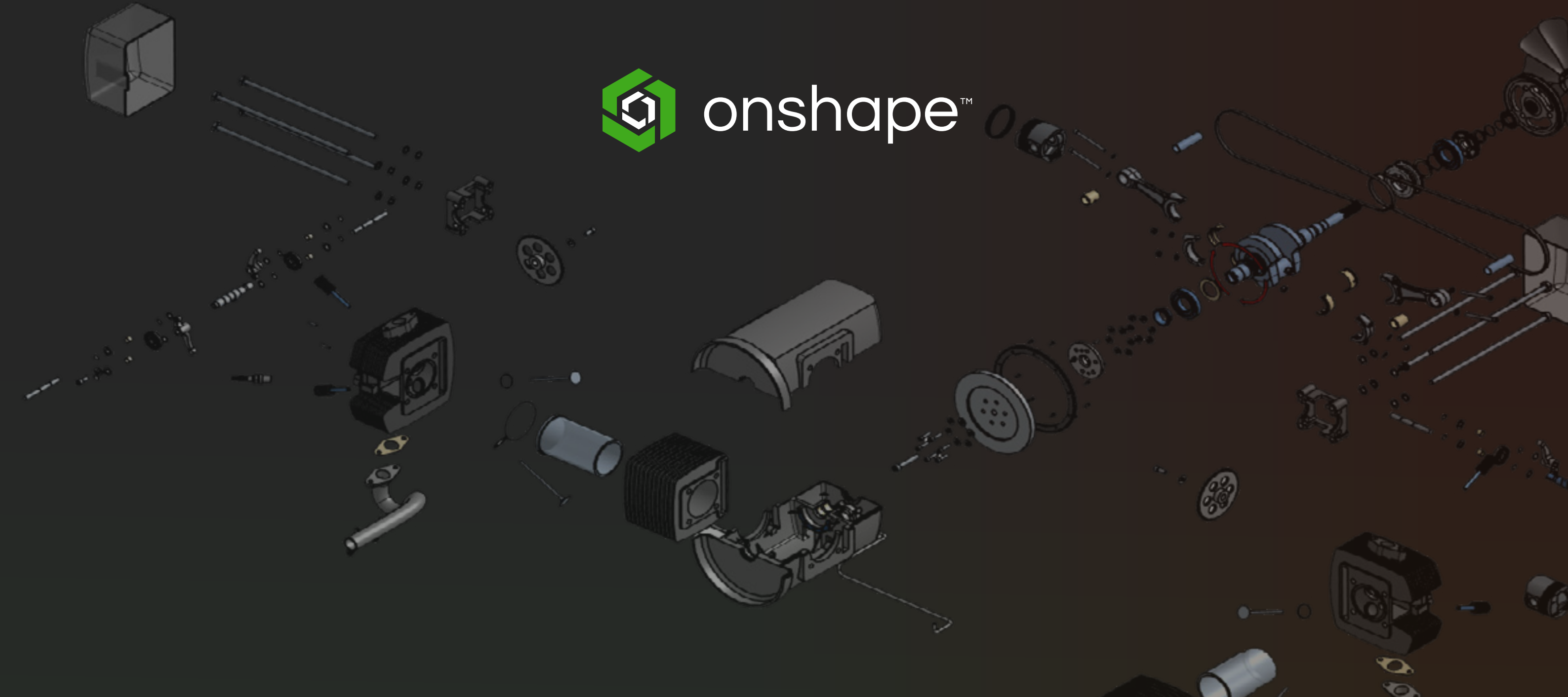
onshape™



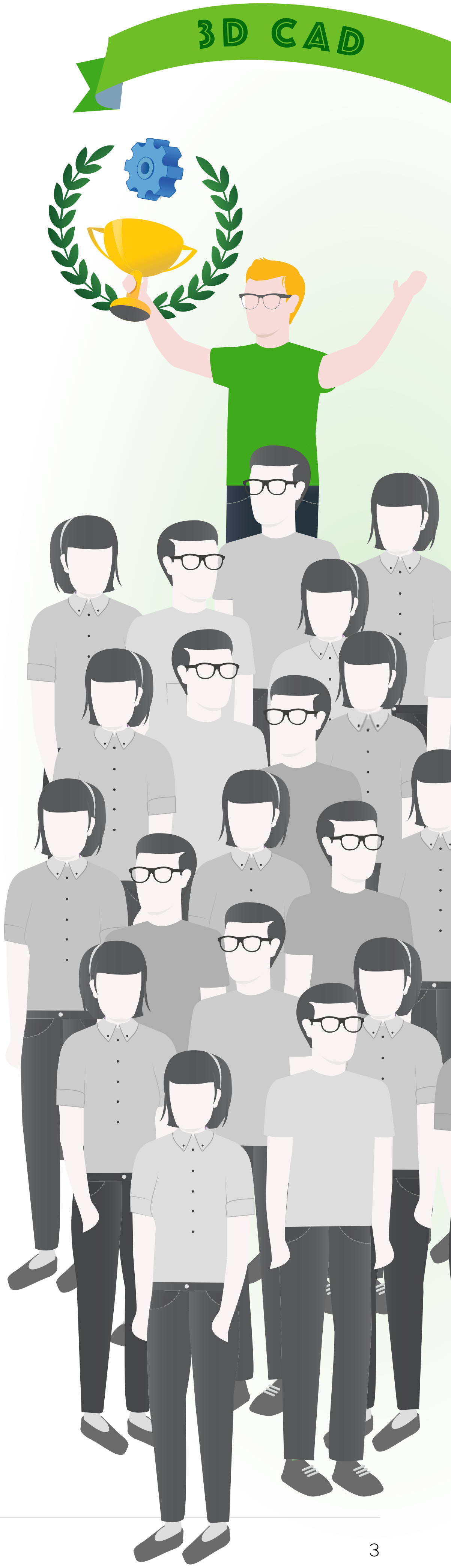
Mastering 3D Modeling

Field-Proven CAD Techniques to Help You Tackle Any Design Challenge

Table of Contents

onshape

Why Are Some CAD Users Better Than Others?



In every company, there is always at least one or two power users who seem to know everything about CAD. Were they born with a special CAD gene? Are they just more talented than the rest of us? No, for the most part, CAD is a learned skill. Your office expert has "been there" and "done that" and learned the hard way – by failing. Through experience and practice, he or she has learned the pitfalls of parametric modeling and how to avoid them.

This eBook aims to dispel the myth that 3D CAD is hard. It's not.

Modeling in 3D is really not that difficult if you just follow a simple set of rules.

No matter which 3D CAD system you use, the more effort you put in, the more you will get out of it. Lots of detail means lots of CAD features which means lots of time spent modeling. But does it have to be this way?

This eBook will focus on methods you can employ to get the most out of your CAD system with the least amount of effort. We'll explore ways you can plan your models ahead of time, tips to create robust models that are easy to modify, and lastly, how moving your modeling to the cloud will help you get your job done in record time.

Of course, it's important to attend your CAD vendor's training sessions to know what buttons to press and in which order. However, these training sessions often don't go into enough detail on the "do's and don'ts" of 3D modeling that can either speed up your momentum or trip you up and block your progress.

Whether you are transitioning from 2D CAD, are an experienced 3D veteran or brand new to CAD altogether, the tips in "Mastering 3D Modeling" will set you up for success!

Breaking Down The Typical Design Process

During a typical design project, there are two main stages: conceptual design and detailed design. **Conceptual design** is completely fluid until the moment you draw a line in the sand, focusing on brainstorming, solving problems and refining your ideas. **Detailed design**, on the other hand, is where much of the hard work and heavy CAD modeling comes in. This is when you have to make your ideas work, make crucial decisions, and sometimes compromise your design to ensure parts fit together and can be manufactured cost effectively. Both of these design stages are highly iterative, so you need the freedom to make changes quickly and easily. A flexible 3D CAD system and a robust modeling method will facilitate this.

Some people like to create conceptual designs on paper and some directly in a CAD system. In reality, it's a mixture of both. Sketches and design ideas may be started on paper (still the fastest way to visualize your thoughts), but it's not until you draw your idea to scale in a CAD system that you can really see if it's going to work. Once the 2D concepts are ironed out, parts are then developed in 3D for feasibility testing and then further developed into the final detailed design. This is when the lines become blurred between conceptual and detailed design. With traditional file-based CAD, it is commonplace for users to keep conceptual and detailed designs separate, sometimes even in different software, starting from scratch with the detailed design because traditional CAD tools were never designed to work this way.

p.s.

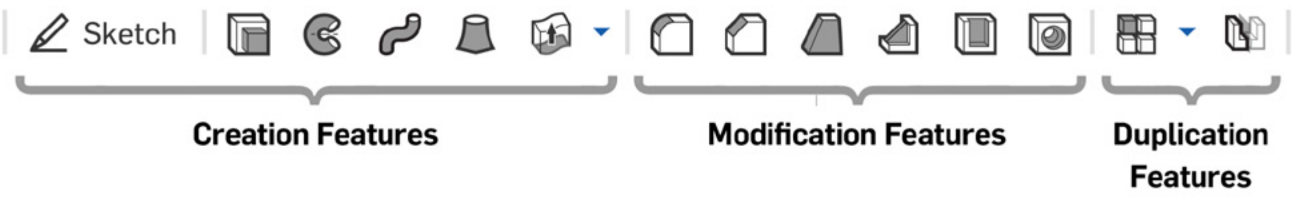
Onshape's Part Studios are designed to let you mix and match any type of design strategy. Since a conceptual design is usually created using a layout sketch, the multi-part architecture of a Part Studio lends itself perfectly to mix both conceptual and detail designs.

This helps you save time and prevents mistakes as your concept evolves into the final design.

Anatomy of a Robust 3D Model

In a feature-based parametric model, there is a process that you can follow that will make you more efficient and more confident that your design will update predictably when you need to make changes.

Typically, feature icons are laid out in groups to represent features that create geometry (sketch, extrude, revolve, sweep, loft, etc.), features that modify geometry (fillet, chamfer, draft, rib, shell, hole, etc.), and features that duplicate geometry (pattern, mirror). There are others, usually required for more complex modeling, but the core tools are represented by this small number of commands.



ll of these features combined in a certain order define the topology of your model. Modification and duplication features can only affect existing geometry, so you must start with an extrude or a revolve or some other geometry creation feature. But before you do that, you need a sketch.

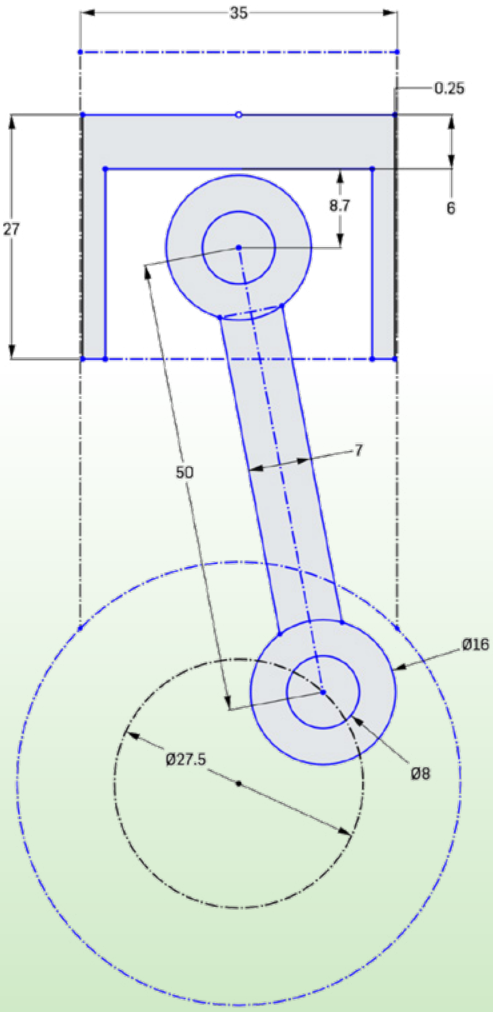
Whether you are creating a conceptual design or a detailed design, everything starts with a sketch. A sketch represents a 2D section view of one or more design features in your model, and can be used to quickly lay out a conceptual design. Geometric constraints help you lock down known elements and even define things like motion, while dimensions set known sizes. Leaving geometry unconstrained (blue) lets you tweak your sketch quickly to test multiple design scenarios. A single sketch can contain the geometry for multiple parts and even features within a part, like holes and cutouts.

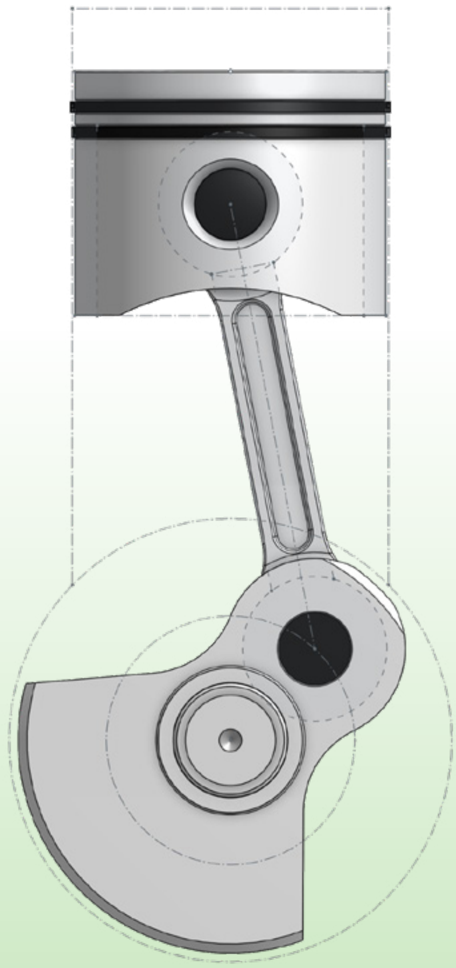
p.s.

One of the main advantages of Part Studios in Onshape is being able to build the final 3D models in-place, relative to the original concept sketch. Any changes to the design specification can be applied at the sketch level and all associated 3D models will update accordingly.

Anatomy of a Robust 3d Model

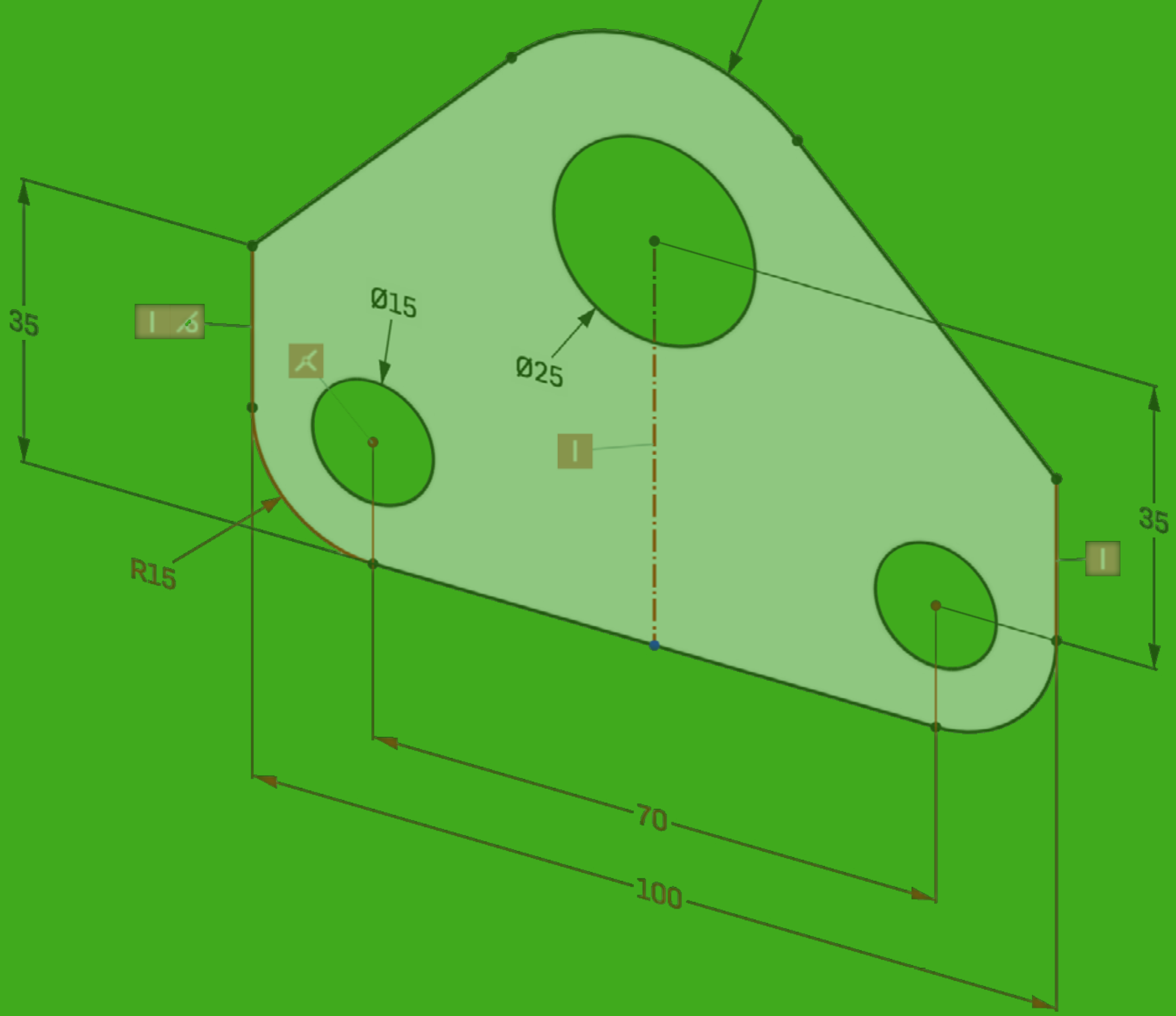
A sketch is the backbone of a robust solid model. Using dimensions and geometric constraints like parallel, concentric and equal, you can add design intent to your sketches so that any future changes will not require any additional thought or rework in order to maintain that intent.





Using a concept layout sketch to build the final assembly

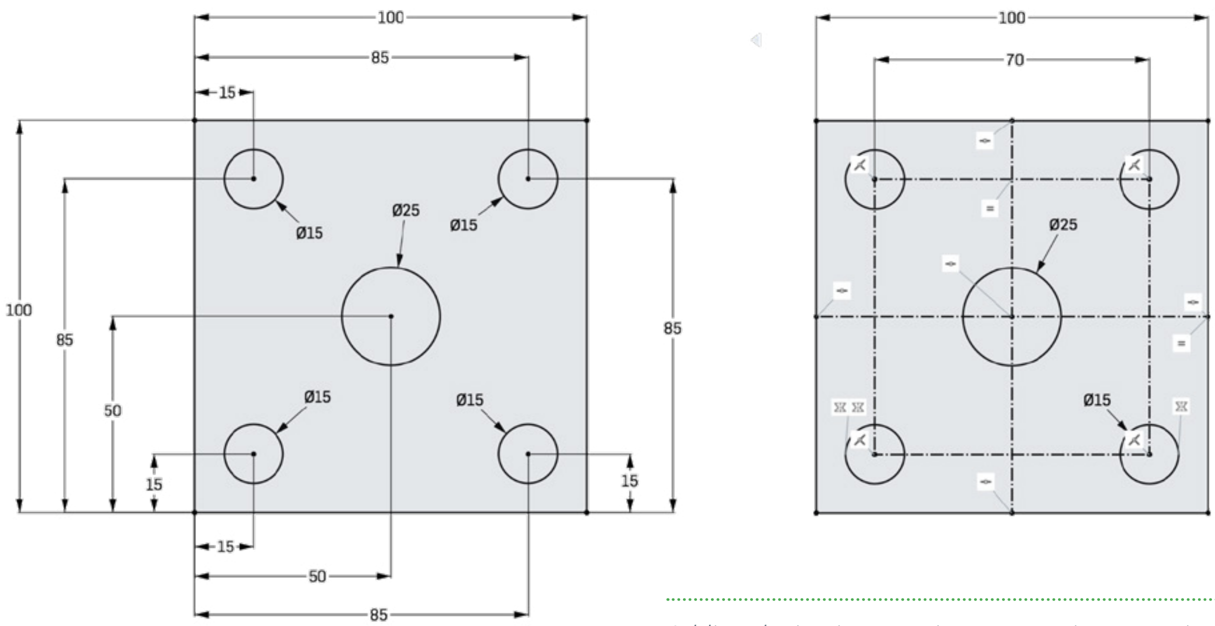
Let's take a look at some frequent mistakes made by CAD users while sketching and how to avoid them...



Common Mistakes to Avoid When Sketching

1. When to Use Dimensions Versus Constraints
2. Avoiding Detail Overload
3. When to Use Sketch Fillets Versus Feature Fillets

1 When to Use Dimensions Versus Constraints



Adding design intent using geometric constraints

Consider these two identical parts. The one on the left is dimensioned from a single datum as it would be on a manufacturing drawing and every hole is called out individually. This is an extreme example, but can you imagine how much work would be required if you had to move the hole centers? That's a lot of clicks and extra manual checking required even on the most simple sketches.

Now take a look at the sketch on the right. Using some extra construction lines and some mirror, equal, and midpoint constraints, you can visually describe your design intent: "a 100mm square plate, with a 25mm diameter hole at the center and 4 x 15mm holes spaced equally, 70mm apart." Now try to describe the sketch on the left. Not so easy.

When creating fully defined sketches for production models (all geometry is black), always prioritize constraints over dimensions. That way you can be sure your sketch will update predictably and preserve your design intent. Manufacturing information can be added to the drawing later.

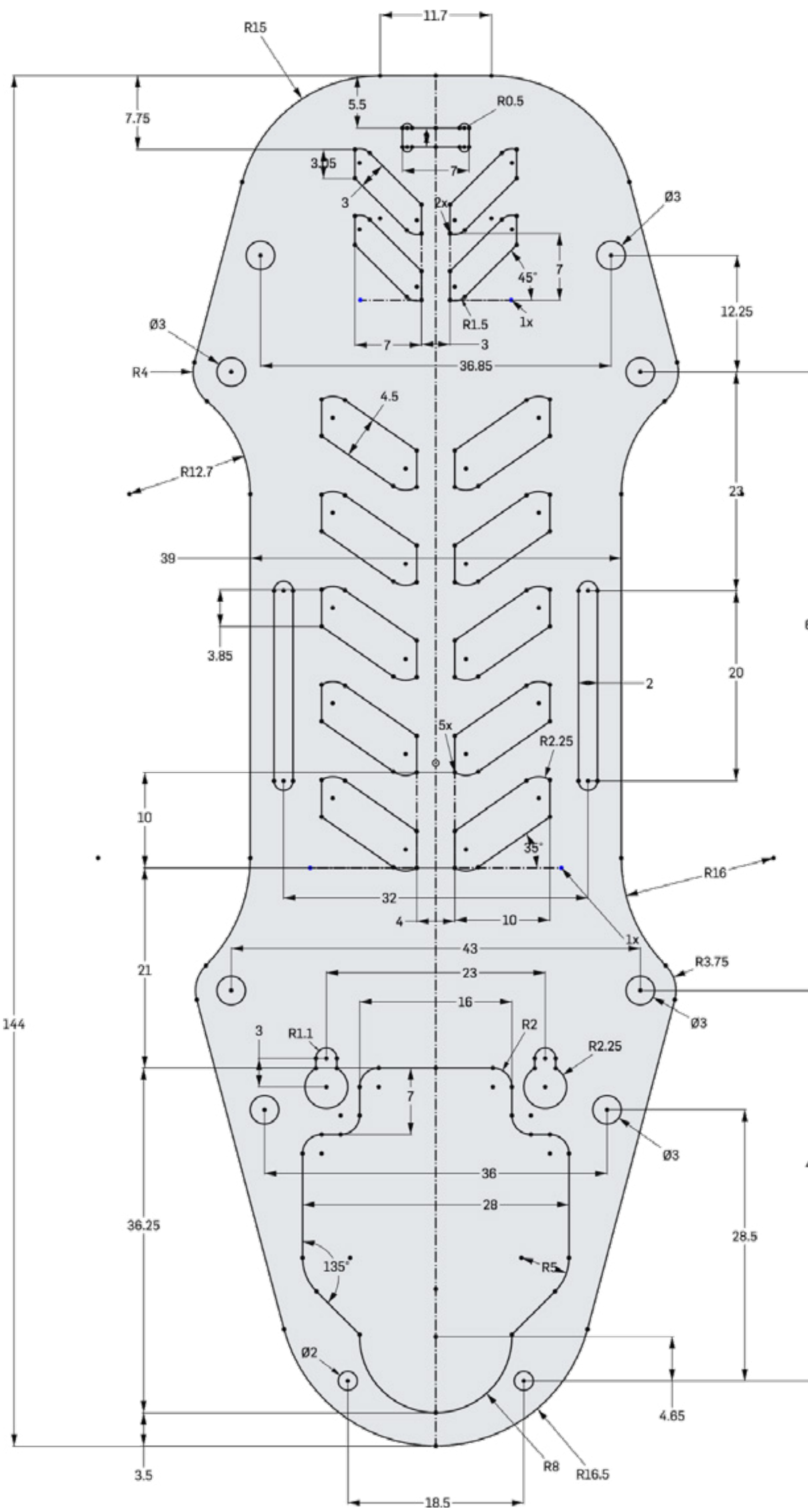
2 Avoiding Detail Overload

One of the biggest mistakes that can be made with a sketch is to make it too complicated. Keep it simple. It's very tempting to add all the detail you need to one master sketch, but once you get over a certain number of sketch entities you'll soon meet the point of diminishing returns. Sketches take longer to create, constraints are harder to add and troubleshoot, and design intent is more difficult to understand. Plus, for every entity, constraint and dimension added, the sketch solver has an order of magnitude more simultaneous equations to solve which has an adverse effect on performance.

If you do need all that detail, break it down into two or more sketches. In the example shown, the mounting holes are the important features and the external profile that surrounds them is secondary. There are also a number of vent features whose size and position are a tertiary consideration.

Therefore, Sketch 1 should contain just the mounting holes, Sketch 2 should contain the external detail, dimensioned relative to Sketch 1, and Sketch 3 should contain the vents, dimensioned relative to the external profile and/or the mounting holes.

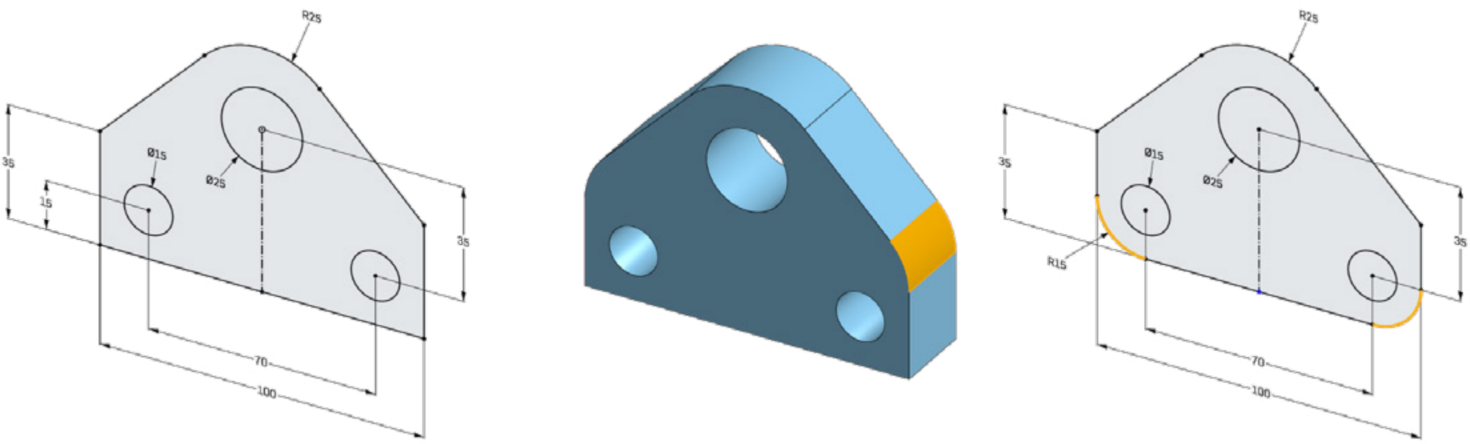
Remember that sketches that drive solid features do not need to be in any particular order. Your first extrude can be driven by Sketch 2, and your cutouts driven by Sketches 1 and 3.



How NOT to create a parametric sketch

3 When to Use Dimensions Versus Constraints

On the theme of keeping sketches simple, should fillets be added to the sketch or as a feature in the solid model? Adding fillets to a sketch is just adding complexity. However, if your sketch is simple and the feature that is derived from it is simple, then there's no harm in including fillets in your sketch. If you work to the rule that functional fillets should be added to the sketch and cosmetic, break-edge fillets should be added to the model, then you'll be in good shape.



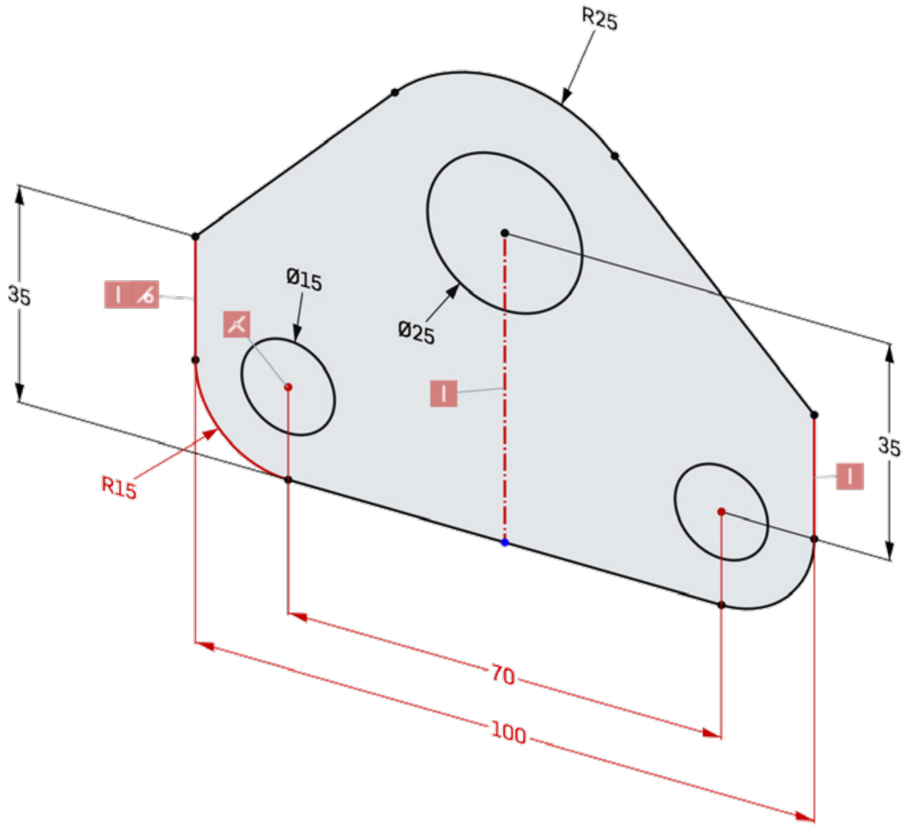
Adding fillets to the sketch or the model

In the first sketch on the left, the arc at the top is concentric to the large hole. This is absolutely a functional fillet because it is concentric to another entity in the sketch. However, the sharp edges at the intersection between the vertical and angled faces is purely for cosmetic purposes.

That should be added as a fillet feature in the model (highlighted orange). However, if fillets are to be added to the bottom corners, and they are to be concentric to the two smaller holes, then you can add them to the sketch if you wish.

If things go wrong, don't panic!

If your sketch turns an evil shade of red, something has gone awry. Don't panic! It's simply the sketch solver letting you know that you have conflicting constraints or dimensions. For example, if you set a circle to be equal in diameter to another and then add a dimension, which should the solver solve? The dimension or the equal constraint? This is where you have to help it out a little.



Which of these dimensions or constraints are not required?

When you have a conflicting constraint, you may expect only those two conflicts to be highlighted. When several appear at once, it can be a bit daunting. This is simply the sketch solver telling you that if you delete any one of the highlighted items, the sketch can be solved. So which one should you delete?

That can be a hard question to answer especially if your sketch is complex (remember, keep it simple). Looking closely, you will see that the fillet in the bottom left corner has a dimension and it has a coincident constraint to the center of the hole. Delete one of those (probably the dimension, as per previous instruction) and you're golden.

Once you've created your first sketch or layout sketch, you can start to build your 3D geometry. But before you do that, let's dig a little deeper into the inner workings of a parametric modeler.

How Does a 3D Parametric Modeler Work?

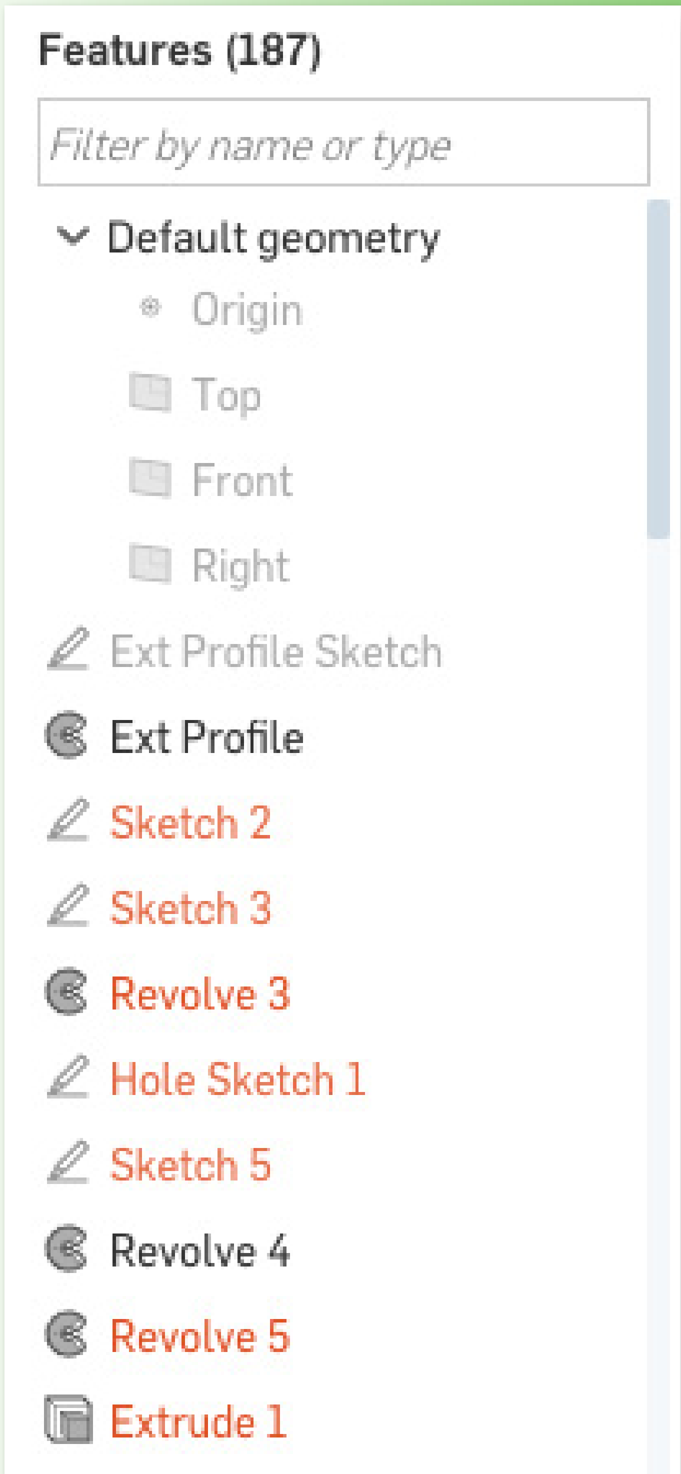
Who cares as long as it does, right? That's absolutely how it should be. However, it's worth having a little knowledge about the inner workings just so you can appreciate why things can sometimes go wrong. Horribly wrong.

You may think you're making a seemingly innocent change early on in your feature history when all of a sudden, your Feature List lights up like a Christmas tree and your entire model blows up.

This has to be the most frustrating aspect of parametric modeling, so it's worth a little effort knowing how to avoid it.

Behind the scenes, every piece of geometry created by a parametric modeling system has a unique ID number. This number is used by subsequent sketches, features, assembly mates, and drawings, to work out where things should be placed relative to the rest of the model.

So if that ID number no longer exists, guess what? That's right, the feature does not know what to do and fails. If a design change makes an edge or face disappear or changes the topology of your model dramatically, it is likely that some downstream features may fail.

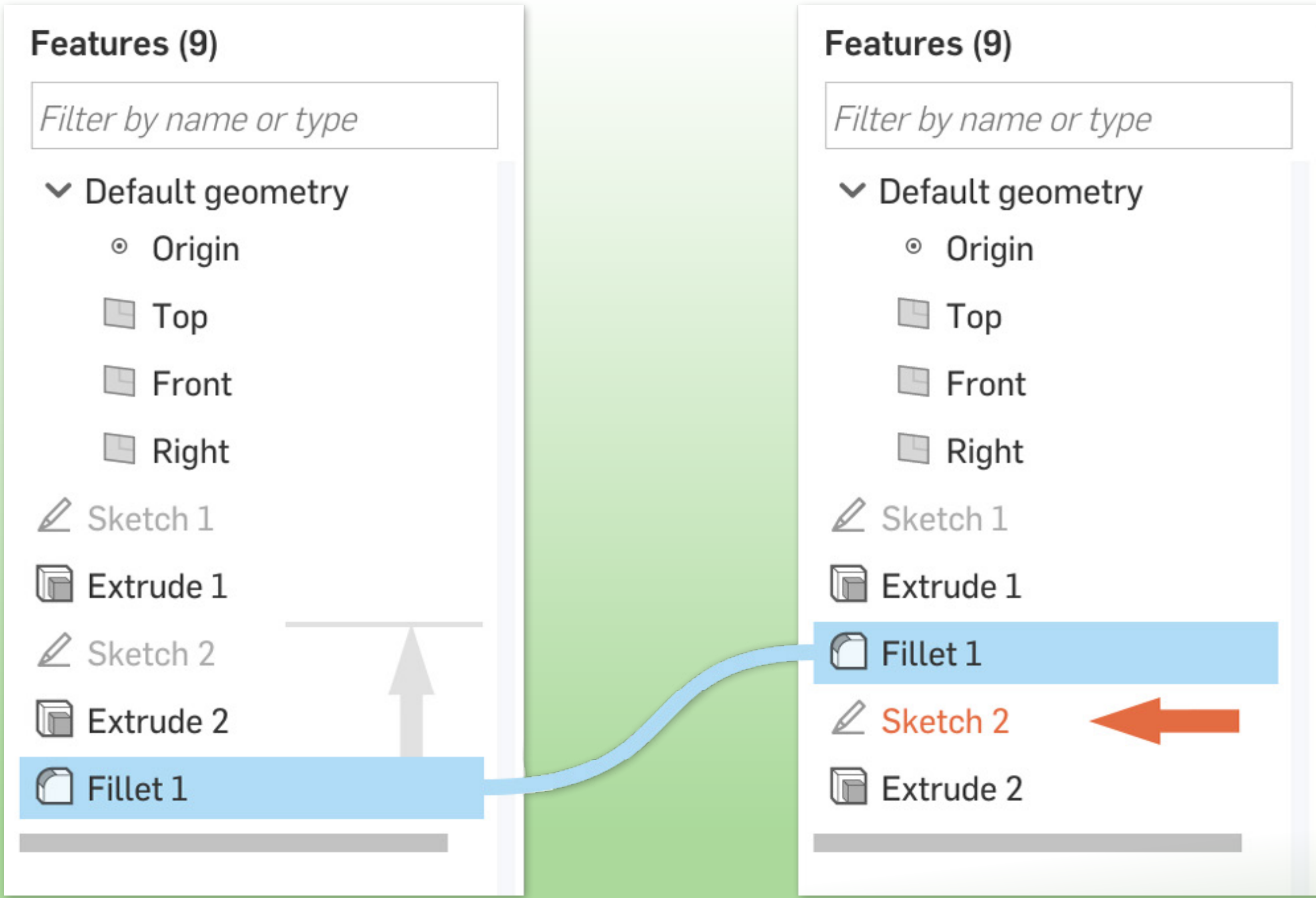


How Does a 3D Parametric Modeler Work?

As you are probably aware, the Feature List down the left of the screen is the order in which each feature is evaluated. Another way to think of it is the recipe that was used to build the model. As your design progresses, you can add new features that build upon the features already present to add more detail. This is what makes parametric modeling systems so powerful, letting you build in design intent so your models will update consistently and predictably.

But it's not the features themselves that are the problem, it's the references you make between them. References are your biggest ally, but also potentially your biggest enemy. A badly considered set of feature references can make your models very fragile.

Here is a simple example. You create a sketch on the face of a solid model and dimension to the edges of the model. Perfectly okay. Later you decide to fillet the edges that the sketch was dimensioned from. If you create the fillet after the sketch, then there's no problem. At the time the sketch is evaluated, the edge ID exists. Now if you reorder that fillet before the sketch, essentially changing the recipe for the model, the fillet removes the edge and the ID number no longer exists. The sketch fails.



Fillet 1 is reordered before Sketch 2 which now fails. Why?

How Does a 3D Parametric Modeler Work?

What is clever though, is that the ID is not lost forever. It is simply masked by the fillet. If you suppress or delete the fillet, the edge that was there reappears and assumes the same ID as it had before. Your sketch rebuilds once more.

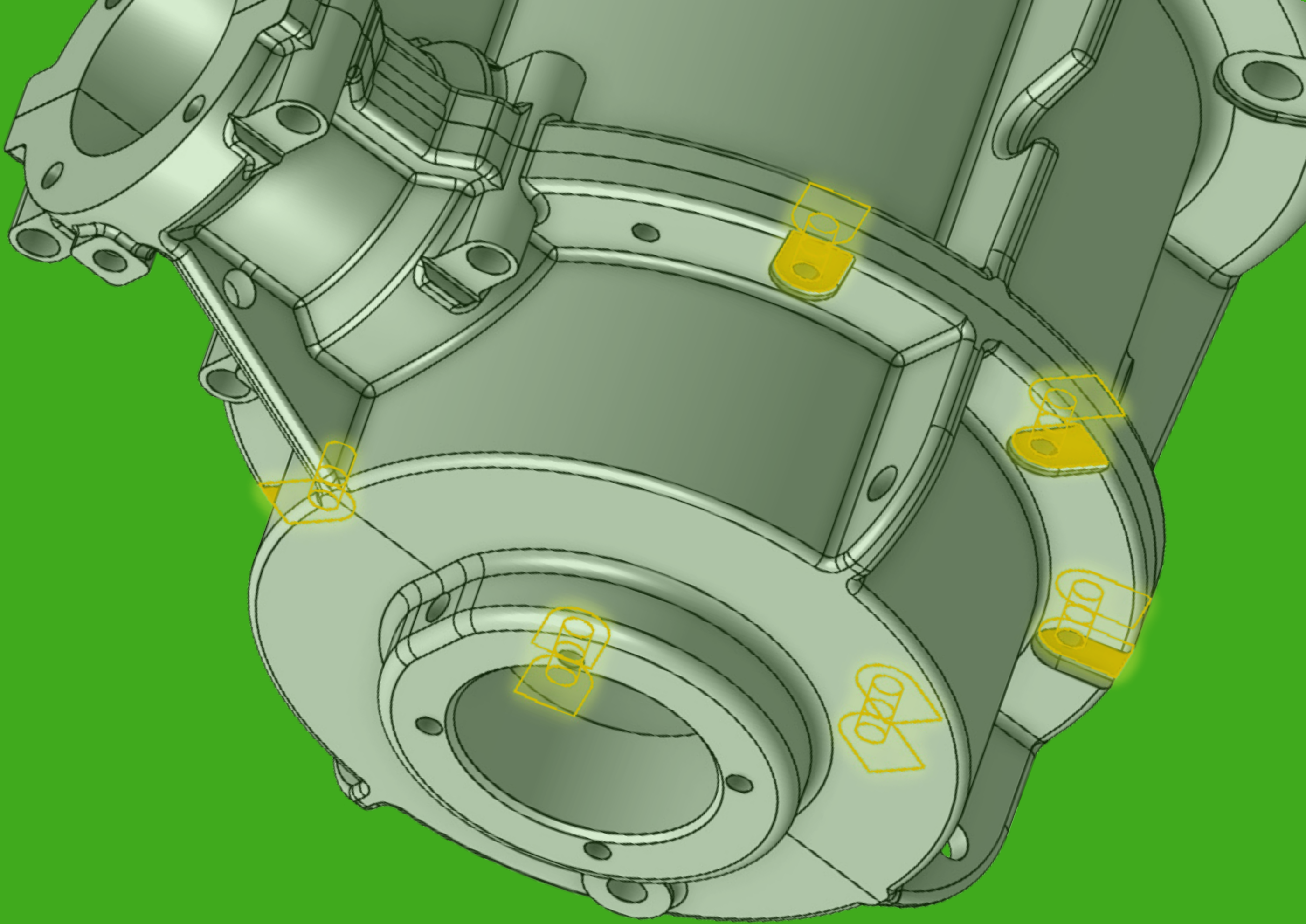
If you had dimensioned to Sketch 1 (that created the original extrude), the ID would never disappear and your model would be bulletproof. With this in mind, consider which references you are using when building features. If your model fails, then now you know why.

If you know things are not likely to change, for example, if you are building a 3D model from a supplier's 2D drawing, then you can be a little gung-ho with your references. But who can predict what will happen during a design project?

With that in mind, here are some iron-clad "Do's and Don'ts" for parametric modeling:

- **DO** use one layout sketch to drive many features across multiple parts
- **DO** dimension to existing sketch entities, planes and the origin
- **DO** Put key driving dimensions early in the tree
- **DO** add draft features early in the design and fillets towards the end
- **DO** dimension to faces perpendicular to the screen if no planes or sketches are present
- **DO** keep your sketches simple
- **DO** name your sketches and features to make them easier to find
- **DO** fully define your sketches once your conceptual design is complete
- **DO** break complex sketches into multiple simple ones

- **DON'T** dimension to an edge created by a fillet or a chamfer
- **DON'T** overcomplicate your sketches
- **DON'T** extrude a cut "blind" when it should extend "thru all"
- **DON'T** use Direct Editing features when an edit to an earlier feature will fix the problem
- **DON'T** model unnecessary features like threads (unless you are 3D printing)



4 Things to Consider Before You Start Modeling

If your design is a brand new concept, it's not always possible to plan ahead before you start modeling. However, there are certain things you will likely know in advance. For example: Is it symmetrical? Which way is up? Knowing just some details about your project in advance can save you a lot of time initially and later if any design changes are required.

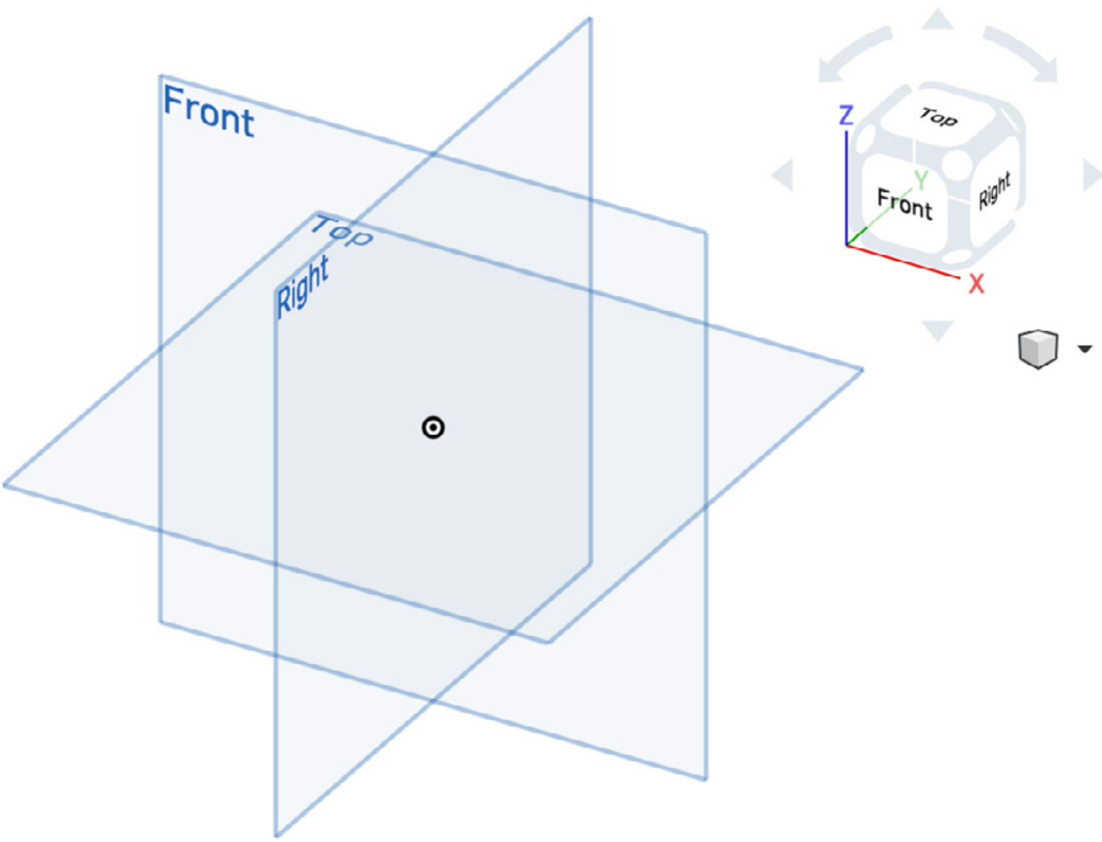
1. Orientation
2. Symmetry
3. Patterns
4. Complexity

1 Orientation

What is the optimum default orientation for your model? It doesn't really matter, but to minimize clicks later on and to communicate your product's purpose with clarity, it's worth considering.

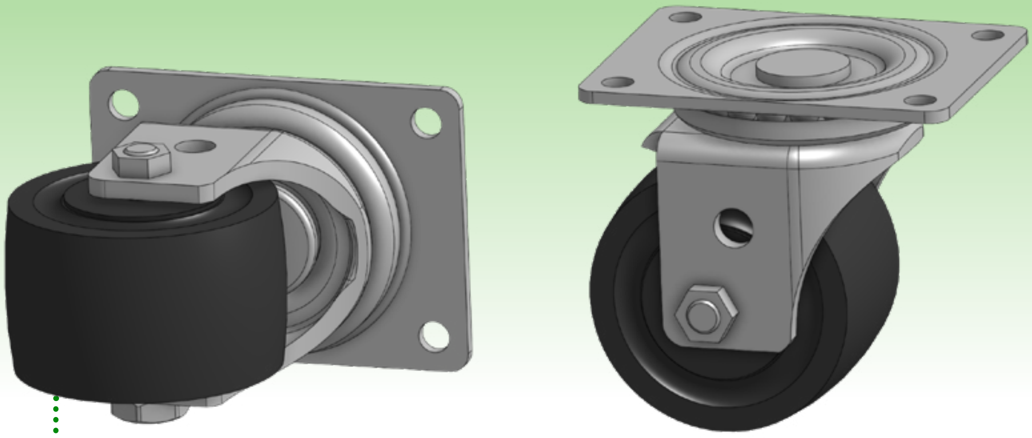
The default orientation is how the model will be shown when it is first opened – in a default isometric view. Looking at the image of the view cube and default planes below, you can see that in its default orientation, front faces front and to the left, right faces right, and top faces top. Clever, eh?

The default orientation of the view cube and default planes



Most of the time, working to these built-in view orientations makes complete sense. So when you want to view the top of the model, you press Top on the view cube. But there are exceptions, most of which arise from using imported geometry.

1. Orientation



The exact same assembly in different default orientations

Consider the caster models in the image above. Both models are identical except for their orientation. Regardless of whether it was imported, or modeled in Onshape, the model orientation on the right is more realistic and makes more sense. The top of the model is in the Top orientation. This makes it clear to the consumer of this model that this is the orientation in which the model will be used. When assembling, the caster will be inserted the right way up making the assembly process much easier.

Which way is up?

There are other considerations, too. Take for example, manufacturing (milling, molding, casting, 3D printing, etc.). In all of these processes, the Z axis points up.

The mounting plate from a monitor stand in the image could be considered as being "Front," but for manufacturing purposes the orientation shown is "Top."

p.s.

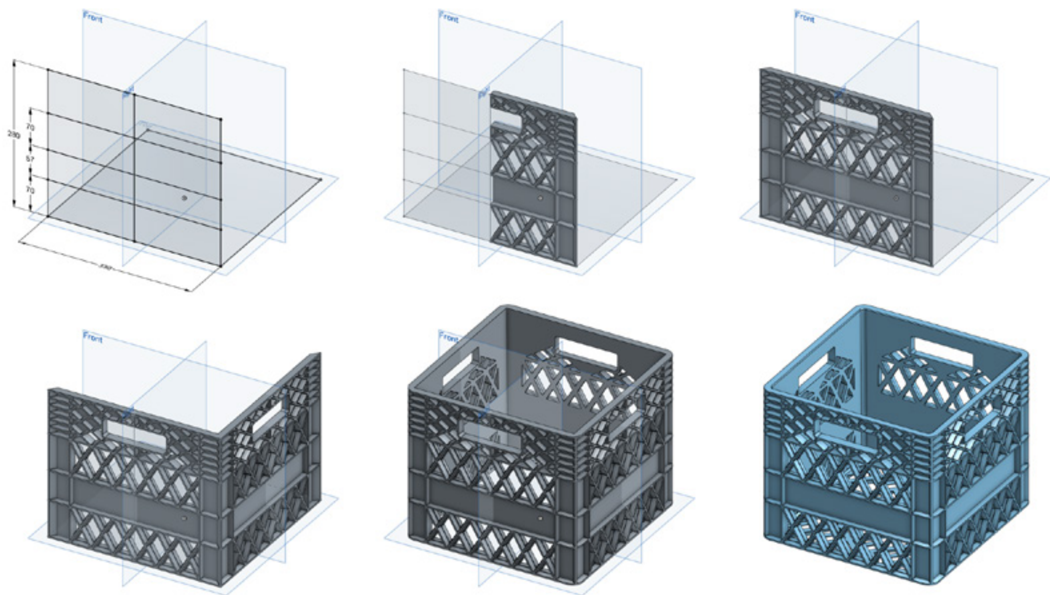
All is not lost, however, if you do import or model a part in the "wrong" orientation. When importing parts from other CAD systems into Onshape (most notably those that have the Y axis as up - such as SOLIDWORKS® and Autodesk® Inventor®) you can set the imported models in "Y Axis Up" coordinates. Once imported, you can also flip the orientation of the model by editing the Import feature and checking the option "Source is Y Axis Up."

All manufacturing software and 3D printing pre-processing software enable you to reorient your part so that it is in Z-up orientation, but getting it right in the CAD software makes the whole process run much more smoothly.

2 Symmetry

Most products have at least one plane of symmetry. While it may feel more natural to model the entire shape as you go, so you can easily review your progress, identifying and taking advantage of symmetry can save you an incredible amount of time when modeling.

Thinking about symmetry ahead of time gives you the opportunity to model only a half or a quarter of your design. Thinking about symmetry will also help you determine where your part should be positioned relative to the origin and the default planes. For most symmetric models, it is desirable that the origin is centered and the default planes are used for mirroring.



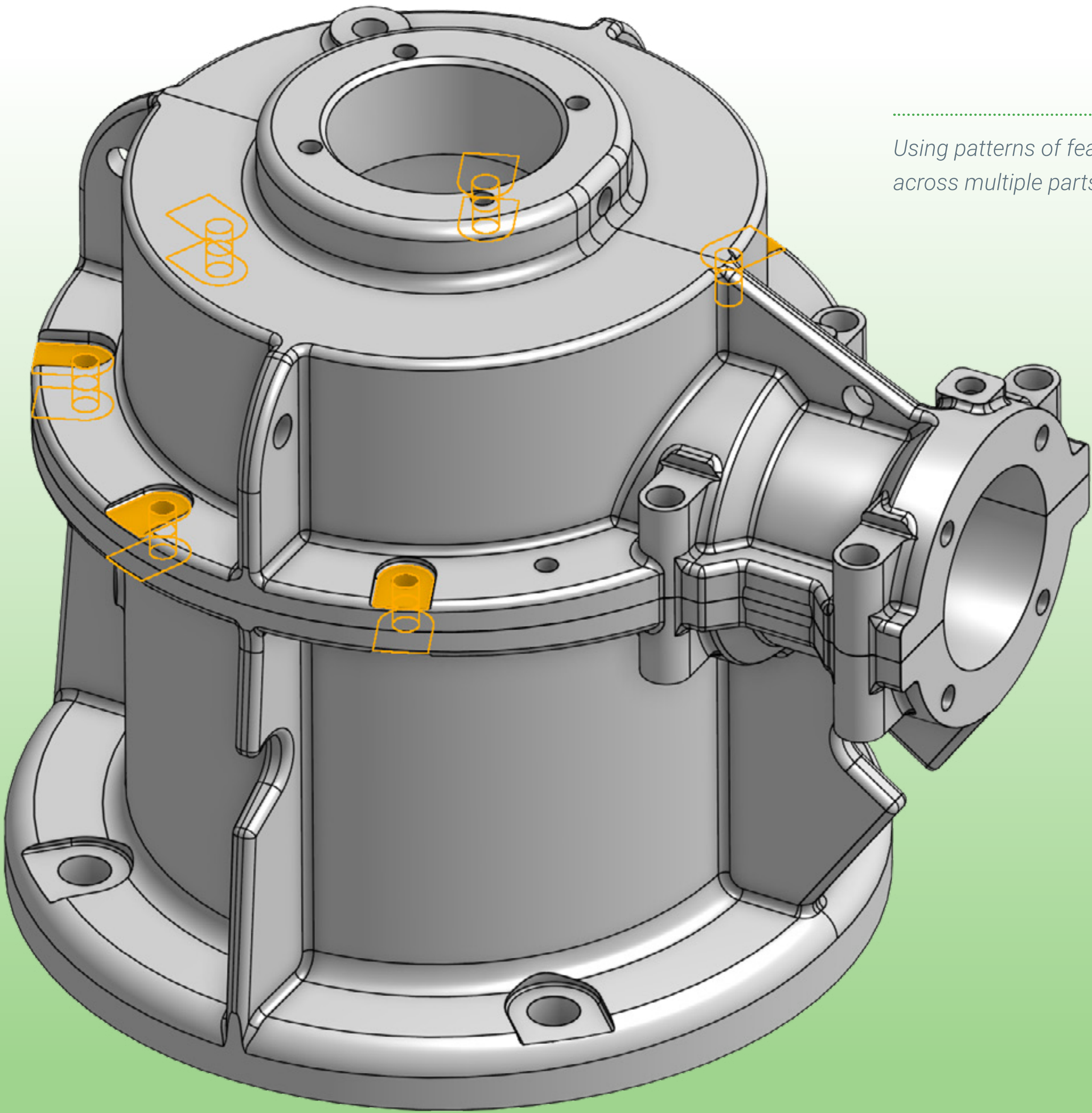
A complex design can be broken down into simple steps by using symmetry.

However, when you're creating your layout sketches you naturally want to sketch and dimension full size, to control and define the specifications of your product. You can still do this, but build your model using only half of your sketch to create one side or one quadrant. You can add multiple mirror features at the end of your feature list to see how your design is looking, then "rollback" your model before the mirror features to continue adding detail. As more detail is added, you can roll your feature list forward and backwards at any time to see how your design is progressing.

A good exercise is to consider the milk crate above. While most people will recognize that each side face is identical, it's easy to miss the fact that only half of a face needs to be modeled and can be mirrored using the angled face in the corner. If only some of the features are symmetrical, model all the symmetry first, then add any other details at the end, like the base of the crate in this example.

3 Patterns

A pattern is a repeating set of features like the ribs in the milk crate example. Identifying patterns of features early in the design and detailing of your product will save you a tremendous amount of time. One of the main benefits of using patterns, other than for speed and convenience, is when it comes to making edits to your design. A change to the original feature propagates through all the pattern features automatically, so large wholesale design changes can be completed in minutes.



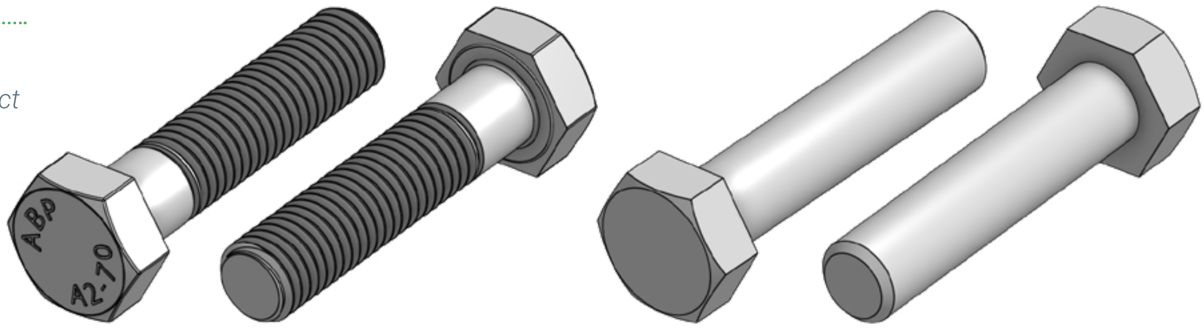
Using patterns of features across multiple parts

4 Complexity

One of the main benefits of 3D modeling is being able to add 100% of the detail required for manufacturing. For every part you design, you should always aim to add every fillet, chamfer, and draft feature to accurately portray the finished model. In doing so, you will have more accurate material properties such as mass, volume, center of gravity, etc., more accurate and complete drawings, and your manufacturing friends will thank you for saving them from having to add those details afterwards.

There is one caveat, however. Standard parts such as nuts, bolts, bearings, motors, etc., should not be modeled to the last finite detail. Why? Because you are adding a level of complexity to your assemblies and drawings that has no benefit whatsoever. This complexity can have a detrimental effect on your CAD system's performance as more graphics power is required to render them.

Adding too much detail can have a detrimental effect on performance.



Consider the two sets of bolts shown above. The bolts on the left look very lifelike. The bolts on the right have been simplified, but have more than enough geometry to accurately represent the parts. To render the helix created by the threads and the text on the head of the bolt, the graphics card needs to render over 150x more triangles in real-time compared to the bolts on the right. Now imagine if you have an assembly with hundreds of nuts, bolts, and holes with threads. Your graphics card is going to go into meltdown!

If you download models from your supplier's website and they contain too much detail, consider asking them for a simplified version (or simplify it yourself using Onshape's Direct Editing tools). The only exception to this rule is if you need to 3D print parts with threads.

Next time you start a new CAD project, take the time to consider these four simple techniques for planning your models in advance. Doing so will yield great rewards. Your models will be easier to understand, easier to assemble, and easier to edit in a manner that delivers more predictable results.

Benefits of 3D Modeling in the Cloud

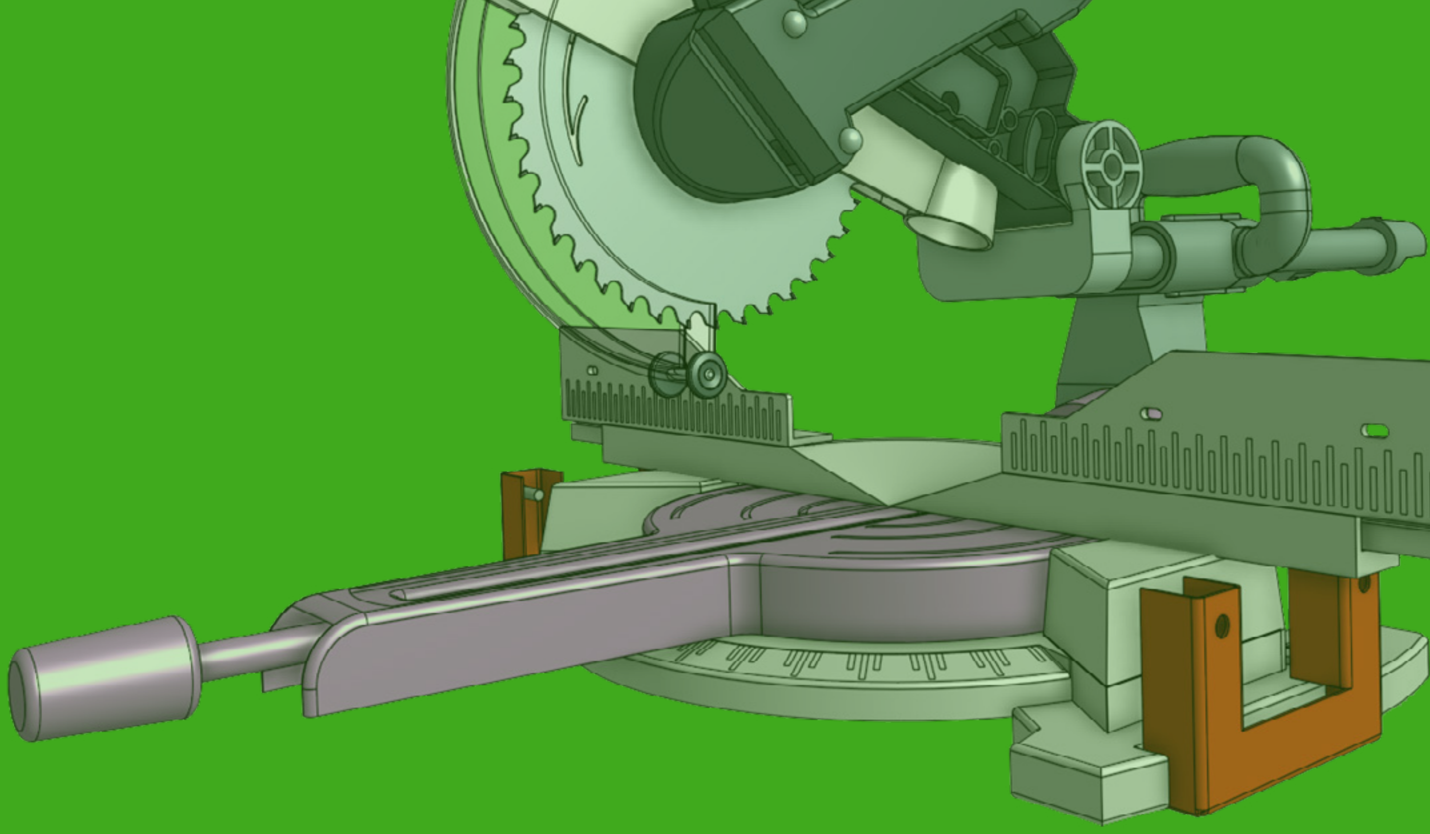
Ironically, engineers and product designers who are the earliest adopters of other new technologies, find themselves running in place with their most important occupational tool – CAD. Traditional file-based CAD systems, which still must be installed locally on each computer, haven't changed much since the 1990s. They cannot offer the massive parallel processing power, virtually unlimited storage and the real-time database architecture that the cloud makes possible.

As the first and only full-cloud professional 3D CAD system, Onshape lets multiple people simultaneously work on the same design from any computer, tablet or phone. Your CAD system and CAD data is stored in one central place in the cloud so there is no confusion over which version is the latest version of your work. Any change to your model made by anyone, anywhere is instantly seen by everyone, everywhere.

Pushing collaboration aside for a moment, full-cloud CAD also allows you to be more agile when you are working alone.

For example, Onshape is the only CAD system that enables you to design multiple parts in the same Document. Using Part Studios, you can create features that span multiple parts, like holes, creating a single pattern feature that adds detail to each part. This is not only a great time saver and a great way to convey design intent, but it also ensures that all related geometry updates predictably. This reduces the chances of making mistakes and causing expensive errors on the shop floor.

In the next section, we'll explore three of the most powerful benefits of using full-cloud CAD – all of which dramatically speed up the design process.



3 Things to Consider Before You Finish Your Design

As talented as we all are at designing perfect products, a fresh set of eyes always helps to make them even better. Whether it is your colleagues or your customers who provide input for your design workflow, sharing and collaboration are essential parts of the whole process.

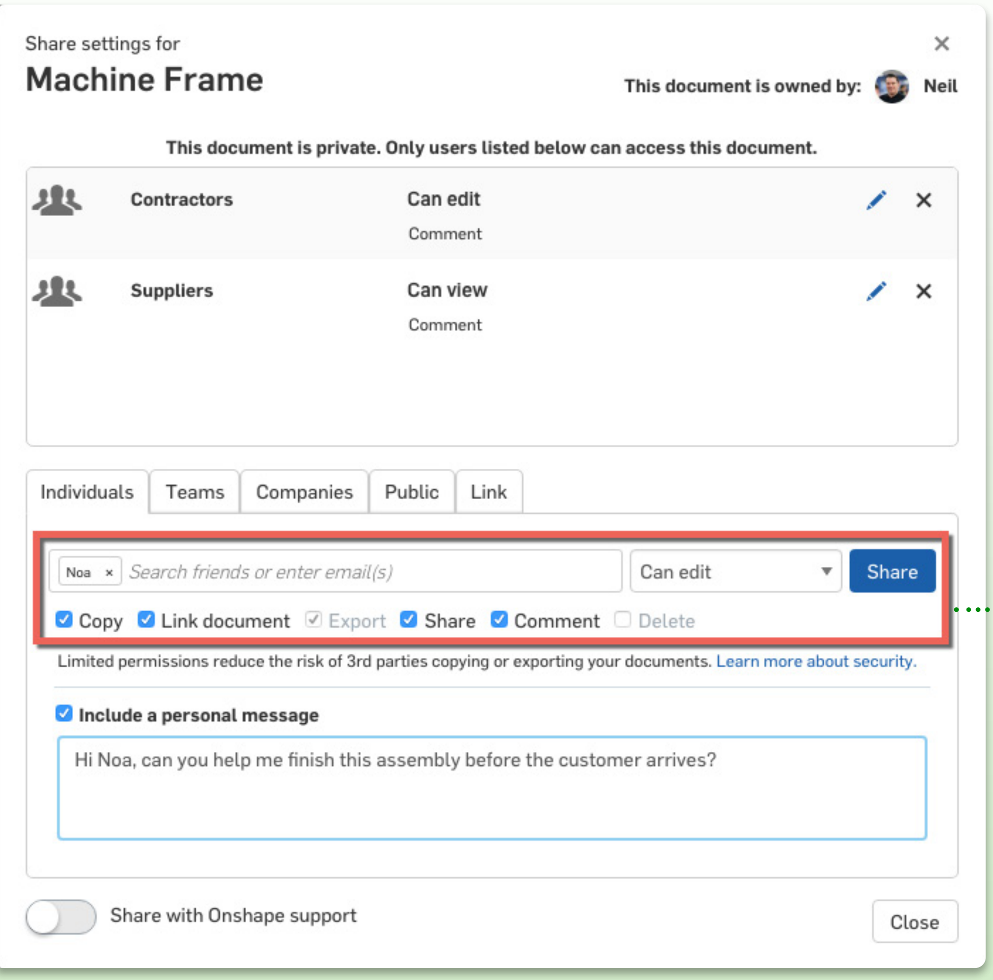
If everyone involved in the development of your products (co-workers, contractors, suppliers, manufacturing, management and customers) are all on the same CAD system, on the same version, and connected to the same PDM system, then that would be utopia. It would also be a miracle. This is where Onshape shines.

1. Sharing
2. Branching
3. Versioning

1 Sharing

We're living in a multi-CAD world. It is very rare these days that a product is designed in total isolation in one CAD system. Manufacturing is often outsourced and many designs incorporate bought-in or off-the-shelf items. It's highly unlikely that your entire supply chain uses the same software as you and the same version, so exchanging data in native or neutral file formats or sending proprietary view-only files is unavoidable. With teams spread across multiple geographies and time zones, and customers and suppliers getting involved from the start, managing the distribution of files and making sure everyone is working on the latest version is the stuff of nightmares.

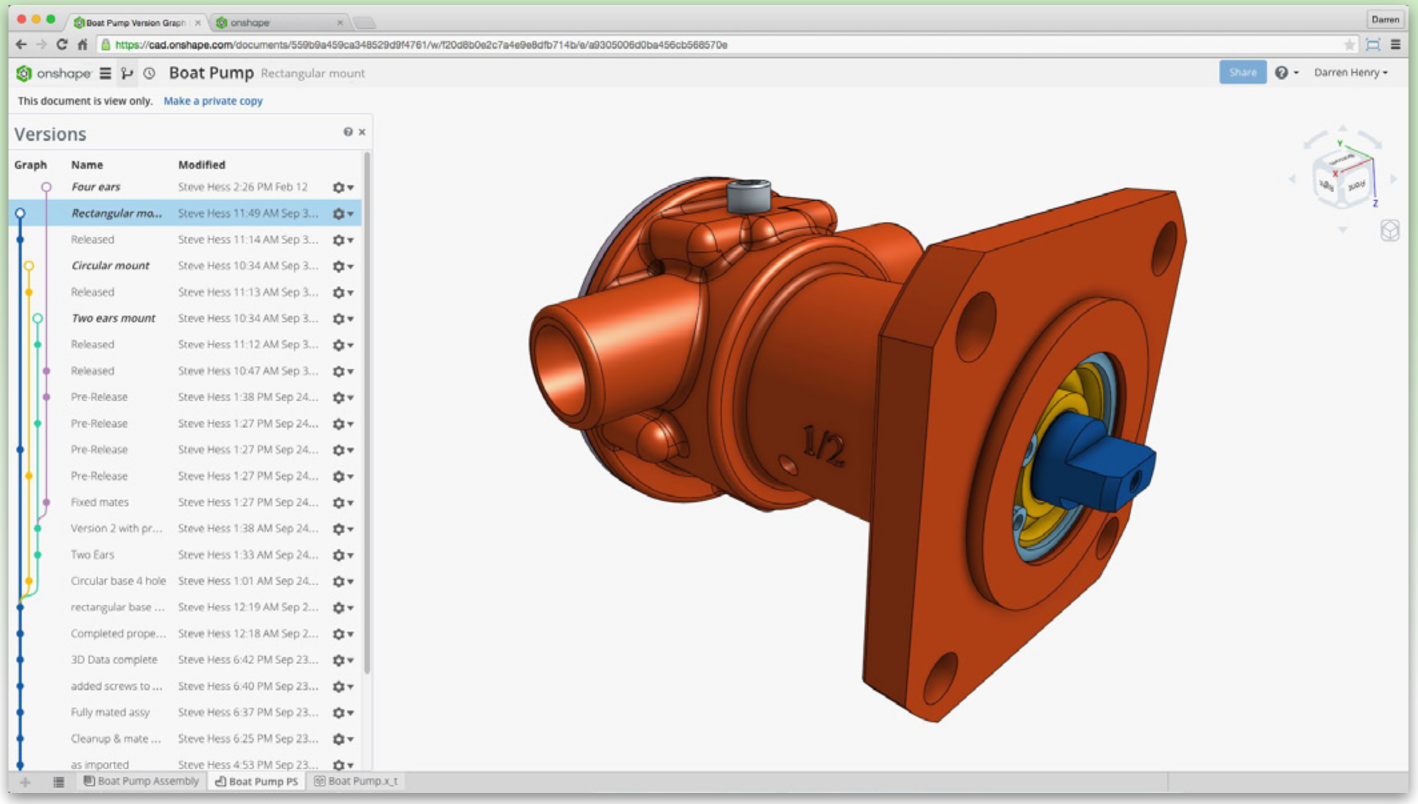
Onshape was built from the ground up to simplify this critical process. Your CAD system and all of your CAD data are stored in one place in the cloud. There are no files, no copies, and no need to send anything by email, FTP or Dropbox. Your data never leaves our servers, so there is nothing that can be distributed or copied without your consent.



This is one of the fundamental differences between Onshape and any traditional CAD.

To share a Document with an individual or a team, just hit the Share button. Since the data is controlled in a database, you can add granular permissions for each individual or team to grant them either view-only or full editing access – as well as tightly control whether they can copy, export, share, or just comment.

Every user with edit permissions can edit the same part or assembly at the same time, and every change updates in real-time for everyone to see. **This is the ultimate in collaboration.**



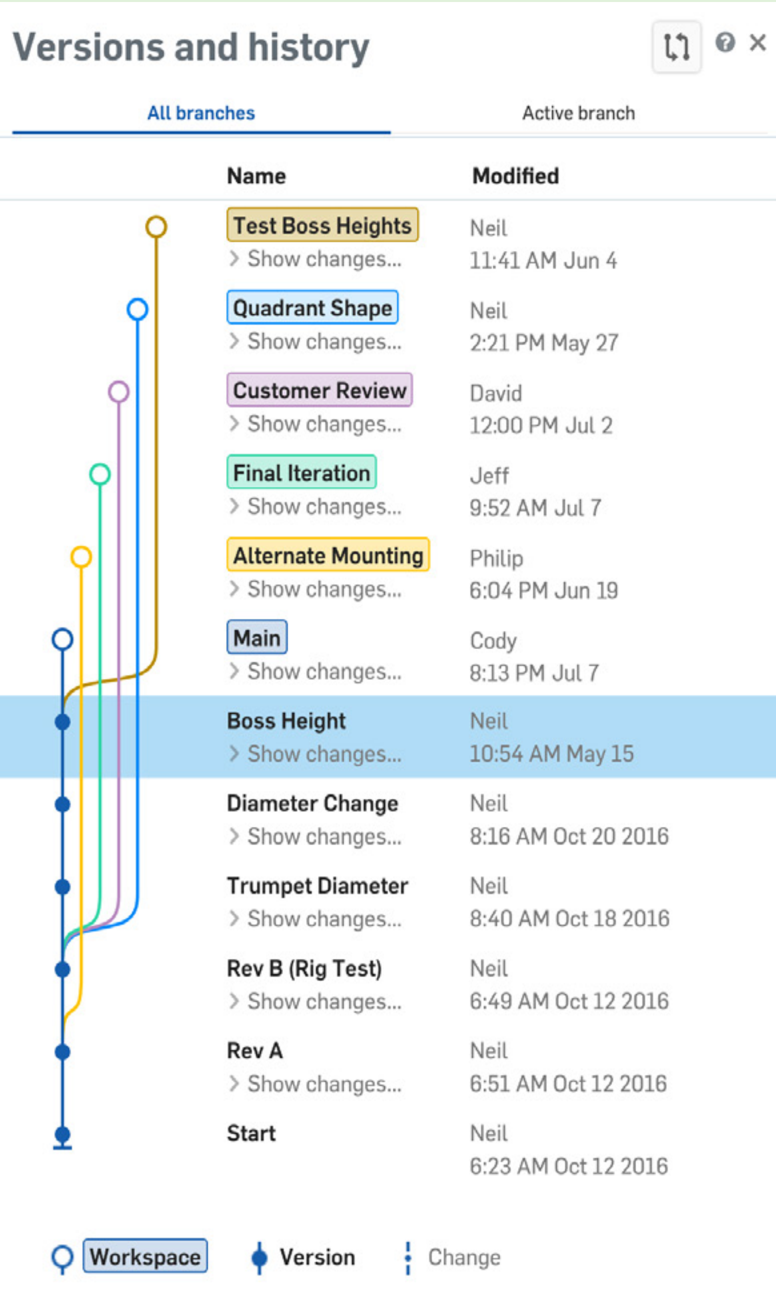
2 Branching

Design is all about exploring different ideas, then turning the best ideas into best-selling products. Whether that exploration is done during conceptual design or detailed design, making multiple iterations and versions of your designs should be easy. With traditional CAD, that means copying files. Many times. Then attempting to manage all those copied files and track the best ideas. Copies can lead to errors and errors can lead to scrap and rework – an unnecessary waste of time and money.

Because of Onshape's database architecture, there are no files. To quickly try out a new idea, you can create a branch within the same Onshape Document. When the branch is created, a snapshot of the original design is added to the Document for you to work on independently. Each branch behaves like a sandbox, letting you try as many different design ideas as you like and any modifications you make have no effect on the original design. This mechanism makes it easy for multiple people to work in the same Document and work on different ideas at the same time.

Once you've arrived at the ideal solution, your branch can be merged back into the main design and all the other unused ideas can be kept for future reference or discarded.

3 Versioning

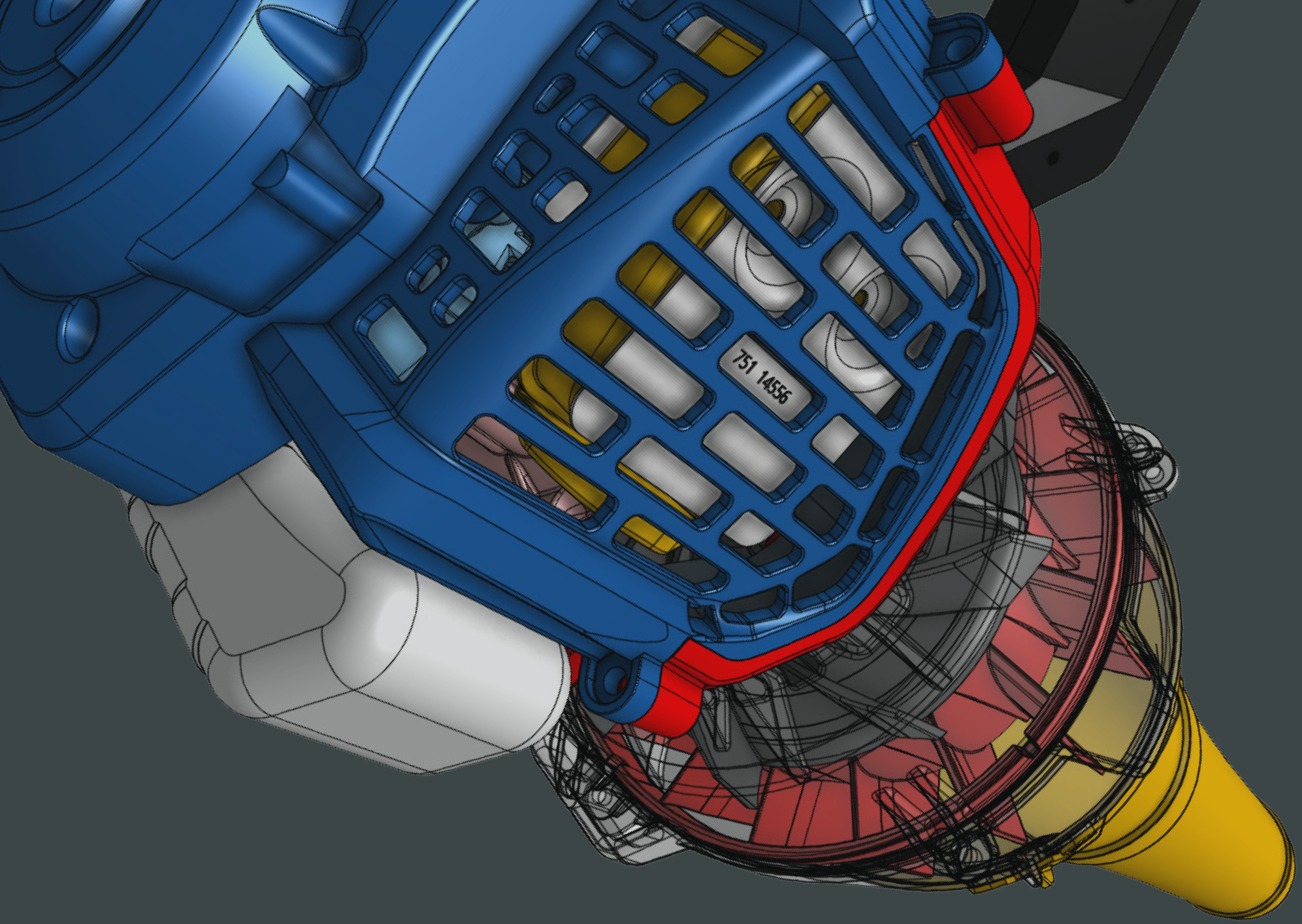


When your design is finally final, you need everyone to know that this is the version to be manufactured.

With a traditional CAD and PDM system, creating a version is fairly easy, but you still have the issue with communicating this to your extended enterprise.

If they don't have the same CAD system and PDM system as you, you have to make and send copies. There is a theme going on here. If you send copies, the files are no longer read-only and may be accidentally modified. Or if you send a proprietary view-only file, it may not contain enough information to manufacture from.

In Onshape, you can make as many versions as you like at any time. A version cannot be changed, but it can be shared. Each user it is shared with can interrogate all the parts, assemblies, and drawings to get the information they need, using just a web browser or mobile device.



onshape™

IS YOUR DESIGN PROCESS FAST ENOUGH NOW?

No matter which parametric 3D CAD system you've been using, the tips in this eBook will help you become a modeling grandmaster. But why stop there?

Using the best practices and taking advantage of the latest technologies will make you even faster – giving you the luxury of time to take more creative risks, explore alternative designs and be more innovative.

Making Onshape your primary CAD system or using it side-by-side with your current system will help dramatically speed up your time-to-market. Start your next project with an Onshape Professional Trial and experience the difference yourself:

Instant CAD Access From Anywhere – Onshape runs on any computer, tablet or phone (iOS or Android). You are no longer tied to the one computer connected to your CAD license.

Part Studios – Design multiple parts together with robust references in the same Document. Because Onshape allows multiple parts to share sketches and features, new levels of efficiency can be realized by building and editing parts together.

Assemblies – Use a modern approach to building assemblies. Higher-level mates enable users to build assemblies with one-third the mating relationships of traditional desktop-installed CAD. Onshape assemblies are easier to build, manage and edit.

Instant Sharing – Collaborate with internal and external partners in real time without copying or sending files, unintentionally risking your IP. Have the ability to instantly revoke individual project access when desired.

Branching Designs – Test multiple design scenarios through separate branches and later choose to merge the best features.

Managed in-Context Editing – This powerful feature edits parts within the context of assemblies to ensure fit and function. In-Context Editing helps designers avoid broken 3D models commonly experienced by traditional CAD users whenever a part is changed or deleted.

Simultaneous Sheet Metal – Featuring simultaneous and synchronized flat, folded and tabular views, these unique design tools enable CAD users to visualize errors and interferences immediately, consider alternatives and ultimately, reduce scrap and wasted time.

Integrated Cloud Engineering Apps – The Onshape App Store includes a full menu of essential engineering tools – CAM, Rendering, Simulation, etc. – each with the same cloud-enhanced productivity as Onshape.

Request an Onshape Professional Trial today

to put any of the workflows highlighted in this book to the ultimate test!

REQUEST A TRIAL