



MFG125102

Fusion Assemblies Explained for the Traditional CAD Designer

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Learning Objectives

- Learn how to work with assemblies in Fusion 360's single-design file context
- Learn how to use distributed design in Fusion 360 when it makes sense
- Learn how to restructure Fusion 360 designs and troubleshoot when restructuring isn't possible
- Determine which assembly scenarios Fusion 360 isn't the best fit for

Description

Maybe you've used CAD for years and modeled parts and assemblies the traditional way. Or maybe you're familiar with the traditional bottom-up CAD design process, and you've noticed Fusion 360 software looks a little different. In this class, we'll compare the assembly design process in Inventor software with the processes you can use in Fusion 360. After all, you want to know all the facts before evaluating what's best for you.

Speaker

I'm a Senior Implementation Consultant for D3 Technologies, an Autodesk Partner and Authorized Training Center. I specialize in Autodesk software training and consulting for a variety of customers across the US. I have been using Fusion 360 for several years, and have helped develop a full live training program for D3 on Fusion 360, including almost 20 different classes.

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Fusion 360 Assemblies Explained for the Traditional CAD Designer

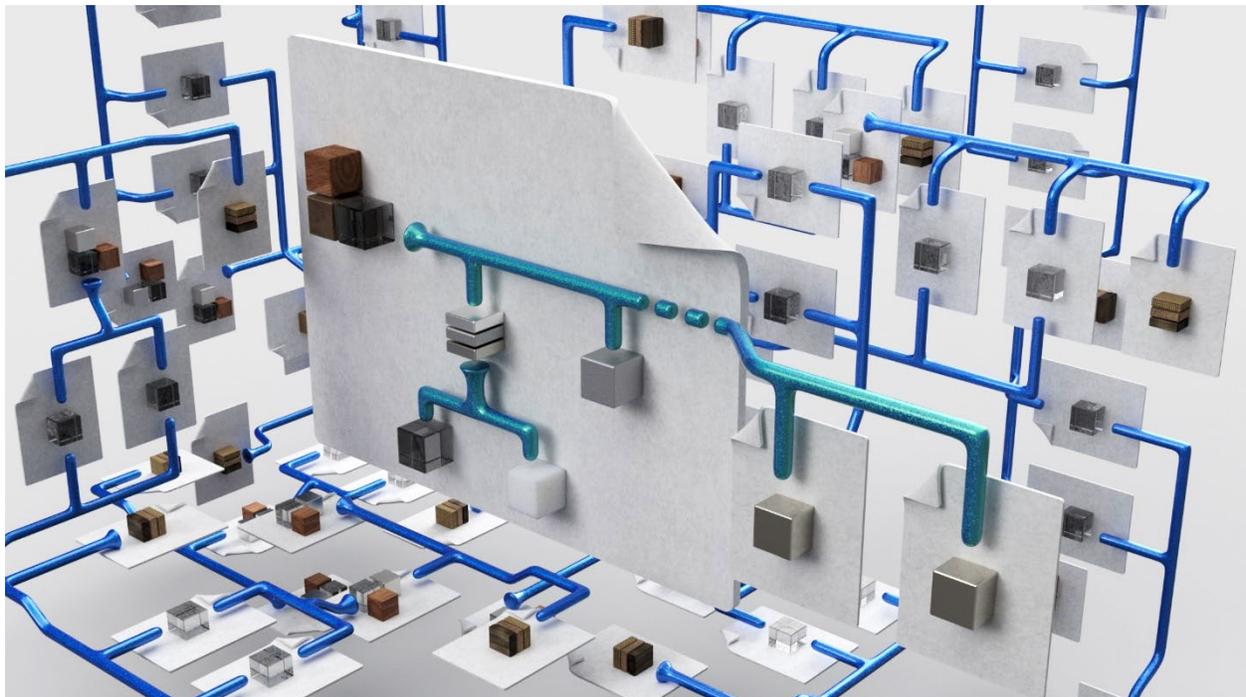
Introduction

Fusion 360 will present you with new ways of looking at your design process. The way assemblies are handled in a single design context **can be a source of innovation or of frustration**, depending on how well you understand it.

This course was designed to allow you to learn about Fusion 360's design environment and to be able to compare it to the more traditional multiple-file design process of most other CAD design software. Hopefully by the end of this course, you'll be better able to evaluate if Fusion 360 could work in your environment. Have a question about a tool or feature discussed here? Feel free to [contact me](#) and ask about it.

Because Fusion 360 is evolving so rapidly, I have to put in a disclaimer here. All the information covered in this handout is time-sensitive. It covers the state of Fusion 360 while I was preparing this course. Since a new version of Fusion 360 is released about every 6-8 weeks, it could change at any time.

Models used in this presentation will be available for you to access by sending me a request to join the Fusion Team project for this class. All images and renderings come from designs modeled or rendered in Fusion 360. [See more here.](#)



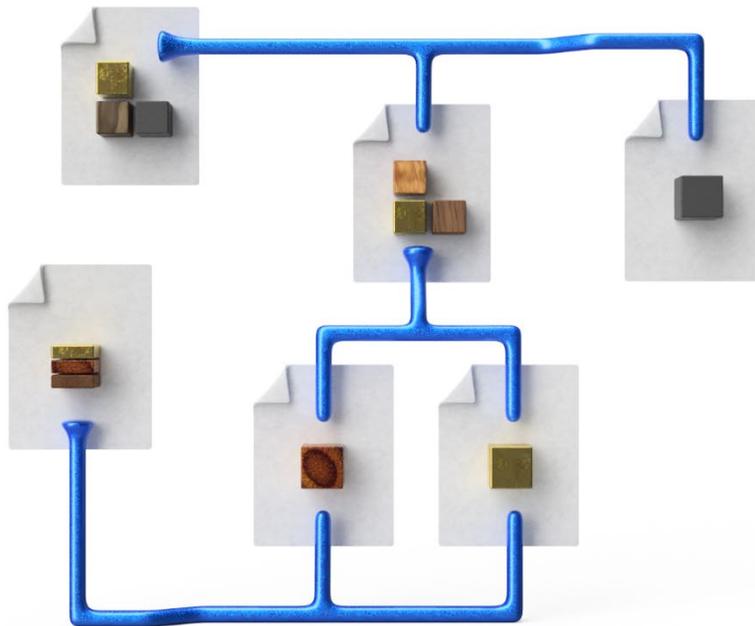
Working with Fusion 360's single design file context

Single Design Context

Our first topic will review [Fusion 360's single design context](#), and compare it to the traditional CAD structure. Jumping straight into the nuances of Fusion 360 workflows could be confusing if you're not already familiar with the reasons behind why we do things the way we do in Fusion 360.

Traditional CAD Structure

We'll start by looking at our first diagram, the design structure from a product like Inventor.



TRADITIONAL CAD FILE STRUCTURE

In this design, each sheet of paper is a separate file. You'll notice the Assembly is a file, referenced to other assemblies and parts, which are also their own files. Every part of the design requires its own file.

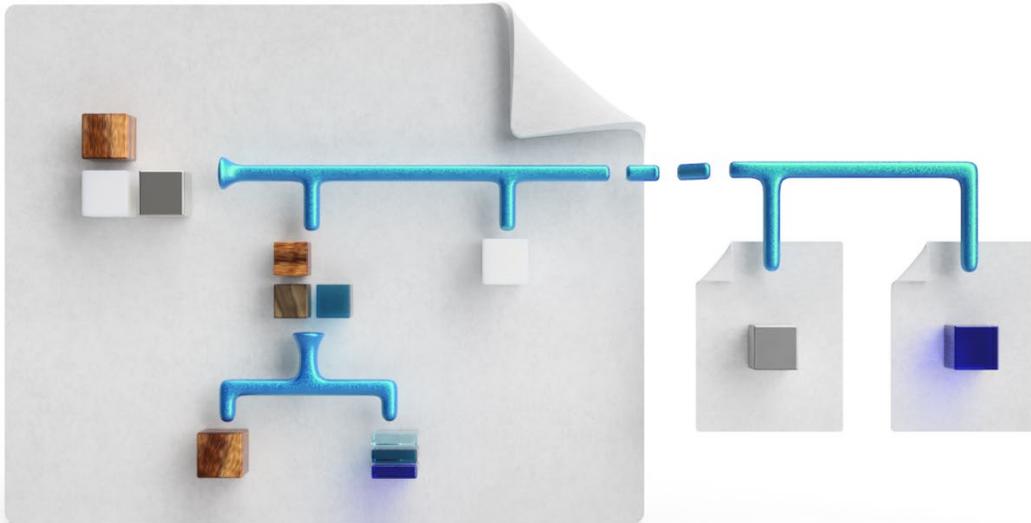
You'll also notice a multi-body part in the lower left corner. This is its own design as well. If you want to convert each body to its own part file, you guessed it, you have to generate additional part files for that too.

This design is effective, but it requires the managing of a variety of separate files, even in if they are all part of an overall design.



Fusion 360 Structure

Next, we'll look at a diagram of the design structure in Fusion 360.



FUSION 360 FILE STRUCTURE

Just like before, each sheet of paper here is a separate design file. Notice the difference? In Fusion 360, all parts of a design are contained in one design file. Regardless of if it's a separate solid body, a separate component, even a separate subassembly, it is all contained in a single design context.

This is really what separates Fusion 360 from the traditional CAD design process. There is no need to manage many files for a single design. There is also no danger of losing a file from the design and getting unresolved reference errors.

You'll notice I do show a few parts referenced in this design that are separate files. Fusion 360 does allow for distributed design, where designs are linked into others as needed. But instead of being forced to use this for every part of a design, Fusion 360 is flexible enough to allow you to decide when and if you'd like to do it.

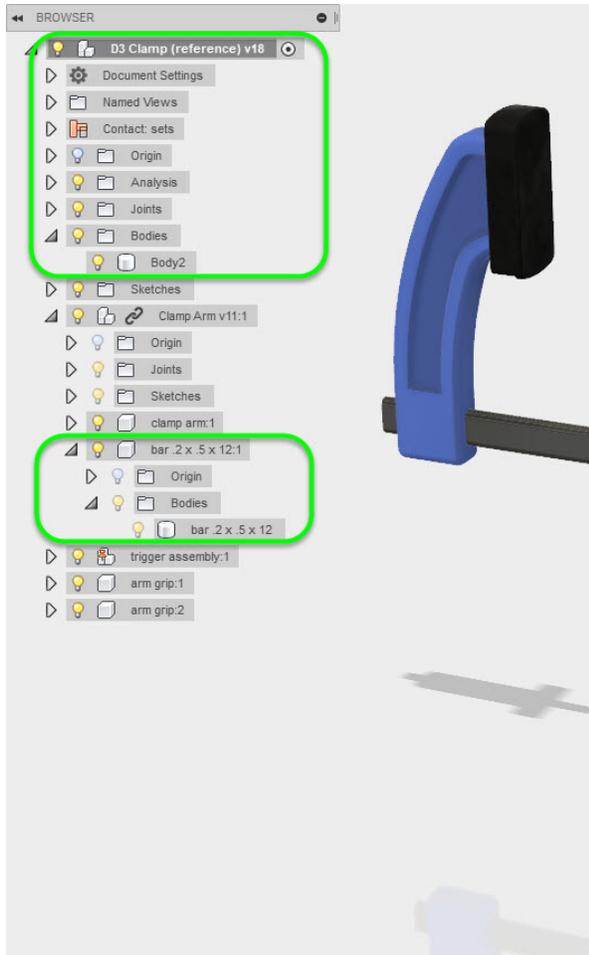
Active Component Filtering

Since Fusion 360 hold so much data in the same design environment, it could easily become unmanageable. This is another reason understanding Fusion 360's design environment is important.

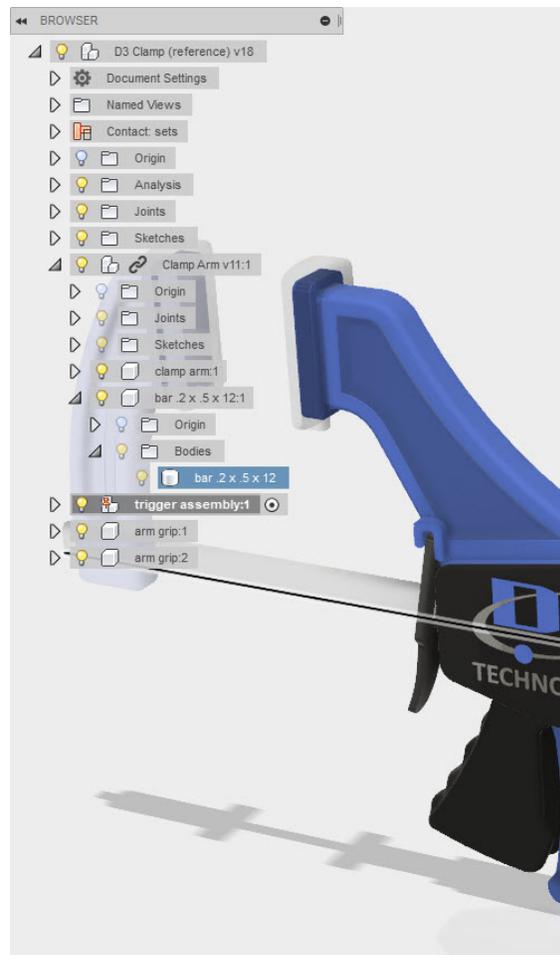
There are two main places to hold 3D solids in Fusion 360. They can be contained in a **body** or in a **component**. Using a component will allow you to filter the design history of that component from others in the design. Thus, components are very important to be aware of when working in Fusion 360.

Bodies are contained in a subfolder for each component in the browser. Components are shown in the browser after all other items. You'll also notice *single component* and *assembly component* icons. The assembly icon indicates a component has other subcomponents contained inside it.

To activate a component, simply click on the radio button next to it in the browser. You'll notice bodies do not give you the ability to filter in this way.



BODIES AND COMPONENTS IN THE BROWSER

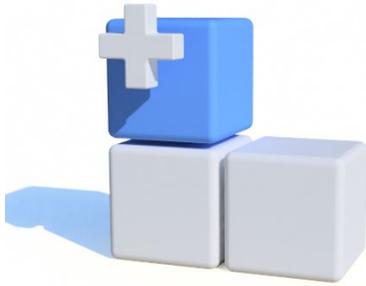


TRIGGER ASSEMBLY ACTIVE

Rule #1

There is a design philosophy that has developed among Fusion 360 users in the community. This is commonly referred to as Rule #1. Simply stated, it says that you should create components early in the design process. Creating components early not only helps organize your design, but it also allows easier re-use of parts of your design in other places when necessary.

Although you can create components whenever you'd like during the design process, following Rule #1 will keep your design organized and make it easier to work with.



To start a new component *first*, before the body, simply use the *New Component* tool in the Assemble panel. This starts a new component inside of the component you choose. If you don't choose a target component, the active component is chosen by default.

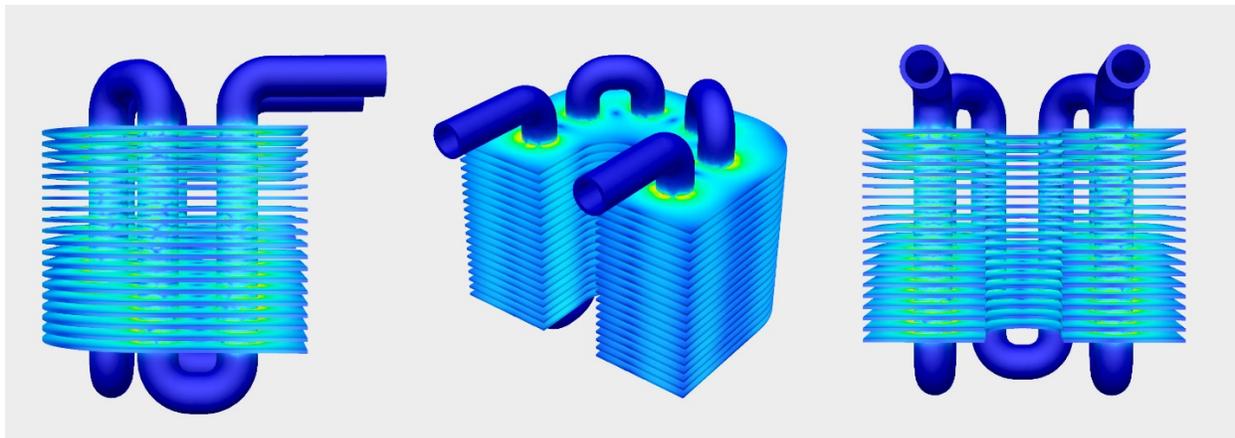
You'll notice once you create a new component inside another, the original component icon changes to an assembly icon, indicating it contains one or more subcomponents.

Multibody vs. Assembly

Although we just considered Rule #1, designs do not always require multiple components. How will you know if you should create new solids as new bodies or as new components? There are several things to consider.

- Components are able to move independently of each other. If your design will contain moving parts and joints, you'll want to be sure you are generating components for those moving parts.
- The BOM will include components as their own line items, but not bodies. All bodies in a component will be considered as part of the same item.
- You can easily move bodies to components and restructure at any time, but Rule #1 suggests creating components as early as possible in the design.

So, although it may be easier to design with multiple bodies, if the end result will be an assembly with motion, components are required.



THIS DESIGN (BUILT MAINLY FOR THERMAL SIMULATION) DID NOT USE MULTIPLE COMPONENTS

Distributed Designs in Fusion 360

Distributed Design

Sometimes you do not want a component to be created in the local design context. For example, maybe you're referencing in a common component you use in many designs. It doesn't make sense to re-model that component in every design where you need it. In this case distributed design will help.



If you have a design you'd like to insert into another, you can open the target design and simply drag and drop the component into it from the Data Panel. Also, you can right-click on a design and choose "Insert into Current Design" to insert it into the active design as a distributed component.

You'll know when a component is referenced into your design because it shows in the browser with a chain link over the component icon. This indicates the design is linked from another file.

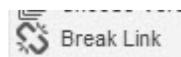
Let's say someone else is working on the source design you've distributed into your model. If a linked design has changed, Fusion 360 will notify you the file is out of date. Simply click the update button in the quick access toolbar to refresh your design with the latest version. You can also choose an old version of the referenced design to use. Just right-click on it in the browser and click "choose version."

Copying Designs

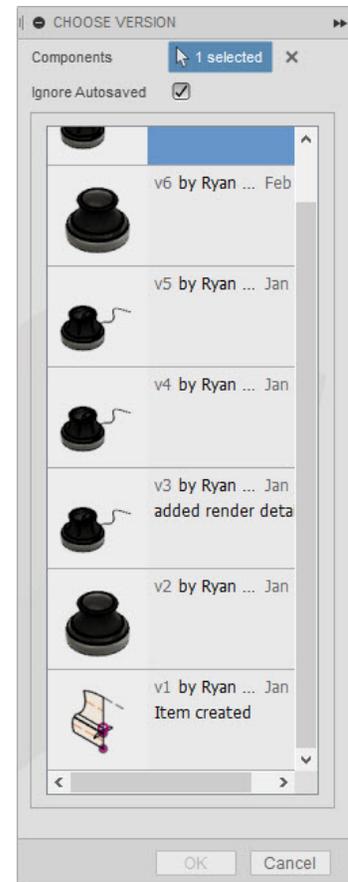
If you've used distributed designs before, you might notice you are not able to modify a distributed design when it's being used as a reference. You'll need to open the original design file to modify it.

If you do so, the design is modified and those changes are reflected in *all locations* where it was referenced. Sometimes you may want to reference a copy of a design instead, so you can make local changes without affecting the original or any other instance where it's been referenced.

There are a few ways to copy designs into other, but the easiest is to start with a distributed design as discussed in the last section, then simply break the link. This removes any association to the original and lets you edit the copy in the new design independently. Just right-click a distributed design in the browser and choose "Break Link."



BREAK LINK IN A DISTRIBUTED DESIGN



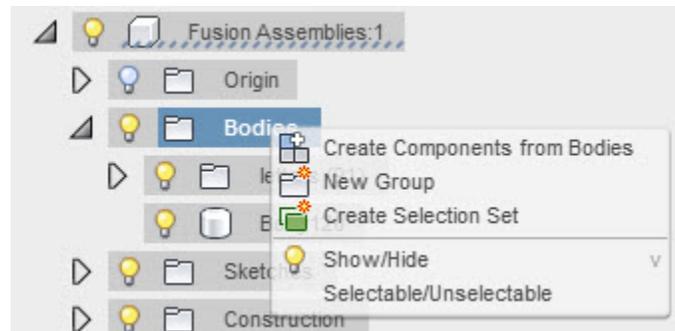
CHOOSE VERSIONS



Restructuring Fusion 360 Designs

Creating Components from Bodies

What if you've already created a multibody design in Fusion 360, and now you want add joints and motion? There's an easy way to convert bodies into components. Simply right-click on a body in the browser and choose "Create Components from Bodies." You can also right-click on the Bodies Folder to convert all bodies it contains into new components.



CREATE COMPONENTS FROM BODIES

Moving Bodies and Components

You can also move bodies and components into other components. For example, maybe you want to restructure the design so two sibling components are instead part of their own subassembly.

To move bodies simply drag them from the bodies folder where they are, and drop them onto another component in the browser. They will be moved into the bodies folder of that component. To move components, drag the component line in the browser and drop it on top of any other component or assembly.

Troubleshooting problems with restructuring

Sometimes you may try restructuring a design and it simply doesn't work. Here are a few things to consider.

Drag and Drop onto the Component Level

When dragging bodies into another component, it's easiest to drop them onto the component level in the browser. There are other options for where to drop it, but if you stick with the component level, Fusion 360 knows what you're trying to do and puts it in the correct place. If you drop it in the wrong place, for example, the sketches folder, Fusion 360 ignores your attempt since bodies cannot live in the Sketches folder.

Simply put, restructuring will remain largely free of frustration if you just drag and drop onto the component level.



Bodies are not the only thing you can restructure. You can also drag and drop parts of the design such as sketches and construction geometry. These work just like bodies, drag and drop onto the component you want them moved to.

Pay attention to the Timeline when moving stuff

Besides moving things in the browser, you can also drag modeling features around in the timeline to reorder them. When restructuring, it is important to know not only where you are moving things, but how it will affect the timeline. You could put it this way: in Fusion 360, it doesn't matter *where* stuff happens as much as *when* it happens.

Although Fusion 360 is very flexible with the restructuring it allows, there are times when it simply won't work. If you've run into a case where restructuring isn't allowed, it is probably because the move you are attempting would result in a feature moving in the timeline into a state where it exists before some other feature that it references. For this reason, it is important when restructuring sketches or other features to note not only *where you are putting it in the browser*, but also *how it will affect the timeline for the component your moving it into*.



Where Fusion 360 might not cut it

Fusion does very good on the design side for things like consumer products and other small to medium assemblies. However, sometimes Fusion 360 might not be the best tool for the job. If we want an honest evaluation of its capabilities, we should not be afraid to discuss both the pros and the cons. Next, we'll discuss disadvantages Fusion 360 has in the following two areas: large assembly creation and design accelerators.

To be clear, even if you don't design your product in Fusion 360 doesn't mean there aren't other great workflows you can accomplish using Fusion 360 together with other design software. For example, Fusion 360 has very advanced simulation and CAM environments. Or, if you're looking for interactive collaboration with others during the design or sales process, Fusion 360 is your tool for that. However, in this course our main focus is on the design side.

Really, Really Large Assemblies

If you get to a point where you might have thousands or even tens or hundreds of thousands of parts in your designs, Fusion 360 may not be the best tool for those designs. Using Rule #1 keeps your timeline organized for most assemblies. But if your design grows too large, it can be hard to manage the timeline and the browser.

If you feel your Fusion 360 designs are getting out of hand, it may not be time to jump ship just yet. Consider the following organization tools:

- *Groups in the bodies folder.* These allow you to group many bodies into a folder so they can be selected for modification with one click.
- *Isolate in the right-click menu for any component or body.* This will hide everything else in the design except for the selection. I commonly use this in conjunction with the activation of a component in larger designs.
- *Grouping in the timeline.* You can group many features together in the timeline from the right-click menu, so they can be expanded or collapsed (shown below).



GROUPING FEATURES ON THE TIMELINE

Design Accelerators

If you are using design accelerators such as Frame Generator or Bolted Connections in Inventor, Fusion 360 will probably not be as efficient at creating those designs. Also, if you're using a design environment like Tube & Pipe or Cable & Harness, Inventor (or your other CAD software) will probably still be able to create those designs more efficiently and with more intelligence than Fusion 360.

While Fusion 360 is certainly capable of creating these models, often when using the design environments in a program such as Inventor, a lot of the work is automatically done. For example, you can simply create a path in a Tube & Pipe design, and Inventor will populate it with the appropriate pieces of piping, elbows, fittings, and whatever else you specify. You don't have to model *or* constrain those parts. Furthermore, the design properties such as pipe length are already calculated. This type of design is simply easier to create in Inventor.

This doesn't mean you can't use Fusion 360 at all in your workflow. Maybe you upload your Inventor designs to Fusion 360 afterwards for collaboration, simulation, or rendering. There is still a lot Fusion 360 can offer to augment these more advanced designs downstream.

Conclusion

Now that you have an overview of the way assemblies work in Fusion 360, what do you think? Hopefully you have a better idea of the design structure in Fusion 360. Maybe you even picked up a few new tips or tricks.

Feel free to connect with me or reach out if you have any questions or just need more information. The following page contains a short list of the commands and tools we've discussed in this handout.



Glossary of tools and terms

Working with Fusion's Single Design Context

Activate Component – makes a component active, filtering the timeline to show only features associated with that component

To use: click the radio button to the right of any component



New Component – creates a new empty component in the design

To use: Assemble Panel > New Component



Rule #1 – a design philosophy: Create components early in the design to organize features better and allow easier reuse of designs

Distributed Designs in Fusion 360

Insert into Current Design – Inserts a distributed design into the current design

To use: right-click another design in the data panel > Insert into Current Design

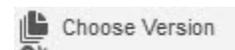
Update Component – updates distributed designs in the current document

To use: click icon on quick access toolbar; in browser, right-click a distributed design > Get Latest; or in browser, right-click on the top design level > Get All Latest



Choose Version – use to specify the version of a distributed design inserted

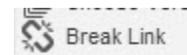
To use: in browser, right-click a distributed component > Choose Version



Restructuring Fusion 360 Designs

Break Link – breaks the link from a distributed design so it can be edited locally

To use: in browser right-click a distributed design > Break Link



Create Components from Bodies – Converts bodies into new components

To use: right-click on any body or body folder > Create Components from Bodies

Managing Large Designs in Fusion 360

New Group – creates a new group for organizing bodies

To use: right-click on the bodies folder > New Group



Isolate – hides all bodies or components other than those selected

To use: right-click any component or body selection > Isolate

Create Group – groups components in the timeline for organization

To use: select multiple adjoining features in the timeline, right-click > Create Group

