

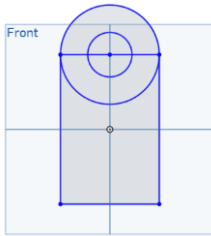
Course Wrap-Up

Congratulations, you've completed the 12-week course on Computer-Aided Design (CAD) using Onshape! Even though this is just an introductory course, you've made some pretty cool models, along with two fairly complex projects. In addition to becoming comfortable with Onshape, you hopefully have a better understanding of CAD and how it has become such an integral part of the Product Development Process. No longer just an "automated drawing tool," CAD is now a platform where designers and engineers can document their design intent, share new ideas with others, and communicate within their team.

In addition, this curriculum was intended to introduce you to the concepts of Design for Manufacturing & Assembly (DFM/DFA), specifically how to use Top-Down Design techniques to design the perfect assembly, manage hardware and other "off the shelf" components such as speakers and electric motors, and consider common manufacturing processes like Plastic Injection Molding, and CNC machining.

To conclude this course, let's briefly review some important concepts that we covered throughout the weeks.

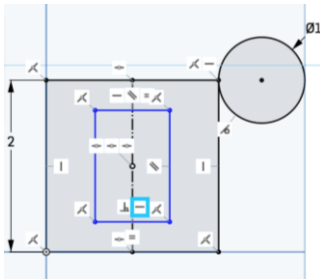
Sketching



When sketching, you have to first define a **sketch plane** in a **Part Studio**. In a sketch you create **sketch entities**, such as lines, circles, arcs, rectangles, splines, and points.

Your sketches will serve as the foundation of your 3D CAD model, so make sure your sketches are clean, purposeful, and fully-defined whenever possible.

Dimensions and Constraints



Enough dimensions and constraints will **fully-define** a sketch, changing the color of the sketch from blue to black. Dimensions refer to the distance and angle values of sketch entities. Constraints refer to the geometric relationships and rules within a sketch, common ones being Coincident, Horizontal, Vertical, Concentric, Midpoint, and Tangent.

A sketch can be **over-defined** when the constraints applied to it contradict other existing constraints and make the sketch invalid. Onshape will notify you if you sketch is over-defined by making the sketch entities red.

Remember that dimensions and constraints exist to define your **design intent**, the practice of developing your project's objectives and requirements even before working on your design. It might be easier to think about design intent by reversing the words to "intended design".

Parts

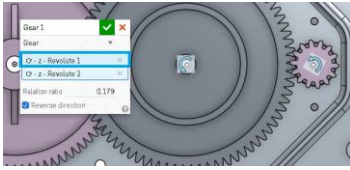


Most objects can be made through the 2D sketch > 3D feature workflow. Recall that the **four foundational features** - Extrude, Revolve, Sweep, and Loft - allow you to create just about any geometry in the world! Of course, you can create 3D geometry without a 2D sketch, such as with Fillets and Chamfers, Thicken, Draft, Shell, Boolean, and Pattern.

Assemblies

In Onshape, **Assemblies** refer to components whose parts move relative to one another. You can make Assemblies in the **Assembly Tab**.

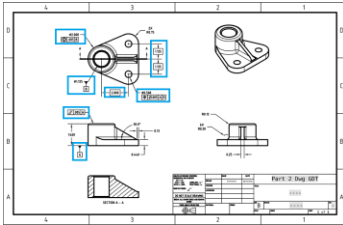
We can define the relative movements between parts through



Mates. In Onshape, you use **Mate Connectors** to determine the relative orientation between two parts, and use Mates to describe the motion. Some common examples of Mates include Fastened, Revolute, Slider, and Cylindrical.

Relations constrain degrees of freedom between two Mates. There are four types of Relations in Onshape - Gear, Rack and Pinion, Screw, and Linear. These Relations rely on pre-existing Mates to define the type of motion that will occur between the two parts.

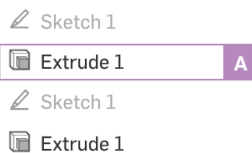
Drawings



Engineering Drawings help document our designs in a clear and concise manner, such that the original design intent is communicated to others. Drawings are especially important when you need to manufacture your design.

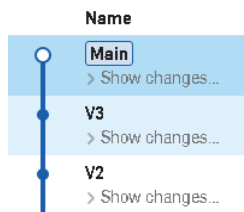
You can use projected, section, detailed, and auxiliary views to capture different components of your part. **Tolerances** state how much variation the dimensions are allowed during the manufacturing process. In addition to standard tolerances in the Drawing **Sheet Format**, some companies use the **Geometric Dimensioning and Tolerancing (GD&T)** method for their drawings, which include datums and geometric tolerances.

Sharing and Collaboration



When designing, you're almost always going to work with other people. Onshape allows you to share your private Documents with people you specify so that you can **simultaneously collaborate** on a CAD model. You can **follow** your teammates by double-clicking on their icon and **comment** on your model using the commenting system.

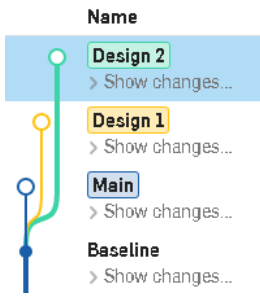
History and Versions



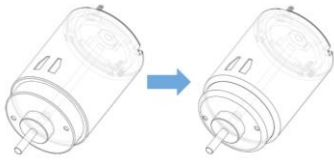
Whenever you make a change in your model, Onshape permanently documents it in its **History**. You can also create **Versions** of your model to document certain milestones of your project. Versions are view-only, but you can always go back, or **restore**, to a previous Version.

Branching, Comparing, and Merging

You can also pursue different versions of a design in parallel using **branching**. Let's say you start with a baseline design. You can branch off from that baseline to create many different versions of the design. Then, you can **compare** designs to see which design is the best, or even **merge** them if you want to incorporate multiple design elements from different versions. Onshape allows you to do all this through the "Versions and history" flyout.



Direct Editing



Onshape allows you to import different types of files, including PDFs, JPEGs, and most importantly, other CAD files (such as from Solidworks). You can make changes to an existing model by using the **Direct Editing** tools, such as Modify Fillet, Delete Face, Move Face, and Replace Face.

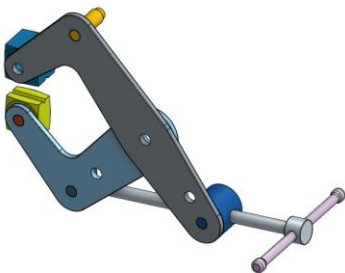
App Store and Onshape Mobile



The Onshape **App Store** has a broad offering of apps for simulations, such as Finite Element Analysis (FEA) and Computational Fluid Dynamics (CFD), Rendering, 3D-Printing and Computer Aided Manufacturing (CAM) for Computer Numerically Controlled (CNC) machining.

You can also access Onshape via your phone. Onshape **Mobile** is commonly used to “respond to comments while on the road” but is also a full-CAD editor, allowing you to edit existing designs or create new documents from scratch..

Top-Down Design

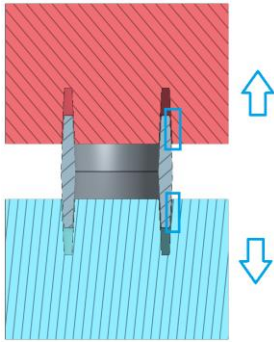


Top-Down Design is when the shape of an overall product is sketched first, and then different regions of that sketch are used to create the lower level parts and their features. Top-Down is a more intuitive way to approach a design because typically we, as designers, envision the final product first, then as time goes on, we refine the concept into finer and finer detail.

There is also **Bottom-Up Design**, where geometry is creating starting with the lower level entities (like 2D lines and circles) up to the hierarchy to the highest level assembly (such as the final product being built), which can be helpful for reverse engineering.

DFM/DFA

Some **Design for Manufacturing (DFM)** techniques we used in our lesson plans include applying **Drafts**, adding **Fillet**s,



and using the **Hole** tool to prepare for common processes like Plastic Injection Molding and CNC Machining.

Some **Design for Assembly (DFA)** techniques we used include managing hardware, using section views to look for interference, analyzing **Mass Properties**, creating assembly drawings, and exporting Onshape documents to different CAD file types.

That's It!

Of course, there are many, many other topics in CAD and design in general but you're off to an excellent start. We're excited to see what you'll come up with!