Onshape

FULL-CLOUD CAD

WORKSHOP 1
AN INTRODUCTION

BLUEPRINT3D

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CONTENTS

WHAT IS ONSHAPE?

WELL...

According to their website, there are 6 things you need to know about Onshape:

- Onshape is the first full-cloud 3D CAD system. It runs in a web browser and on any mobile device.
- Onshape uses cloud-native Documents, not files.
- You create parts in Onshape Part Studios and can have as many Part Studios as you need in a single Document.
- Onshape Assemblies use a new, simplified approach to mating parts.
- Onshape changes the way you collaborate. You can instantly share Documents and simultaneously work with your peers.
- Onshape streamlines data management with built-in version control

NOW IN ENGLISH...

Onshape is “CAD of the future”.

It runs in a web-browser, which means it can be used on any Mac or PC.

It saves your files for you inside your Onshape account, so you don’t need to worry about saving or losing files.

It is multi-user compatible, like Google Docs, which means you can invite others into your files to see what you’re doing or collaborate.

You never need to download large files, install programs or worry about licensing.

GETTING STARTED

CREATE AN ACCOUNT

Because Onshape is online, web-based and uses cloud-storage for your files, you will need to create an account to login and use it. To create an account, visit www.onshape.com.

Creating an account is free, but there are a few things we should note about the free account:

- Onshape uses a public/private file system (see below for explanation of public vs. private files).
- As a free user, you can have as many public files as you like, and have unlimited storage space. However, you're limited to 10 private files with a total storage allowance of 100mb.
- If you are somehow related to the education industry (I.e, you are a teacher or student of any kind enrolled in an education institution such as a school or university) you can sign up for Onshape Education account (onshape.com/edu) and are allowed unlimited private files and storage.

PUBLIC VS. PRIVATE FILES

Public files can be viewed by any Onshape user. They can view your workspace and make a copy of it to edit it for themselves, but cannot edit, change, remove or otherwise alter your original workspace.

Private files only appear on your "My Documents" list, no one else can view them or edit them at all, unless you share it with them through use of the share function.

CREATING A FILE

Once you have created your account and logged in at cad.onshape.com you will see some standard tutorials. You can do these in your own time if you like. At this point, click

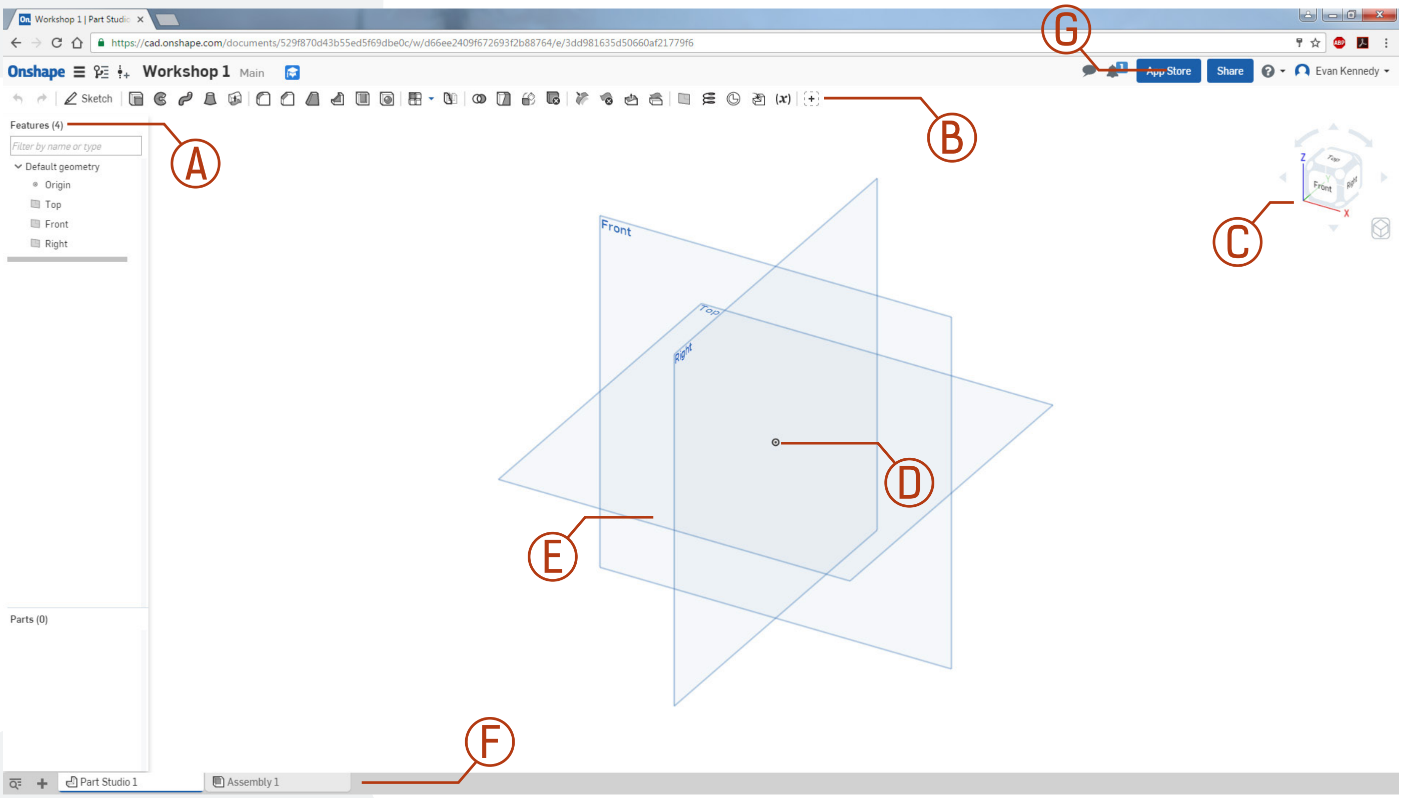
Select private or public file, it doesn't matter all for this workshop, and you can remove private files or change them to public later if you'd like. Give it a name. Once you've created a file, you will see your new Onshape CAD User Interface.

USER INTERFACE OVERVIEW

WHAT AM I LOOKING AT?

If you've never done any 3D modeling or CAD before, this may look a bit daunting, so let me break it down for you.

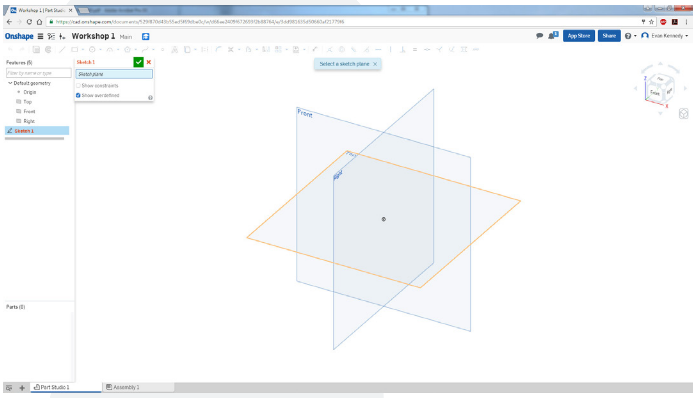
A Feature Tree
B Main Tool Bar
C View Control Box (VCB)
D Origin
E Base Reference Planes (BRP)
F File Management Bar
G Settings, Share, Community, Chat, Help



BASIC SKETCHING

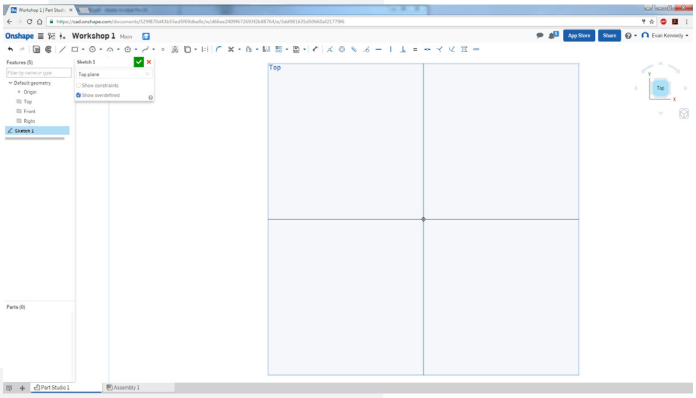
LET'S START MAKING STUFF!

Follow these steps to begin making your first part.



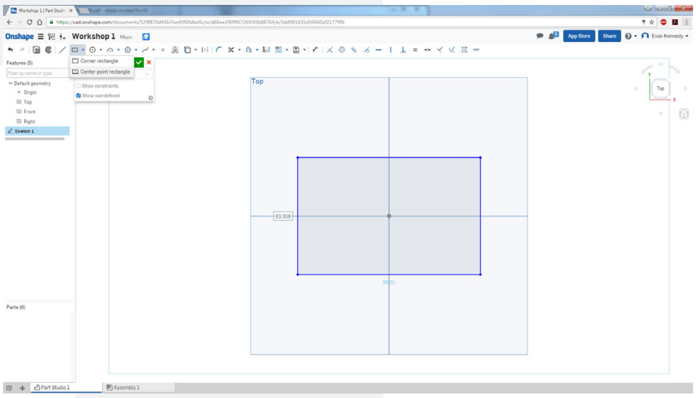
The functionality of CAD programs relies on creating "Sketches" and using various tools to either add or remove a solid object in reference to them.

Click the "Sketch" button on the Main Tool Bar, and you will be asked to "Select a sketch plane". Note that you can create a sketch on any BRP, Plane or flat surface of an existing object. A surface or plane will turn orange when you hover over it. Lets choose the Top BRP.

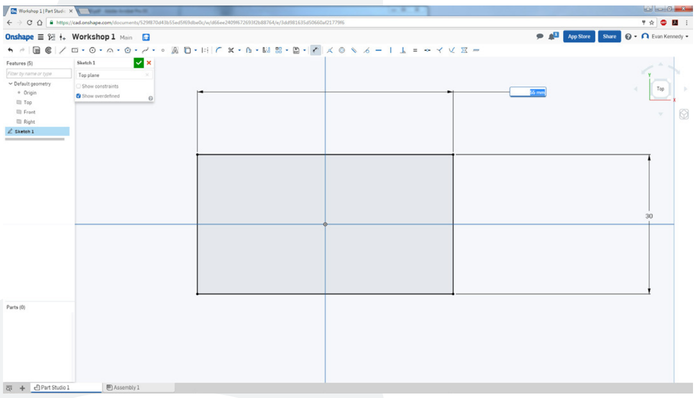


To make it easier to see what we're doing, let's change our view to the "top view", by selecting the "Top" surface on the VCB.

At this point, notice that the tools available on your Main Tool bar have changed. This is because you are now in Sketch Mode, and these are the tools we will use to draw stuff.



Let's start by drawing a rectangle.
Click the drop-down arrow next to the rectangle tool on your Main Tool Bar, and you'll see "Corner rectangle" and "Center point rectangle".
Experiment with both if you like.
To keep our shape centered around the Origin, to make things neater for later on, let's use the "Center point rectangle" and draw a rectangle of any size.

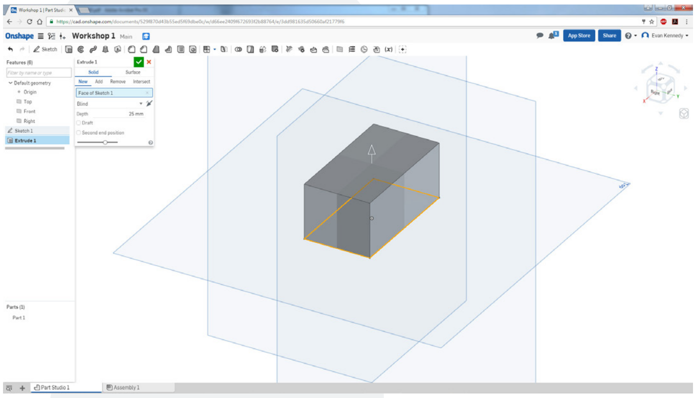


Dimensioning!!!!!!!!!!!!!!!!!
Dimensioning is the most important part of CAD, it's what separates it from regular 3D Modeling, and lets us input measurements which we can use to manufacture these parts.
Use the Dimension tool, to click the right hand side of your rectangle, then click out in the open space somewhere, and change this to 30mm.
Now change the top line to 55mm.
Press the green tick button to tell Onshape you're done with your sketch. Or you can not... Whatever.

EXTRUDING

TO TURN YOUR SKETCH INTO SOMETHING 3D

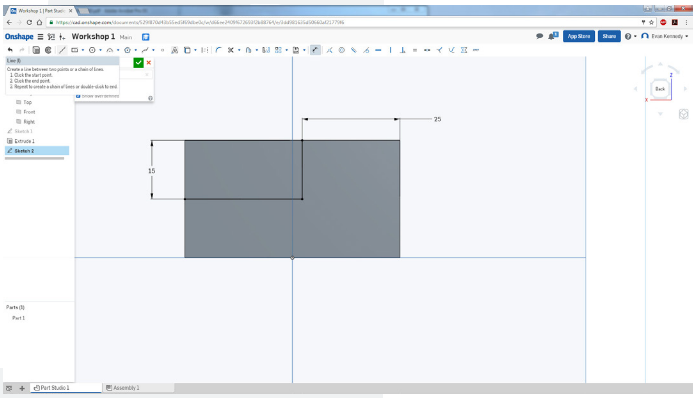
Follow these steps to extrude your sketch into a 3D shape, and then cut stuff away from it



To Extrude your sketch, click on the “Extrude” button on the Main Tool Bar. It looks like a white cube with a little grey cube sticking out the side. If you clicked the green tick in the last step, at this point you’ll need to select your sketch, either by clicking on it on the main screen, or on the Feature Tree.

Change the view either with your mouse or by clicking a corner on the VCB.

Change the “Depth” to 30mm, and click the tick.

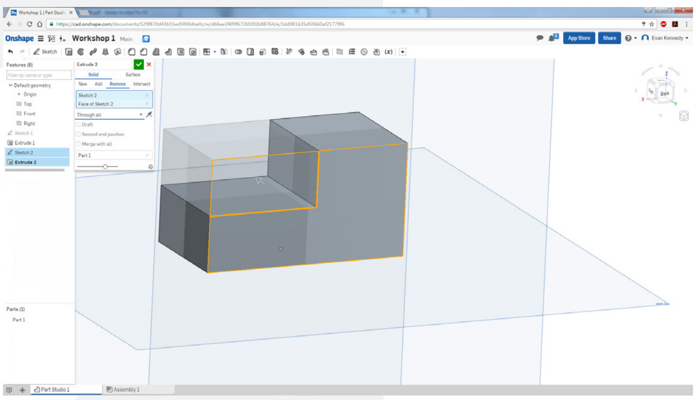


Create a new sketch on one of the large side faces of your box, and use the line tool to draw a shape like the one pictured here.

The dimension can also be used to specify distance between two things, rather than just length of one thing.

Use the dimension tool, click on the vertical line, and then click the right hand edge of your box, make it 25mm.

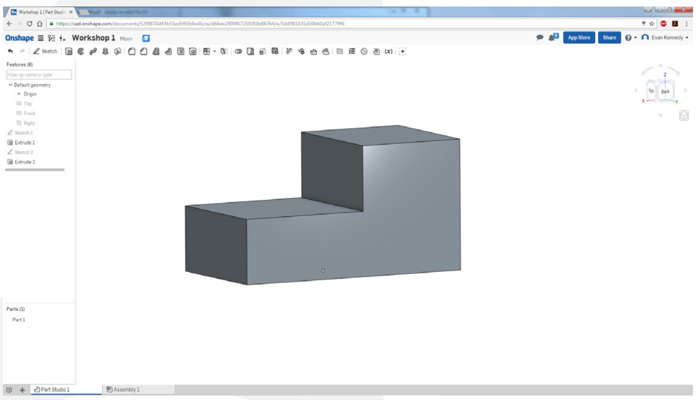
Do the same for the horizontal line and the top of your box, make it 15mm.



Use the Extrude tool again. This time, click on “Remove” in the Extrude options panel, and change it from “Blind” to “Through all”.

Because our sketch has divided the face into two shapes, a small rectangle and a backwards L shape, you’ll need to click on the face you’d like to extrude. If you accidentally select both, you can delete one by clicking it again, or by pressing the small X next to it in the Extrude options panel.

Click the Tick.



Were done for this step. But, now that more things are going on, lets turn off the vision of the BRP’s just to clean up the screen a bit.

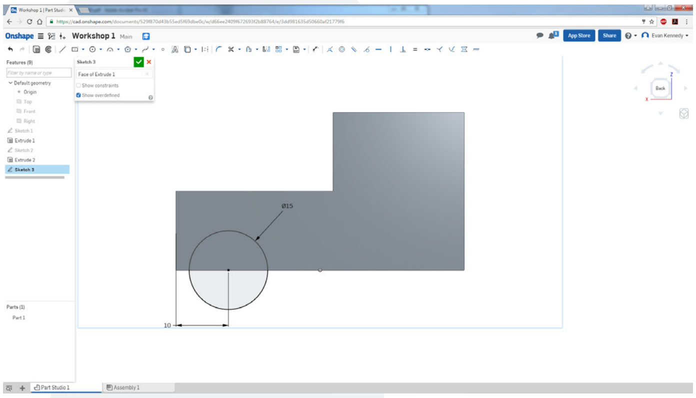
On the Feature Tree, hover over each of the BRP’s (Top, Front, Right) and little eyeball will appear, click it and the BRP will vanish.

You can still use the BRP’s for stuff by selecting them on the Feature Tree, and you can turn the vision of them back on the same way that we just turned it off.

PATTERNING INSIDE A SKETCH

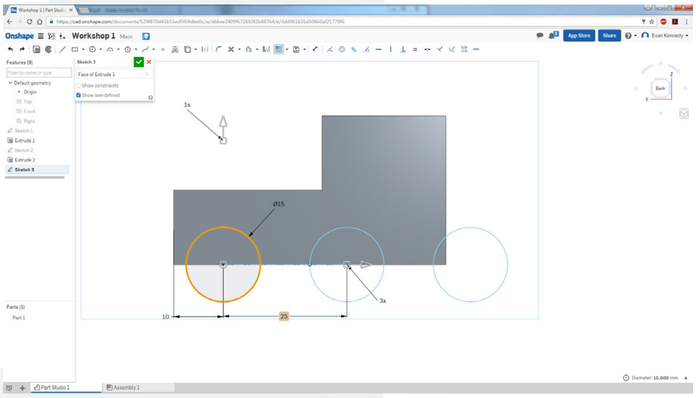
SOME MORE ADVANCED SKETCH STUFF

Lets make a sketch, and pattern it across the shape, inside the sketch.



Make a new sketch on the same side. Draw a circle using the “Centre point circle” tool, that attaches to the bottom edge of your shape. Use the dimension tool to set the distance between the circles center point, and the left hand edge of your shape to 10mm (make sure you select the circle’s center point, not the circle itself).

Now use the dimension tool to change the diameter of the circle to 15mm. To do this, click on the edge of the circle, and then somewhere in open space, and input 15mm.

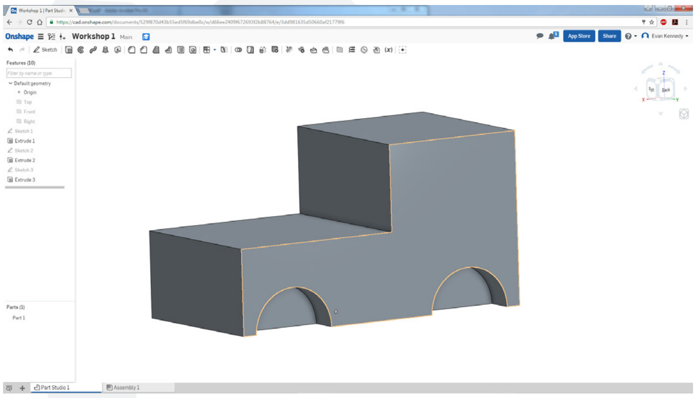


Click the “Linear Pattern” tool on the Main Tool Bar (4 small squares), and then click on your circle.

This will initiate a patterning sequence, which we can control by changing some variables. Our variables are amount of vertical and horizontal duplicates, and distance between them.

By default, vertical duplicates is set to 1x, horizontal duplicates is set to 3x, and distance between them is set to 25mm. Lets change it to 1x, 2x, 35mm, respectively.

To finish your pattern, press Enter on your keyboard, and then click in open space (weird, I know...). When you’ve finished your sketch, press the tick!



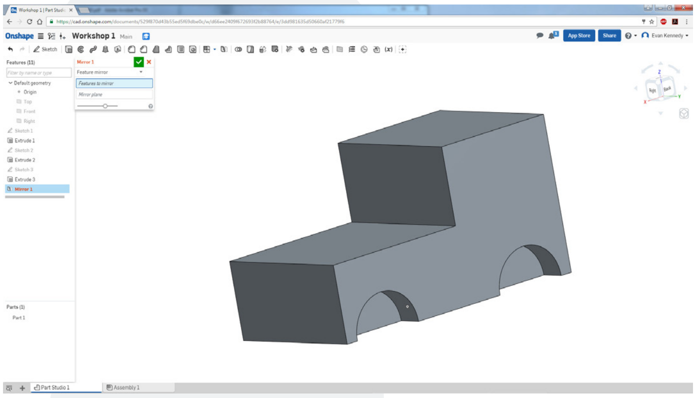
Use the Extrude tool again to cut both of these circles out to a depth of 5mm.

Remember to change your Extrude to “Remove”, or you’ll end up with things sticking out, instead of holes.

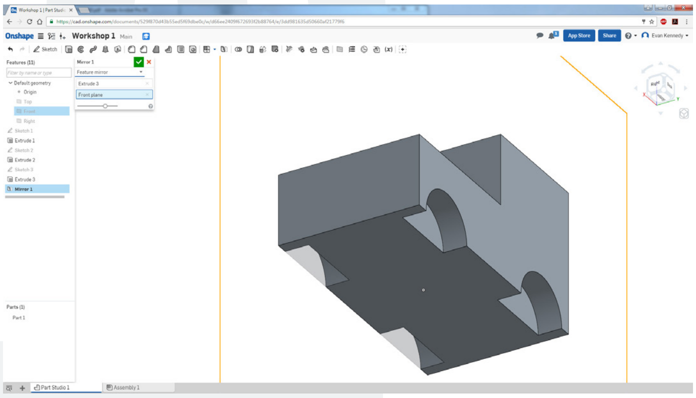
MIRRORING FEATURES

LETS KEEP THIS SHAPE SYMMETRICAL!

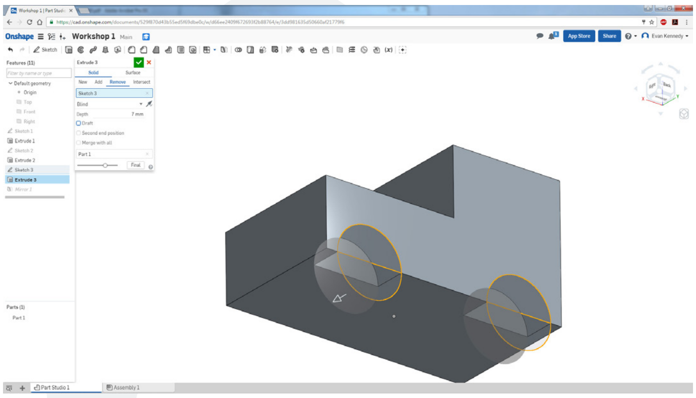
Lets use the Mirror tool, to copy our holes to the other side of the shape, and then learn about feature editing.



Choose the Mirror tool on the Main Tool Bar.
When the Mirror options panel pops up, immediately change “Part mirror” to “Feature mirror”, if you miss this, you’ll end up duplicating your entire shape.



Onshape now wants to know two things.
First, it wants to know which feature(s) it’s mirroring. On your Feature Tree, select “Extrude 3” (or whatever yours is called). You can also select the extrude by clicking on the holes we cut on your shape instead.
Second, it wants to know where the Mirror is. We know that a mirror is the surface smack-bang in the middle of us and our reflection. So, we need a plane or surface that is in the middle of our original extrude and where we want it’s new reflection to be.
We’ll use the Front BRP! Press the Tick.



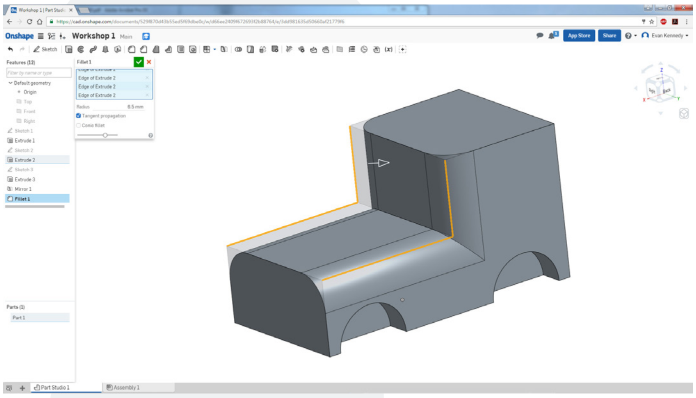
At any point in time, you can go back and edit sketches, extrudes, or any other step on your Feature Tree.
Right click on “Extrude 3” (or whatever yours is called) on your Feature Tree and select Edit. Lets change the Depth from 5mm to 7mm.
Press the tick.

Because “Mirror 1” on our Feature Tree, uses “Extrude 3” and mirrors it, you’ll notice the reflection of Extrude 3 has also just updated to the new depth.

FILLET AND CHAMFER

A SNEAKY WAY TO MAKE THINGS LOOK GREAT.

But seriously, they do look great.

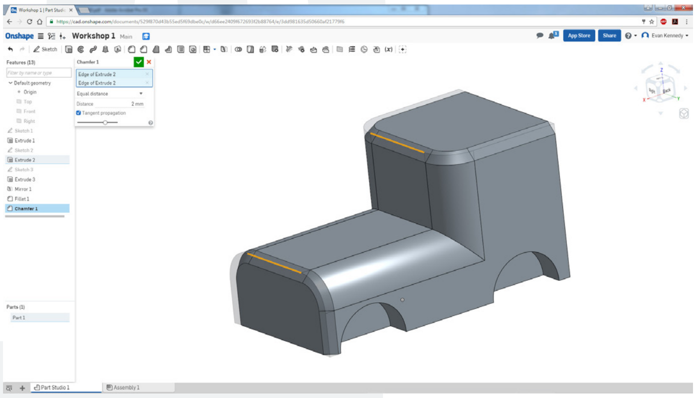


Click the “Fillet” tool on the Main Tool Bar. It’s a cube with a rounded edge.

Select an edge, and will become rounded. I chose the two long horizontal edges, and the two vertical edges in the middle of our shape.

Change their Radius to 6.5mm.

Press the tick.



Similar to above, let’s use the Chamfer tool (next to the Fillet tool) to angle some edges.

Select the top, horizontal lines that are parallel to the green Y-axis line on your VCB.

By default, the Chamfer tool makes a 45degree cut along the edge you select. You can change this by changing “equal distance” to “two distances”.

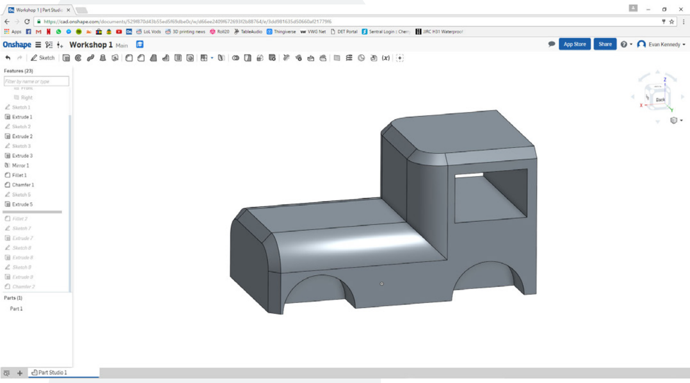
Let’s leave it as “equal distance” and change the Distance to 2mm.

Press the tick.

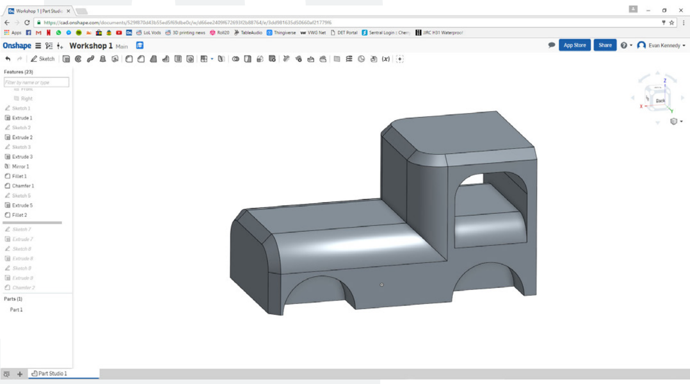
AND SOME MORE STUFF

INCASE YOU DON'T SEE IT....

We're making a sort of blocky, old school steam train. Without step by step instructions, see if you can do what I've done below. **Ask as many questions as you need...**

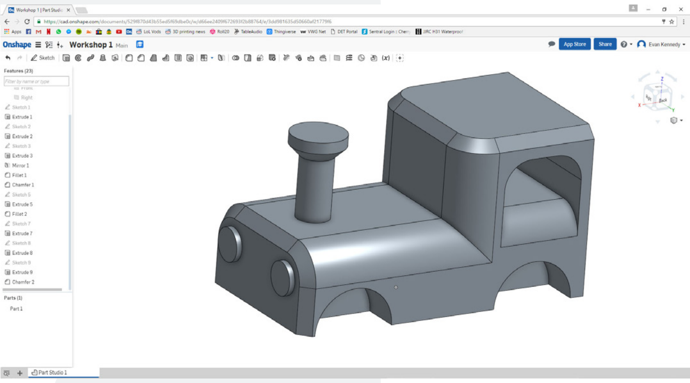


I've cut out a window hole.
The window is 15mm wide, 10mm high, sits 4mm down from the top of the train roof, and sits 2mm from the back of the train.



I've made my window look cooler somehow...

Hint: 5mm.



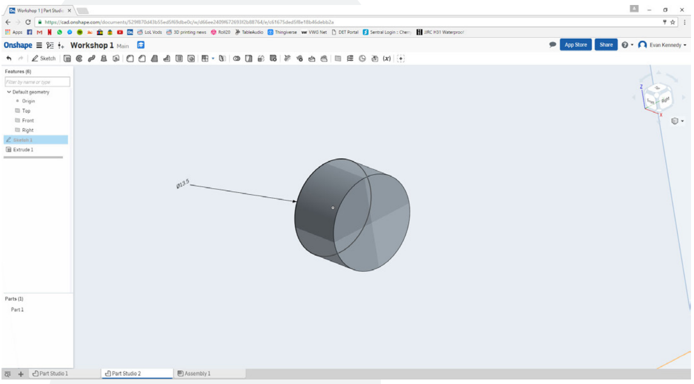
I've added headlights. 6mm diameter. 1mm depth. They share the centrepoint with the curved corners of the front of the train.

I also made a chimney flute thing. The long part is 6mm diameter, and it's center point is 20mm away from the windshield. It's 15mm tall.
The top part is a backwards extrude, 4mm deep, of a 10mm diameter circle, with a 2mm chamfered bottom edge.

MORE QUICK STUFF

LETS ADD SOME WHEELS

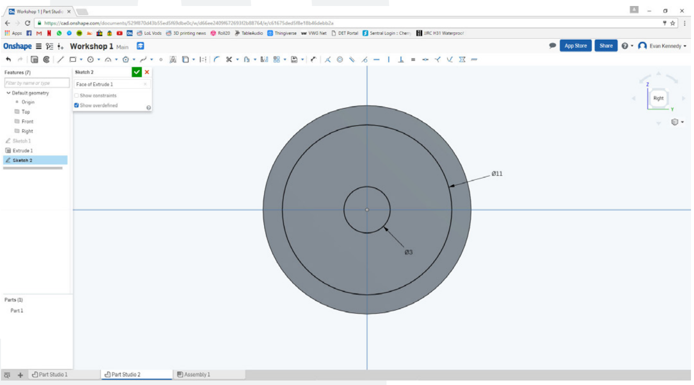
We'll use the circular pattern tool to make some cool wheels before we assemble them onto the train.



On the File Management Bar, choose the "+" and then "Create Part Studio".

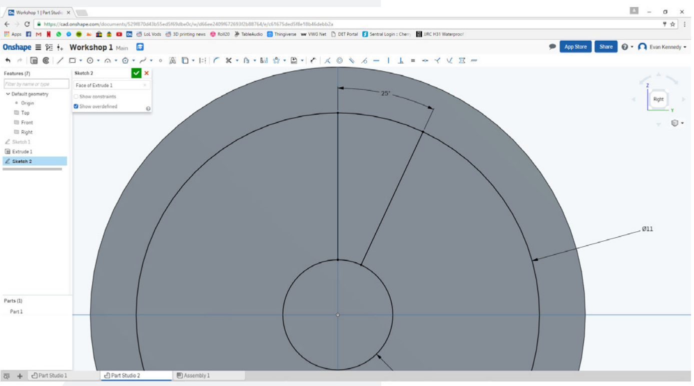
Inside Part Studio 2, let's make a wheel for our train.

Make a cylinder that is 13.5mm diameter, and extrude it 8mm out from the Right BRP.



Now lets make it look like a wheel, with some spokes.

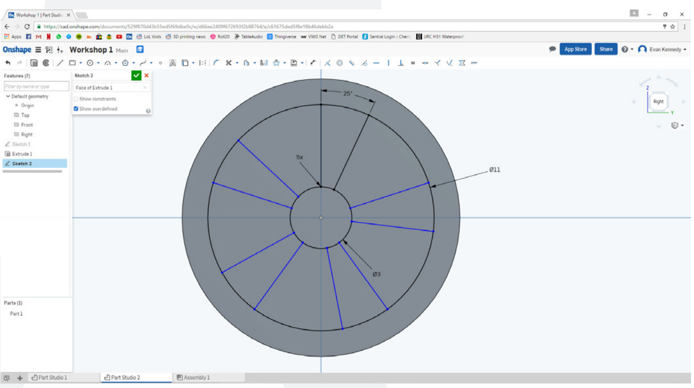
Make a new sketch on the face of the wheel, and draw two circles that are coincentric with the wheel. One at 3mm, and one at 11mm.



Draw a line from the top/center of the smaller circle, to the top/center of the larger one.

Draw a second line from the centrepoint of the circles, to the larger circle, just to the right of the first line, at about "1 o'clock".

Now, lets use the "Dimension" tool to set the angle between the lines to 25 Degrees. Then use the "Trim" tool to snip off the bottom part of the line.



Choose the "Circular pattern" tool on your Main Toolbar (in the dropdown box next to the rectangle pattern tool).

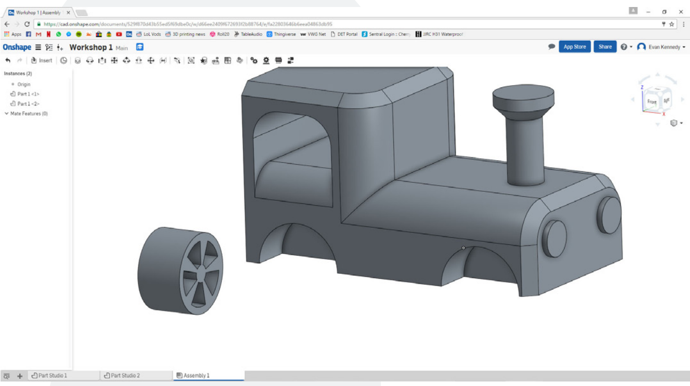
Select the two lines, and they will start duplicating and spinning around the circles they are attached to. You can control their quantity by changing the "3x" box to however many you want, let's make it 5x.

Click away from the drawing to accept the pattern and now extrude/remove your spoke holes!

ASSEMBLING

LETS JOIN OUR WHEELS

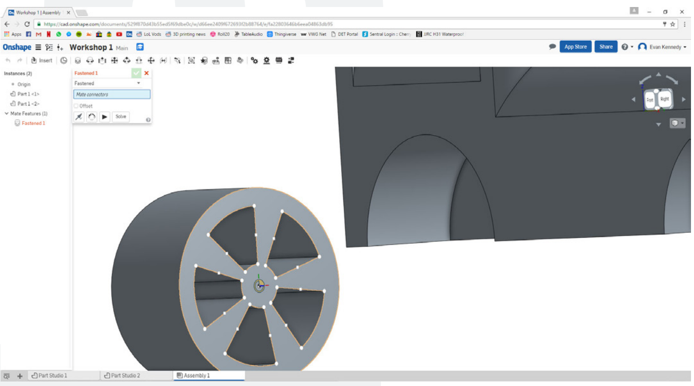
Let’s use the assembly and mate functions to join our wheels to the train.



On the File Management Bar, access “Assembly 1”.

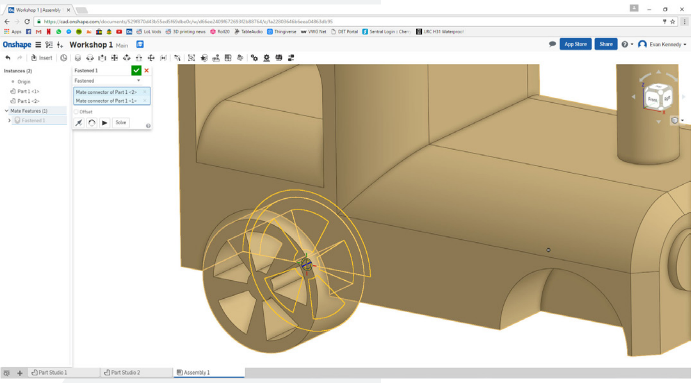
Use the “Insert” tool on the Main Toolbar to insert your trian body once, and your wheel once. It’s okay if they are placed in the wrong orientations.

Select the “Fastened Mate” tool from the Main Tool bar (looks like a cylinder with a straight join in the middle).



The mate tools ask us to select two points, which Onshape will join together for us.

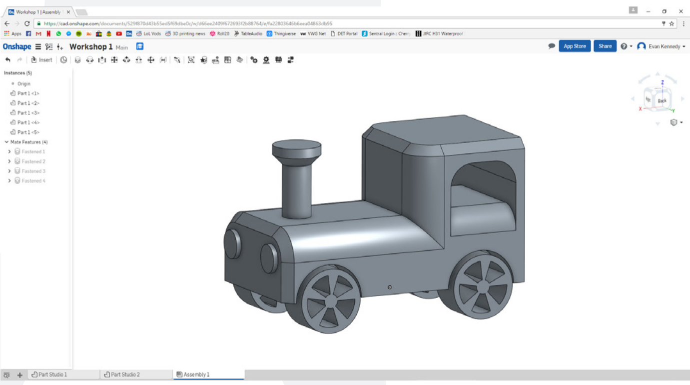
To select a point, hover over your objects and a few white dots will appear. You can hover over these white dots and a little Pacman logo with some coloured lines will appear. When selecting a point for the mate, pay close attention to which way the BLUE line on the funny Pacman logo is facing. When we select a second point, it will join the two blue lines together.



Select the centre point of the wheel. The blue line should be sticking out perpendicular to the face of the wheel.

Rotate your camera around so you can see inside the wheel arc. Select the centrepoint of the wheel arc (the middle of the line at the bottom of the wheel arc).

If the wheel is now facing the wrong way, choose the “Flip Primary Axis” tool on the mate popup window.

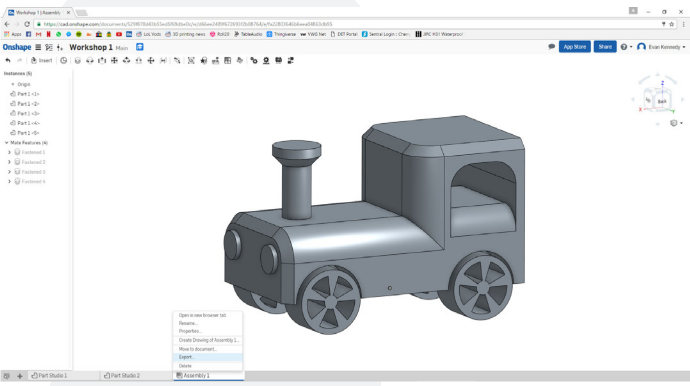


Now insert another 3 copies of the wheel, and mate them to the other wheel arcs in the same way.

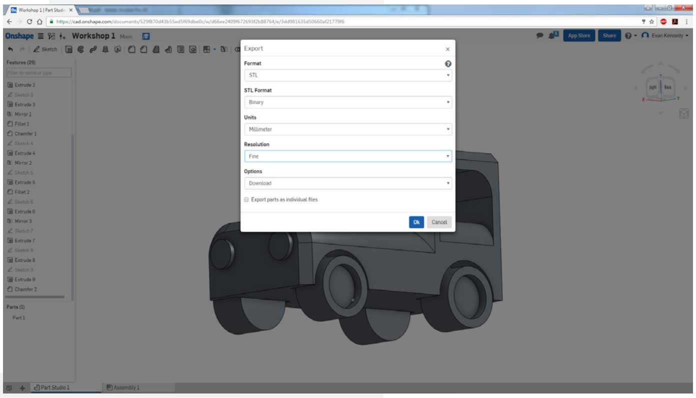
EXPORTING

WE'RE DONE!

Lets just export this train for 3D Printing, and then call it a day....



On the File Management Bar, right click on Assembly 1, and choose "Export".



Change the "Format" to "STL".

Then, change the "Resolution" to "Fine". When printing, this severely effects curves. This resolution, for example, dictates how many tiny straight edges make up your curve in the STL file.

Click "Ok", and in a few seconds, it will download the STL file into your downloads folder, ready for you to import into a 3D Printer slicing program... A lesson for another day.