

NEW
chapters cover the app
store and additional features

Getting Started with Onshape®

Second 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 2016 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

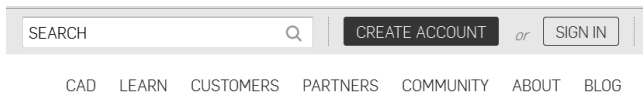
Estimated Time: 10 minutes

Objectives:

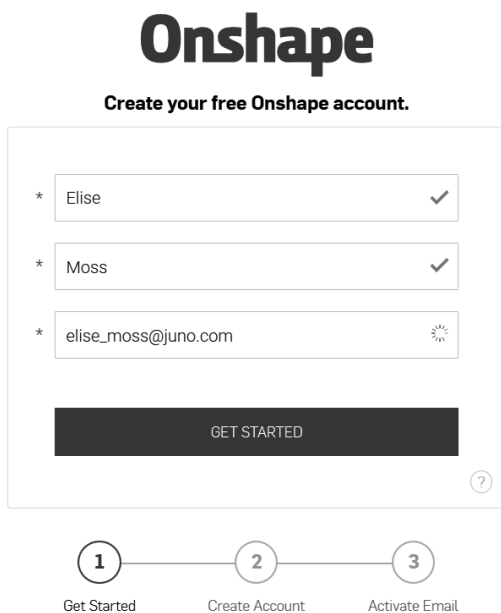
- Create an Onshape user account

To get started, go to onshape.com.

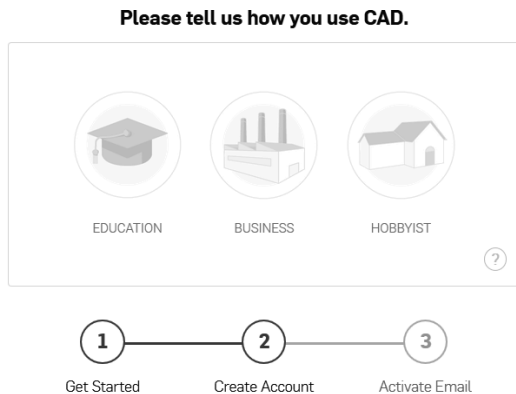
1. Select the **Create Account** link.



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



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



4. Fill out the form appropriately with your information.

* Educator ▼

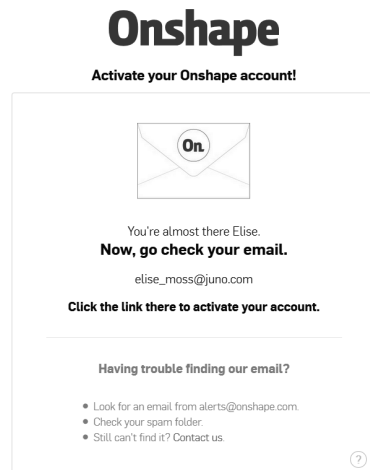
* Laney College ✓

* College or University ▼

* Adjunct Faculty ✓

CREATE FREE ACCOUNT

5. 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.



6. Select the link that is provided in the email.

7. 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
smoss@peralta.edu

Password

Name
Susan Moss


Role
Student / Educator ▼

Company
Laney College






Sign up

8. You start in the Tutorials and Samples area.

Your subscription: **Free** [Learn more](#) 2 of 10 Private documents 54 MB of 100 MB Private storage

[Create](#) 

Tutorials & Samples

	Name	Workspace	Modified ▼	Modified by	Owned by	Size
Recently opened	 Tutorial 1 - Onshape Tour	Main	5:57 PM Mar 19	Onshape	Onshape	155 MB
My documents	 Tutorial 2 - Build a Part	Main	5:56 PM Mar 19	Onshape	Onshape	3 MB
Created by me	 Tutorial 3 - Make an Assembly	M...	5:56 PM Mar 19	Onshape	Onshape	18 MB
Shared with me	 Tutorial 4 - Create a Drawing	Main	5:55 PM Mar 19	Onshape	Onshape	19 MB
Public	 Tutorial 5 - Import and Export	Main	11:11 AM Mar 18	Onshape	Onshape	3 MB
Tutorials & Samples						
Trash (5)						

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.

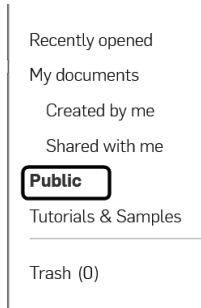
Navigating Documents and Workspaces

Estimated Time: 30 minutes

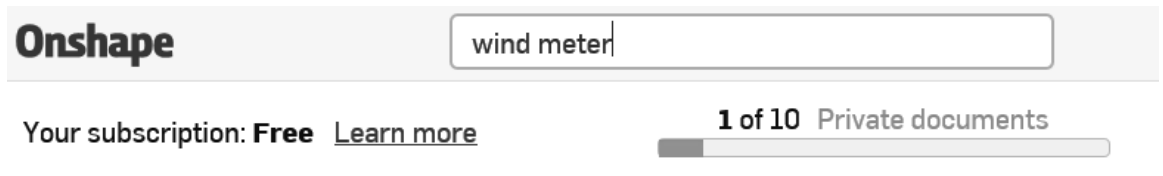
Objectives:

- 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. The Public area includes tutorials as well as projects from other users.

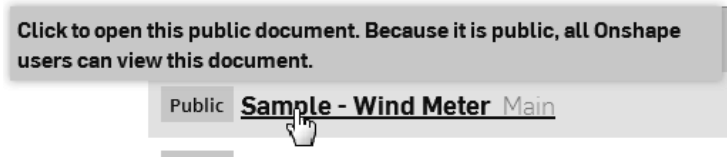
Click on the **Public** link.



2. Type **wind meter** in the search box at the top of the screen.



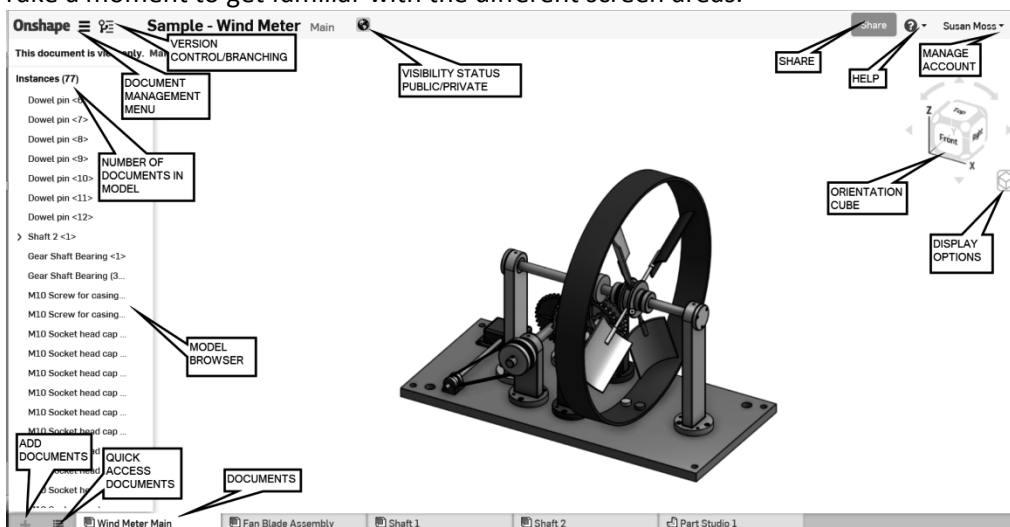
3. If you hover over one of the files labeled Sample, a small balloon appears indicating the file is a public document. Locate the Wind Meter document and select it.



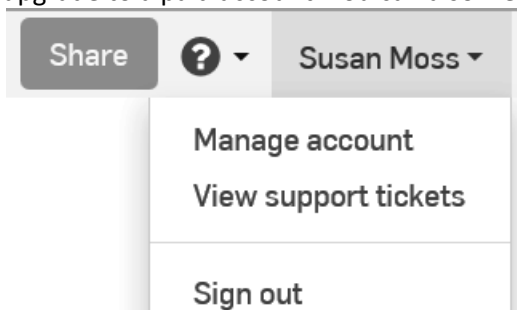
- When you left click on the link there is a pause while the model is loaded into your workspace.



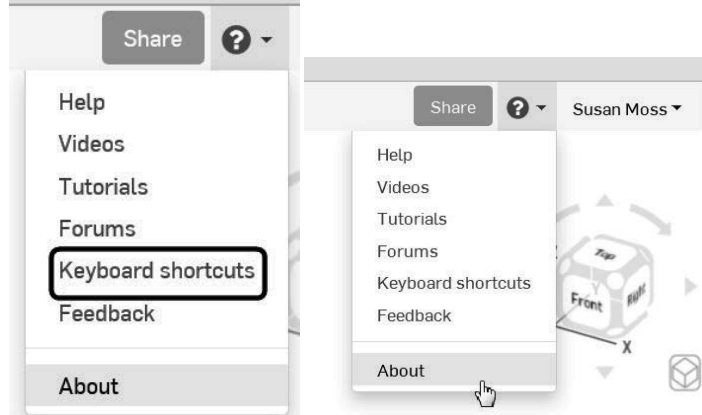
- Take a moment to get familiar with the different screen areas.



- 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.



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



8. A nice table of shortcuts appears.

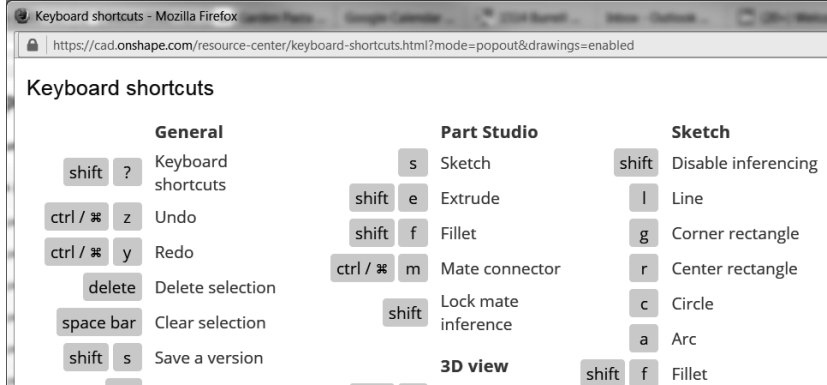
Keyboard shortcuts

General	Part Studio	Sketch	Drawings
shift ? Keyboard shortcuts	s Sketch	shift Disable inferencing	shift z Zoom in
ctrl / ⌘ z Undo	shift e Extrude	l Line	z Zoom out
ctrl / ⌘ y Redo	shift f Fillet	g Corner rectangle	f Zoom to fit
delete Delete selection	ctrl / ⌘ m Mate connector	r Center rectangle	w Zoom window
space bar Clear selection	shift Lock mate inference	c Circle	p Projected view
shift s Save a version	3D view	a Arc	d Linear dimension
esc Cancel command	shift z Zoom in	shift f Fillet	shift r Radial dimension
enter Accept command	z Zoom out	m Trim	shift d Diameter dimension
shift enter Accept & repeat command	f Zoom to fit	x Extend	n Note annotation
shift click Open in new window	w Zoom to window	o Offset	ctrl q Update drawing
ctrl / ⌘ click Open in new tab	rotate	u Use	l Line
ctrl u Feedback	shift rotate	d Dimension	ctrl s Display sheet menu
Assembly	shift 1 Front view	i Coincident	pg dn Next sheet
shift Lock mate inference	shift 2 Back view	b Parallel	pg up Previous sheet
ctrl / ⌘ c Copy	shift 3 Left view	t Tangent	home First sheet
ctrl / ⌘ v Paste	shift 4 Right view	h Horizontal	end Last sheet
	shift 5 Top view	v Vertical	
		e Equal	

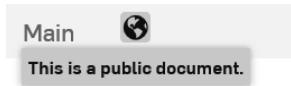
9. Select the Arrow in the top right corner of the keyboard shortcuts window.



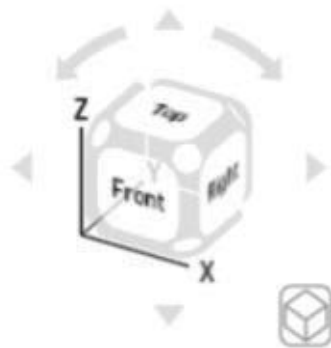
- 10. A webpage will appear that can be printed out. Simply press Ctrl+P and select the desired printer. You can then pin this up to your cubicle wall or keep it handy for easy reference.



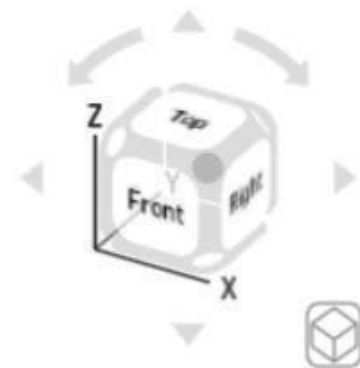
- 11. The Share button in the upper right corner will not work when you are in a public document. The icon next to the active document name at the top of the screen indicates whether the document is public or private.



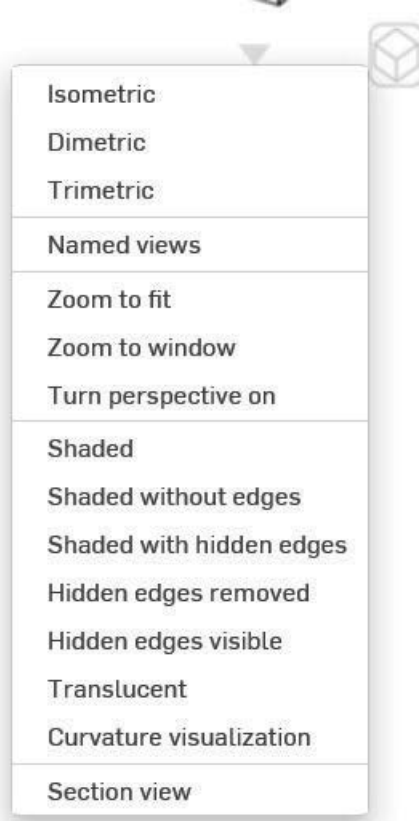
- 12. The Orientation Cube located in the upper right of the display window allows you to modify your view easily.




- 13. 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.

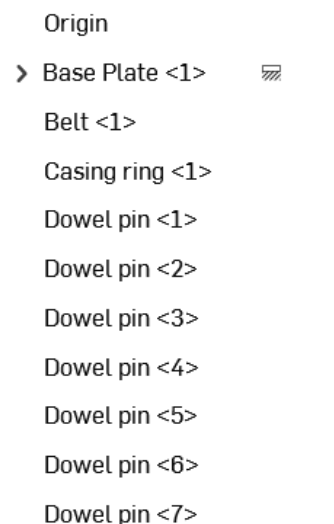


14. If you left click on the smaller cube (Display Options cube), you will get a shortcut menu where you can change the different display options as well as default orientations.



15. 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)



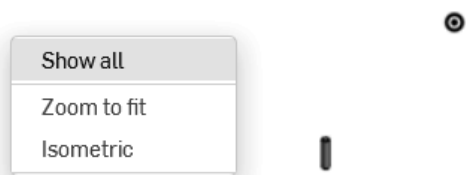
16. Highlight the Dowel pin part. Right click and select **Hide other parts**.



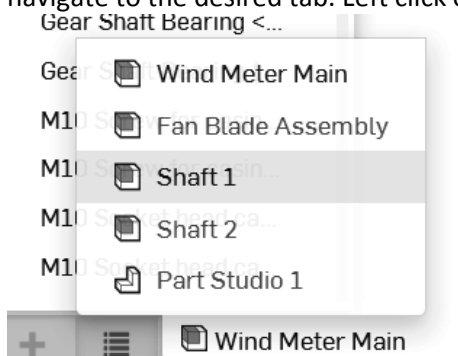
17. Visibility of all the parts is turned off. The small dot towards the right is the origin.



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



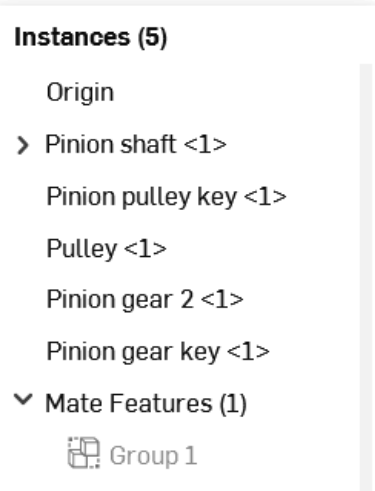
19. In the lower left of the screen, there is a small menu icon. If you left click on the icon, you get a quick list of all the tabs available in the model. This allows you to quickly navigate to the desired tab. Left click on Shaft 1.



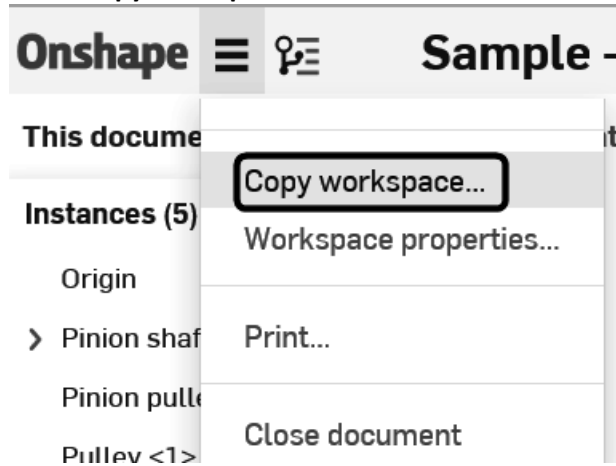
20. The Shaft 1 tab opens.

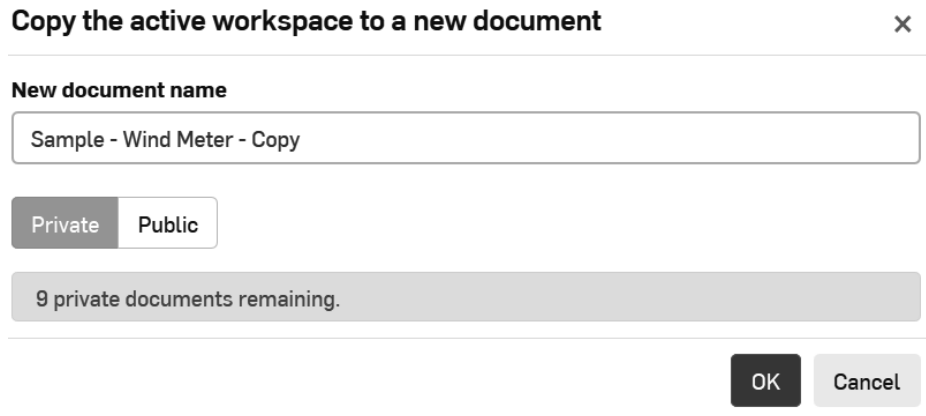


21. In the model browser, five parts are listed.



22. Select **Copy Workspace...** from the Document menu.





23. Enable **Private** to set the copy of the document to Private. You can only make changes to documents which you own. You can own documents which are private or public. Press **OK**.

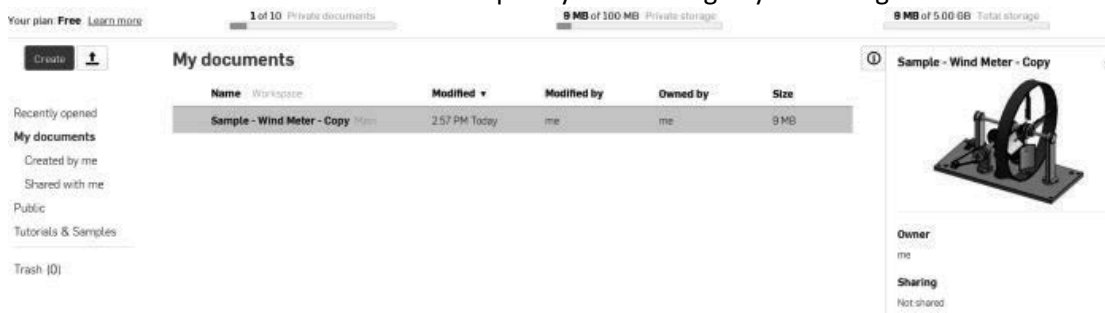
24. You now have a copy of the document that you own. You may modify the copy and share it with other users.



25. 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.



26. 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.

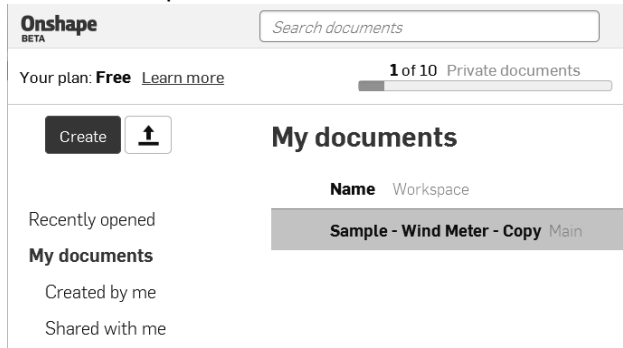


Sharing a Document

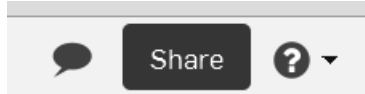
Estimated Time: 5 minutes

Objectives:

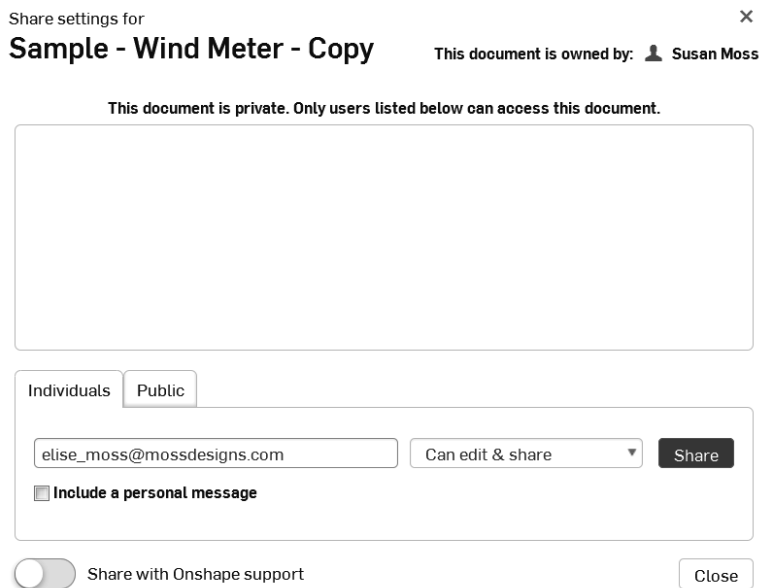
- Collaboration
1. Login to Onshape. Locate the Wind Meter which was copied to the My Documents area. Left click to open the document.



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**.



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

x

Individuals
Public

Can edit & share

Share

Include a personal message

Share with Onshape support

Close

Individuals
Public

Can edit & share

Share

Include a personal message

Share with Onshape support

Close

Owner

Can edit & share

Can edit

Can view, comment, copy & export

Can view, copy & export

Can view

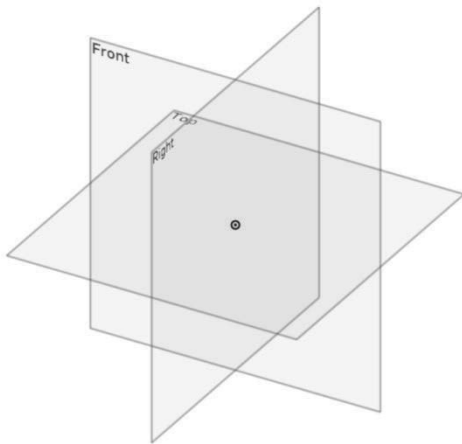
If you want to transfer ownership to another user, select **Owner** from the drop-down list. Then left click on the Share button.

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

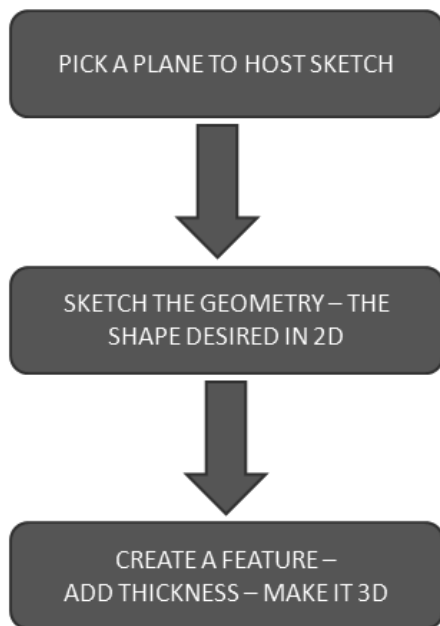
Front

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:

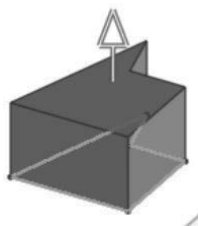
- Extrude
- Revolve
- Sweep
- Loft




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



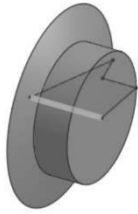
Common objects



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.



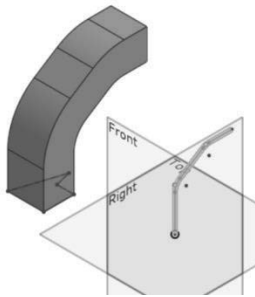
For a Sweep, you create two sketches. One sketch is the profile and one sketch is the path. The

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

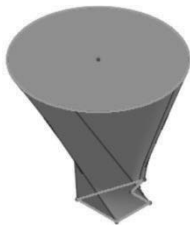


The two sketches are usually perpendicular to each other. The path would be on one plane, and the profile would be on a different plane.

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.

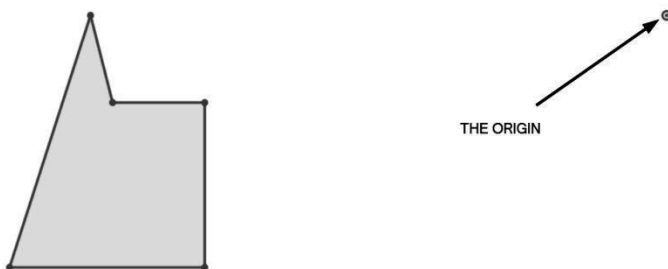
When you create a sketch, you draw the geometry – lines, arcs, and circles. Then you use sketch constraints to determine how the elements interact. There are two types of sketch constraints – geometric and dimensional. Geometric constraints are horizontal, vertical, perpendicular, parallel, concentric,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

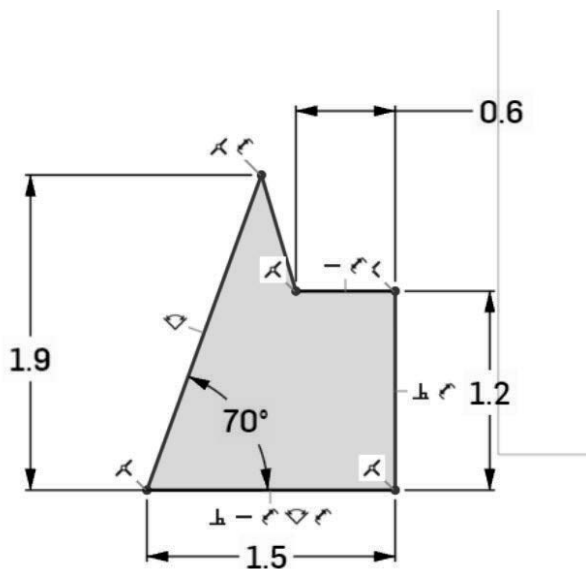


Because the sketch is located in 3-dimensional space, the elements always need to be located in relation to the origin in order to be fully defined. Otherwise, they are just floating in space.

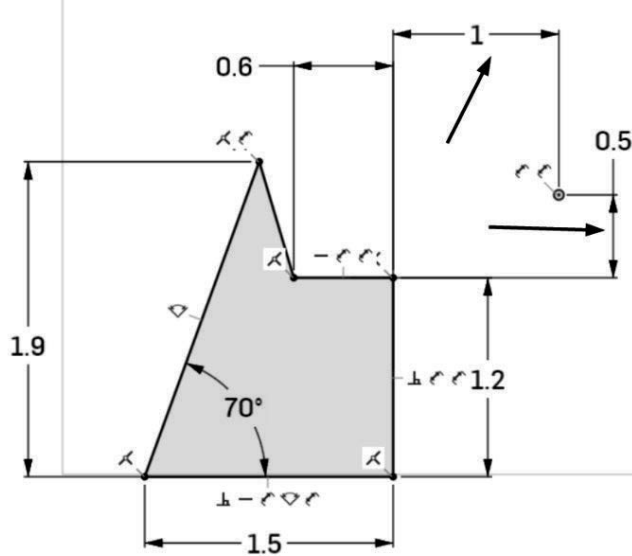
As dimensions and geometric constraints are placed, the geometry changes color.

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

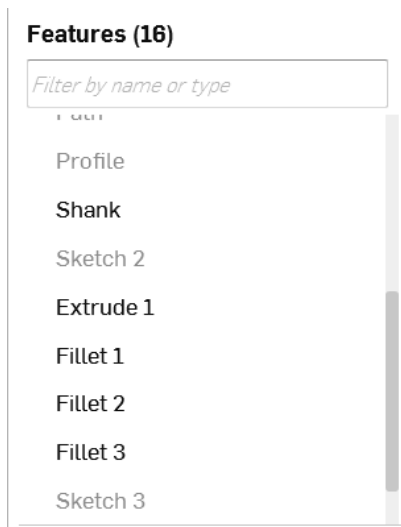
Some of the elements in this sketch are displayed as blue because it still isn't indicated where it is located in relation to the origin.



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.

