

CP10167

Fusion 360 Large Model Management for productivity and performance

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Learning Objectives

- Learn how to use productivity-enhancing tools in Fusion for large assemblies
- Learn how to use distributed designs workflows in large designs
- Learn how to avoid modeling pitfalls that affect large design performance
- Learn how to get the most out of the Fusion interface, options, and settings in any situation

Description

Presented by Fusion 360 Quality Assurance, this class will focus on the challenges presented by larger models, such as performance, clutter, and informational or screen overload. Discover Fusion 360 software tools to help you stay organized. Learn how to prevent performance problems as we explore solutions in the user interface, workflows, hardware configurations, and data management. We'll also demonstrate the commands and tools to increase productivity and extend the power of your assemblies and designs in Fusion 360 software.

Your AU Experts

Phil Eichmiller has worked with Autodesk Inventor as a product designer for 15 years and for the last 8 years teaching Inventor for the CAD program at Portland Community College. He also currently is a senior SQA Engineer for Autodesk on the Fusion 360 team where he enjoys testing, presenting, and teaching about Fusion internally for Autodesk, at local maker spaces, and for high school kids involved with STEM.

Joel Palioca has been a Software Quality Assurance Engineer at Autodesk for the past 4 years working on Autodesk Product Design Suite, and currently Fusion 360. His time with CAD has allowed him to experience different perspectives on design, manufacturing, and engineering, as well as virtualization of applications. Joel has been using CAD for the past 15 years, across a variety of different solutions. His desire to continue learning, and assisting users is what drives Joel every day to produce quality products.



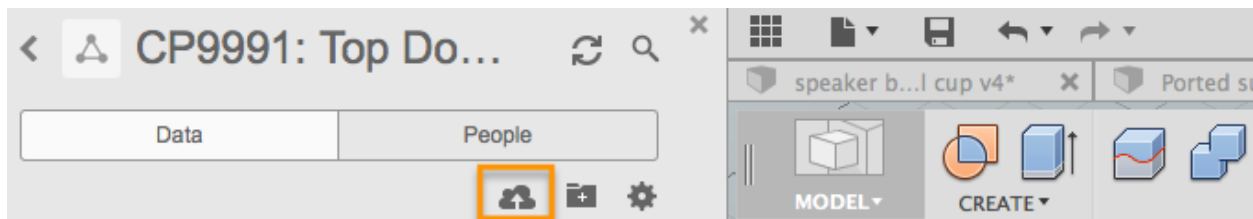
Fusion 360: Productivity-enhancing tools for large assemblies

What is a “Large Assembly” for Fusion 360?

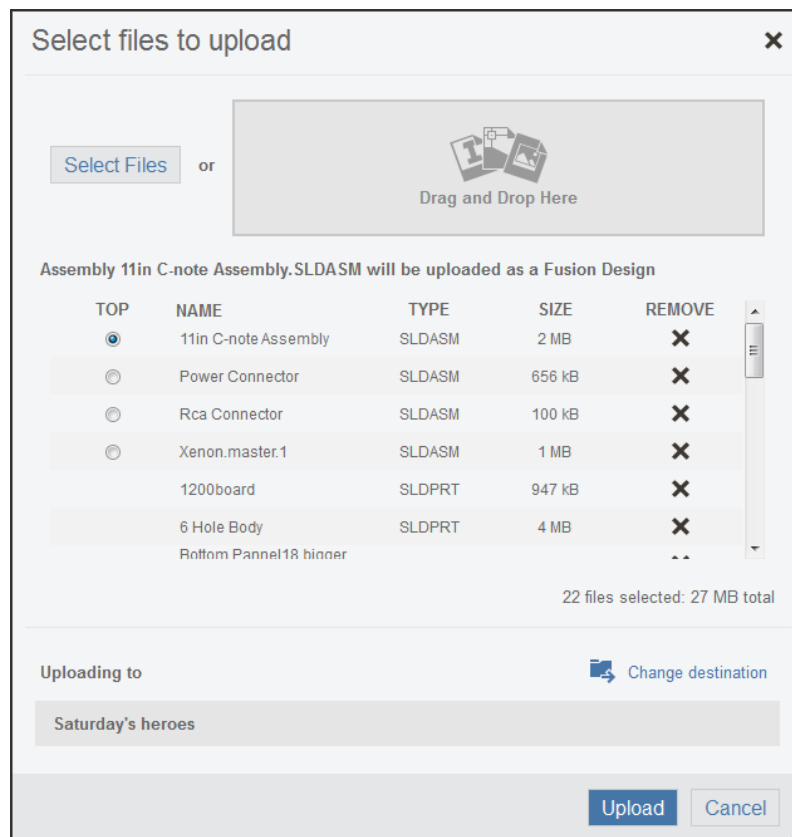
Currently the upper limit for Fusion assemblies, depending on your hardware, is above 2000 parts. In reality, and for the purposes of this class, a “large assembly” is any assembly that has a browser longer than the screen, and a timeline wider than the screen, with multiple child components. These conditions introduce the need to use Fusion 360’s tools and methods for staying organized and efficient. So let’s first talk briefly about how assemblies are created inside Fusion by *uploading*, *inserting*, or *modeling*.

Uploaded Assemblies

Uploading your assemblies inside Fusion is easy.



For Autodesk Inventor or SolidWorks assemblies it’s helpful to start with a flattened data view. You need to select all the parts and assembly files at once. Fusion will prompt you to select the top level assembly.

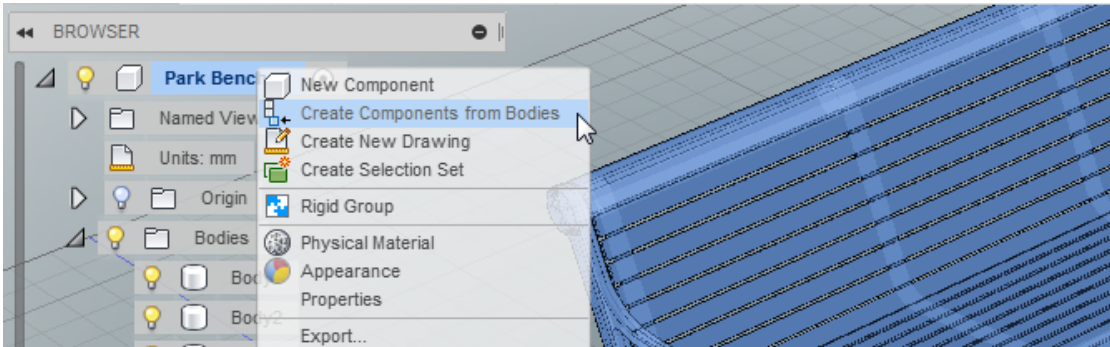


THE TOP LEVEL ASSEMBLY IS SELECTED DURING UPLOAD



What about SAT or STEP files?

SAT (.sat) files made from assemblies will place all components into a flattened structure under the Bodies folder in your Fusion browser. Names, materials, and appearances will be lost and you will need to manually create components from the bodies.



STEP (.stp) files retain component structure and names. You will need to manually apply assembly structure such as Joints for motion, Rigid Groups for non-flexible sub-assemblies, and grounding for components that don't move. These commands are covered in depth in later sections.

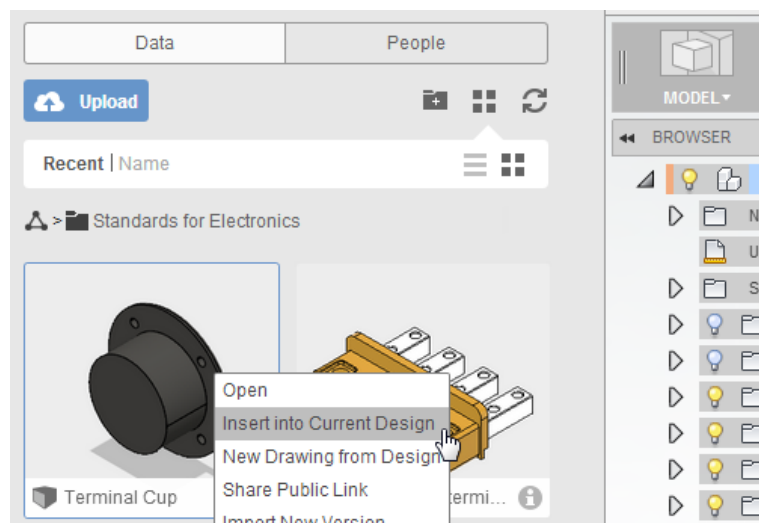
Modeled Assemblies

You will also create assemblies inside Fusion. Since there is no special file type for assemblies, each Fusion design can be a *single part* with only bodies, or an *assembly* with jointed components, or a *mix* of the two. This is managed with workflows like promoting bodies with Create Components from Bodies, using the New Component command, and dragging bodies and components into other components which is a cut/paste operation.

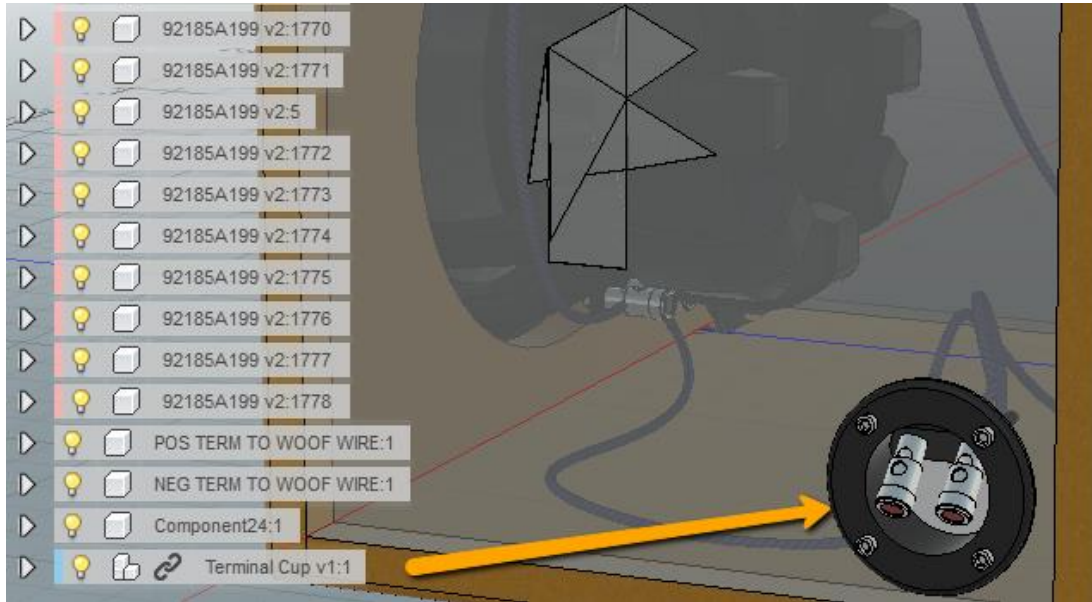
There are parametric considerations to keep in mind as you create assemblies using cut/paste. These are covered in depth in later sections.

Inserted Assemblies

A very powerful workflow is to insert other designs into your assembly.



The inserted components can remain Linked, so any changes to the linked component outside of the current design will be shown. While linked the inserted design is read-only, however joints will be flexible. You can also break the link to make the inserted component into an independent copy within your current design; this will also bring the parametric history of the inserted design into the current design. *This is covered in depth in the next section, p16.*



THE INSERTED COMPONENT SHOWS THAT IT'S LINKED TO THE EXTERNAL DESIGN WITH A CHAIN-LINK ICON

Immediate steps to take after uploading, modeling, or inserting your assembly

Whether you uploaded, created, or inserted your assembly information, there are a handful of essential tasks that you should consider immediately. These steps will help organize your design and help with modeling as you complete the design. These steps cover a few critical areas: movement of jointed mechanisms, material appearances, and organizing sub-assemblies using the browser and timeline.

Arranging Timeline and Browser

The browser and timeline helps organize components. The later sections go into more detail for these areas. The tips below are for getting started with imported assemblies.

Browser: Drag and drop components into other components to create sub-assemblies. It's helpful to first create an empty component and then drag all the browser nodes into it.

[Video demonstration](#)

For imported SAT assemblies this is particularly needed because they have no component structure when imported.

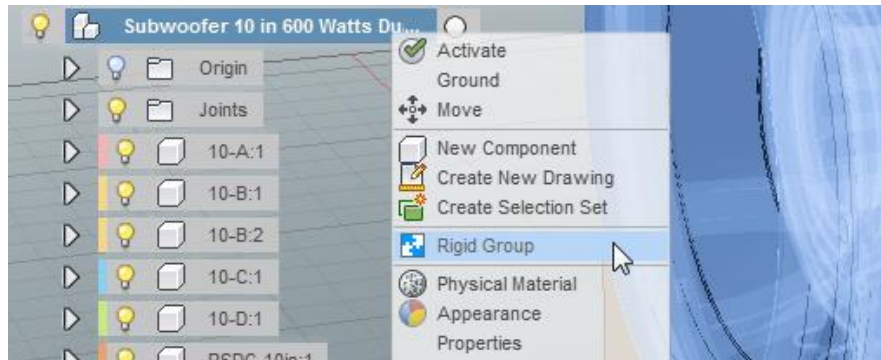
Timeline: Drag items that belong together logically together in the timeline. This also re-orders the browser items to reflect the new structure.

[Video demonstration](#)



Creating Rigid Groups

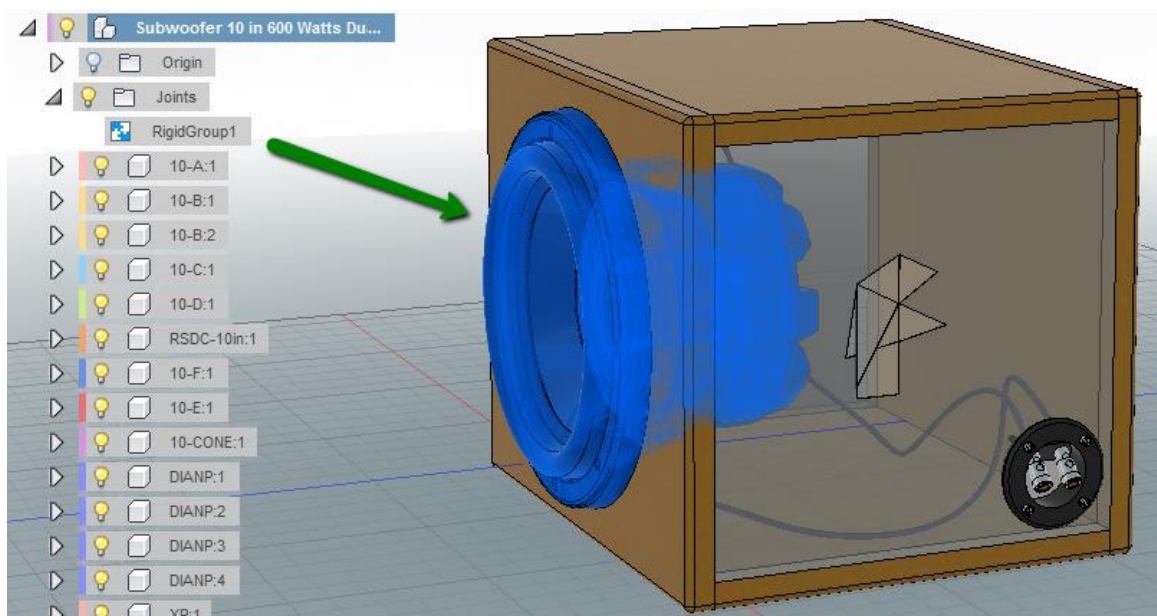
Parts grouped in logical sub-assemblies, unless affixed by joints, are free to move and interact with any other component in the design. When your intent is that a group of components will not move relative to each other use Rigid Group to “glue” them all together instead of manually placing dozens of joints. They will now react in the design as a single rigid group.



RIGID GROUP IS A RIGHT-CLICK OPTION FOR SUB-ASSEMBLIES

In this example a speaker design was uploaded from Inventor and inserted into the subwoofer box design.

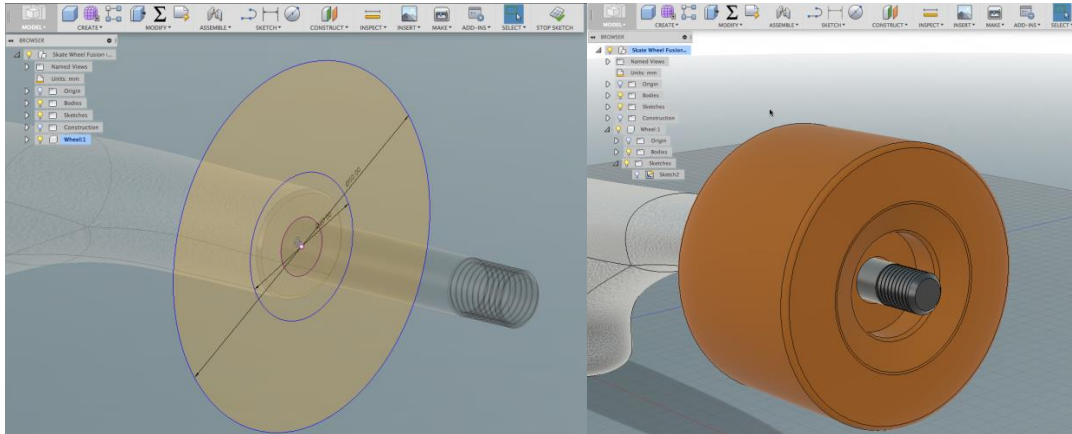
- There are no joints in the imported model.
- There is no need for the parts to move relative to each other.
- Rigid Group is used. The speaker components can be jointed to the box as a single unit.



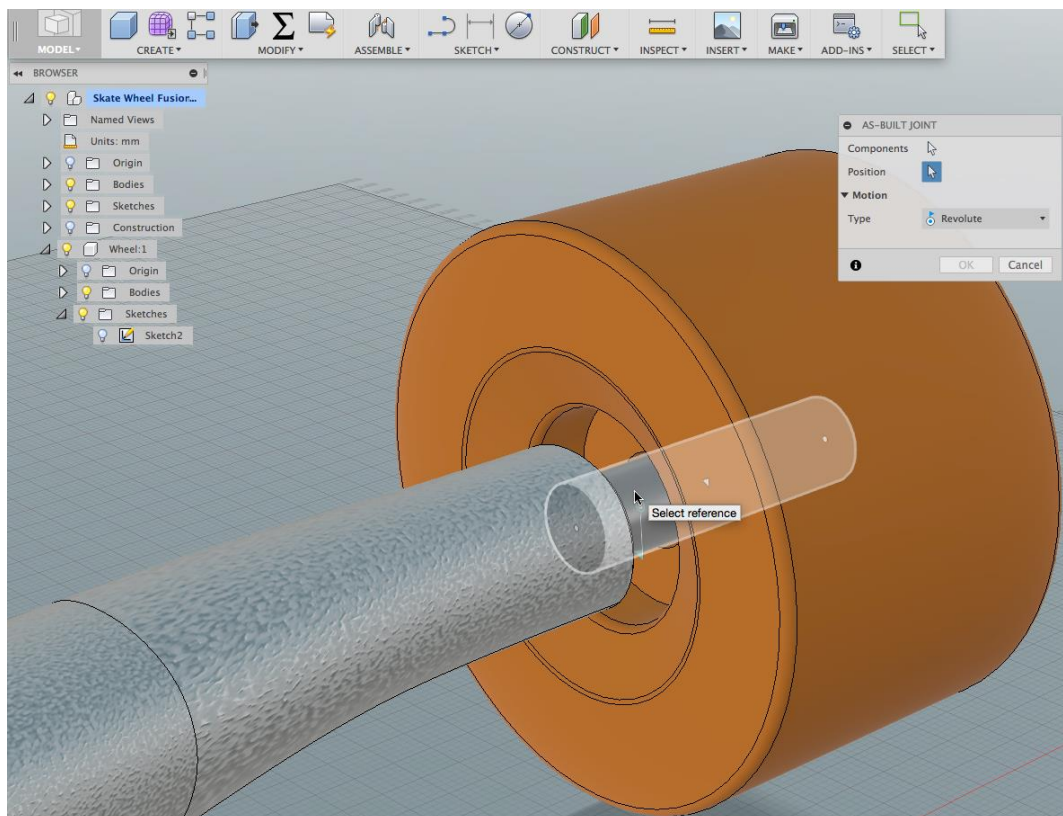
AFTER INSERTION THE SPEAKER ASSEMBLY IS MADE INTO A RIGID GROUP

Using As-Built Joints

When components that require joints are already positioned correctly relative to each other, a handy tool is the As-Built Joint. This command applies a regular joint between the components without forcing you to move them apart. This reduces useless Snapshots and keeps the timeline uncluttered.



A: THE WHEEL COMPONENT IS CREATED IN PLACE



B: THE JOINT FOR THE WHEEL IS CREATED IN PLACE TOO, WITH AS-BUILT JOINT



Appearances matter!

After uploading a large assembly, use physical material and appearance material to differentiate the parts. The workflow is easy and powerful. A side benefit is you can render your progress at any time! This is a great way to keep your visual view of the data organized and coherent.

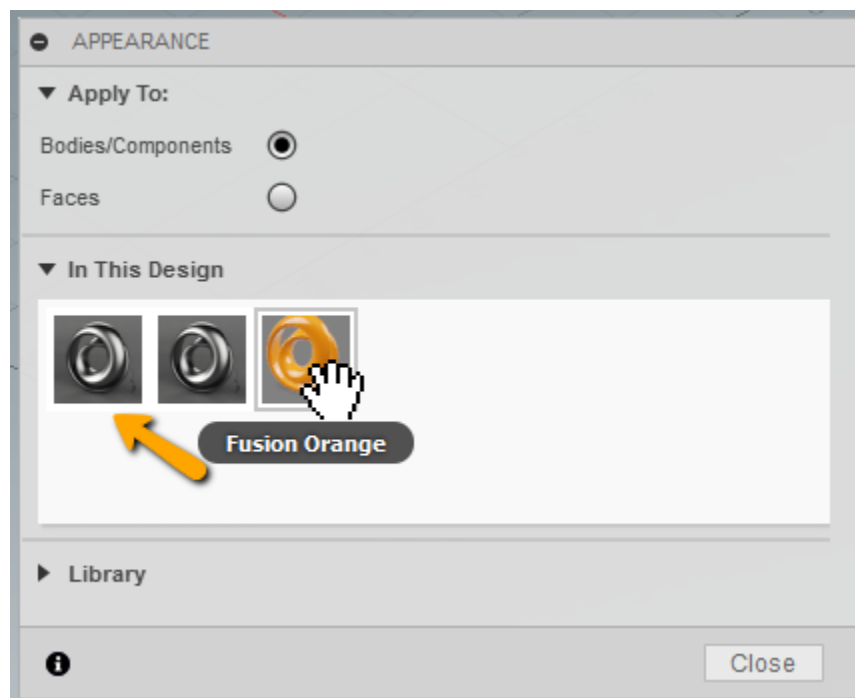
First a couple definitions:

- **Appearance** refers to the color of a body, face, or component and is independent of physical material. The use of appearances will override the physical material.
- **Physical Material** refers to the material that components are made of.
 - For example: A physical *material* aluminum part can have a red anodized *appearance*.
 - This information is used in the design Properties, such as mass.
- References in this document to “materials” or “appearances” are generally referring to either unless specified.

Here are a few tips for using materials in larger assemblies.

Replacing materials

Drag new materials/appearances onto the existing swatches that are shown in the dialog section: In This Design. This is a quick way to replace materials on many components at once. The existing appearance (or material) is replaced with the new one for all parts that carry it.



EVERYTHING THAT HAS STEEL COLOR WILL GET FUSION ORANGE APPEARANCE

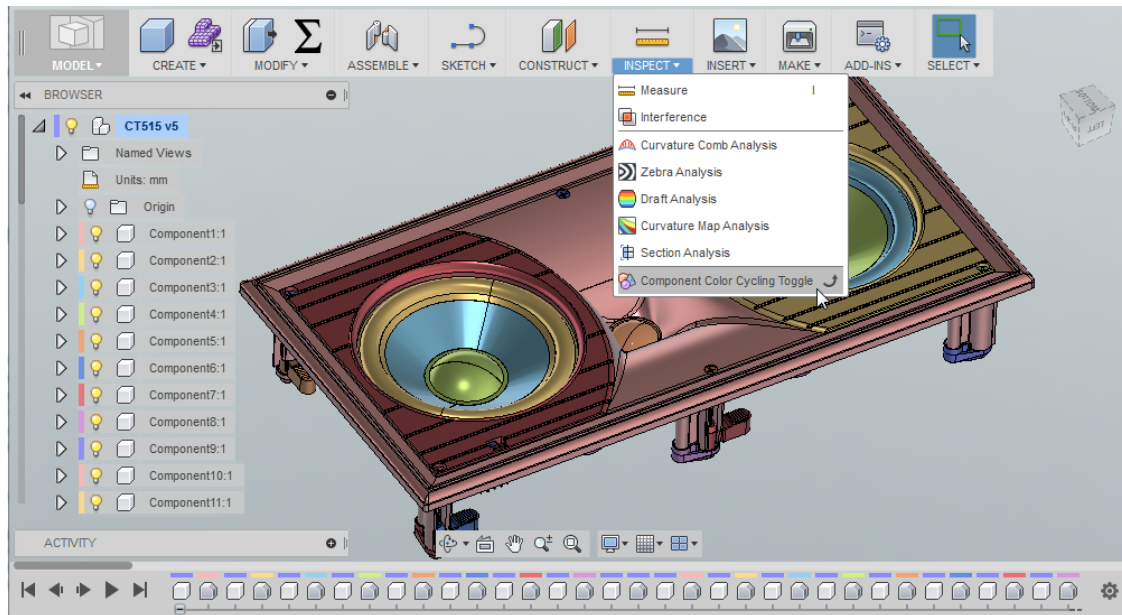
Drag swatches onto components and sub-assemblies in the browser. This applies the materials to all components at once in the sub-assembly.

[Video demonstration](#)



Component Color Cycling Toggle

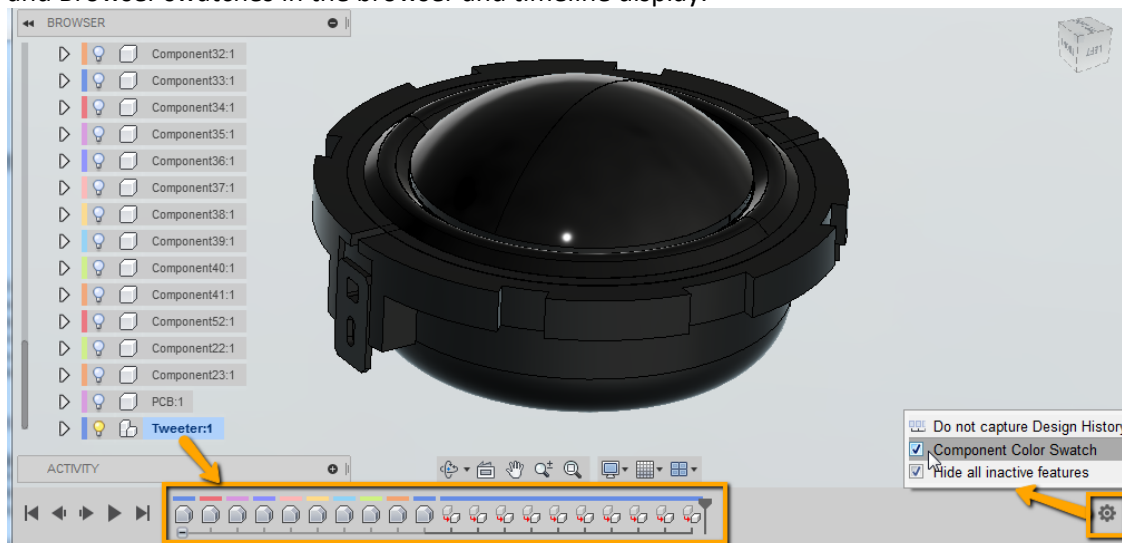
Found in the Inspect tools, this toggle applies a temporary random color to your components for the purpose of visual distinction when many components share the same material/appearance.



THE TIMELINE SWITCHES CORRESPOND TO THE ASSEMBLY COMPONENTS

Component Color Swatch

Independent from Component Color Toggle, but works in the same way for Browser and Timeline objects only. This means you can keep the normal model colors, but still use the small swatch colors. You cannot use the Component Color Toggle without also getting the Timeline and Browser swatches in the browser and timeline display.



THE TIMELINE SHOWS THE BLUE SWATCH BAR OVER ALL THE TWEETER TIMELINE OBJECTS



The assembly is now ready to participate in the design

Now that your assembly is behaving as you intend for this design, and your modeling has started, you will begin to use the tools in the UI (user interface) that are there to help keep you organized and productive. This next section will discuss three UI areas, and how to use them to your advantage as your model increases in size and complexity.

Timeline

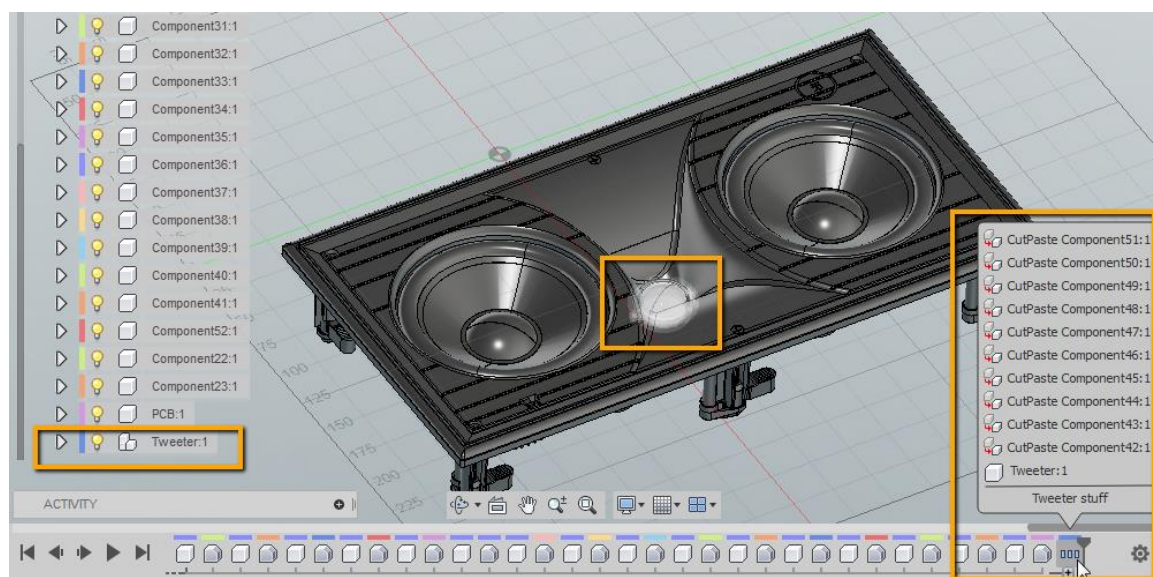
While we were preparing our assembly with the basic organizing steps above, we got a glimpse of the timeline features. This section goes farther by showing detailed workflows and features of the Timeline.

Creating Timeline Groups

Another way to keep the Timeline organized is by use of Groups. Timeline nodes can be placed in groups using the Create Group command. This works for all timeline features and related operations, but is also handy for assembly components that naturally should be grouped, such as sub-assemblies. *Note! Timeline is only shown in parametric designs!*



THE TIMELINE NODES FOR TWEETER ARE SELECTED AND RIGHT-CLICK MENU IS USED TO CREATE GROUP



THE NEW TWEETER COMPONENT AND ASSOCIATED OPERATIONS ARE GROUPED AS "TWEETER STUFF"

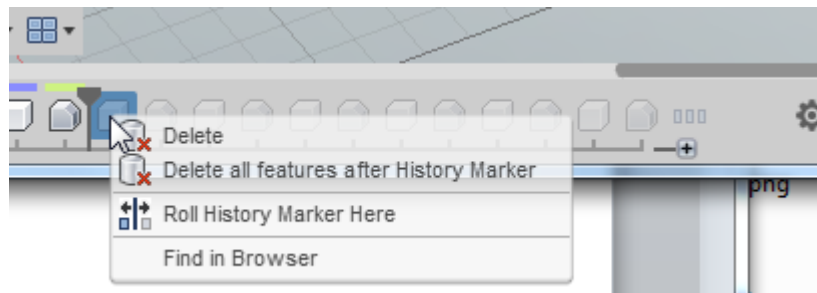


- Groups reduce timeline clutter.
- Groups can be renamed by a right click command.
- Groups expand and contract by the + and – nodes under them.
- Groups can be used in Selection Sets. Great for managing visibility. (see Browser: Selection Sets)
- By default uploaded STEP files with subcomponents are shown as groups.
- Also inserted designs where you subsequently use **Break Link** are groups in the timeline.

More Timeline tools

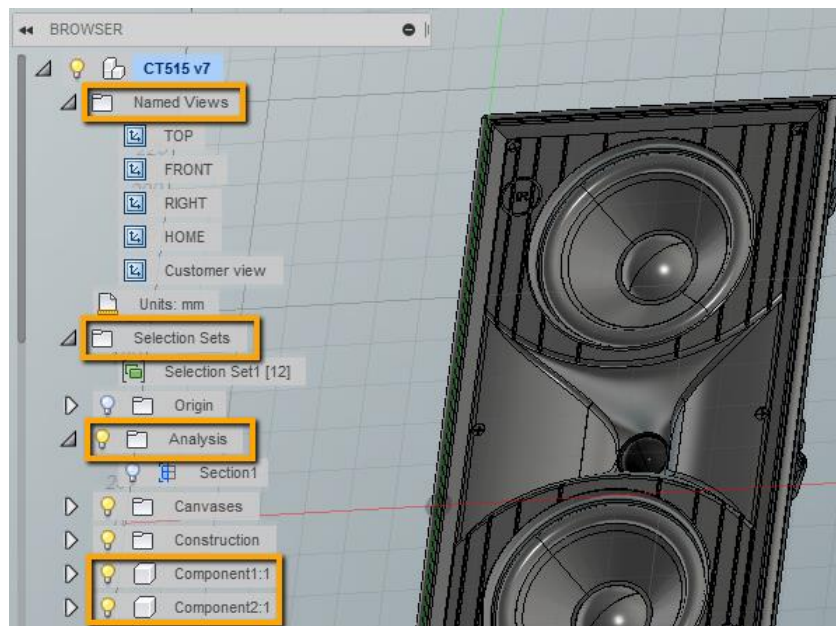
This class does not cover all things Timeline; rather, we are just focusing on productivity in large assemblies or models.

There is one more tool to show you and it helps when pruning the timeline. If you need to remove items from the end of your timeline, slide the marker to the left of the items to remove, right click on the grey icons, and pick **Delete** or **Delete all features after History Marker**.



Browser

The browser shows objects in the design. Named Views, user defined Selection Sets, and Analysis are tools you can use to navigate, select, and organize your larger models or assemblies.



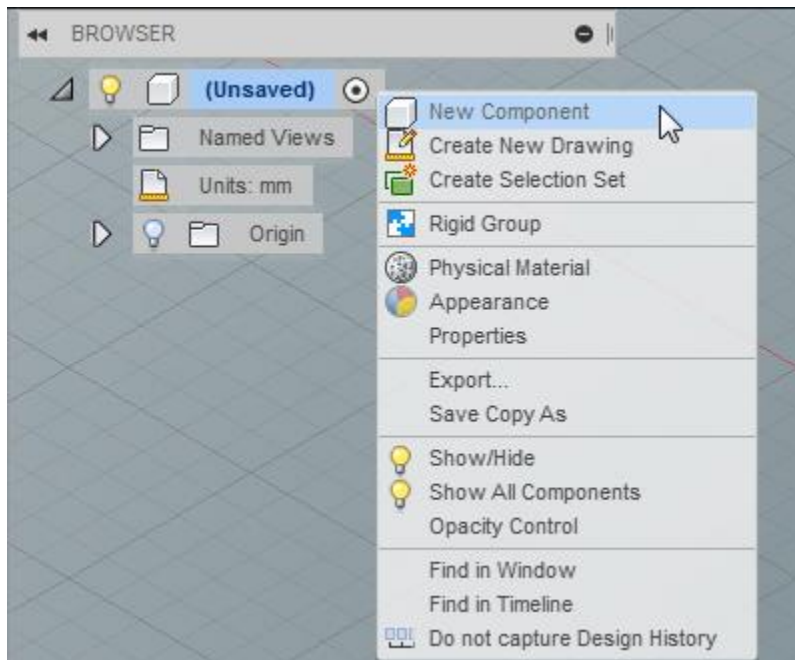
THE BROWSER: NAMED VIEWS, SELECTION SETS, ANALYSIS, AND COMPONENTS



Right click on browser items to find options for and perform editing of the items. Use browser items in tandem to create specific effects, such as combining a user defined Named View and Section Analysis to quickly navigate to an area of interest in a large model. The section below describes the tools and right-click options that help with assembly management working from the top of the menu down.

Right click Browser top node – New Component

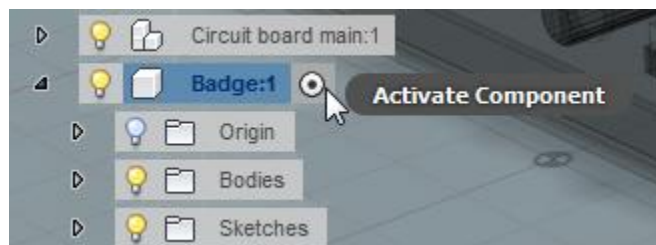
The top node of the browser, when the design has components, and is thus an assembly, has options in the right click menu that are discussed widely in this document. The most relevant item is right at the top, New Component. When this is used the design instantly becomes an assembly.



AMONG MANY USEFUL TOOLS FOR ASSEMBLIES: NEW COMPONENT

Activate Component

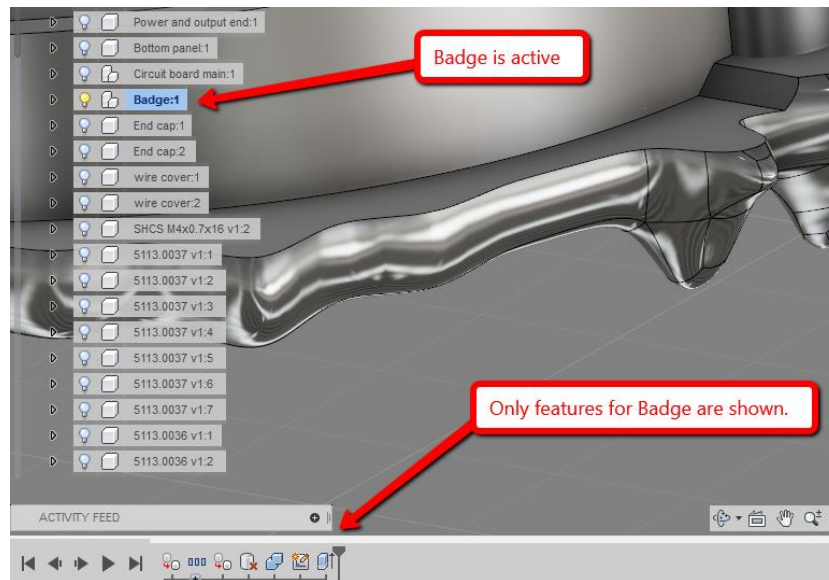
When working in assemblies with components, the use of Activate Component is critical to keeping information specific to individual components contained within that component, such as sketches and bodies. This is covered in detail in the class session *CP9991: Fusion 360 Workflows for Design Success: Top Down Versus Bottom Up*.



FIRST ACTIVATE THE COMPONENT, THEN CREATE SKETCHES AND BODIES



Tip! When components are active, the timeline only shows the features of that component.



Cut-Paste, or drag? Making sub-assemblies

You may need to create sub-assemblies by arranging components that already exist in the timeline/browser. This is done, in the browser only, by right-click and using Cut-Paste to move one component into another, or you can simply drag components into each other. The icon changes from a single box to a stack. This cannot be done in the timeline.

TIP! It is critical to use the Activate Component workflow for component creation, well before Cut-Paste operations on components. Why? This ensures that the entirety of the parametric history, including sketches and bodies will travel with the component to its new sub-assembly structure. Consider this workflow as an important best practice.

Named Views

When your model is large in size or complexity often it can be a pain to navigate to the details you want quickly. Create Named Views to speed this up. You can easily create a custom named view any time you are zoomed to an area of interest.

Workflow: As you work on an area of detail, right click on the Named Views folder and use the option New Named View. Select the named view item and rename it for easy reference. Just select it when you want to return to that view. [Video demonstration](#)

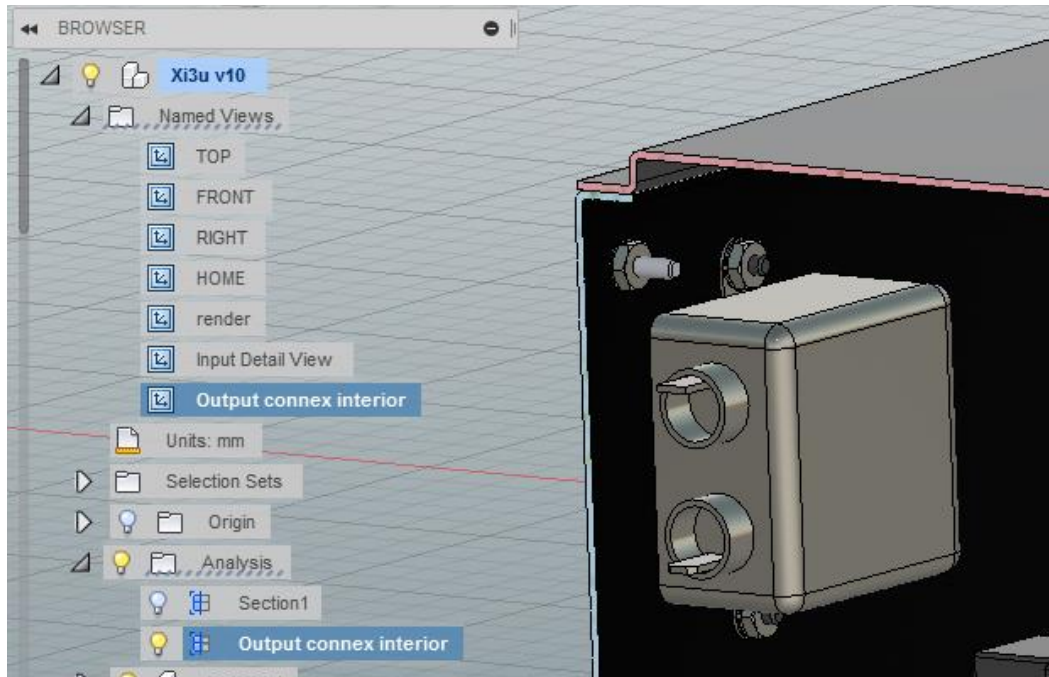
Named Views are also useful for Design reviews, detailing in relatively small areas, renderings.

Extending Named Views

Use Named Views along with Section analysis to zoom in and also remove visual obstacles.

[Video demonstration](#)





TIP: USE THE SAME NAME FOR YOUR CUSTOM VIEW AND SECTION ANALYSIS

Selection Sets

Selection Sets are user defined and allow you to select any number of objects with a single click. The workflow is similar to Named Views in that it's very convenient to capture Selection Sets on the fly. When you have a bunch of objects selected, and you know you'll need to select them again later, just pause to right click and use Create Selection Set.

Extending Selection Sets

Use Selection Sets and visibility controls, or also Named View, to quickly toggle visibility of many objects at once and get to that detail view quickly every time.

[Video demonstration](#)

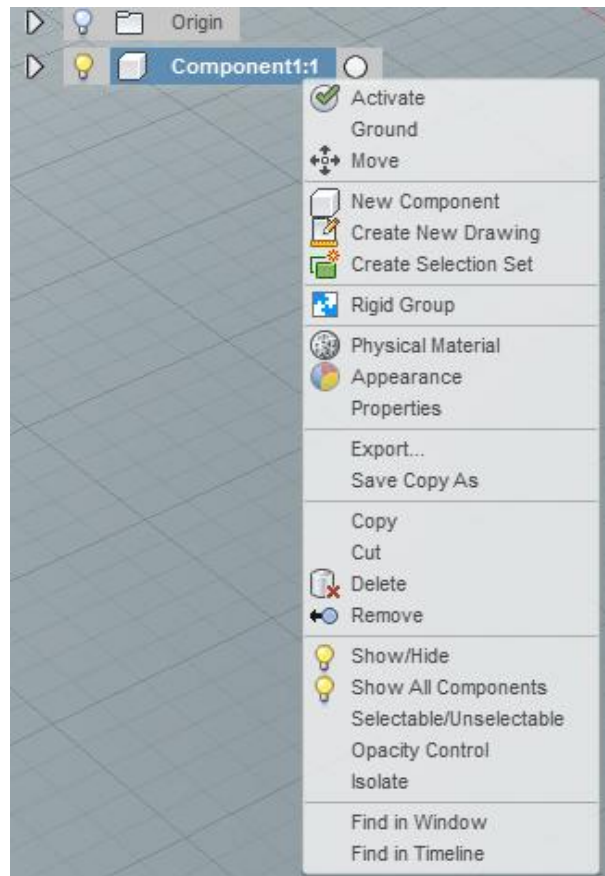
Working with components in the browser

Right click on any component in the browser. You will find a long list of options and commands you can apply to that component. Several have been discussed already, but here a few more that help with selections, visibility, and deletion of parametric data.

- **Selectable/Unselectable:** while Unselectable, component highlighting is also muted.
- **Show All:** when you need to see all the components again.
- **Opacity:** makes components semi-transparent when they visually get in your way.
- **Isolate:** quickly shuts visibility of all other components. Un-isolate returns all components to fully visible.
- **Remove/Delete:** these both delete components, but Remove preserves the parametric information associated with the component. Delete will destroy all parametric associations in the model while deleting the component.

TIP! Use Remove instead of Delete when bodies and sketches refer to the component.

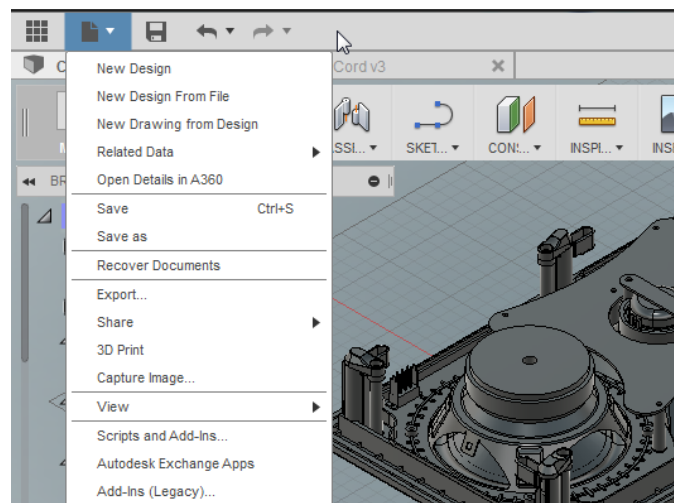




THE FULL RIGHT CLICK MENU FOR COMPONENTS

File Menu

File Menu commands that help with assembly management: *Related Data*, *Open Details in A360*



THE FILE MENU: RELATED DATA, OPEN DETAILS IN A360



Related Data

This menu shows any related drawings or linked designs.

Open details in A360

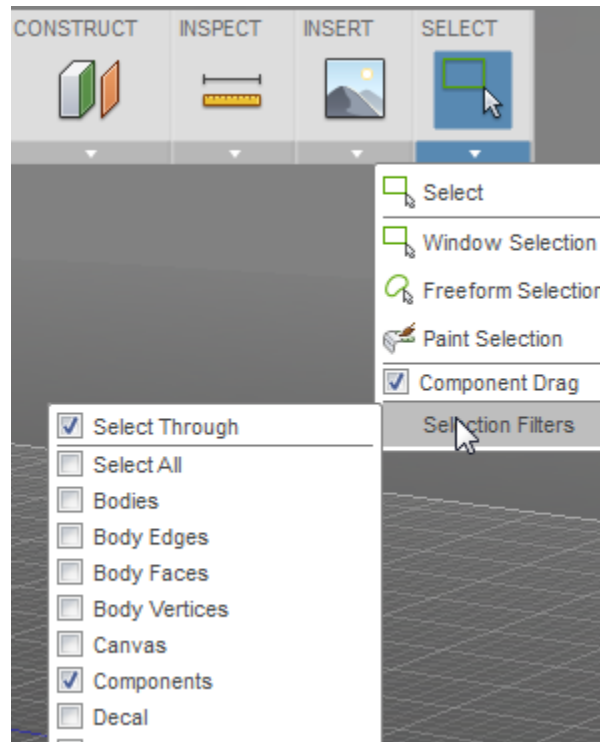
This opens a web browser and displays the detail page for the design.

Other tools in the user interface (UI)

The last two tips for using tools in the UI are definitely not “least”, but are often overlooked.

Selection Filter

Use the selection filter when your workflow involves a lot of selections of a specific type of object or objects. For instance, when you want to select several components to create a Selection Set or apply Appearances, it helps to set the filter to Components only for the duration of the workflow.



THE SELECTION FILTER SET TO COMPONENTS

Setting the pivot

Large assemblies can make view rotation a pain when the pivot point is not near the area of focus. There is a simple key-combo that sets a new pivot point on the fly. Simply press Shift + click middle mouse wheel.

[Video demonstration](#)



Fusion 360: Using Distributed Designs in Assemblies

What is a Distributed Design?

Like most CAD systems, Fusion 360 allows you to insert components into your designs. These components are usually used in multiple designs, hence the need to store them externally to your design. Hardware items like screws and hinges, or common purchased components like switches or motors are typically inserted.

Benefits of using Distributed Designs

Using linked files speeds design workflow, provides less UI clutter, and increases compute performance.



THE LINKED DESIGN TAKES UP VERY LITTLE TIMELINE SPACE

Performance Matters! Use Distributed Designs

Each time you add a feature to your design, a compute is performed on the model. When assemblies are fully developed and have hundreds of features, this can begin to impact performance greatly. One of the most important benefits of linking designs is the fact that these designs are Read-Only when inserted **and do not participate in the compute cycle**.

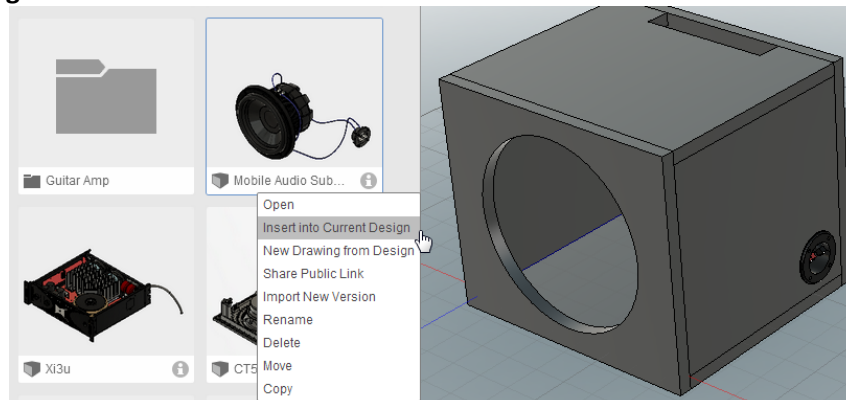
Linked files take less Timeline space

Another benefit is that inside the large assembly, the linked components take up very little Timeline space. The node is just a single icon, rather than all the accumulated commands that make it.

Collaboration

When you work with other Fusion 360 users, linked files can enable your team to work together on a single design. Once the design is linked to the master assembly, another user can work on it separately and simultaneously. When the linked file is saved by the other user, the master assembly is updated to reflect the change.

Distributed Design workflows



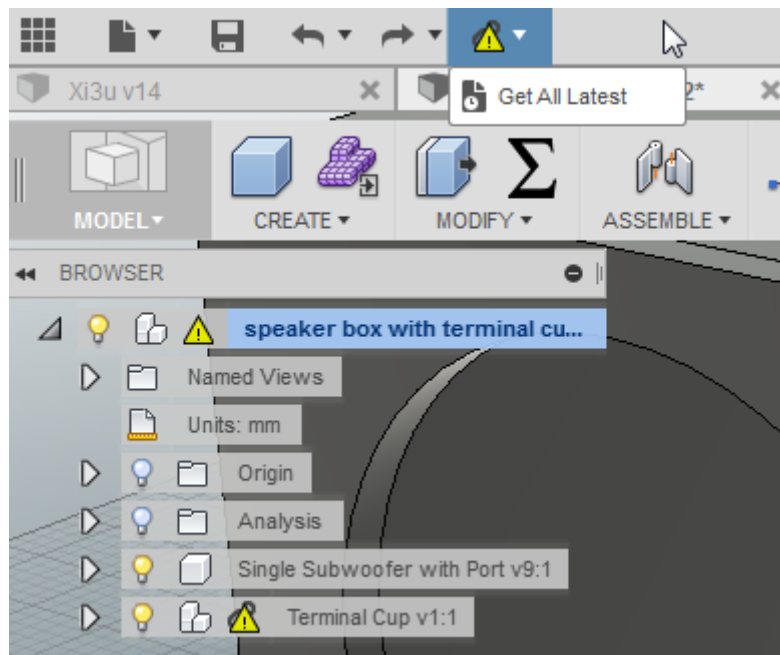
To insert a design into another design: open the data panel, right click on the design and pick Insert into Current Design.

If you wish for the inserted design to no longer be linked, right click on the browser node and pick Break Link. The link is removed and a new component is made. A group is automatically created that contains the parametric history of the new component.

[Video demonstration](#)

Get all latest

When a change is made to a linked design, all referencing designs will detect the change from any edit to the linked file. A warning appears that shows the referencing design should be updated. Use the command Get All Latest to update your referencing design.



THE YELLOW WARNINGS SHOW THAT THE LINKED FILE IS OUT OF DATE

Fusion 360: Avoiding modeling pitfalls that affect assembly performance and usability

Tips and best practices for modeling or assemblies

These tips are by no means a complete guide. Rather, we are focused on Assemblies here so these are just a couple tips for avoiding problems in Assemblies. However, generally this advice applies to any design in Fusion 360.

Usability pitfalls

Body visibility: Do not turn off body visibility; you will regret it when it's time to look for the body in a nested sub assembly. There is no automatic way to show all hidden bodies. Instead use component visibility, it's much easier to see in the browser and fix by using Show All Components.



A corollary to this is to be careful when using the hotkey V (visibility). Here's why: When your selection filter is set to "all", Faces take selection priority, but Face selections promote to Body selections when a non-face command is used. Faces select first in this situation because it's more useful when creating sketches or doing Push-Pull operations. Face selections promote to body when you use V because you cannot make a face invisible without making the body invisible! So if you start selecting faces and typing V you are turning off body visibility and that is harder to recover from, you must manually find the bodies to turn them back on.

Copying parametrically: Try to use the activate component workflow described in this document as much as possible. This enables copying components into other designs, or using Save Copy As, and ensures you will get a healthy and parametrically complete copy of your sub-assembly or sub-component.

Performance pitfalls

- Avoid unnecessary snapshots. If you make more than one while moving a component, you can delete all but the last one for any particular component move.
- Avoid unnecessary body moves. Design the bodies in place when possible.
- Be careful about inserting linked files with many bodies and components. Graphics performance will suffer eventually. See section below about optimizing graphics.
- Define components early. Create them in place.
- Avoid unnecessary references between components. Use origin reference planes or work planes based on the origin rather than convenient but illogical connections.
- Fix errors when they happen. This can add up during compute cycles.



THERE ARE UNNECESSARY MOVES AND UNFIXED ERRORS HERE

Fusion 360: Optimizing for performance

Performance in Compute vs. Graphics

Other than good modeling practices, such as fixing errors and using linked files as described earlier in the document, there isn't currently much way to improve the compute cycle. Graphics performance, however, can vary widely depending on hardware, and settings.

CPU vs GPU

- Externally referenced designs reduce CPU load. When a command requires a compute, the linked designs do not participate.
- Large numbers of any kind of object increase CPU load. You need to make them un-selectable via right click menu.
- Textured appearances, effects like Ambient Occlusion, the number of edges, faces, etc. all affect GPU load. Turn off effects (see p19) or, in extreme cases, turn off visibility of objects when not needed.



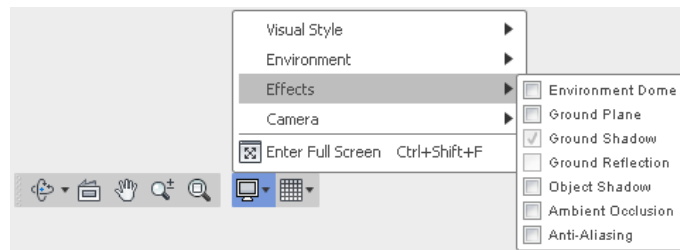
Use Selection Sets to manage the scene for CPU and GPU performance

1. Select all the components you wish to turn off and on visibility or select-ability.
2. Right click, make a selection set.
3. Use the selection set by picking the "select" button attached to it.
4. Right click and use the visibility toggles or selectable/unselectable toggle.
5. [VIDEO](#)

Graphics settings inside Fusion

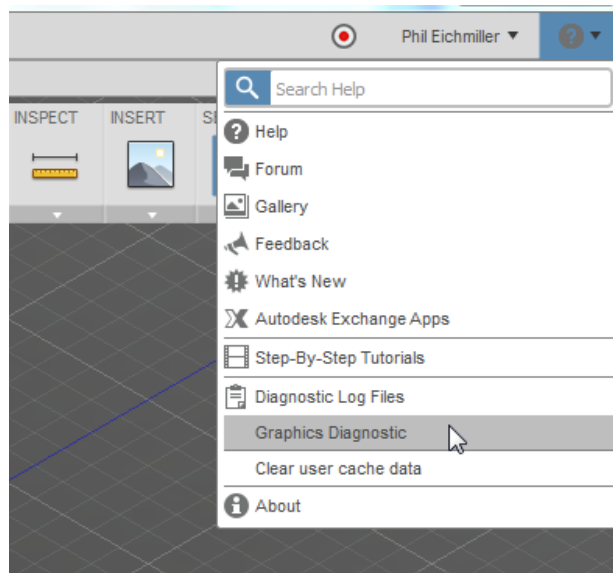
Fusion 360 defaults to showing many graphics effects. It looks great and usually does not cause problems. In a large model, when performance becomes a concern, what is more important? Great shadows or getting the job done?

Use Effects settings to turn off the heavier effects while you are modeling, then turn them back on when you need to see the model in full realism. Control them with the Effects settings.



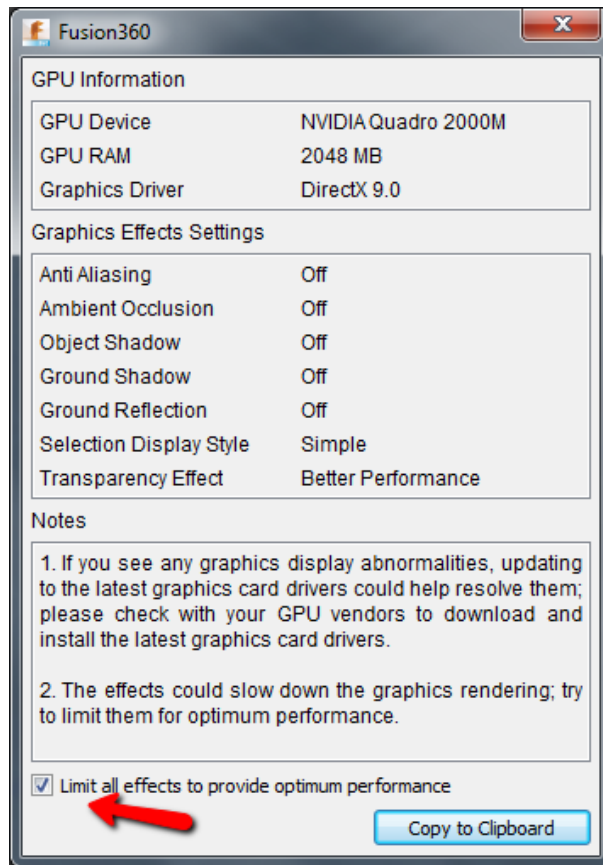
ANTI-ALIASING, AMBIENT OCCLUSION, REFLECTION ARE THE HEAVIEST LOAD ON YOUR GPU

The Graphics Diagnostic has a toggle for all effects, and can be useful for analyzing your hardware and graphics issues.



ACCESS THE GRAPHICS DIAGNOSTIC





USE THE TOGGLE TO CONTROL ALL EFFECTS AT ONCE

Graphics settings outside Fusion

Windows Only

To adjust all visual effects for best performance:

1. Open Performance Information and Tools by opening the Control Panel. In the search box, type Performance Information and Tools, and then, in the list of results, click Performance Information and Tools.
2. Click Adjust visual effects. Administrator permission required If you're prompted for an administrator password or confirmation, type the password or provide confirmation.
3. Click the Visual Effects tab, click Adjust for best performance, and then click OK. (For a less drastic option, select Let Windows choose what's best for my computer.)

Laptops Only

Some laptops have power saving settings that will force the laptop to use a weak chip-set graphics adapter rather than the dedicated video card. Performance can suffer greatly in this case.

This problem may be caused by NVIDIA Optimus and AMD Enduro or "Dynamic Switchable Graphics" video configurations, which are available in newer laptops. Both NVIDIA and AMD have introduced technology that uses a discrete graphic card in addition to the integrated Intel



HD GPU. This allows the system to automatically switch between the two in order to save battery power or when more graphics processing is needed.

In some cases, the system preferences for conserving energy can impact the behavior of the application. In other cases, the switching may be the root cause of the issue.

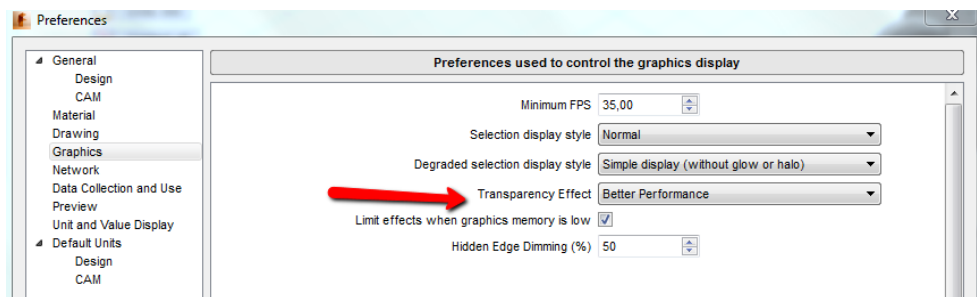
Outdated drivers as well as system settings may also lead to failures in graphics switching.

Steps for managing graphics cards in laptops are spelled out in this Autodesk Knowledge Network article.

<https://knowledge.autodesk.com/support/fusion-360/troubleshooting/caas/sfdcarticles/sfdcarticles/Poor-Performance-in-Fusion-360-on-laptops-with-dual-graphic-cards.html>

Preferences for graphics

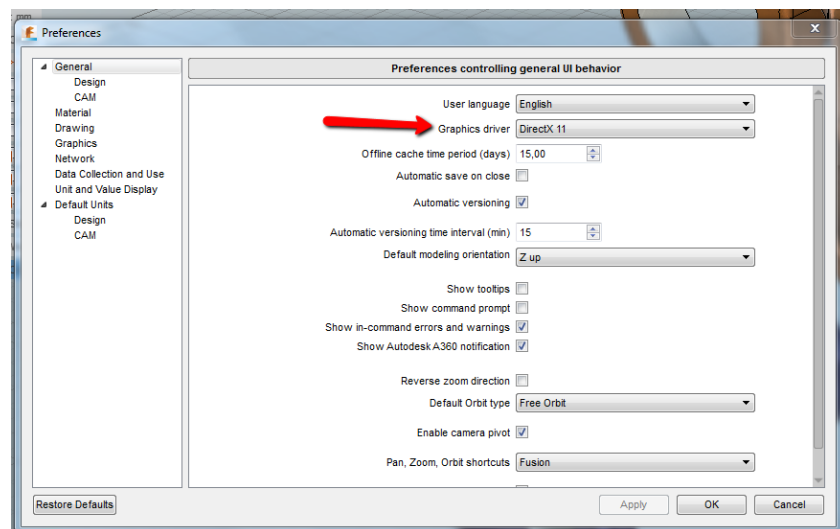
Preferences contains two important settings for graphics performance.



SET TRANSPARENCY EFFECT TO BETTER PERFORMANCE

Windows Only

Sometimes switching the Fusion 360 preference for graphics driver from DX11 to DX9 can solve Graphics problems, including performance.



CHANGING GRAPHICS DRIVERS INSIDE FUSION 360



CP9991

Fusion 360 Workflows for Design Success: Top Down vs. Bottom Up

CP10165

Joints, So Much More than Rigid

