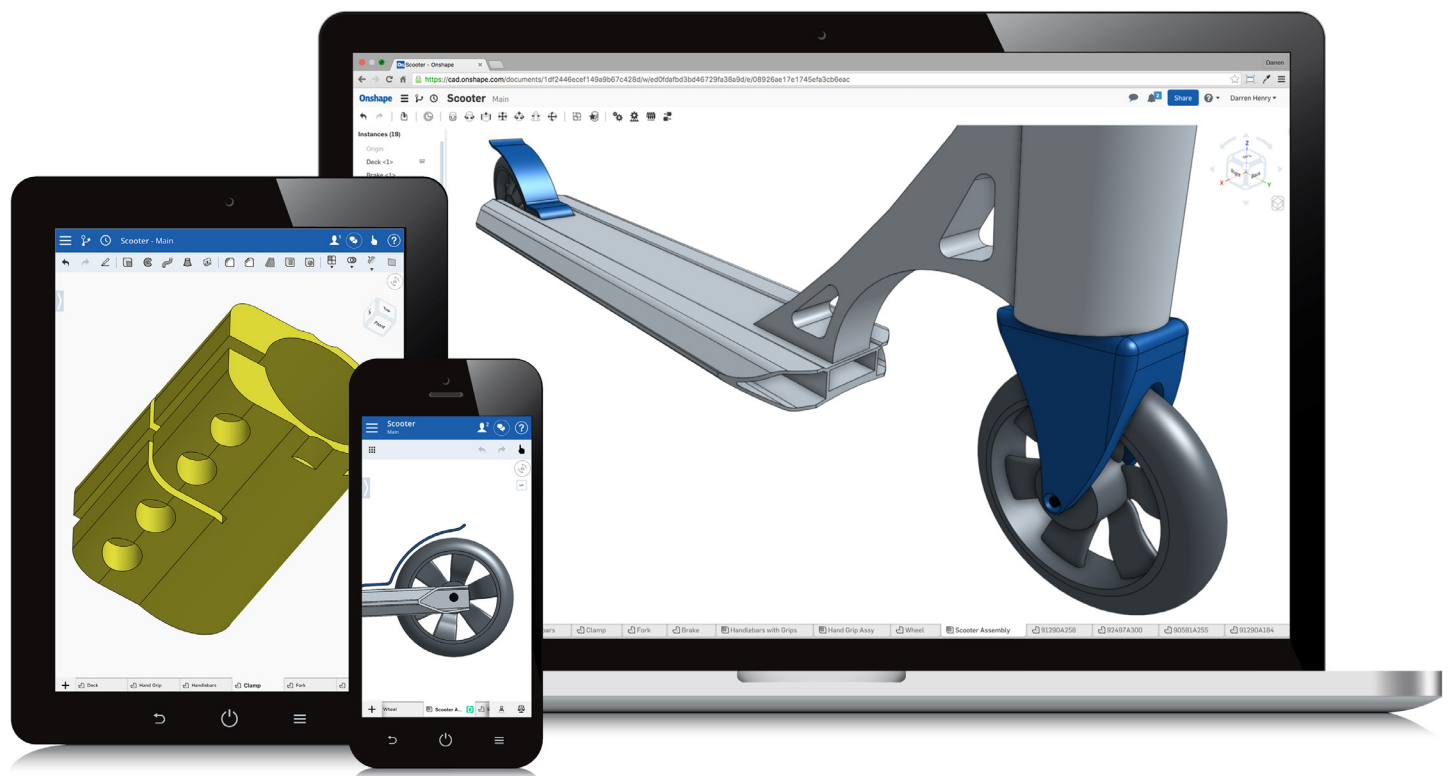


Getting Started with Onshape®

First Edition



Elise Moss,
Authorized Onshape Partner



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Getting Started

Explanation of how Onshape Works

When I started working in CAD in 1982, the emerging software – the new kid on the block – was AutoCAD created by a company headquartered in San Rafael, California named Autodesk. The interface consisted of a screen menu. You used a keyboard and short cut keys – like L for Line – to create your technical drawings, which consisted of lines, arcs, and circles. There was no rendering. There were no layers. There was no color. You drew in white on a black background. At the time, many drafters and designers said that CADD (Computer Aided Drafting/Design) software would never replace pencil and paper.

So, here we are in 2015 and how the world has changed. Today most CADD software is moving to the cloud. This means that users will download a small software program that allows them to log in to an account and save their work on remote servers. So, users are no longer tied down to a workstation or an office. You can access your work from any device anywhere in the world – as long as you have an internet connection.

Onshape takes it one step beyond this.

Firstly, Onshape is entirely 100% cloud-based. This means you don't have to install any software. You don't have to get permission from your IT department to put the software on your workstation, and you don't use up any space on your hard drive or jump drive or wherever you like to store files. Everything happens on Onshape's servers.

Because Onshape is entirely cloud-based, it is accessible from any device that uses a browser, like Firefox or Chrome. This means that you can pull up Onshape with any internet connection on your tablet, smart phone, laptop, or work station. Onshape even offers apps for Android and iOS devices.

Secondly, Onshape offers a *free* version to users. The free version is perfect for students and hobbyists. You can open an account without a credit card using an email address. The free account limits the amount of space you are provided, but you can always delete a project to free up space and start over.

Thirdly, because Onshape is entirely cloud-based, you are always using the latest version of Onshape. Onshape is regularly updated to fix bugs and add new features. Because Onshape is constantly being updated some of the screenshots you see in this text may look different from what you see on your screen. Try to ignore any differences. The tools should work the same way even if new features have been added.

Onshape works best using Chrome or Firefox browsers. It will not work at all with Internet Explorer. Onshape requires WebGL, which IE does not currently support.

Onshape does use "cookies," so make sure you set your browser to accept cookies. If the browser interface is not working properly, perform a hard refresh. (Hold down Command and Shift, then press R in any Apple or Mac operating system. For a WINDOWS operating system, press Ctl+F5.)

Setting Up an Account

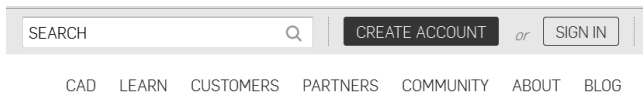
Estimated Time: 10 minutes

Objectives:

- Create an Onshape user account

To get started, go to onshape.com.

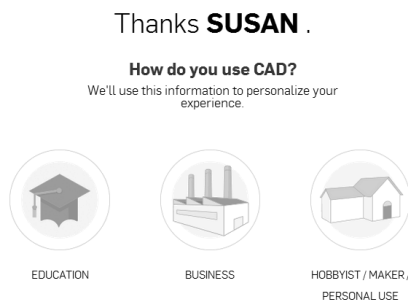
1. Select the **Create Account** link.



2. Fill in the form with your name and email address.

A screenshot of the account creation form. It consists of three input fields, each with a checkmark on the right side, indicating they have been filled. The first field contains the email address 'smoss@peralta.edu'. The second field contains the first name 'SUSAN'. The third field contains the last name 'MOSS'. Below these fields is a dark grey button with the text 'NEXT' in white.

3. Select the market segment that applies to you. I selected EDUCATION.



4. Fill out the form appropriately with your information.

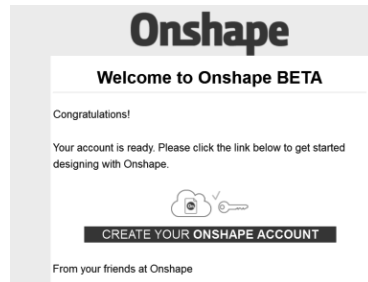
A screenshot of the account creation form. It consists of four input fields. The first field is a dropdown menu with 'Teacher' selected. The second field contains the text 'LANEY COLLEGE'. The third field is a dropdown menu with 'College or University' selected. The fourth field contains the text 'ADJUNCT FACULTY'. Below these fields is a dark grey button with the text 'Create Free Account' in white.

- Be sure to add alerts@onshape.com to your contacts or “whitelist” or their email might end up in your junk folder. If you can’t locate the confirmation email in your junk folder or access it, use the Reset Password link to get a new email sent to you after you have added Onshape to your contacts list.

THANK YOU FOR CREATING YOUR ACCOUNT

Check your inbox for an email from alerts@onshape.com. Click the link there to verify your address.

- Select the link.



- Set up a password for the account. Be sure to write it down somewhere! Press the Sign Up button and Onshape will immediately launch.

Email

Password

Name

Role

Company

- You start in the Tutorials and Samples area.

The screenshot shows the Onshape web interface. At the top, it says 'Your plan: Free' and '0 of 10 Private documents'. Below that, there's a 'Tutorials & Samples' section with a table of documents. The table has columns for Name, Workspace, Modified, Modified by, Owned by, and Size. The first document is 'Tutorial - Google Chrome Tips' with a size of 100 KB. Other documents include 'Sample - Wind Motor', 'Tutorial - Collaboration', 'Tutorials - Assembly and Mating', 'Tutorial - IRS Shutter', 'Tutorials - Sketching and Dimensioning', 'Tutorials - Sweep, Fillet & Chamfer', 'Tutorial - Extrude and Revolve', 'Tutorials - Draft, Shell and Direct Editing', 'Tutorial - Documents and Tabs', 'Tutorial - Version Control & Edit History', 'Tutorials - Mobile', 'Sample - Jergens Pro 5 Axis Workhold', 'Sample - Engine', and 'Tutorial (Assembly) - Woodworking Vise'.

On the right side, there's a 'Tutorial - Google Chrome Tips' window with a 'No preview available' message. Below that, there's a sidebar with 'Owner', 'Sharing', 'Created by', and 'Last modified by' information.

Onshape has two document states: Public and Private. If a document is public, anybody with an Onshape account can view and download the file. If you are the owner or creator of the document, you can share it with other users, but the users must have an Onshape account.

Both public and private documents can be shared. Sharing a document allows anyone who has been provided access to make modifications to the document. More than one user can edit a shared document at a time, similarly to how Google Docs works.

If you are not the owner/creator of a document, you cannot share it. If you find a public model that you like, you can send the URL to a friend, so they can see it. Since neither you nor your friend own the model, neither you nor your friend will be able to modify the file. Alternatively, you can make a copy of a public document and own the copy. Once you own the copy, you can share it with anyone you like.

You can transfer ownership of a document to another user. If you are finished working on a project, you can hand it off to another user and let them take over.

Navigating Documents and Workspaces

Estimated Time: 30 minutes

Objectives:

- Getting familiar with Onshape user interface
1. On the left side of the screen there is a list of links. These links organize your documents. The tutorials and samples area provides videos and sample models you can use to help you build your skills.

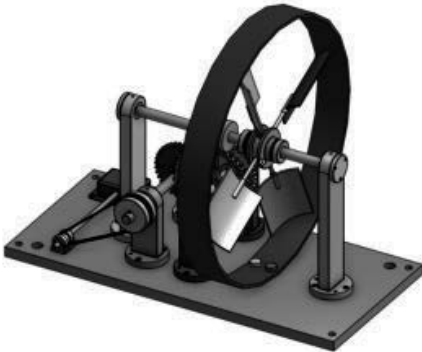
Recently opened
My documents
 Created by me
 Shared with me
Public
Tutorials & Samples
Trash

2. If you hover over one of the files labeled Sample, a small balloon appears indicating the file is a public document. Locate the Wind Meter document and select it.

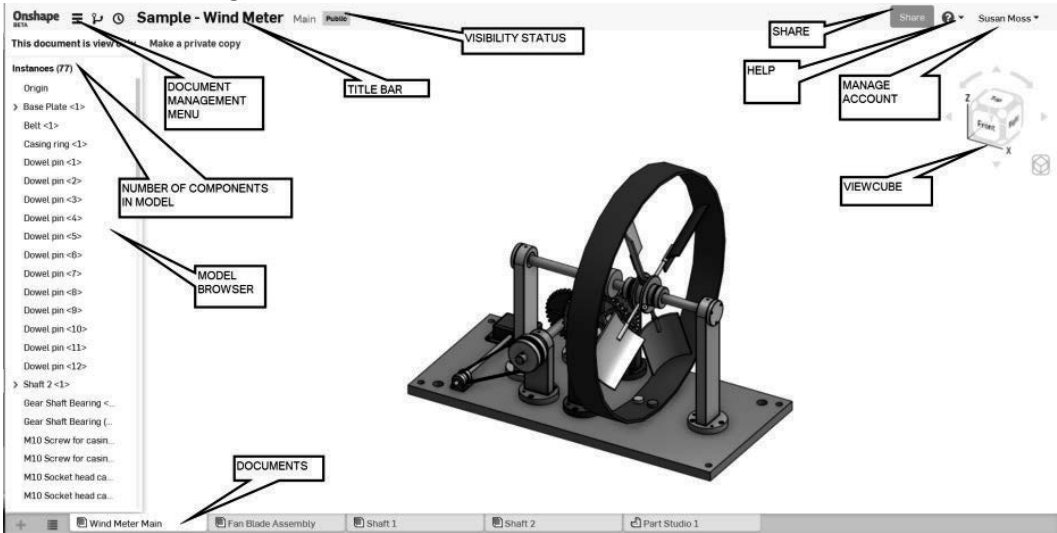
Click to open this public document. Because it is public, all Onshape users can view this document.

Public **Sample - Wind Meter** Main

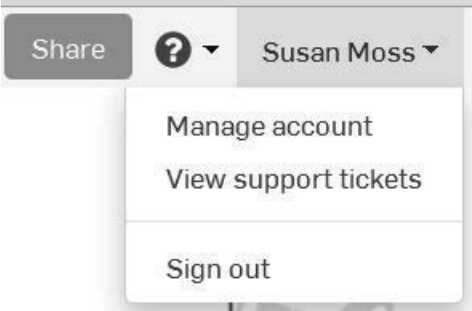
- 3. When you left click on the link there is a pause while the model is loaded into your workspace.



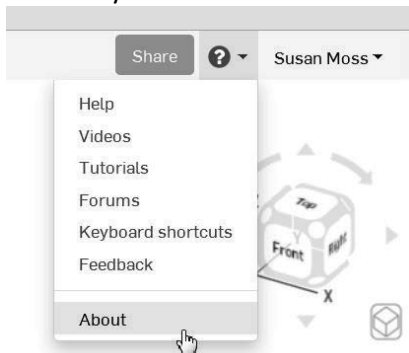
- 4. Take a moment to get familiar with the different screen areas.



- 5. In the upper right corner, you see your user name. If you select the down arrow next to your name, you can manage your account – which allows you to change the password or upgrade to a paid account. You can also view any support tickets.



- If you select the down arrow next to the Question mark, you are in the Help menu area. Select Keyboard shortcuts.



- A nice table of shortcuts appears.

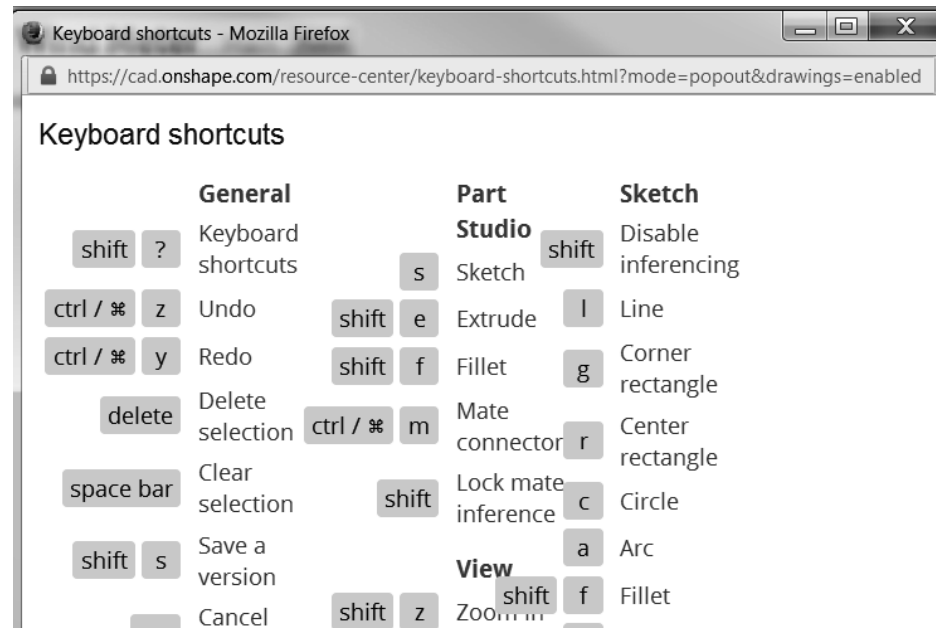
General	Part Studio	Sketch	Drawings
shift ? Keyboard shortcuts	s Sketch	shift Disable inferencing	w Zoom window
ctrl / ⌘ z Undo	shift e Extrude	l Line	d Linear dimension
ctrl / ⌘ y Redo	shift f Fillet	g Corner rectangle	shift r Radial dimension
delete Delete selection	ctrl / ⌘ m Mate connector	r Center rectangle	shift d Diameter dimension
space bar Clear selection	shift Lock mate inference	c Circle	n Note annotation
shift s Save a version	View	a Arc	ctrl q Update drawing
esc Cancel command	shift z Zoom in	shift f Fillet	l Line
enter Accept command	z Zoom out	m Trim	ctrl s Display sheet menu
shift enter Accept/repeat command	f Zoom to fit	x Extend	PgDn Next sheet
Assembly	← → ↑ ↓ Rotate	o Offset	PgUp Previous sheet
shift Lock mate inference	shift ← → ↑ ↓ Pan	u Use	Home First sheet
ctrl / ⌘ c Copy	shift 1 Front view	d Dimension	End Last sheet
ctrl / ⌘ v Paste	shift 2 Back view	i Coincident	
m Mate	shift 3 Left view	b Parallel	
ctrl / ⌘ m Mate connector	shift 4 Right view	t Tangent	
i Insert dialog	shift 5 Top view	h Horizontal	
s Enable snap mode	shift 6 Bottom view	v Vertical	
a Flip alignment	shift 7 Isometric view	e Equal	
q Change quadrant	n Normal to		
	p Hide/show planes		

- Select the Arrow in the top right corner of the keyboard shortcuts window.

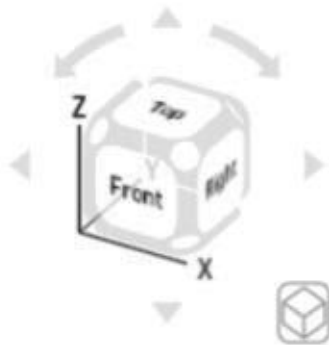


- A webpage will appear that can be printed out. Simply press Ctrl+P and select the desired printer. You can then pin this up to your cubicle wall or keep it handy for easy

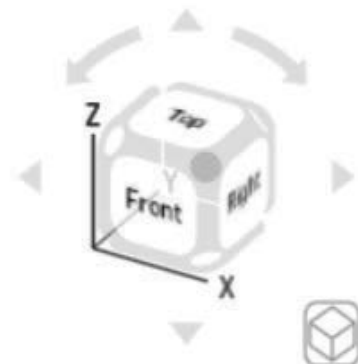
reference.



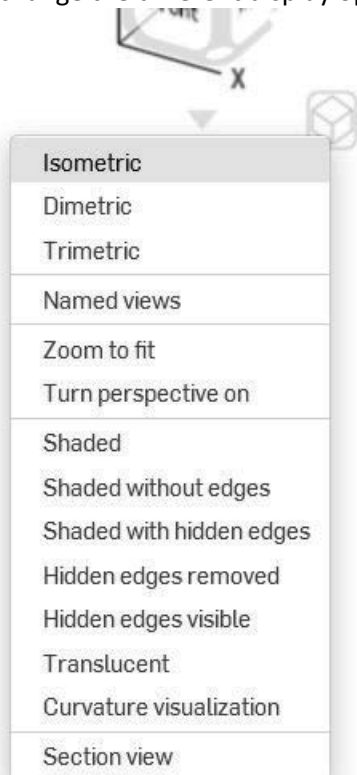
- The Share button in the upper right corner will not work when you are in a public document.
- The orientation cube located in the upper right of the display window allows you to modify your view easily.




- Notice that as you hover over different areas of the orientation cube it will highlight. Left click to select the shaded area as the new view orientation.




13. If you left click on the smaller cube (display cube), you will get a shortcut menu where you can change the different display options as well as default orientations.

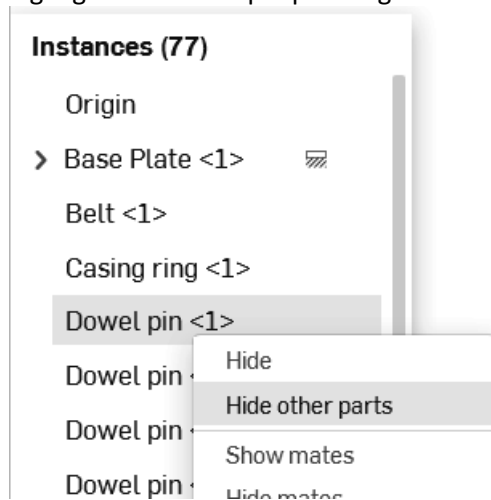


14. In the model browser, you see a list of parts and sub-assemblies. The Base Plate has an icon next to it to indicate it is grounded or fixed in place.  The Base Plate, belt, casing ring, and dowel pins are parts.

Instances (77)

- Origin
- > Base Plate <1> 
- Belt <1>
- Casing ring <1>
- Dowel pin <1>
- Dowel pin <2>
- Dowel pin <3>
- Dowel pin <4>
- Dowel pin <5>
- Dowel pin <6>
- Dowel pin <7>

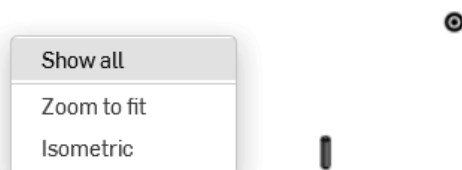
15. Highlight the Dowel pin part. Right click and select **Hide other parts**.



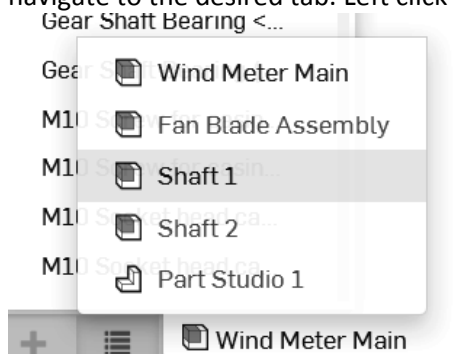
16. Visibility of all the parts is turned off. The small dot towards the right is the origin.



17. Right click in the display window and select Show All to restore visibility of the model.



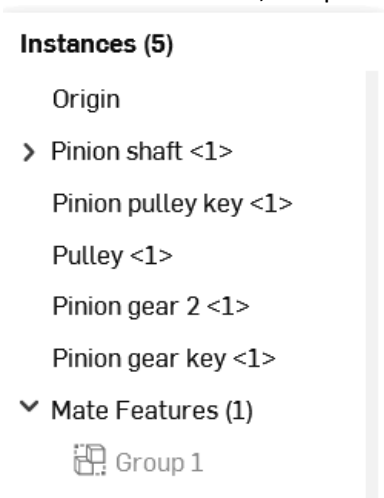
18. In the lower left of the screen, there is a small menu icon. If you left click on the icon, you get a quick list of all the tabs available in the model. This allows you to quickly navigate to the desired tab. Left click on Shaft 1.



19. The Shaft 1 tab opens.



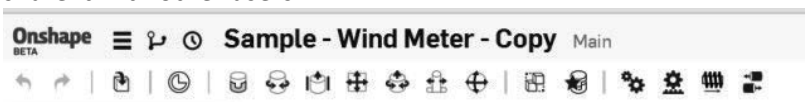
20. In the model browser, five parts are listed.



21. Select Make a private copy at the top left corner of the screen.

This document is view only. Make a private copy

22. You now have a copy of the document that you own. You may modify the copy and share it with other users.



23. Select the Onshape logo located on the upper left of the screen. A popup will appear that says Close document. Left click to close the document.



24. The copy is listed under My documents. You can see the file size and that it is currently not shared. You can also see how much space you are using in your storage area.

Your plan: **Free** [Learn more](#) 1 of 10 Private documents 9 MB of 100 MB Private storage 9 MB of 5.00 GB Total storage

Create

My documents

Name	Workspace	Modified	Modified by	Owned by	Size
Sample - Wind Meter - Copy	Main	2:57 PM Today	me	me	9 MB

Recently opened

My documents

- Created by me
- Shared with me
- Public
- Tutorials & Samples
- Trash (0)

Sample - Wind Meter - Copy

Owner: me

Sharing: Not shared

Sharing a Document

Estimated Time: 5 minutes

Objectives:

- Collaboration
1. Login to Onshape. Locate the Wind Meter which was copied to the My Documents area. Left click to open the document.

Onshape BETA

Your plan: **Free** [Learn more](#) 1 of 10 Private documents

Create

My documents

Name	Workspace
Sample - Wind Meter - Copy	Main

Recently opened

My documents

- Created by me
- Shared with me

2. Left click on the Share button.

3. Enter your instructor's email address or the email address of someone with whom you wish to share the document. *Anybody you share a document with is called a*

Collaborator. Press **Share**.

Share settings for ×
Sample - Wind Meter - Copy This document is owned by: Susan Moss

This document is private. Only users listed below can access this document.

Individuals Public

Can edit & share Share

Include a personal message

Share with Onshape support Close

4. The Collaborator's email is now listed. Press **Close**.

This document is private. Only users listed below can access this document.

elise_moss@mossdesigns.com Can edit & share ×

Individuals Public

Can edit & share Share

Include a personal message

Share with Onshape support Close

Individuals Public

Can edit & share

Include a personal message

This document is shared with Onshape support

- Owner
- Can edit & share
- Can edit
- Can view & comment
- Can view

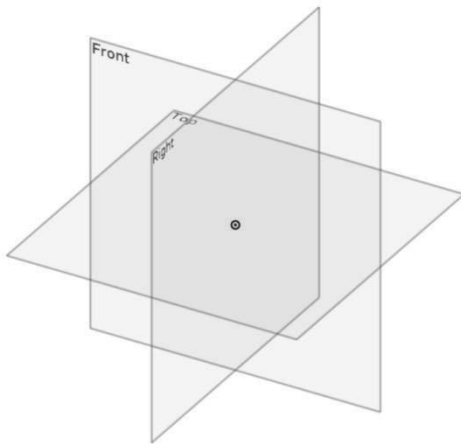
If you want to transfer ownership to another user, select **Owner** from the drop-down list. Then left click on the Share button.

For the first project, we will design a custom signet ring that can be rapid prototyped. Many schools have their own 3D printers. If you do not have access to a 3D printer at your school or office, you can download your model file in an stl format onto a jump drive and take it to your local office supply store, such as Staples or Office Depot. These stores have 3D printers available and can print your model for a small charge.

There is a tutorial that walks you through how to create the stl file once you are satisfied with your design.

When you first create a document, there are three planes in the display window. The planes are named Front, Top, and Right. These correspond with the front, top, and right views in a 2D drawing.

The planes are displayed as thin rectangles. Planes are infinite, but there really isn't a good way to display an infinite plane. Think of the plane as a limitless piece of paper.



In the browser, under Default geometry, the Origin and the three planes are listed.

▼ Default geometry

Origin

Top

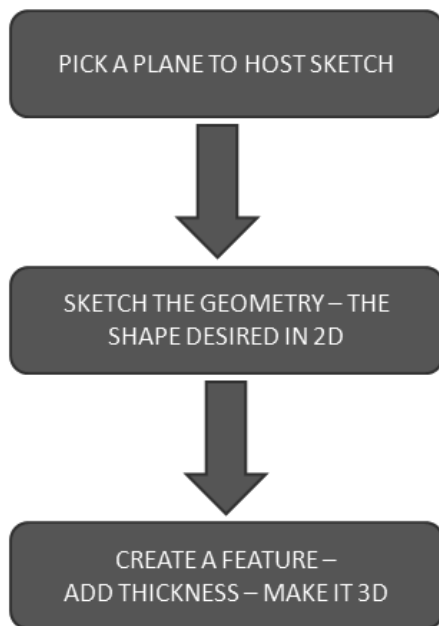
Front

Right

The origin is the 0,0,0 ($x=0$, $y=0$, $z=0$) point in the 3D universe where you are creating the design.

The origin basically allows you to locate the geometry in relation to 3-dimensional space. If you don't provide a location for the lines, circles, and arcs, it could move around and the results would not be pretty.

Each part is created by drawing a sketch on a plane, then creating a feature from the sketch, and then adding on.



There are four primary feature types:

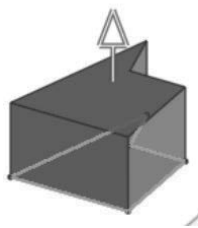
- Extrude
- Revolve
- Sweep
- Loft



For an Extrude, you place a sketch and then add height or thickness. Common objects that use extrudes are boxes.



Common objects

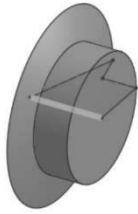


For a Revolve, you place a sketch and then revolve it around an axis. An axis can be an edge or

line.



Common objects that use revolves are wheels and gears.



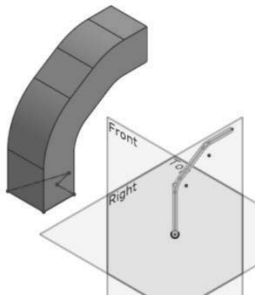
For a Sweep, you create two sketches. One sketch is the profile and one sketch is the path. The

profile travels along the path to create the 3D shape.

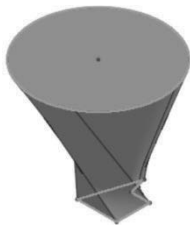


The two sketches are usually perpendicular to each other. The path would be on one plane, and the profile would be on a different plane.

Common objects that use sweeps are handles, ducts, and cords.

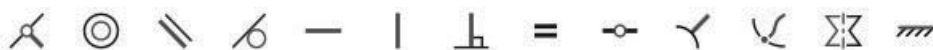


A Loft is also called a blend. A loft is created by placing two or more sketches on offset planes. Each sketch is usually a different shape. The loft blends the sketches together to create a unique 3D feature. Common objects that use lofts are faucets and bottles.



When you model a part, you usually create more than one feature. A part might include an extrude, a revolve, and a loft or it might just be a single sweep. It depends on how complicated the part is. As you work through this text, some parts will be fairly simple and other parts will be quite complex.

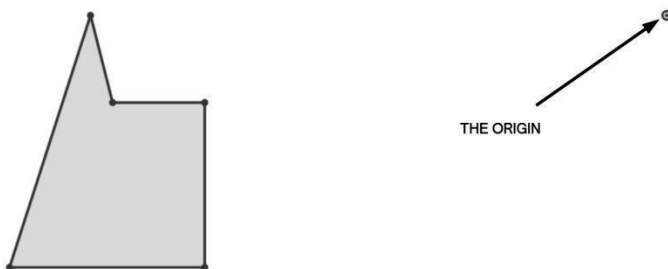
When you create a sketch, you draw the geometry – lines, arcs, and circles. Then you use sketch constraints to determine how the elements interact. There are two types of sketch constraints – geometric and dimensional. Geometric constraints are horizontal, vertical, perpendicular, parallel, concentric, coincident, tangent, equal, midpoint, symmetric, etc.



Geometric constraints are located on the sketch ribbon. To add a sketch constraint to geometry, select the element(s) and then select the desired constraint.

Sketches can be under-defined, fully-defined, or over-defined. Under-defined means the sketch does not have all the information – dimensions or geometric constraints – to fully constrain or define all the elements. Fully defined means that all the geometry has been fully designated. Over-defined means that there is duplicate information in the sketch.

When a sketch element is displayed in blue that indicates it is not fully defined. When a sketch element is defined in black, it is fully defined.

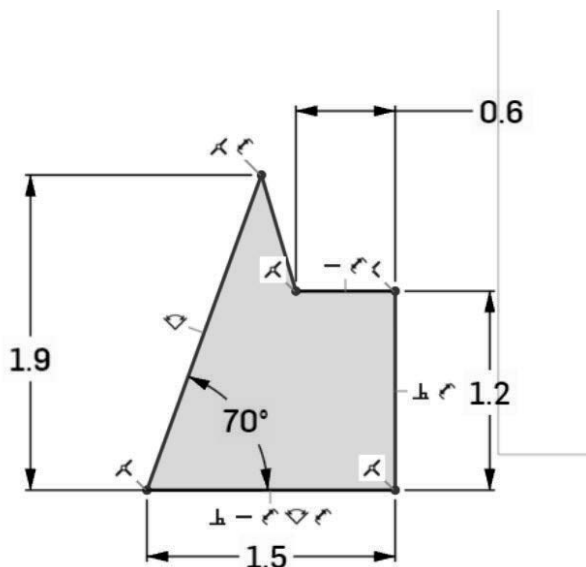


Because the sketch is located in 3-dimensional space, the elements always need to be located in relation to the origin in order to be fully defined. Otherwise, they are just floating in space.

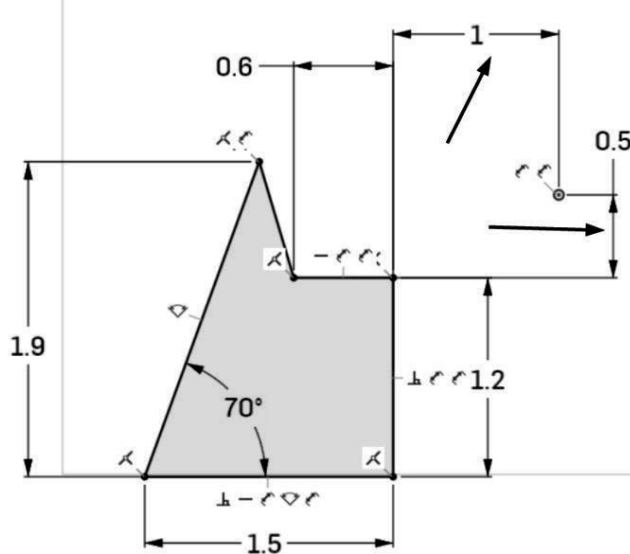
As dimensions and geometric constraints are placed, the geometry changes color.

Black represents geometry that is fully defined and blue indicates geometry that is under defined.

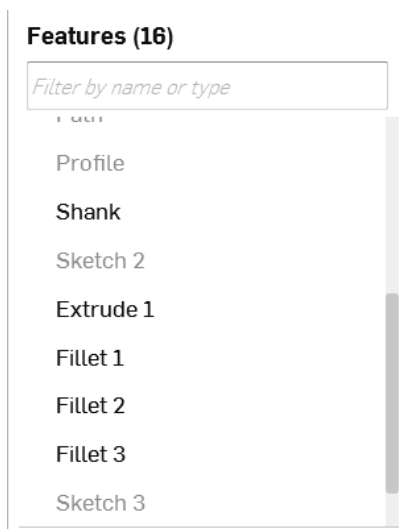
Some of the elements in this sketch are displayed as blue because it still isn't indicated where it is located in relation to the origin.



Once the two dimensions are added to indicate where the elements are located relative to the origin, the sketch will display in black – fully defined.



The browser keeps track of the sketches and features as they are placed. You can go back and modify sketches and features at any time. Sketches can be moved to a different plane, if desired as well.



The Onshape document consists of multiple tabs.



Right now there are three types of tabs that can be generated:

- Part Studio – where parts are defined
- Assembly – where parts are assembled
- Drawing – 2D drawings to be annotated

There is no limit to the number of tabs in any single document. So, even with a free account, you could use a single document to store all your parts and not exceed the document limits of a

free account.

