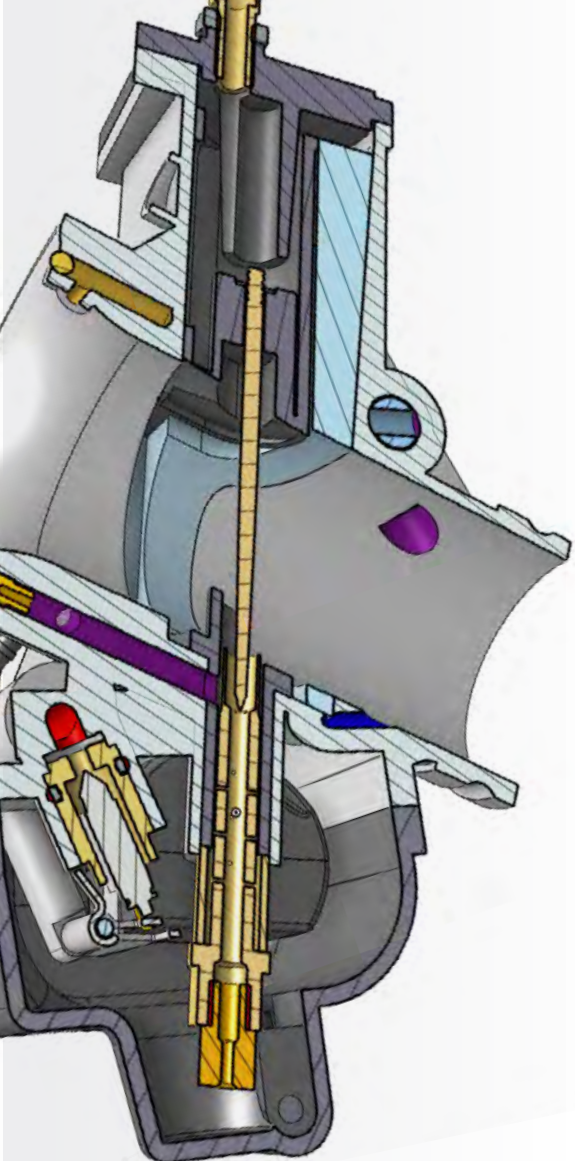


# Let's Get Technical: Transitioning to Onshape from SOLIDWORKS

Answers to the 10 most common  
questions asked when transitioning  
to Onshape from SOLIDWORKS





For product designers and engineering teams, switching to a new computer-aided design (CAD) system can seem like a daunting task.

Luckily, the Onshape team is here to help. After years of assisting new users, we've come across quite a few commonly asked questions specific to the transition from legacy CAD systems, like SOLIDWORKS, to Onshape.

In this guide, we've put together step-by-step instructions to answer those questions, all while highlighting the cloud-native CAD platform's most important and unique features.

The Guide is broken into easy-to-read "How Do I" overview sections:

#### **"HOW DO I":**

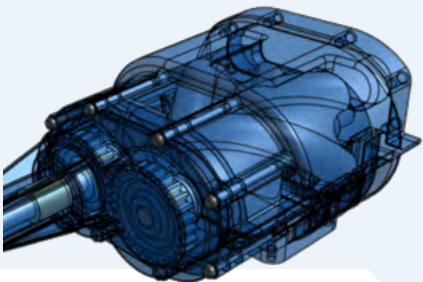
1. Migrate my data to Onshape from SOLIDWORKS?	page 03
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# 1 | How do I migrate my data to Onshape from SOLIDWORKS?

While customers come to Onshape for many reasons, the process of beginning is the same and simple for all.

## Expert Advice for Getting Started:

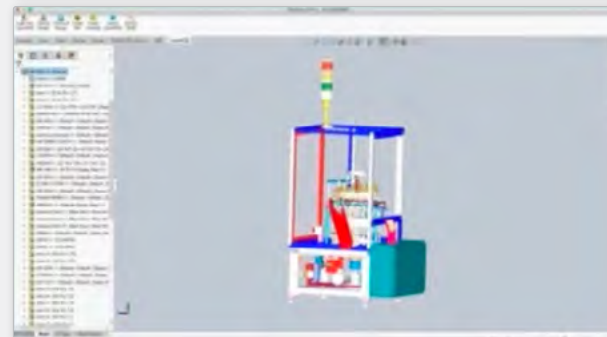
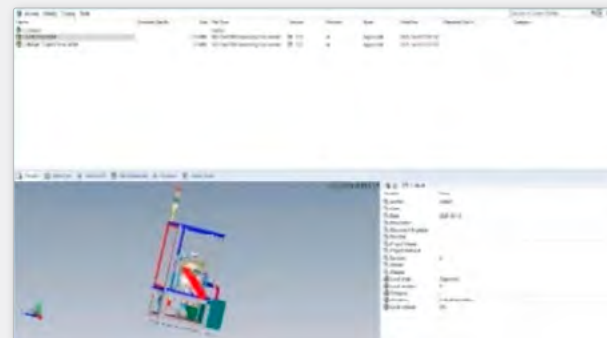
- **Transfer your Most Important Data When You Need it.**  
Choose your first Onshape project and import necessary data. Additionally, transfer your common and purchased parts into Onshape.
- **Learn the Basics of Onshape Through our Onboarding Program.**  
Register for either self-guided or instructor-led [Onshape bootcamp](#) to learn how to successfully organize and build your Onshape Documents.
- **Accelerate Your Product Development with our Tech-Tips**  
Read our tech tips, such as [10 Essential Onshape Tips](#), to enhance your skills within Onshape.
- **Explore Partner Products to Assist Cloud Adoption**  
CADSharp have created [Monarch](#) and [SolidTranslate](#) to handle both large scale migrations, and feature-to-feature conversions. Click [here](#) to learn more.



### Technical Brief: Importing Data

Learn more including best practices and supported formats for importing data into Onshape.

LEARN MORE



Onshape supports the import of SOLIDWORKS files such as:

- ..... Parts (.sldprt)
- ..... Assemblies (Via a Pack and Go.zip)
- ..... Drawings (DXF)

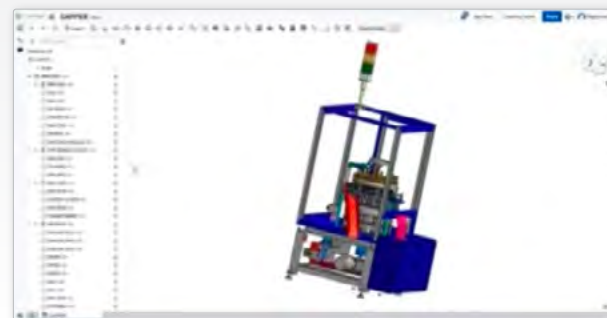
And additional data including

- ..... Images
- ..... Videos
- ..... PDF

And more.



For a full list of supported file formats see :  
<https://cad.onshape.com/help/Content/translation.htm>





## 2 | How do I share data with supply chain partners?

Onshape allows users to share design documents with external partners quickly and easily.

Onshape provides secure, easy sharing of documents with external partners using permission-based sharing:

- The document owner, who is either an administrator or has share permission, can grant various levels of permission-based access (read-only, editing rights) to other users.
- Documents are sent to recipients with an intuitive, easy-to-use weblink.
- Recipients don't need any specialized hardware or software to view files.

### SHARING A DOCUMENT

*To share a document:*

1. On the Documents page, right-click on the document to share and select "Share."
2. The Share Settings dialog will open allowing the user to add email addresses for chosen users.
3. In addition to sharing with select individuals, the document owner can share with broad groups of individuals quickly and easily.

The document owner determines the level of access granted to each recipient – from view-only to full editing rights. The document owner can change a user's permission level at any time. For example, the two settings for an individual or an entire team can be changed at any time.

*These are:*

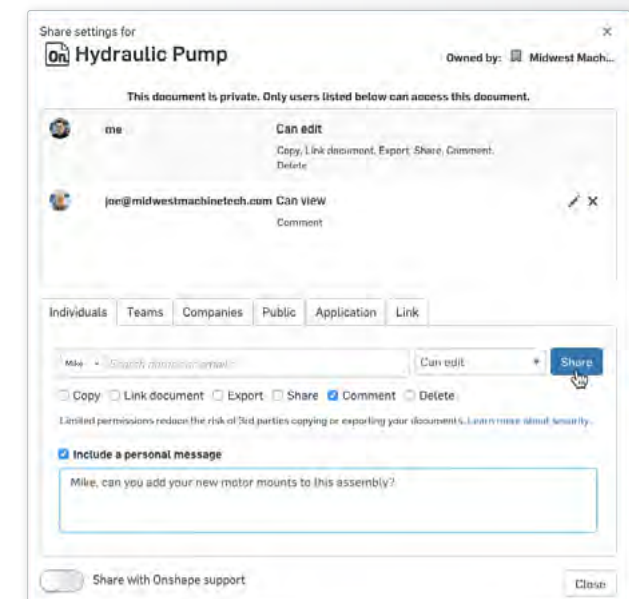
- **Can edit:** Permission to edit, copy, link, export, share, and comment on a Document.
- **Can view:** Permission to view, copy, link, export, and comment on a Document.

### LINK SHARING

A quicker method to share documents is to use link sharing.

Based on permission settings, with a single link, both Onshape and non-Onshape users can access a document as read-only. Permission can be revoked at any time in the Share Settings dialog.

| *The Share Settings dialog.*



### 3 | How do I export and import data to manufacturing with Onshape?

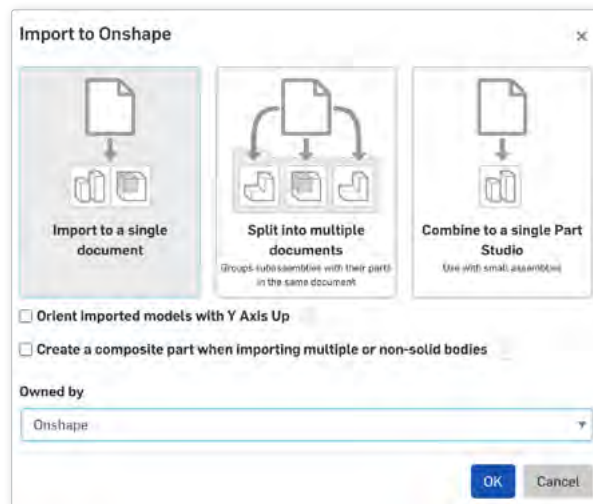
Onshape makes exporting data simple and quick. **To export data:**

1. Right-click on the tab or items to export and click "Export."
2. Select the desired file format for exported data.
3. A tab will be created with the translated file, a direct download, a password-protected email link, or a link to a cloud storage system of choice.


**note:** Onshape supports a wide range of standard CAD file formats that can be translated and viewed inside of a browser.

It is important for new Onshape users to note that these files above are CAD files that can be imported and viewed from within a browser, but any file can be uploaded to an Onshape Document. This makes each Onshape Document a container for the entire project and not just CAD files. Certain file types like PDFs, images, and video files can be viewed from within Onshape.


*This Image shows the Import to Onshape display screen where different methods of importing Onshape Documents can be initiated.*




**Onshape can import the following types of Part Studios:**

 Parasolid, ACIS, STEP, IGES, CATIA v4, CATIA V5, CATIA V6, SOLIDWORKS, Inventor, Pro/ENGINEER, Creo, JT, Rhino, STL files, OBJ, NX


**Onshape can export Part Studios in the following formats:**

 Parasolid, ACIS, STEP, IGES, SOLIDWORKS, STL, Rhino, Collada

**Onshape Assemblies can be imported from:**

 Parasolid ACIS, STEP, PRO/ENGINEER, Creo, JT, Rhino, NX, SOLIDWORKS Pack and Go ZIP files


**Onshape Assemblies can be exported to:**

 Parasolid, ACIS, STEP, IGES, STL, Collada

**Onshape Drawing files can be imported from:**

 DWG, DXF

**Onshape Drawing files can be exported to:**

 DXF, DWG, DWT Template, PDF

## 4 | How do I set up drawing templates in Onshape?

When first creating a drawing of a part in Onshape, you can choose to select an existing template or make a custom one. You can also import your existing template from SOLIDWORKS.

*To create a Custom Drawing Template in Onshape, follow the steps below:*

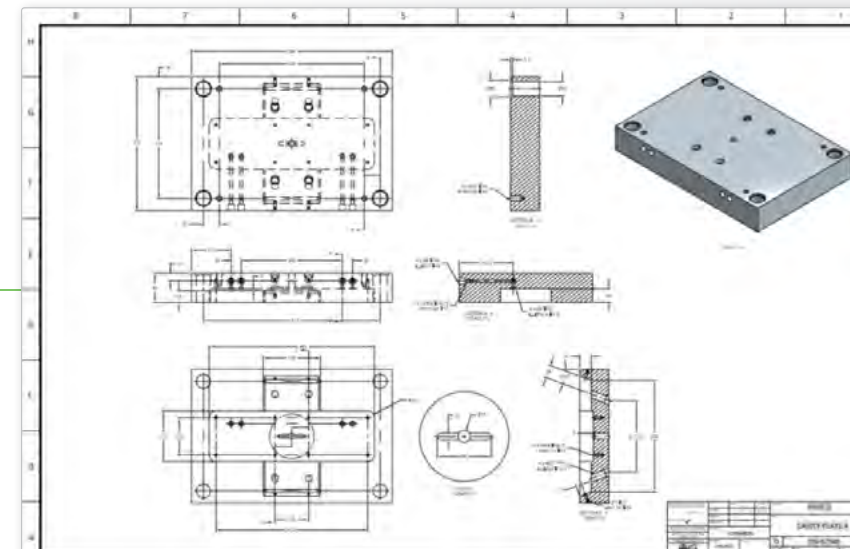
1. At the template step, select Custom Template.
2. Select Do not include the borders.
3. Select Do not include the title block. Set size, standard and other characteristics as needed.
4. Click OK and close the open dialogs in the Drawing.
5. Import your exported traditional CAD drawing (in DWG/DXF file format) into Onshape (through Menu, then Import).
6. Open the empty Drawing you created.
7. Use the Insert DWG/DXF tool to insert the file just uploaded.
8. To tweak the fonts, select the text and change the font to an internally supported font of your choice.
  - a. To add a Revision Table or Block functions:
    - i. Generate a Drawing using the newly uploaded DWG/DXF file.
    - ii. Place a table where needed (for example, the upper-right corner).

- b. To include a company logo in the Drawing template:
  - i. Upload the logo through the Insert Image command.
  - ii. Insert the newly uploaded image, properly place and size it in the Drawing.
- c. Insert drawing fields as necessary in the appropriate areas of your Drawing:
  - i. Add a note to the Drawing.
  - ii. Select the Insert field button.
  - iii. Use the Move to command on the View's context menu (right-click) to add elements to the: Title block, Border frame, and Border zones. (Elements will be added to the corresponding layer: title block, border frame, or border zones.)

*Once the tasks above are completed, follow these steps to apply the new template:*

1. Export the drawing as a DWT file.
2. Upload the new DWT file into an Onshape Document (company-owned is preferred).
3. Test the Drawing template before using it.

*A completed  
Drawing Template*



## 5 | How do I create a purchased Parts Library in Onshape?

Commercial-off-the-shelf items (COTS) make up a large percentage of designs for some companies, especially those that work with large machine designs.

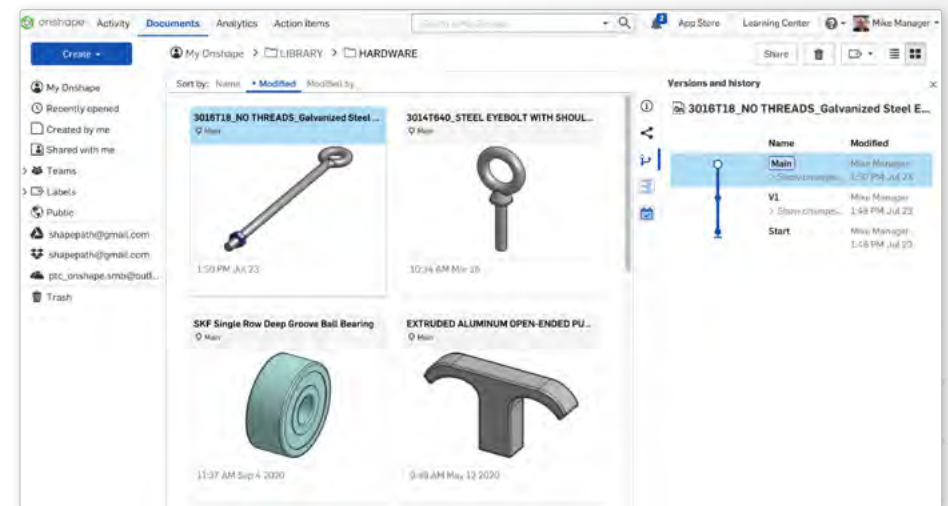
These standard parts and assemblies can be designed natively in Onshape, imported from your current Windows file system or cloud drive, or imported into Onshape directly from nearly any component supplier on the internet.

The best practice for a company using Onshape is to create a folder at the top level of the Documents page and share it with all the design engineers that need access to COTS items. Permissions can be set so that the COTS library administrator(s) can import and edit design data and design engineers can use that design data.

*Here are the steps to get your design information into Onshape:*

1. On your local file system, locate the files you wish to import. The entire list of file types Onshape can import [can be found here](#).
2. On the main Onshape Documents page, choose the Create button in the upper-left corner and select "Import." There are two choices to pull the data:
  - a. Most users will choose "Import Files," which will guide the user to their local file system.
  - b. If the files are stored on a cloud drive, choose "Import From," which will pull directly from Dropbox, Google Drive or Microsoft Onedrive into Onshape. Note that you can shift-select the files to import more than one at a time.

3. Onshape will then upload the files to a Document and process the conversion of CAD data with our built-in translation service. The status of the upload and translation can be found by clicking the Bell at the top-right corner of the Onshape screen. Note that larger assembly files could take a few minutes to translate and appear.
4. To use the items that were imported into a COTS library folder, simply be in an Assembly tab in an Onshape Document and choose "Insert."
5. Then navigate to the "Other Documents" tab and go to the COTS folder you set up or use the built-in search tool to find it by name, part number or any other custom property you may have added in a library.

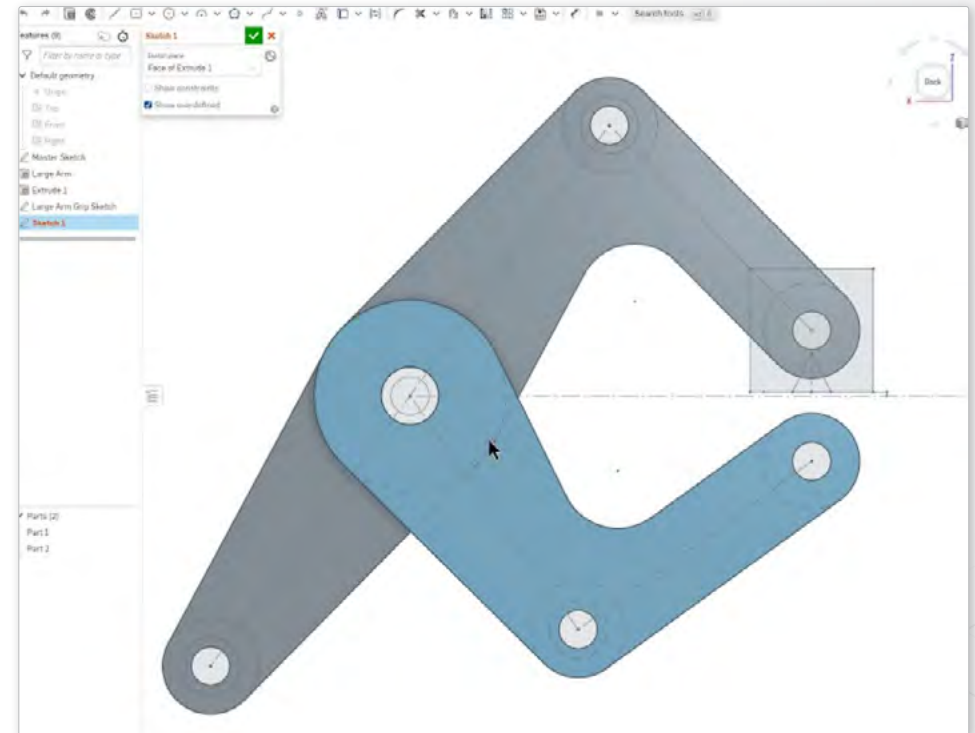


*This image shows an Onshape parts library where standard parts are grouped together in an Onshape folder.*

## 6 | How do I understand the difference between a Part Studio in Onshape and a Part File in SOLIDWORKS?

Onshape functions much differently than other CAD systems because of its cloud-native architecture. SOLIDWORKS has separate files for each part, assembly, and drawing, plus spreadsheets for design tables. Onshape is able to group contextually related parts together into a Multi-Part Studio.

In SOLIDWORKS, when creating multi-body part files, a user is required to separate the files into individual part files when finishing a design. In Onshape, users are able to design the parts that they want in a single-part studio. Users may then drop the parts into an assembly for use. There is no separate “split feature” for the master-model approach. This means that there is no need to deal with multiple part files with complex interrelated cross-references, which is a big advantage for ensuring document organization in Onshape.



| A sketch in Onshape



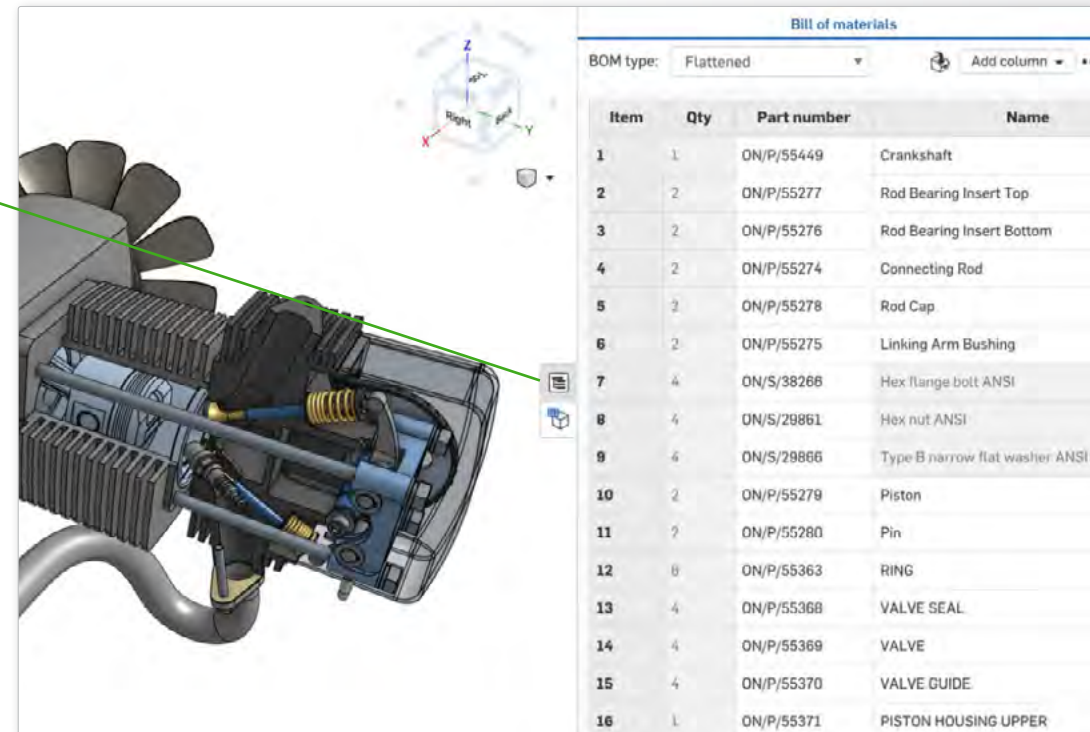
## 7 | How do I create a Bill of Materials in Onshape

Onshape's Bill of Materials (BOM) is automatically generated as part of the Assembly process. Objects in the BOM reflect real objects that are in the Assembly, not simply a table with a separate Drawing.

Open the BOM table by clicking on the BOM table icon on the right side of the screen. This table gives users an entirely new way to interact with their Assemblies.

Onshape users have the ability to add columns to the BOM with new properties if needed. Designers can edit part names to be more descriptive directly in the BOM, as well as any other property. Selecting anything in the BOM, including a row or specific part, will automatically highlight the item in the graphic, making various aspects of an Assembly easily identifiable in the BOM. The Assembly, along with specific properties of a Part, can be edited straight from the BOM as well.

The BOM can be viewed as either a flattened view or structured view. The default flattened view provides a simple list of parts by item number with no indication of subassemblies. The structured view provides a list of parts and subassemblies in an indented format (with a small right-facing caret). The BOM is easily shared, just like any other [Onshape Document](#).

The image shows a 3D assembly of an engine block with various components highlighted in different colors. A green line connects the BOM table icon to the table. The BOM table is titled "Bill of materials" and has a dropdown menu set to "Flattened". It contains 16 items, each with a quantity, part number, and name. The table is as follows:

Item	Qty	Part number	Name
1	1	ON/P/55449	Crankshaft
2	2	ON/P/55277	Rod Bearing Insert Top
3	2	ON/P/55276	Rod Bearing Insert Bottom
4	2	ON/P/55274	Connecting Rod
5	2	ON/P/55278	Rod Cap
6	2	ON/P/55275	Linking Arm Bushing
7	4	ON/S/38266	Hex flange bolt ANSI
8	4	ON/S/29861	Hex nut ANSI
9	6	ON/S/29866	Type B narrow flat washer ANSI
10	2	ON/P/55279	Piston
11	2	ON/P/55280	Pin
12	8	ON/P/55363	RING
13	4	ON/P/55368	VALVE SEAL
14	4	ON/P/55369	VALVE
15	4	ON/P/55370	VALVE GUIDE
16	1	ON/P/55371	PISTON HOUSING UPPER

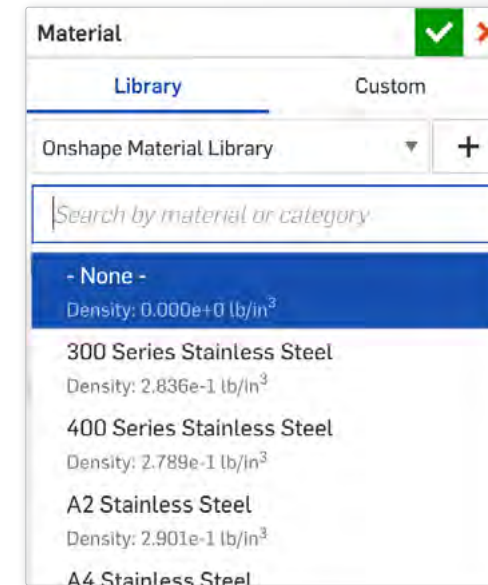
The Bill of Materials table can be accessed on the right in an Assembly tab.

## 8 | How do I customize the material library in Onshape?

Customizing the material library in Onshape is a very useful tool for designers. There are over 150 materials integrated into Onshape that can be used to assign the correct density to chosen parts. Select one or more parts from the parts list, then right-click and select “assign material.” A designer can add more materials, if needed, and share them with other members of the company.

Onshape’s material library is listed in a table. Users can search by category or by name to locate materials. One tip is to create a new document that will act as a container for the material library and give it a name. The list can then be edited in a CSV file for easy formatting and shared with the entire company with one click.

If you need more information on material libraries, visit our [Help Documentation](#).



*The Onshape material library is an easy way to assign materials to parts.*

## 9 | How do I use version control tools for designs in Onshape?

Onshape's version control tools capture the state of every tab in a workspace each time an edit is completed, which allows users to review revision history. Each edit by any user in a given Onshape Document is captured and saved, which compiles a complete list of every edit made in case a user needs to go back to a previous Document version. Onshape provides a way for users to capture an immutable version of a Document at any time.

The name of a workspace, or the place where Onshape users work, is displayed in gray next to the Document name in the Document title bar. Documents may comprise many different workspaces or versions, so workspace/version naming allows users to locate the workspace or version needed.

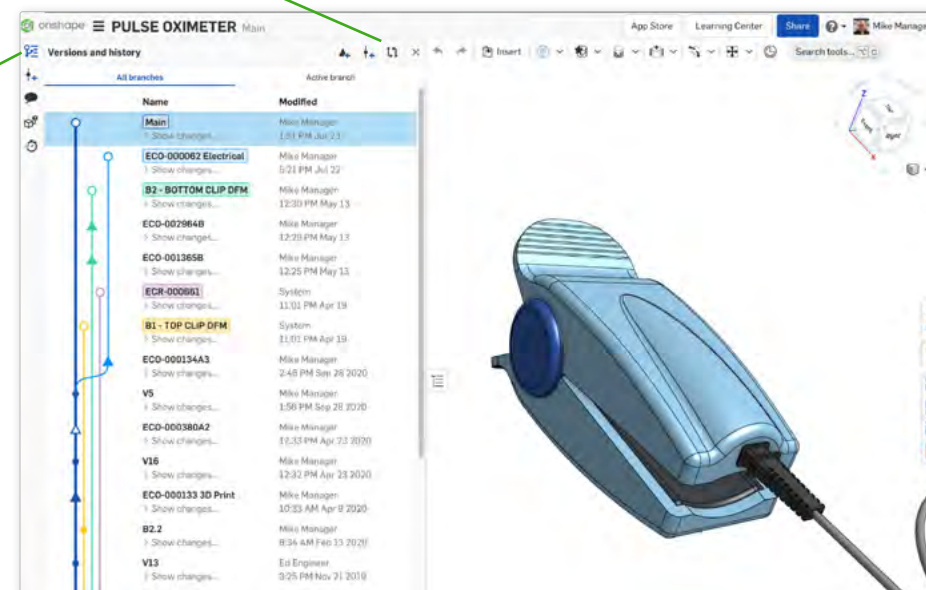
**Note:** Versions can only be viewed. Workspaces are where edits are made.

To view the history of edits made to an Onshape Document, users can click the **versions and history icon** in the Document panel on the upper left. Designers can click "show changes" to see the full list of changes for a particular version, workspace, or release. Hovering over any entry or edit shows a user when and by whom an edit was made. Designers can right-click on any of the Document history changes to complete other actions such as comparing workspaces/versions, viewing entries, restoring to main versions, or creating a new version.

*Users can navigate through previous versions and the history of an Onshape Document through the versions and history tab.*

Comparing allows users to identify how their work differs between workspaces or versions. Onshape allows users to compare two entities at a time, for example, two workspaces, two versions, or one of each.

Comparisons can be made by using the **compare icon** in the top-right corner. By clicking the features list on the left side, another dialog will open allowing users to view a list of the history of features created or edited. Onshape also has a "restore to workspace" option which allows users to bring workspaces back to their original form. Designers must right-click on any change point in the "version and history" in the Document panel and select "restore to <name>."



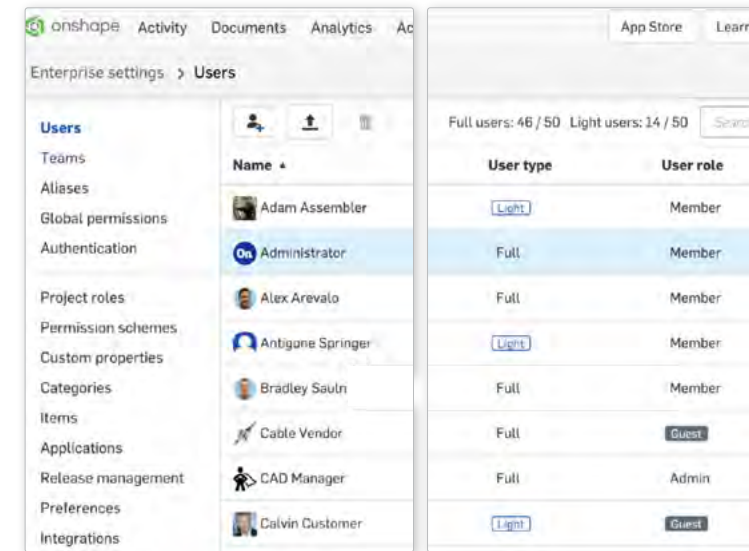
## 10 | How do I add users to Onshape? Where would you find the function to add new users?

Adding users to Onshape takes only a few simple steps. Administrators can add users by heading to their user profile and clicking on “company settings.” Users are added by entering new member email addresses as long as enough subscriptions are available to the company.

To purchase more subscriptions, an administrator can either contact their Onshape customer success manager or account executive or purchase more in-app if the original purchase was also in-app. Administrators will find an option to increase the number of users they are paying for and click “Update” under company settings.

New Onshape users will be specified as either a member or an administrator. Administrators have the permission to add new users, change the role of other members, delete Onshape Documents, and change other company settings in Onshape.

Members are more limited than administrators, but have the ability to both share documents with the rest of the company and receive documents that are shared by others. Documents created and shared in Onshape are owned by the company, not by a specific administrator or member. It is important to note that designs remain secure as users leave the company and new users are added.



New users can be added through the user tab in an Onshape Document.



## Ensuring a Seamless Transition

A smooth transition between CAD systems is important for product designers, engineers, and manufacturers across the globe.

We're ready to help. At Onshape, our in-depth set of [helpful tools](#), a friendly customer success team, and a busy [Onshape Forum](#) will ensure new users are successfully set up in Onshape so they can focus on what's important: Designing products better and faster.

For more assistance with any of the topics above, visit the [Onshape Resource Center](#).

