

OnShape: Extrude Tutorial Part 1

Single Profile Extrude

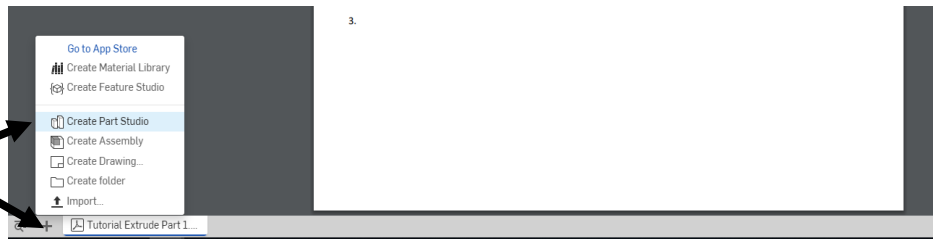
By: Matthew Jourden

Brighton High School

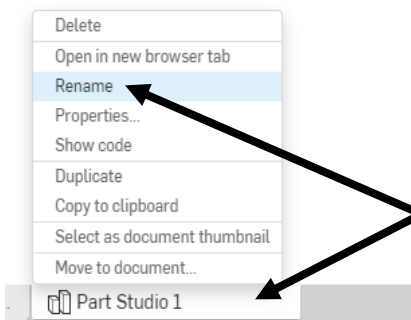
Brighton, MI

1. Open a Web Browser > Navigate to brightonk12.onshape.com > Login using your Brightonk12.com email address > Select Tutorial: Extrude Part 1
2. Go to Student Folder > 01. Open Tutorial Extrude Part 1- "Last Name"
3. Adding a New Tab: Select the Plus Sign next to Tutorial Tab > Select Create Part Studio > New Tab will appear

1. Select + Sign
2. Select Create Part Studio



4. Change Tab Name: Place Cursor on Tab Part Studio 1 > Right Mouse Button > Select Rename > Rename to Extrude Tutorial 1



1. Right Mouse Button On Part Studio 1
2. Select Rename > Change Name



5. Mouse Navigation

View settings

Creo ▼

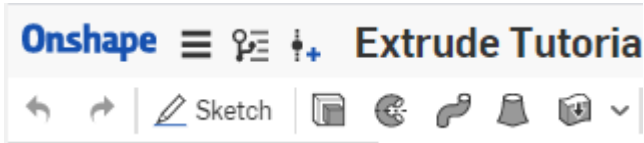
	3D Part & Assembly	2D Drawing
Rotate	Middle mouse button drag	Not applicable
Constrained rotate	Alt + Middle mouse button drag	Not applicable
Pan	Shift + Middle mouse button drag	Middle mouse button drag
Zoom	Scroll wheel in/out or Ctrl + Middle mouse button drag	Scroll wheel in/out or Ctrl + Middle mouse button drag

Reverse scroll wheel zoom direction

Save mouse controls

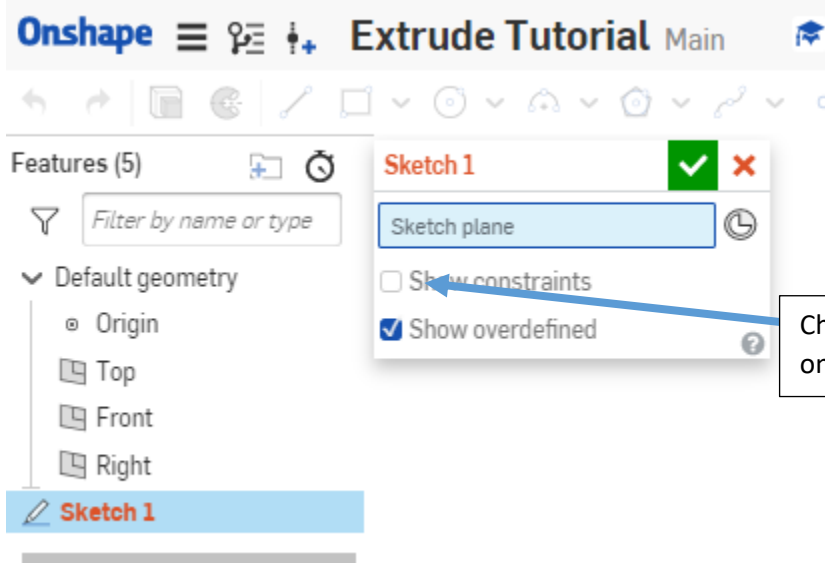


6. OnShape a 2D Sketch will be created first before choosing the 3D Model Function (Extrude, Revolve, Sweep, etc.)
- a. Select Sketch Icon



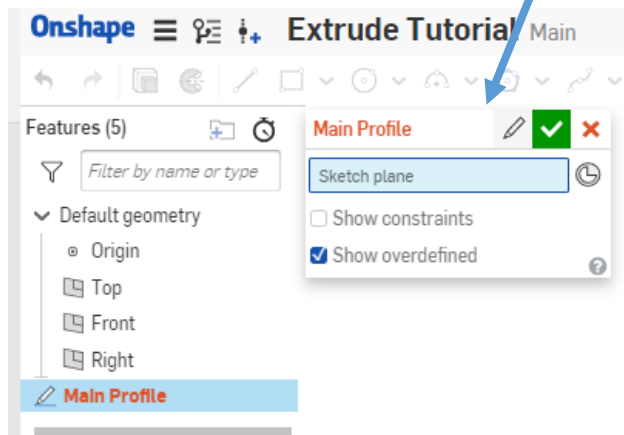
- b. Sketch Pop Menu will Appear

- a. Rename the Sketch by select Sketch1 to Main Profile



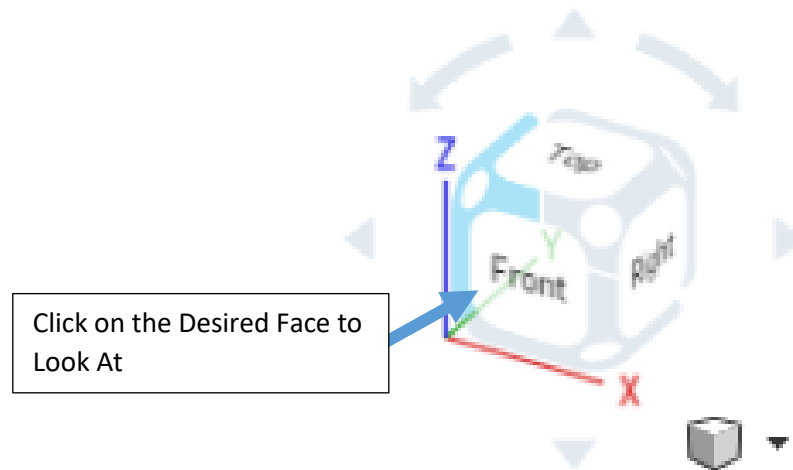
Check Box to Constraint Icons on your part similar to Creo

Select Pencil to change Sketch Name1 to Main Profile



b. Select Front Plane (Datum) > Rotate View into 2D State > Using one of the Options below rotate perpendicular to the Front Plane


Option 1: Click on the desired View on the Navigator Rotator

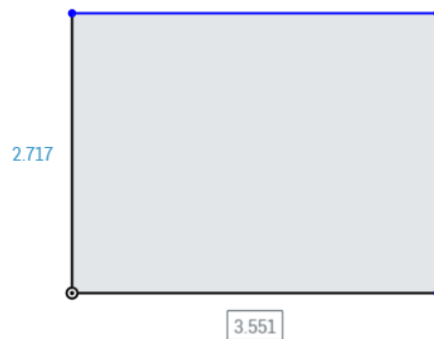
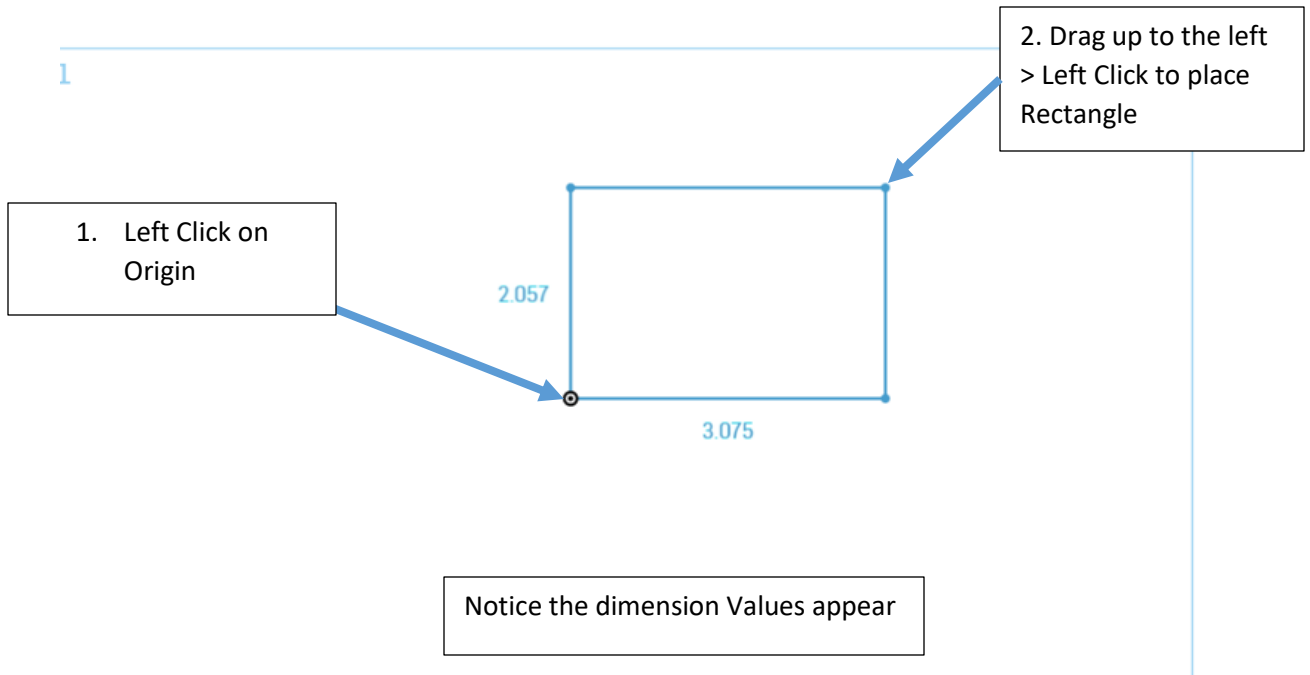
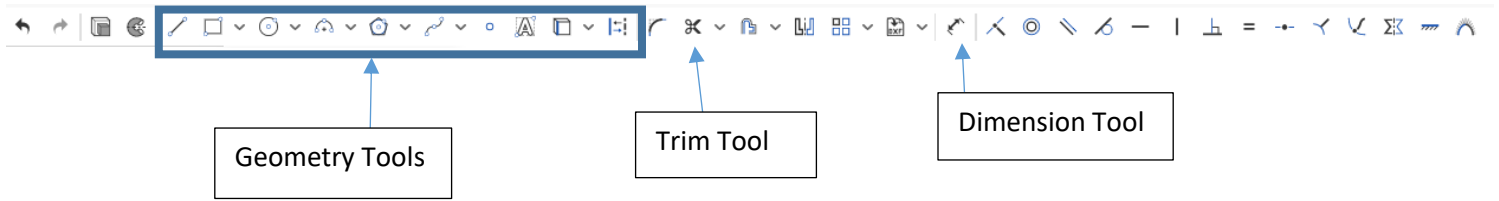


Option 2: Hotkeys

- i. Shift + 1 = Front View
- ii. Shift + 2 = Back View
- iii. Shift + 3 = Left View
- iv. Shift + 4 = Right View
- v. Shift + 5 = Top View
- vi. Shift + 6 = Bottom View
- vii. Shift + 7 = Isometric View
- viii. n Key = View Normal to Activate Plane (Datum)

c. Press p Key to turn off other workplanes (datums)

d. Tool Bar should appear with the Sketching Tools > Select Rectangle Tool  > Left Click at the Origin > Drag Cursor up to the right (Stay in Quadrant 1) > Left Click to set opposite corner of the rectangle



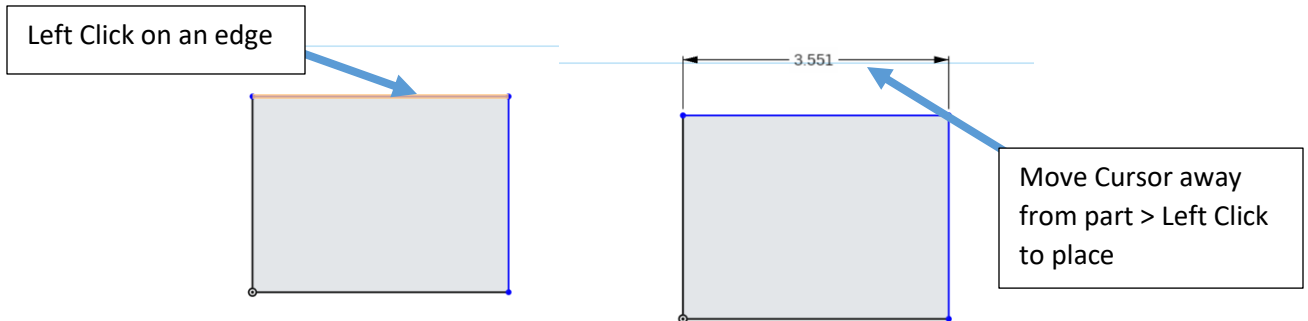
After making the 2nd left click user can type in a value when the white box is around the dimensional value.

NOTE: If any mouse click is performed the dimensions will disappear and the dimension tool will need to be used to place


e. Setting Dimensions

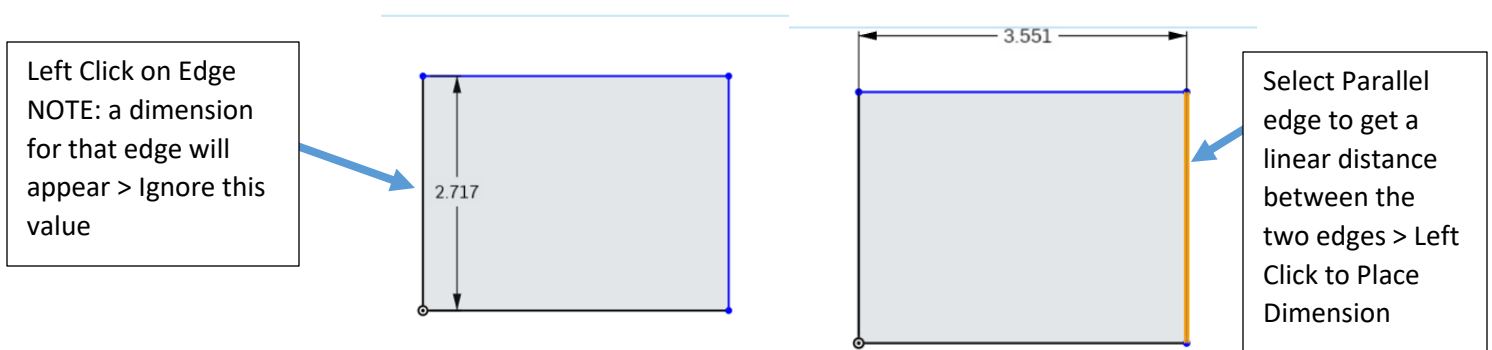
Note: OnShape, like many CAD software's, does not automatically provide Weak Dimensions. The user will place all desired dimensions. OnShape will tell the user if the part is Over Dimensioned/Over Constrained.

- a. Option 1: Immediately after making the final left click for the desired geometry type in the sizes
 - i. Length of Rectangle (Horizontal comes 1st) > Enter 5 > Press Enter
 - ii. Height of Rectangle (Vertical comes 2nd) > Enter 3 > Press Enter

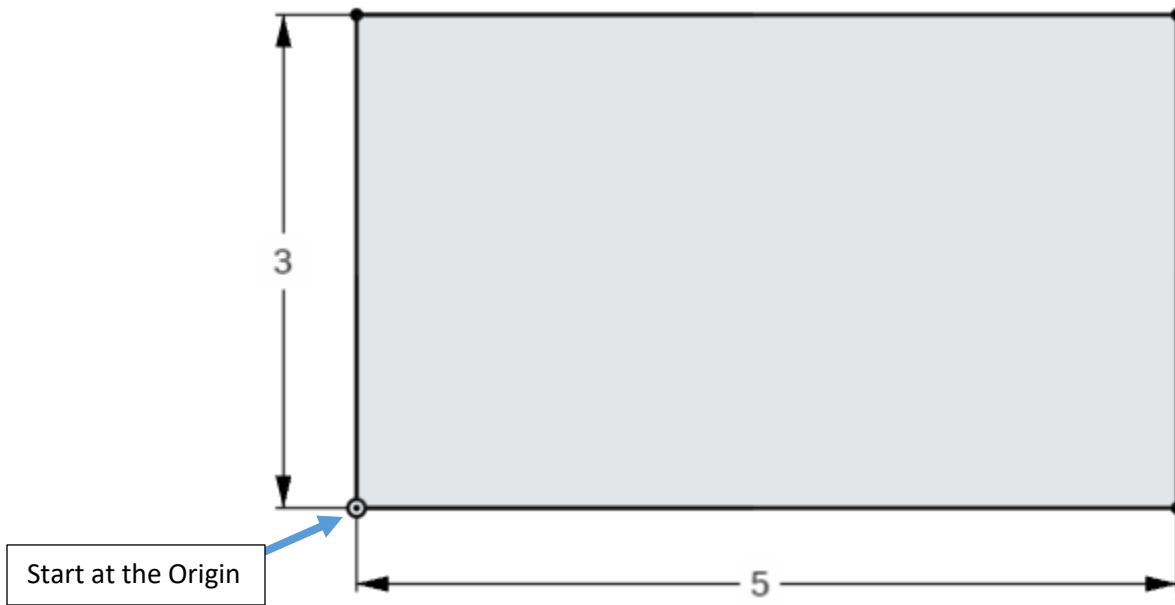


b. Option 2: Using the Dimension tool

- i. After Placing the Sketch Geometry
- ii. Select Dimension Tool 
- iii. Dimension is similar to Creo where the user may
 1. Option 1: Left Click on an Entity (Edge, arc, circle, etc.) (NOTE try to select in the middle of the entity NOT the endpoints) > Move cursor away from the edge > Left Click to place > Type in Value > Press Enter
 2. Option 2: Left Click on Two Entities > Move Cursor away from part > Left Click to place > Type in Value > Press Enter

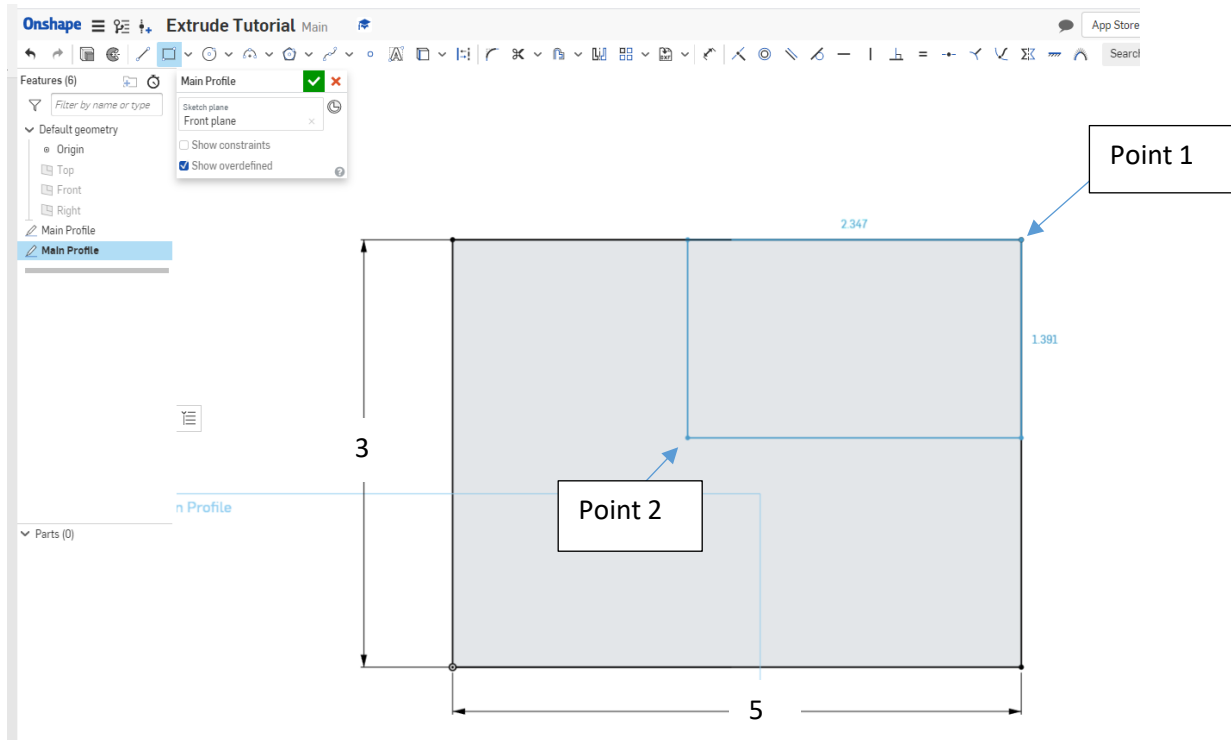


- iv. Change Values to the following
Change Values by double clicking on the value > Enter Value

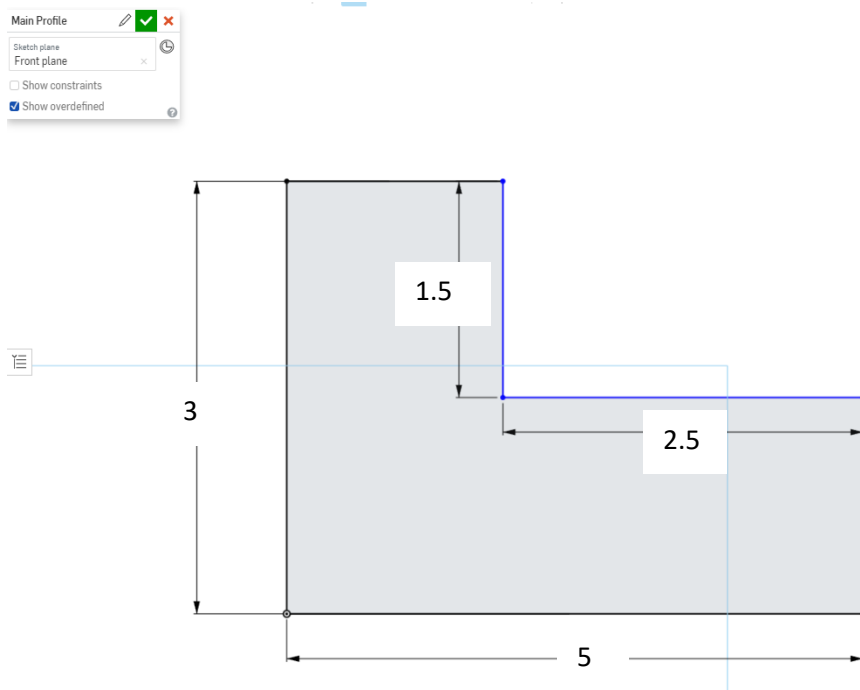


Rectangle should be 5 x 3 > Press f Key to autoscale

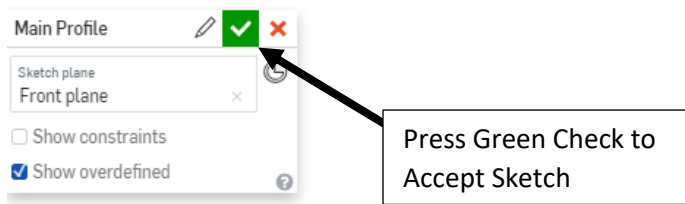
- f. Place a 2nd Rectangle starting in the top right corner of the 1st Rectangle > Drag Cursor for 2nd point in the 1st rectangle
- Set Length = 2.5
 - Set Height = 1.5



- g. Use the Trim Tool to remove the two edges shown to create the L-Block
- Note: Unlike Creo; OnShape does not have a dynamic trim tool. User will need to click on each edge one at a time to delete undesired line segments
- Part should look as follows



h. Press Green Check on the Sketch Block to accept Sketch



i. Press Shift + 7 to switch to Isometric View or Rotate part using Middle Mouse Button

j. Note User can go back into a Sketch at any time by

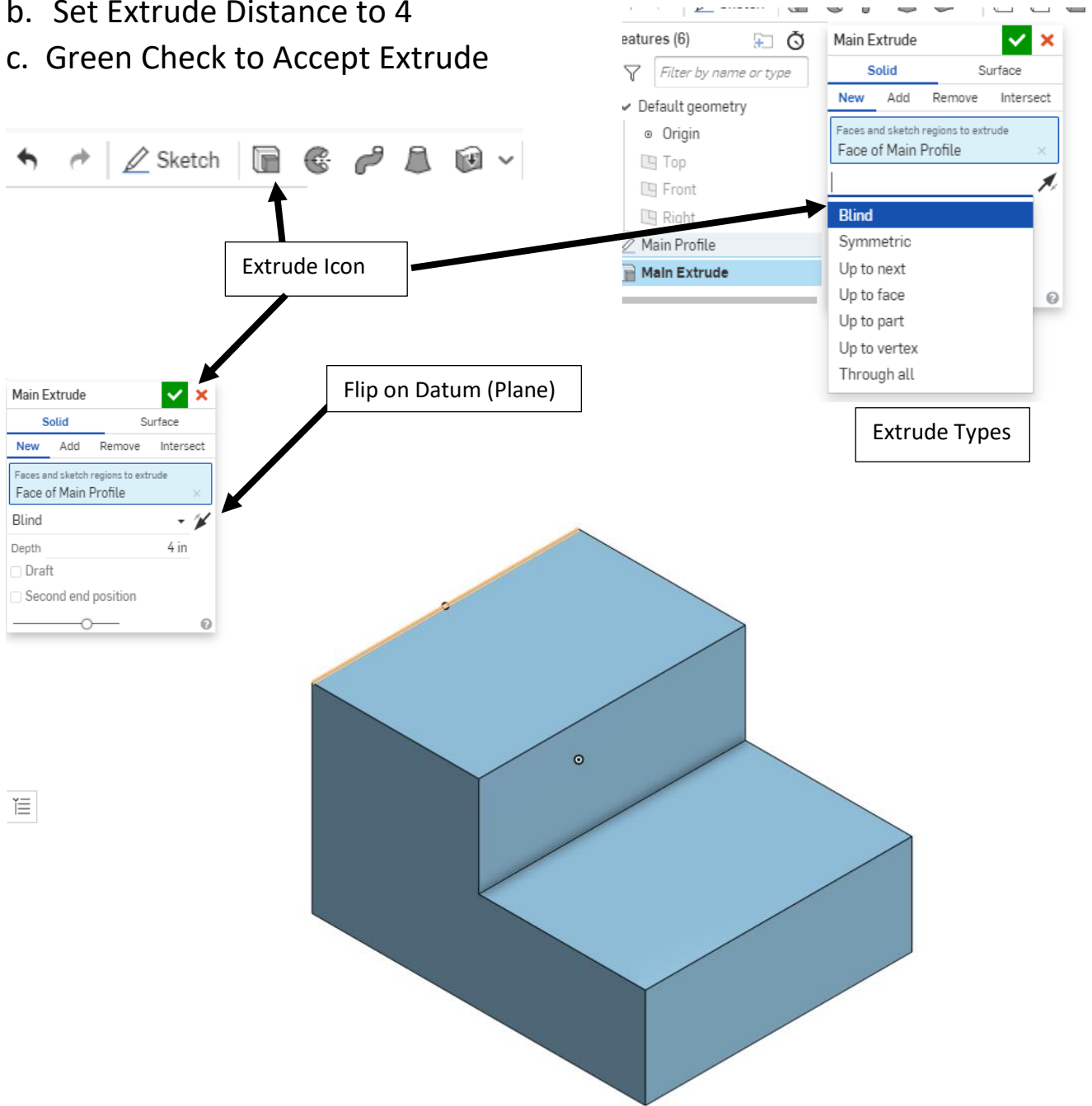
a. Option 1: Double Clicking on the Sketch in the Model Tree (Right Hand Side of the Screen)

b. Option 2: Left Click on Sketch in work area > Right Click > From Pop-up Menu Options > Select Edit Main Profile

Note: Green Check to accept and Exit the Sketch Profile

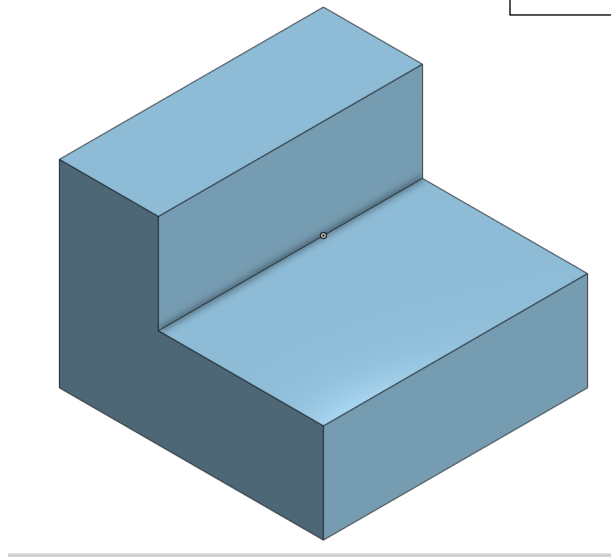
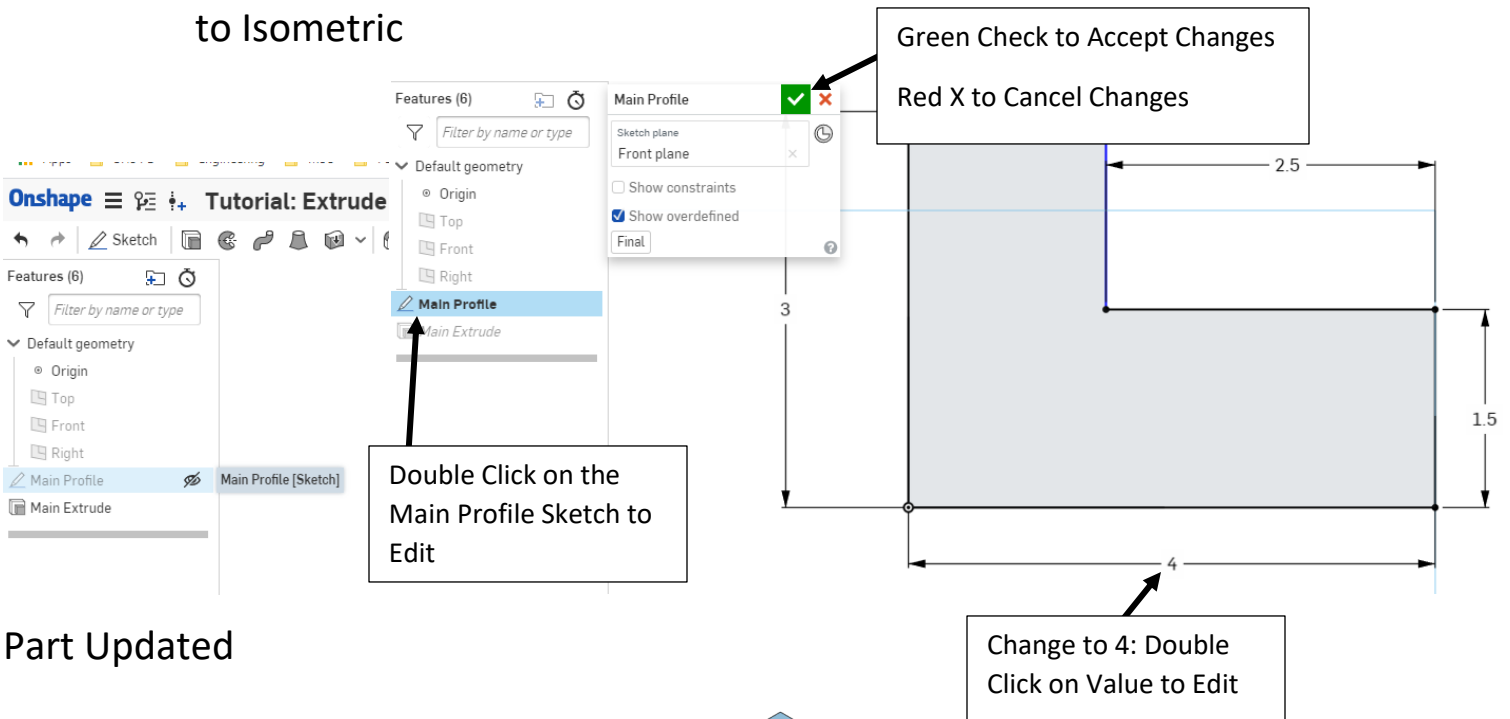
7. Extrude Profile: Select Sketch: Main Profile (either from model tree or in the work area) > Press Extrude Icon > Set the Following

- a. Change Name Extrude 1 to Main Extrude
- b. Set Extrude Distance to 4
- c. Green Check to Accept Extrude



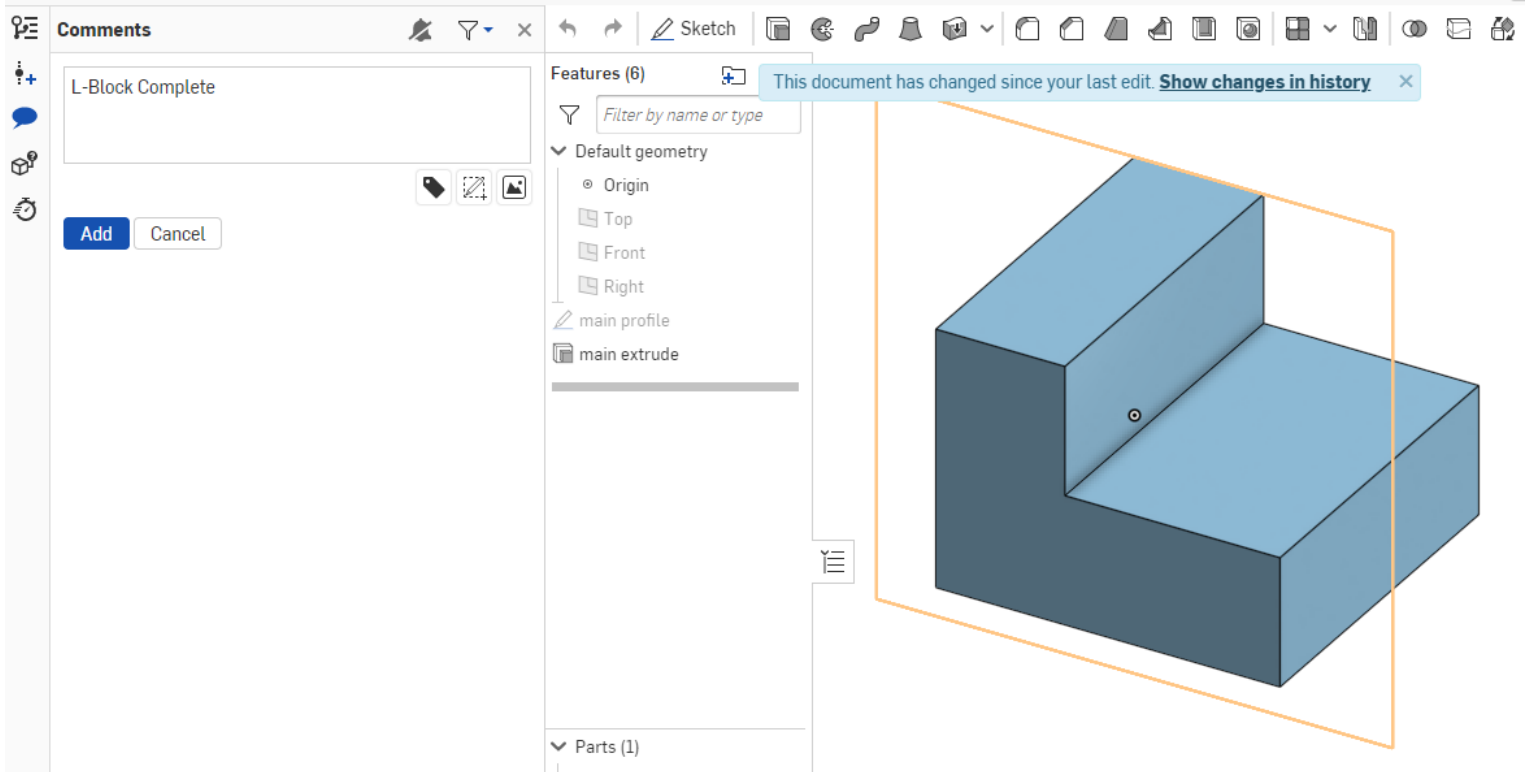
8. Changing Dimensions

- Double Click on Main Profile Sketch from the Model Tree > this will open the Sketch Profile > Press N to rotate perpendicular to the plane > Change the Overall Length to 4 > Green Check to Accept Change > Press Shift+7 to rotate to Isometric



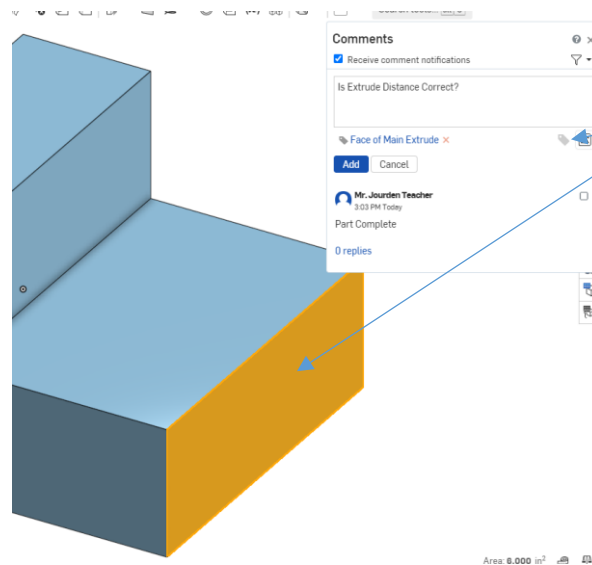
7. Comments: Comments can be used to ask questions about the part or to submit the object.

- a. Part Submission: Select the Comment Tool at the Top of the Screen > Pop Out Menu Appears > Check Receive Comment Notifications (this will provide a message on your activity wall that there is a comment to view) > Type in the Comment “L-Block Complete” > Select Add > Press X in top Left Corner to close Comment Menu



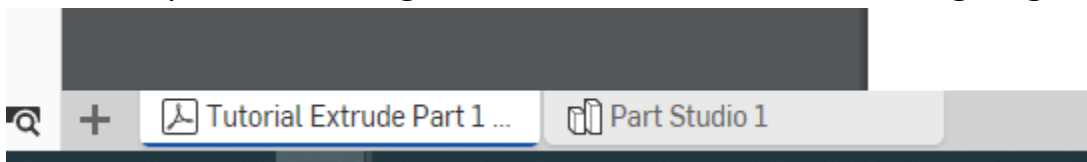
b. Feature Question: If the user has a question about a specific feature do the following:

Select Comment Tool > Check Receive Comment Notifications >
Select Tab Entity > Select the Feature the comment is about >
Type Question or Comment > Select Add

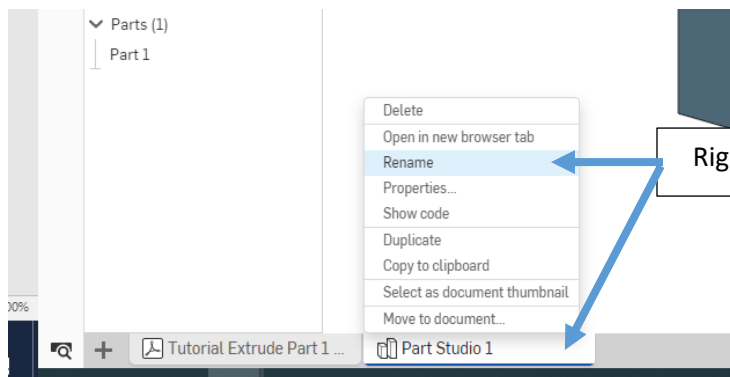


Select Tag Entity >
Select Feature to Tag >
Type Comment > Select
Add

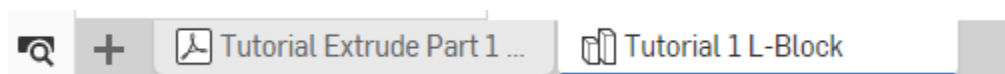
9. Renaming Tabs: It is recommended to provide unique names for all features within a CAD Part (i.e sketch names, Extrude Names, Tab names, etc.). The Default Name does not provide enough information to know what is going on with that feature.



a. Change Part Studio 1 Tab Name > Right Click on Tab > Select Rename >
Change Name to the following



Right Click on Tab > Select Rename



Project: Part Creation

1. Within the Tutorial Extrude Part 1 Create a New Part Studio > Rename Student Design
2. Sketch an object of user choice that uses a minimum of 10 lines and forms a closed profile
3. Extrude Profile
4. Create a Comment Describing what the object.

Submission: Share with Teacher > Create a Comment Stating the Tutorial is Complete

The image shows a screenshot of the Onshape software interface. At the top, a callout box labeled "Share Documents with Teacher" points to the "Share" button in the top right corner of the software. Below the software interface, the "Share settings" dialog box is open for a document titled "Assembly Tutorial". The dialog shows the document is private and owned by Matthew Jourden. A callout box labeled "Give Teacher Editing Rights" points to the "Can edit" dropdown menu. Another callout box labeled "Select Share" points to the "Share" button. A third callout box labeled "Type in Email Address" points to the input field containing "jourdem@brightonk12.com". The dialog also shows checkboxes for "Copy", "Link document", "Export", "Share", "Comment", and "Delete".