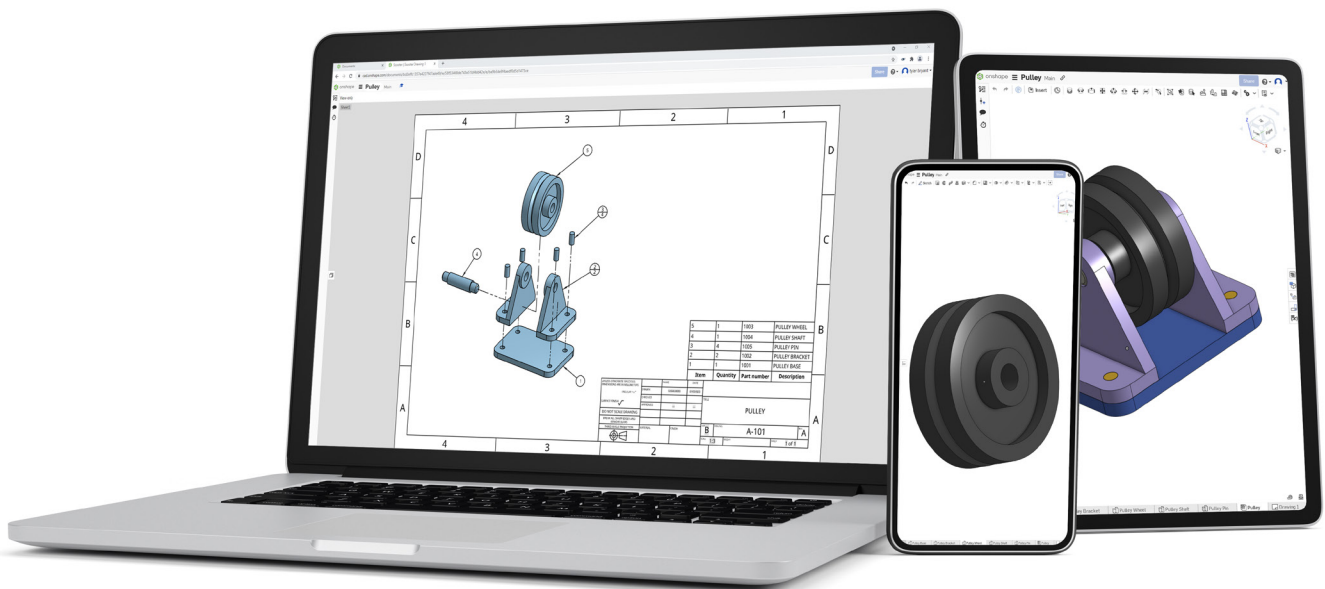


Getting Started with Onshape®

Fourth Edition



Elise Moss
Authorized Onshape Partner

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[BARNES & NOBLE](https://www.barnesandnoble.com)

Chapter 1: 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 2023 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. 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 workstation. 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.

In 2020, when the COVID-19 pandemic hit, many schools were forced to turn to distance or remote learning. Students were issued Chromebooks because they are cheap and easy to use. Onshape – being entirely browser based – became the software of choice for engineering and design students.

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

Onshape currently supports these tested and approved browsers:

- Google Chrome
- Mozilla Firefox
- Safari (Mac OS only)
- Opera
- Microsoft Edge

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.)

Onshape offers an Education Enterprise subscription which can be used for programs with more than 50 students. The Enterprise program provides greater ease of setting up the software for use by multiple students. It doesn't require that each student set up their own account. Instead, the instructor or IT department can create accounts for registered students or create a single sign-on, such as LaneyCollegeStudent with a password, so more than one student can sign into the account at a time. Instructors can create groups or teams to work on a project. Instructors have access to analytics, providing insight into how much time a student is spending using the software as well as how they are using the software. This can be especially useful with remote learning to monitor how engaged each student is in the assigned work. Instructors can also assign tasks and create an engineering release process. This might be useful when working in a Fab Lab or machine shop. Students can create designs to be 3D printed or machined and then submit those designs for approval and released to be machined or printed.

Whether students are working on an individual educational subscription or an enterprise subscription, the actual tools within Onshape used to create designs and drawings are identical.

Onshape does host regular design challenge contests for students and provides support for First Robotics and Vex. If you are an educator with a student team competing in a design challenge using Onshape, reach out to Onshape for stickers and other goodies for your team.

Setting Up an Account

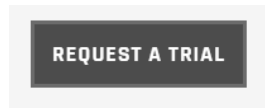
Estimated Time: 10 minutes

Objectives:

- Create an Onshape user account

To get started, go to onshape.com.

1.



Click **Request a Trial**.

2.

Request a trial of Onshape Professional

- Free 14 day trial of Onshape
- Free built-in PDM
- Free real-time team collaboration
- No downloads
- No credit card required
- Content and tutorials to get you started

Ready to buy? [Talk to sales ->](#)

Student or Educator? [Visit the EDU Plan ->](#)

Hobbyist? [Get Free Onshape for Makers ->](#)

Students should sign up for the EDU plan.

Makers or hobbyists should sign up for the free public plan.

The free public plan means that all your files are public and available for anybody to use. If you want to secure your files, you will have to pay to use Onshape.

Click on **Visit the EDU Plan**.

3.

Onshape is a professional-grade, cloud-native CAD platform that students and educators can access for FREE on any device, anywhere, anytime.

CREATE EDU ACCOUNT

Click on **CREATE EDU ACCOUNT**.

4. **Sign up for Onshape for Students and Educators**

First Name *

Last Name *

Email *

Are you a student or educator? *

School level *

* I acknowledge that my personal details will be processed in accordance with Onshape's Privacy Policy [P.1](#). PTC will not use student email addresses for any marketing purposes.

CREATE EDU ACCOUNT

- **Affiliates:** Onshape may share your personal information with PTC Inc.'s Affiliates, where it is necessary to support the provision of the Services or it is necessary for the Affiliates to process the personal information to support their legitimate business interests. All of PTC Inc.'s Affiliates are required to honor this Privacy Policy.
- **Authorized Resellers and Partners:** We may also share your personal information with our authorized resellers and partners for the purpose of further supporting your use of Onshape, and for co-marketing activities. All such authorized resellers and partners are required by Onshape to provide you with a means of opting out of some or all of their communications with you.
- **Aggregated and Non-Identifiable:** Onshape may share aggregated or deidentified personal information with our partners or others in our sole discretion, including for business or research purposes.

Fill in the dialog box.


Click **CREATE EDU ACCOUNT**.

Note: You have to initially opt in to receive spam, but you can opt out afterwards.

It is up to each individual user to manage their opt-out settings every time you receive an email from an Onshape partner.

This may or may not be a concern, but you should be aware.

Students who are minors are protected from receiving spam.

 **Step 2**
Tell us about how you plan to use Onshape

* Laney College ✓

* laney.edu ✓

* Oakland ✓

* California ✓

* United States ✓

* Adjunct Faculty ✓

* [REDACTED] ✓

* Machine Design ✓

Do you use CAD in your class? Yes No

* Machine Design ✓

* I am a current faculty member or volunteer at an accredited educational institution.

You will need to fill out an additional form about where you go to school or teach.

Then click **CREATE ACCOUNT**.

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.

Select the link that is provided in the email.

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

To ensure you are able to communicate properly with Onshape, you can whitelist this domain and IP address:

- **Domain:** outbound-email.onshape.com
- **IP address:** 167.89.77.213

Many corporate environments will whitelist or proxy the network traffic to enforce where users on the network can navigate. In order to use Onshape within this network configuration, the following domains must be added to the “allowed” list: **cad.onshape.com**

In addition, the region of the network as well as any collaborators you might be collaborating with should also be added to the “allowed” list:

- North America: **cad-usw2.onshape.com**
- Europe: **cad-euw1.onshape.com**
- Singapore: **cad-aps1.onshape.com**
- Australia: **cad-aps2.onshape.com**
- Japan: **cad-apn1.onshape.com**

If the network configuration supports domain wildcards, a single entry can be added to the “allowed” list: *.onshape.com

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.

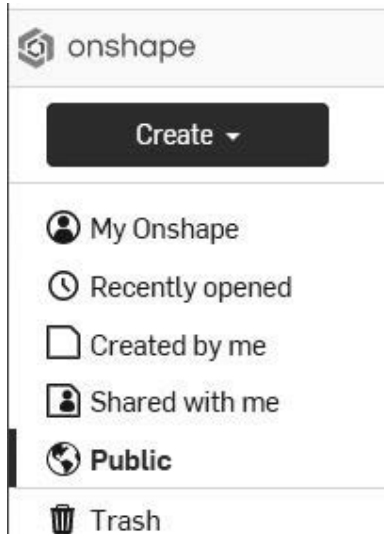
Navigating Documents and Workspaces

Estimated Time: 30 minutes

Objectives:

- Getting familiar with the Onshape user interface

1.



On the left side of the screen there is a list of links. These links organize your documents. 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.



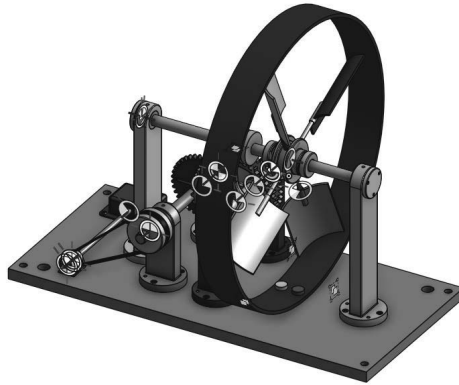
Click on the search icon.

3.



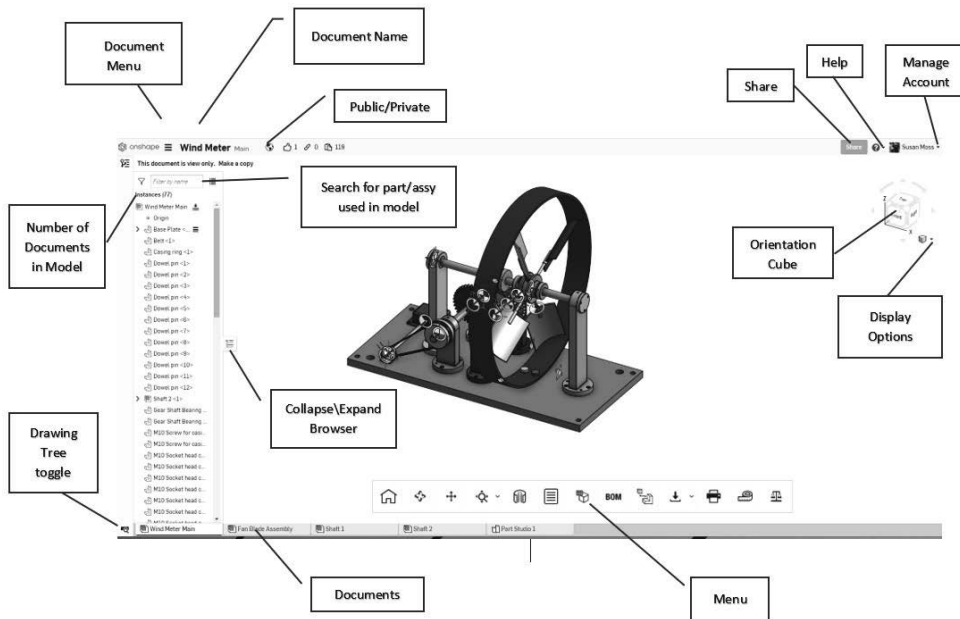
If you hover over one of the files labeled **Wind Meter**, a small balloon appears indicating the file is a public document. Locate the **Wind Meter** document and select it.

4.

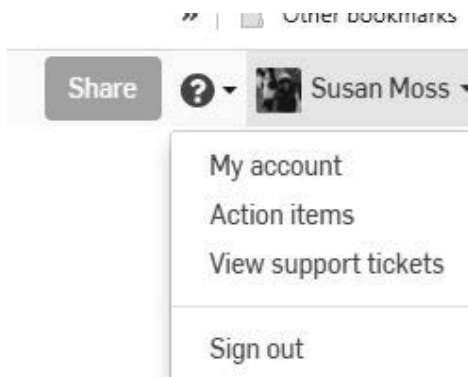


When you left click on the link there is a pause while the model is loaded into your workspace.

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



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.

6.

Action items

Type

- Release
- Comment

Role

- Any
- Assigned to me
- Created by me

Status

- Any
- Open
- Closed

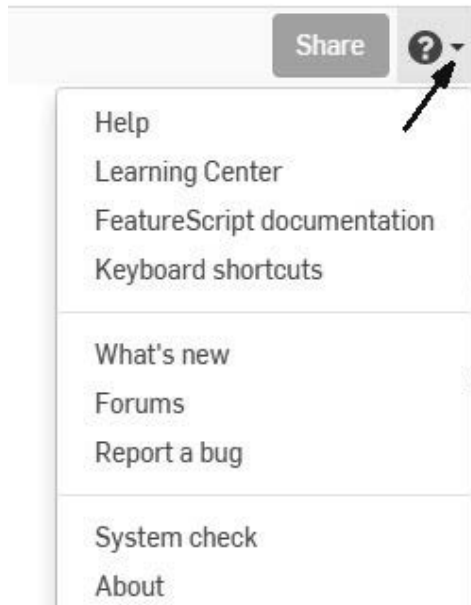
Sort

Oldest first ▼

Action Items allows you to keep track of your work. You may be tasked to review someone else's design or drawing. You may be asked to create a part that is being used in a team project.

If you are a solo user, you can use Action items to help you keep track of where you are in a project.

7.



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

Select **Keyboard shortcuts**.

8.

General	Part Studio	Sketch	Drawings
SHFT ? Keyboard shortcuts	SHFT S Sketch	SHFT S Suppress inferring	SHFT Z Zoom in
Ctrl/J Undo	SHFT E Extrude	Line	Zoom out
SHFT V Redo	SHFT F Fillet	X Corner rectangle	Zoom to fit
SHFT R Rotate selection	SHFT M Mate connector	C Center point rectangle	Zoom to window
Spacebar Clear selection	SHFT L Lock mate inference	C Center point circle	Projected view
Select other	K Flip primary axis	3 point arc	Dimension
SHFT R Rotate selection	R Reverse secondary axis	SHFT F Fillet	Maximum/minimum dimension
ESC Cancel command	X Hide/show mate connections	SHFT T Trim	SHFT R Radial dimension
Enter Accept command	3D View	X Extend	SHFT D Diameter dimension
SHFT Enter Accept & reset command	SHFT Z Zoom in	SHFT O Offset	SHFT Q Quad & Mid snap points
Alt C Search tools	SHFT Z Zoom out	U Use	None
SHFT S Shortcut toolbars	SHFT F Zoom to fit	SHFT C Consistent	SHFT U Update drawing
SHFT O Open in new window	W Zoom to window	SHFT P Parallel	Line
SHFT/SHFT O Open in new tab	R Rotate	SHFT T Target	SHFT D Display sheet menu
SHFT F Feedback/Report a bug	SHFT R Rotate rotate	SHFT H Horizontal	SHFT N Next sheet
SHFT S Search tabs	SHFT R Rotate rotate	E Equal	SHFT P Previous sheet
SHFT Spacebar Quick tab switching	SHFT P Pan	SHFT T Toggle construction	SHFT L Last sheet
SHFT L Lock mate inference	SHFT F Front view	SHFT I Isolate	SHFT/SHFT C Copy annotation
SHFT/C Copy	SHFT B Back view		
SHFT/V Paste	SHFT L Left view		
M Mate	SHFT R Right view		
M Mate connector	SHFT B Bottom view		
SHFT I Insert dialog	SHFT T Top view		
SHFT S Snap mode	SHFT B Bottom view		
SHFT H Show mates mode	SHFT I Isometric view		
F Flip primary axis	SHFT N View normal to		
R Reverse secondary axis	SHFT H Hide/show planes		
H Hide/show mates	SHFT P Hide all construction		
H Hide/show mate connections	SHFT V Named views		
	SHFT S Section view		
	SHFT H Hide selected part		
	SHFT V Show unselected part		
	SHFT M Make transparent		
	SHFT I Isolate		

A nice table of shortcuts appears.

9.



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.

10.

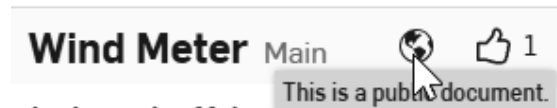


Click the X icon to close the window.

11.



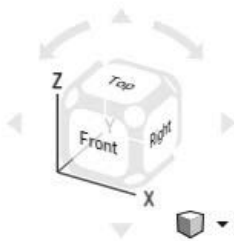
The Share button in the upper right corner will not work when you are in a public document. You will see a little NO symbol if you try to share a public document.



it is view only. Make a copy

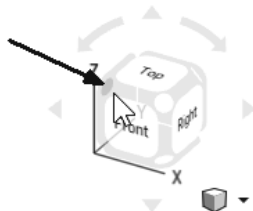
The icon next to the active document name at the top of the screen indicates whether the document is public or private.

12.



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

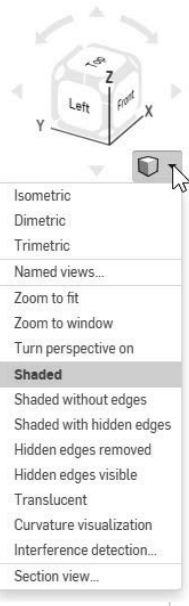
13.



Notice that as you hover over different areas of the orientation cube it will highlight.

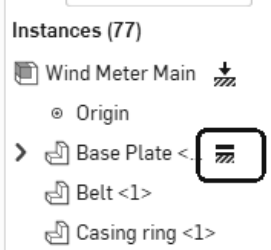
Left click to select the shaded area as the new view orientation.

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.

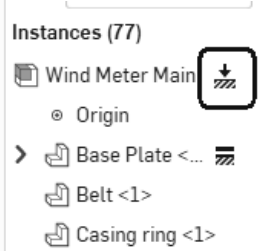
15.



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.

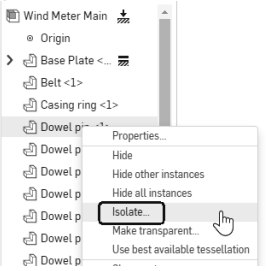
16.



The main assembly also has an icon next to it.

This icon indicates that some of the parts within the assembly are grounded or fixed in place.

17.



Highlight the Dowel pin part.

Right click and select **Isolate**.

18.



All the parts that are not the selected part will become transparent. You may need to orbit the model around to locate the dowel pin.

To rotate the model, use the **ROTATE** tool on the toolbar.

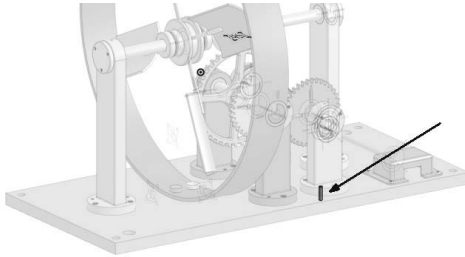
19.



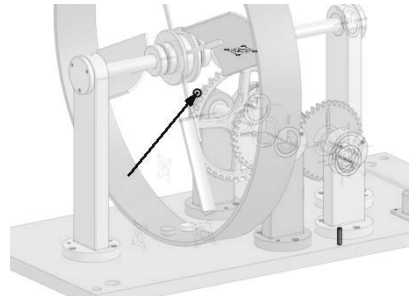
To pan or move the view, select the PAN tool on the toolbar.

The PAN tool doesn't change the position of the model. It adjusts your position relative to the model – like panning a camera.

20.



Can you locate the dowel pin?



The small dot is the origin.

21.



Highlight the Dowel pin.

Select **Hide other instances**.

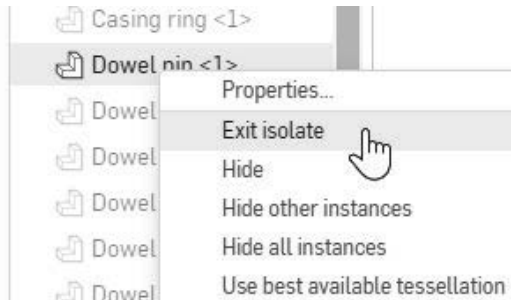
22.



All the parts are hidden.

You should see the dowel pin, the origin, reference points and reference planes.

23.



Highlight the Dowel pin.

Right click and select **Exit isolate**.

You still don't see all the other parts.

24.

Right click in the window and select **Show all**.

Visibility of the model is restored.



25.



At the bottom of the screen, there are several tabs. The tabs list the names of the sub-assemblies in the model.

This allows you to quickly navigate to the desired tab.

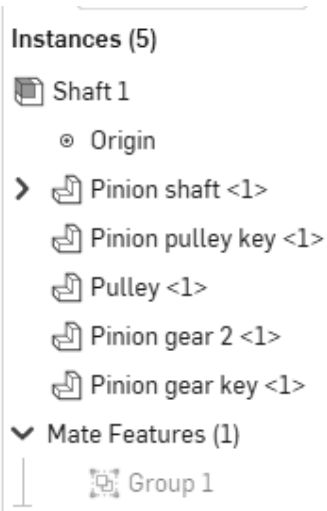
Left click on **Shaft 1**.

26.



The sub-assembly is displayed.

27.



The model browser lists the parts used in the assembly.

Note that the browser displays (5) instances and there are five parts used in the sub-assembly.

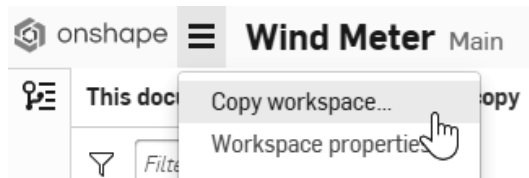
28.



If you hover the mouse over a tab, you will see a preview of the part or sub-assembly.

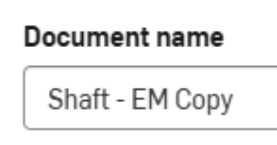
This allows you to quickly locate the sub-assembly or part you want to review.

29.



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

30.



Type the Document name, your initials, and Copy.

Click **OK**.

31.

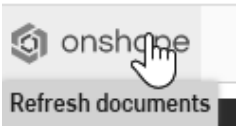


You now have a copy of the document that you own.

You may modify the copy and share it with other users.

Notice that the Public icon is not displayed on the document.

32. Click on the OnShape logo. This will close any open documents.



33. Notice that your copied document is owned by **me**.

Name	Modified	Modified by	Owned by
Folders			
121.x.1	8:42 PM, Jan 9 2018	me	me
Documents			
Shaft - EM Copy # 0 views	3:42 PM Today	me	me

34. If you click on Created by me, you see the copied document.

- My Onshape
- Recently opened
- Created by me**
- Shared with me
- Public
- Trash

Sharing a Document

Estimated Time: 5 minutes

Objectives:

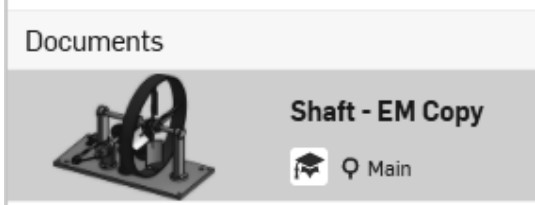
- Collaboration

1. My Onshape
- Recently opened
 - Created by me
 - Shared with me
 - Public
 - Trash

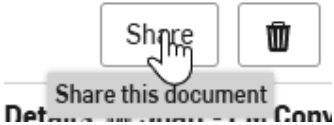
Log in to Onshape. Locate the Wind Meter which was copied to the **My Onshape** area.



2. Highlight the document.



3. Click **Share** located in the upper left of the screen.



4. 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.

5. Copy Link document Export Share Comment Delete
Limited permissions reduce the risk of 3rd parties copying or exporting your documents. [Learn more about security.](#)

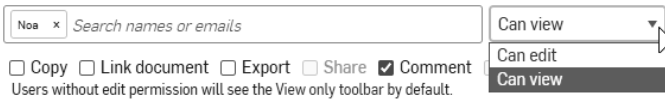
Enable the permissions you would like the receiver to have.

- Enable **Copy**.
- Enable **Export**.
- Enable **Share**.
- Enable **Comment**.

If Copy is enabled, the receiver can make their own copy of the model to work on.
 If Export is enabled, the receiver can export the file to be 3D printed or used in a different CAD software.

If Share is enabled, the receiver can share the model with anyone else.

If Comment is enabled, the receiver can review and add comments to your document.

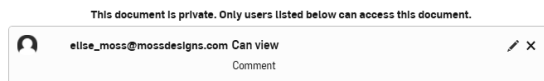
6. 

In the drop-down list, set the permission to **Can view**.

Notice that when you changed the setting to **Can view**, the recipient can only Comment on the document. All the other permissions are disabled.

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

7. Click **Share**.



The Collaborator’s email is now listed.

You can edit the collaborator’s permissions or delete them.

New document shared with you

This is the email message the collaborator will receive.

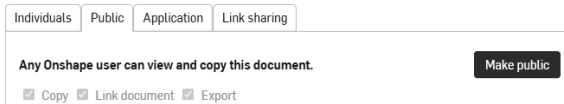
Hello Elise Moss,

An Onshape document has been shared with you by Susan Moss.

Click to access the document, Shaft - EM Copy

GO TO YOUR ONSHAPE DOCUMENT

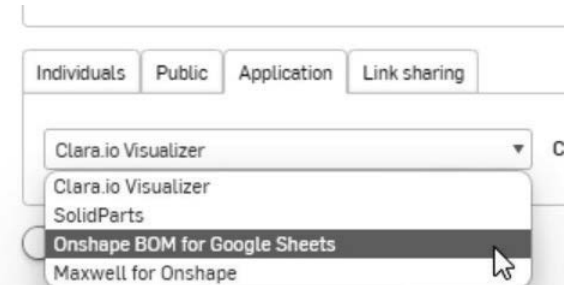
8.



If you click on the Public tab, this is where you can share your document with the Onshape community.

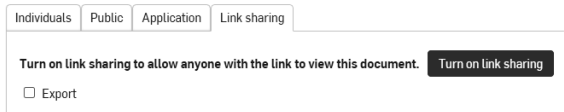
Do not do this if you are working for a private company or you don't want your work to be copied.

9.



If you click on the Application tab, you see a list of Onshape partners that will import your model.

10.



If you click the Link sharing tab, you can create a link that you can email or text to one or more people so they can view and comment on your document.

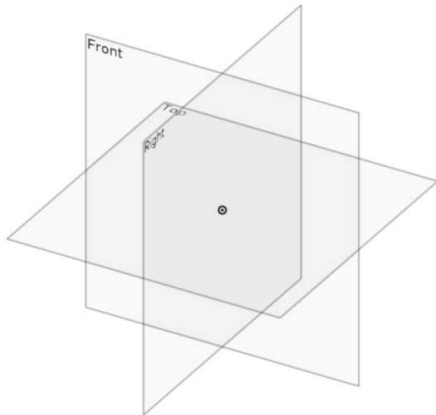
11.

Close

Click **Close**.

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

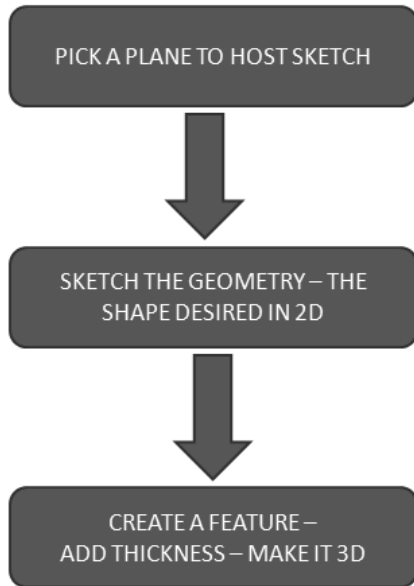
Front

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:

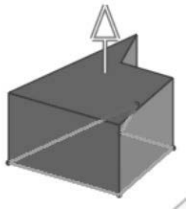
- Extrude
- Revolve
- Sweep
- Loft



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



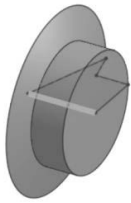
Common objects



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.



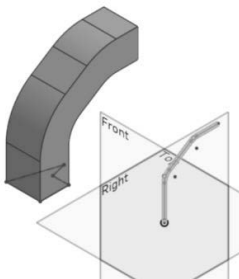
For a Sweep, you create two sketches. One sketch is the profile and one sketch is the path. The

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

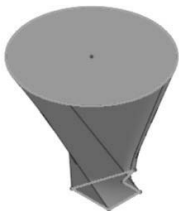


The two sketches are usually perpendicular to each other. The path would be on one plane, and the profile would be on a different plane.

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.



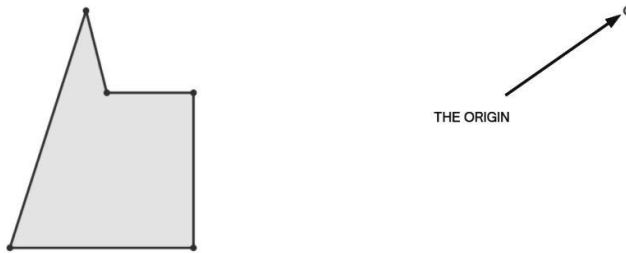
When you create a sketch, you draw the geometry – lines, arcs, and circles. Then you use sketch constraints to determine how the elements interact. There are two types of sketch constraints – geometric and dimensional. Geometric constraints are horizontal, vertical, perpendicular, parallel, concentric,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

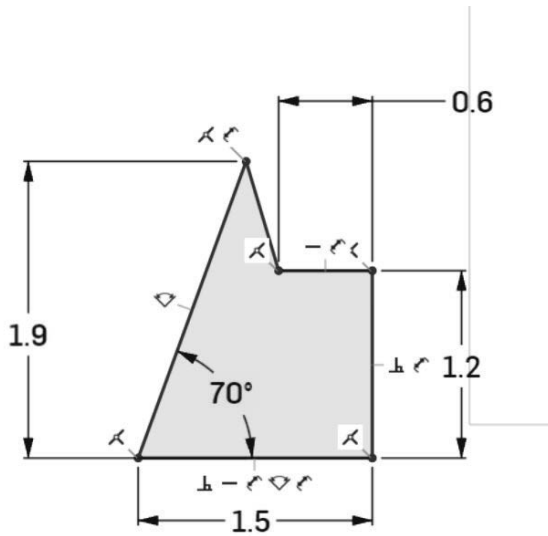


Because the sketch is located in 3-dimensional space, the elements always need to be located in relation to the origin in order to be fully defined. Otherwise, they are just floating in space.

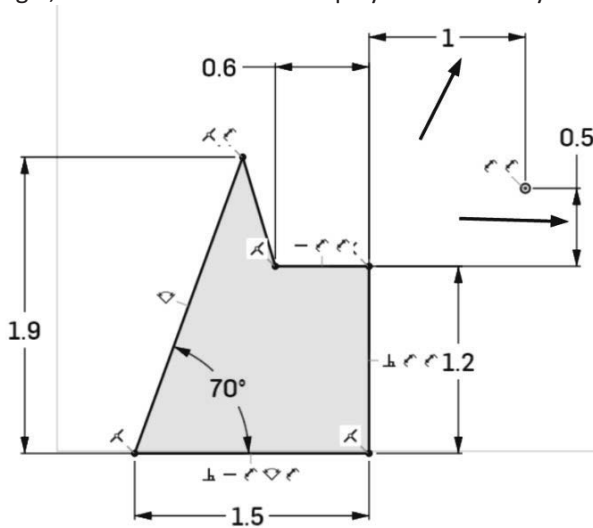
As dimensions and geometric constraints are placed, the geometry changes color.

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

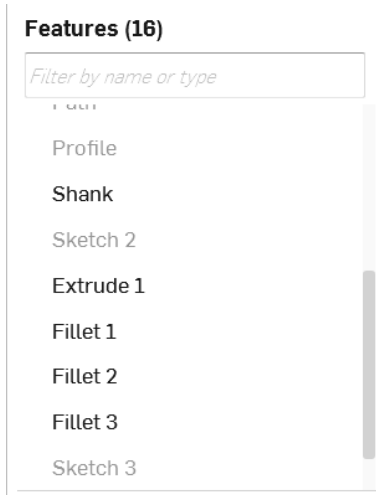
Some of the elements in this sketch are displayed as blue because it still isn't indicated where it is located in relation to the origin.



Once the two dimensions are added to indicate where the elements are located relative to the origin, the entire 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.



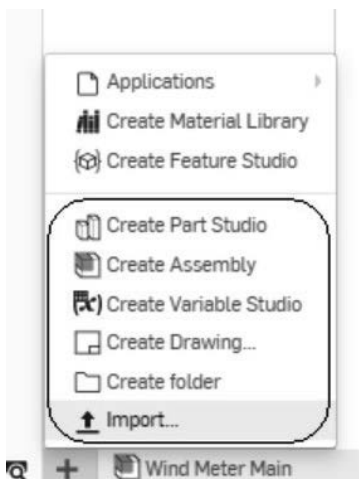
The Onshape document consists of multiple tabs.



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.



A Variable Studio is a table. This is useful when creating table-driven parts.

A table-driven part is a part that has similar features, but different sizes.

For example, you can use the variable studio to create screws of different sizes or panels with different sizes and openings.

Notes: