

Onshape Education



INSTRUCTOR GUIDE

Onshape
A PTC Business

Tips, best practices and resources for instructors using the Onshape Education Plan

Contents:

About this Guide	2
Introductory Webinars & Learning Resources.....	2
Manage Students in Onshape by Creating an Onshape Team	2
Organize Documents with Folders.....	3
Sharing Documents with Students Individually or as Teams	3
Evaluating and Grading Work with Versions	3
Importing Data from your old CAD System	5
Collaborative Learning and Working	6
Standard Content.....	7
Onshape isn't just for CAD	7
IT Administrators.....	8

About this Guide

This is a guide for current and new Onshape instructors. Onshape is a completely cloud-based design platform that requires no downloads or installation, and can run on any type of device with an internet connection (Windows or Mac, Android, Chromebook, etc.).

Introductory Webinars & Learning Resources

The Onshape Essentials webinar is a great place to start for those who are brand new to Onshape. You will learn the ins and outs of Onshape which will quickly prepare you for some of the more advanced sessions.

You can also review Onshape for the K12 Classroom, a webinar that reviews a basic demo of Onshape for Education and also some sample classroom curriculum used by K12 educators.

More webinars and learning resources can be found in the Learning Center.

Additionally, you can find discussions focused on Onshape curriculum, best-practices, classroom setup and more in the [Onshape Instructor Forum](#).

Manage Students in Onshape by Creating an Onshape Team

The Onshape Teams feature can be used to group students in a single class, a group project, or both. Use the steps below to create student teams:

1. If they don't already have them, have students create their own accounts [using this link](#).
2. After the students have created their Education accounts, go to your **My Account** preferences and select [Teams](#) to create a team and add them to a team via email address.
 - a. If you have a spreadsheet of emails, you can copy and paste a column of emails from your favorite spreadsheet editor, or a comma-separated list.
3. Make your students **Members** of the team
 - a. Students added to a team will receive an email confirming they have been added to a team.
 - b. Members receive notification emails when they are added and removed from a team and are able to belong to more than one team at a time.
 - c. Team members can see other team members.
 - d. Team members are able to remove themselves from a team
 - e. A team admin may delete the team at any time. Upon deletion of the team, all documents shared with the team become unshared from the team members and removed from their Documents lists.

[Read more about Teams here in the Onshape help.](#)

Organize Documents with Folders

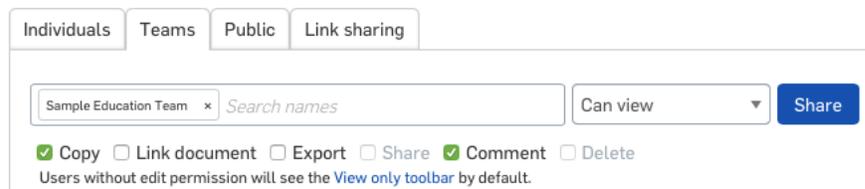
It's important to understand that all content in Onshape are stored in documents. Documents are collections of parts, assemblies, drawings, uploaded models and even PDFs, images or video.

The best way to organize documents in Onshape is by using **Folders**

- Folders can be created inside of other folders, but only the top-level folder can be shared with individuals and teams
 - Sub-folders and documents within folders inherit the sharing settings of the folder they are in, so sharing a folder with a student or team shares everything within by default
- Documents within folders and sub-folders can be shared with both individuals and/or teams

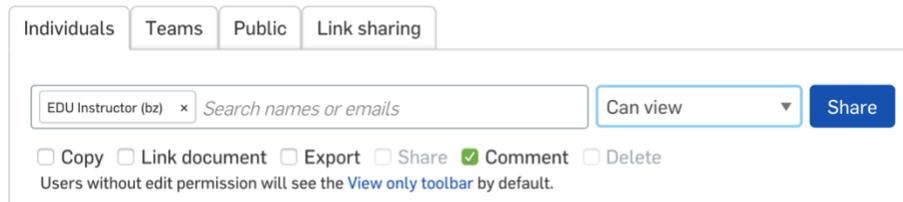
Sharing Documents with Students Individually or as Teams

1. To Share a document, an instructor can create a new folder and share it with the team or individual student using **View** permissions.
2. The creator of the folder will be designated the owner of that folder (and all documents within).
3. By using View permissions, students will have to make a copy of the documents in order to edit them – make sure to check the **copy** option to allow that.



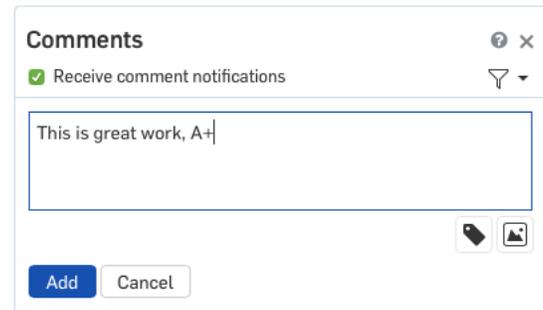
Evaluating and Grading Work with Versions

1. When a student finishes an assignment, they should:
 - a. Make a version
 - i. This will make a bookmark in time; if students keep working on the **Main** workspace, the link to the version won't change, so instructors can still see what was done on time.
 - b. Share the document with the Instructor, with **Can view** and **Comment** permissions



2. An Instructor can:
 - a. Open the document and view Versions and History
 - i. Not familiar with versions? [Read about Versions in the Onshape Primer](#)
 - ii. Review version time stamps to ensure completion was by the due date/time
 - b. Comment to leave a grade or feedback
 - i. To add a comment, right-click and select **Add comment**.
 - c. Comments can be added to Features in Feature lists, Mate connectors, Mates, Entities in a sub-assembly and Drawings

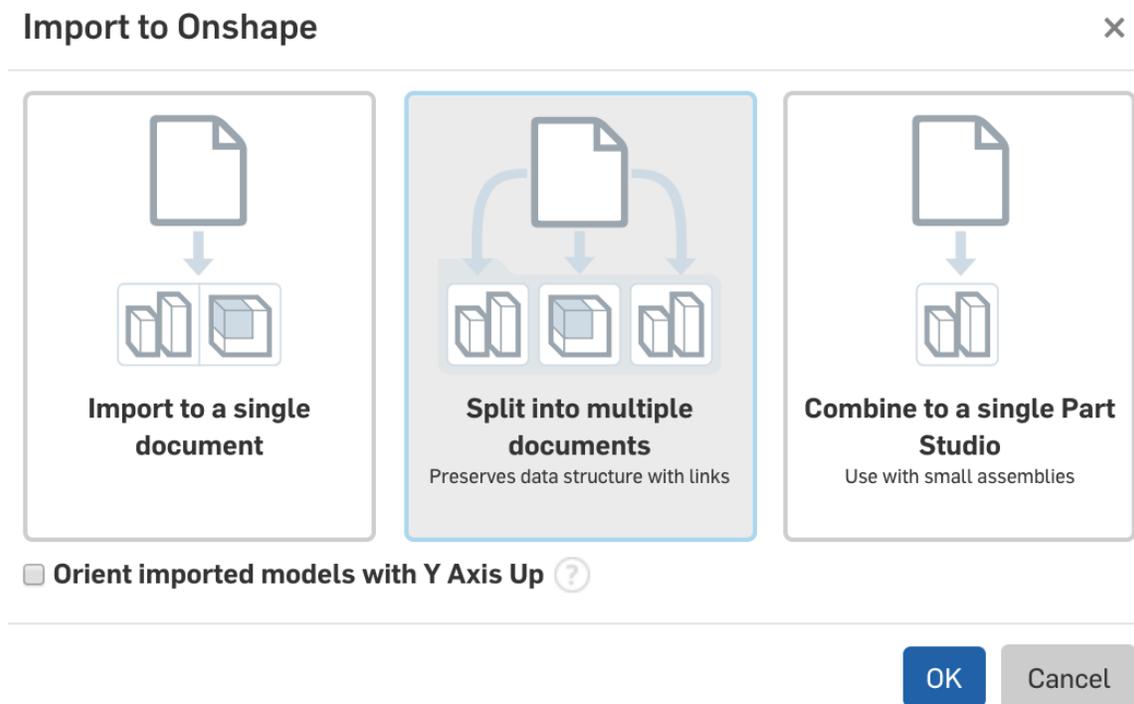
Note: This can also help ensure honesty, as the entire history of the document can be viewed in the same place, so an instructor can see who did what work and when.



Importing Data from your old CAD System

When importing a CAD file into Onshape, you have the following options:

- **Import all assemblies and parts to a single document** - Part Studio and Assemblies are created as needed, depending on the contents of the imported file and named according to the names in the imported file.
- **Split into multiple documents** - If the file is an assembly, this option creates a separate document for each part or subassembly and places those documents in a new folder whose name is the imported file name. The data structure and its links are preserved.
- **Combine to a single Part Studio** - If the file is an assembly, or contains an assembly, you have the option to import it as only a Part Studio. In this case, the assembly is stored as a set of parts in a single Part Studio. There will be duplicate parts created whenever a part is instanced more than once. Be aware that this choice should only be made if the assembly is small, otherwise, performance will be affected.



If you haven't done this before, check out [Importing Files in the Onshape Primer](#) for a quick overview.

For more on migrating data from your old CAD system, be sure to read the [Importing Data](#) technical briefing in the Learning Center.

Collaborative Learning and Work

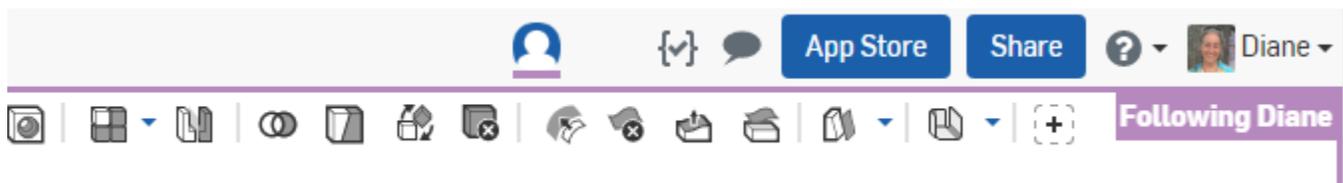
Making sure everyone is on the same page at the same time is crucial to success. Onshape is a collaborative tool designed for project-based activities. Your team can work on the same part or assembly at the same time – without any fear of data loss – everything you do in Onshape is updated in all open Document in real-time.

In a group project, multiple people can simultaneously work on [Onshape Documents](#) together. Different students in a group can build different components, drawings and assemblies all within one Onshape Document, and can watch each other work and work on the same Part Studio.

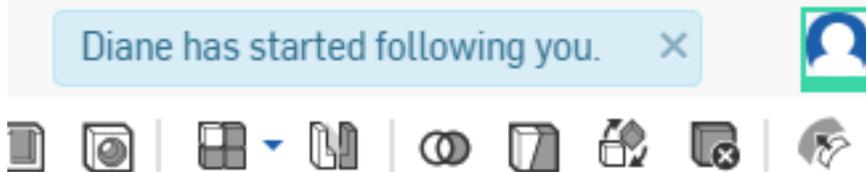
- You can give design feedback and even tag other users in the comments to give design feedback.
- You can put links or QR codes on PDFs or parts for quick access to parts or assemblies.
- The Onshape mobile app makes it easy to show and debug problems, even when you can't be at your computer.
- When users are collaborating in a single document, they have the ability to choose to follow another collaborator. This allows the follower to see the actions of the other collaborator.

For teaching CAD, one of the most powerful features of Onshape is Follow Mode, where a user may follow a collaborator across browser and mobile. This means a user on a browser may follow a collaborator on a browser or mobile device or vice-versa.

To follow someone, double-click their social cue icon in the toolbar at the top of the window. Notice that the workspace is outlined to indicate to you are following the collaborator:



The user being followed sees the follower's social cue outlined to indicate the follower is using Follow Mode:



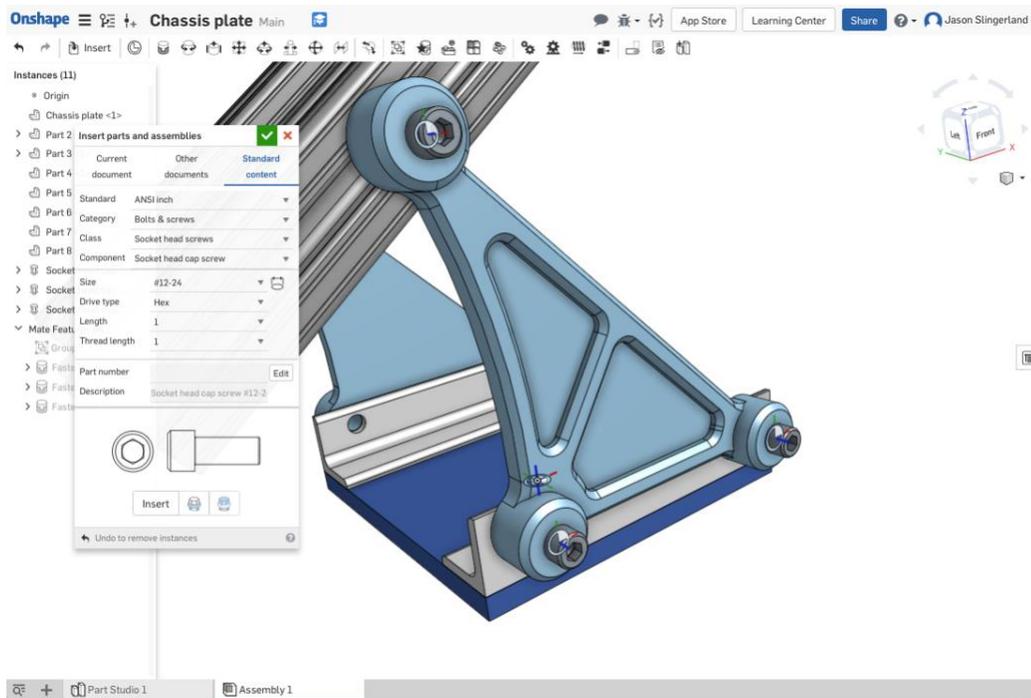
To stop following, click anywhere in your browser window.

Read more about [how to use Follow mode](#).

Standard Content

There is no need to design all of the nuts, bolts and washers from scratch or import them from a supplier. Onshape Standard Content is a collection of fasteners that you can insert inside of your assembly.

1. After you have some parts modeled and assembled in Onshape, select **Insert** and then the **Standard Content** option on the far right.
2. Once you have found a component you are interested in, select **Insert**.
3. There is also an auto-size button if you forgot the size of the hole.



Onshape isn't just for CAD

The Onshape App Store offers many tools to help design teams achieve their goal.

A few recommended apps for team-based projects include:

- [Link tab for Onshape](#) This app allows you to pin webpages inside of Onshape. You can now have a Google sheet with your Gantt chart or the rules page right inside of your Onshape Document for easy access.
- [SIMSOLID](#) offers a 14-day free trial if you need to run some structural analysis on parts to ensure you are loading them correctly, and so they won't break when you need them most.
- [VisualCAMc](#) (14-day free trial) and [Kiri:Moto](#) (free forever) both offer CAM (computer-aided manufacturing) tools to make toolpaths for anything from a ShopBot® to a HAAS VF-2.

IT Administrators

If your school IT department needs URLs, IP addresses and emails to whitelist so your team can work with school emails and on-campus networks, please contact [Onshape Support](#).

If there are any other questions you or your team have, please check out the [Forum](#), our [Help documentation](#), [Learning Center](#) or reach out to [Onshape Support](#).