.04.01.00] On-Shape CAD