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

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Learning Objectives

- Discover the differences between top-down and bottom-up design in Fusion 360
- Learn how to upload and use various design data: parts, assemblies, dxf, and images
- Learn how to prepare and use components in distributed designs
- Explore the differences of and how to use Body versus Component in a design

Description

Are you starting with a blank screen? Or do you have a library of components? This is the top-down versus the bottom-up design question. Most designs use both. Your workflow should start with a strategy to get the most out of these styles of design. This class will dive into these different workflows in detail, using Fusion 360 software. Explore top-down design, including the body or component—what's the difference and why you should start with either; workflows to create in place components—starting off on the right foot; and when and how to reuse the designs you're making. Learn about techniques for bottom-up design, including upload—what's the best way to get large or small designs from anywhere into Fusion 360 software; file management via Fusion 360—keeping track of it all; design preparation—how to better prepare designs for insert and use; and distributed design workflows—inserting and using the parts you already have.

Your AU Experts

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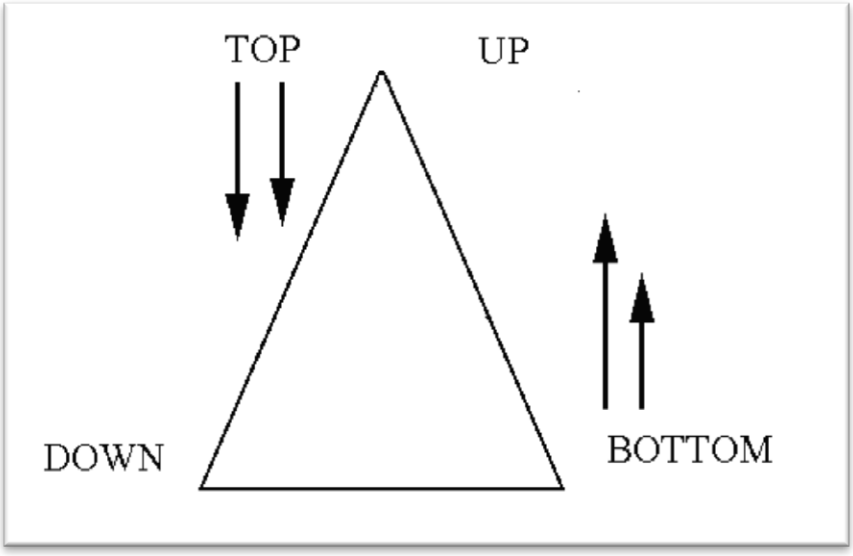
Sachlene Singh *is a Technology Evangelist at Autodesk. This gives her the unique opportunity to work with users of design software and make them successful. With a more recent focus on emerging technology companies, Sachlene has played the role of mentor, design advisor, engineer and team member to help startups bring their products to market. She is a Mechanical Engineer, has a Master's Degree in Mechanical Engineering and has several years of experience in the industry and at Autodesk.*

The differences between top-down and bottom-up design in Fusion 360

Some definitions
The terms top-down and bottom-up and even "middle-out", when talking about design, describe workflows that seem logically opposed to each other. It's important to start by understanding that these approaches to design are not exclusive of each other. Most design is a combination of these workflows.

Top-Down:
An approach to design that begins at the highest conceptual level and works down to the details.

Bottom-Up:
An approach to design that begins with details and works up to the highest conceptual level.



THE TOP-DOWN OR BOTTOM UP PYRAMID OF DATA

Top-down vs. bottom-up design in CAD
When it comes to CAD modeling these competing concepts are expressed in distinct workflows. At any point in your design you are deciding to use one or the other. This document is about 'when and how'.

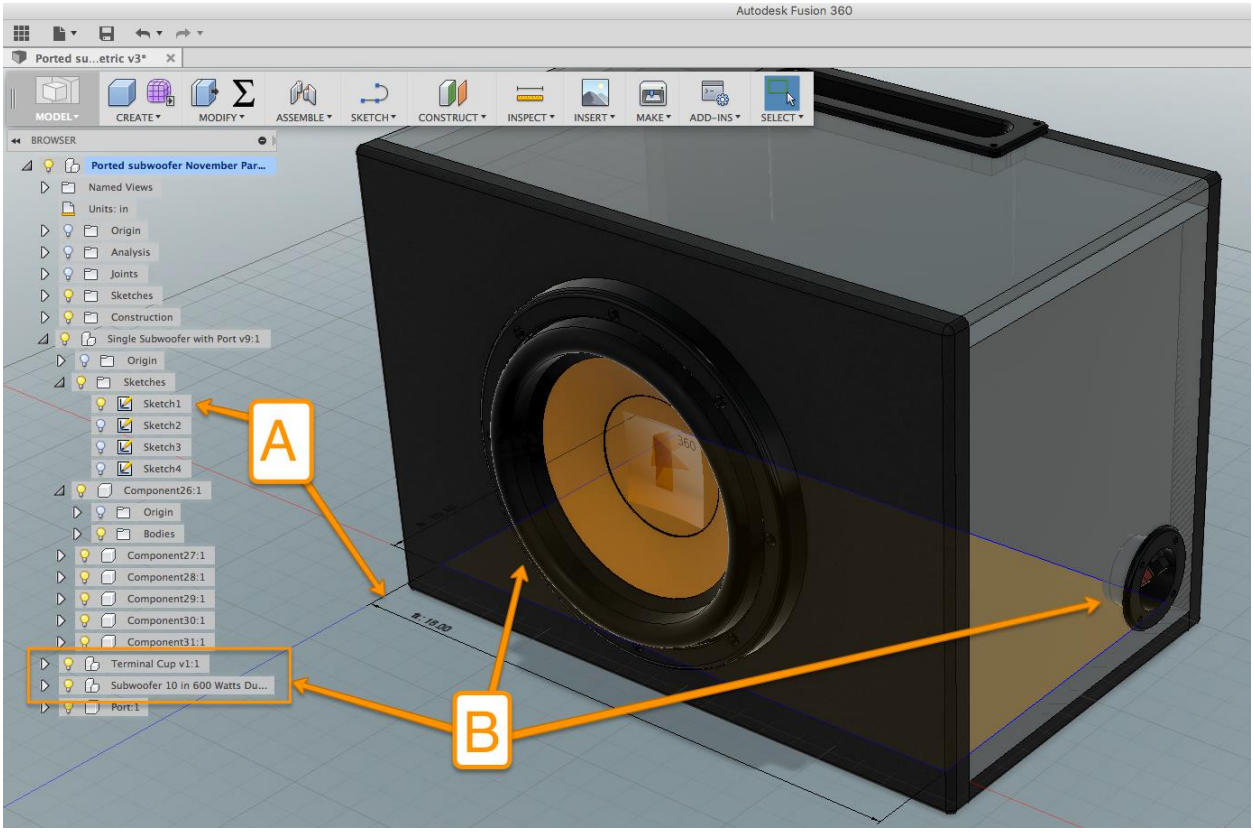
Top-down
For 3D modeling, this means starting out with the design goal, but nothing on the screen, and then building all design aspects in place. The only references needed are that of the design as they are created. The parts refer to each other. The problem is being solved at the conceptual level, and a model is being created that addresses the needs of the solution. This can also mean creating a conceptual framework, such as a skeleton sketch, or using a reference image.

Bottom-up

When the design is based on known properties of known components the challenge is different. In this case the designer has a tool kit of existing parts and must configure them together to solve the problem. Legos are a vivid example, but more realistically think about conveyor design for factories, or any kind of modular components such as your cubicle walls. There is plenty of design to be done; you must make the system work! But you start from a set of knowns rather than concepts, such as a set of fully developed CAD models instead of a skeleton sketch.

Real work is a combination

Rarely is the entire design just one or the other. These are general approaches to solving the problem at hand, but the top-down skeleton sketch based design will also use purchased components and the bottom-up conveyor system will also require custom components to make it all work. This document attempts to make sense of the two approaches as they are expressed in Fusion 360 workflows, describing how to do each in its place as your design requires.



TOP DOWN MEETS BOTTOM UP DESIGN

A) SKELETAL SKETCH, DRIVEN BY PARAMETERS, CONTROLS 80% OF THE SHAPE OF THE BOX.
B) PURCHASED COMPONENTS. THE DESIGN MUST REACT TO THESE PRE-DEFINED OBJECTS.

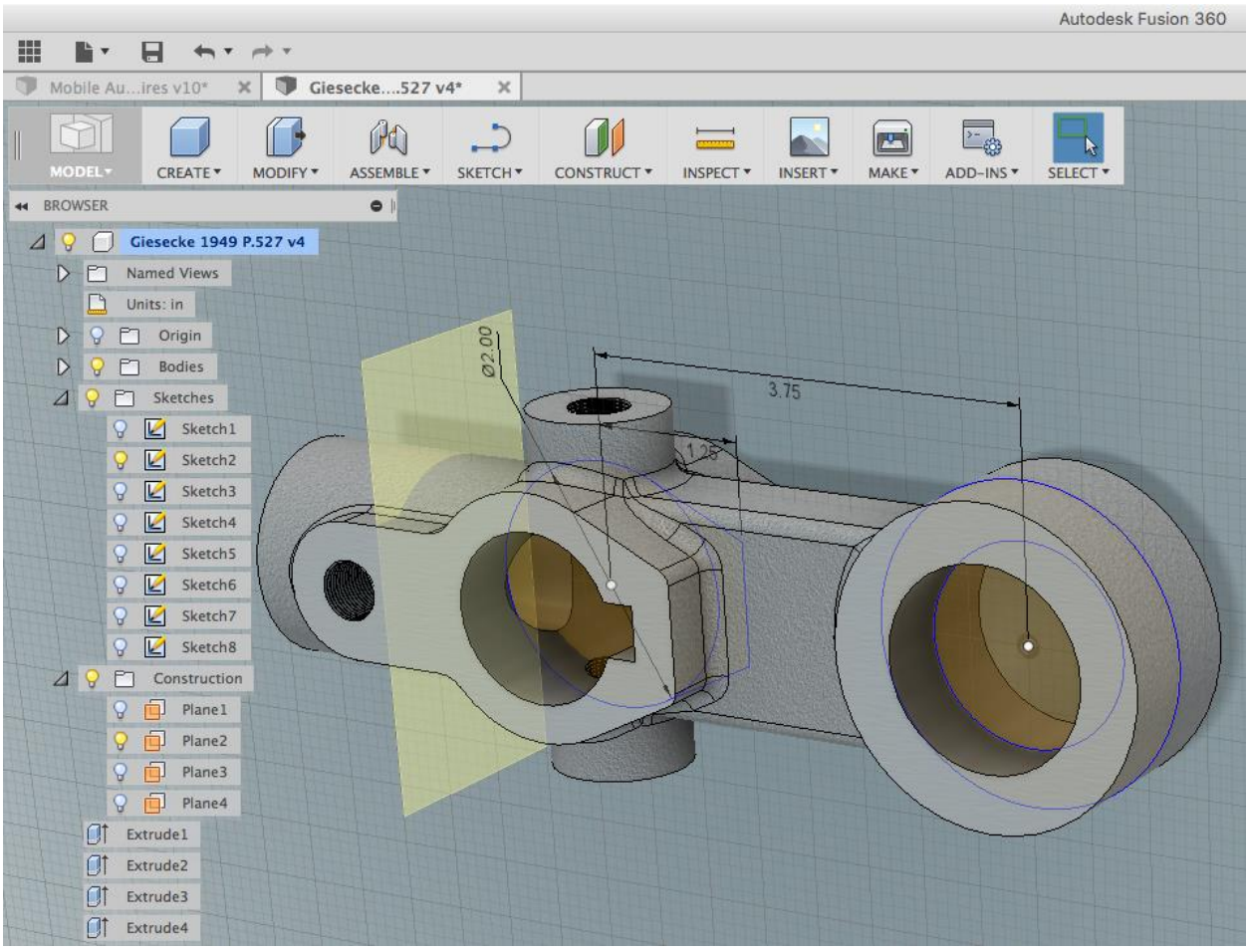
Top-down options and workflows in Fusion 360 modeling

Fusion 360's major strength and focus is top-down design, so much of this document deals with this concept. These are some of the techniques, workflows, and things to consider in top-down design using Fusion 360.

Creating bodies

What is a body? It is a single container for a contiguous 3D shape in your design. By default, when you create your first 3D shape it will become a single body in a component that represents one part. Additional bodies may be created and later joined together, but in a single component design, all bodies represent distinct regions of only one single component.

Bodies are also used as modeling tools. You may need to create a body that is used to cut or split another body. Interactions between bodies happen inside components.



EXAMPLE OF A DM (NON-PARAMETRIC), SINGLE BODY COMPONENT

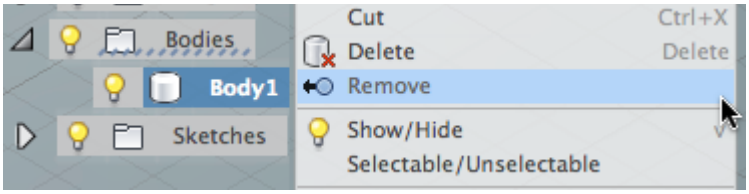
Workflow:

- If your intent is to model a single part that has one body, after initially creating your first body, continue by using the Join option in commands like Extrude, or make more bodies and join them later with commands like Combine.
- If instead your intent is to create an assembly of components, it is best to create components first, and then add bodies inside the components they belong to from the start. Use Activate Component before creating bodies.

LEFT TO RIGHT: BOUNDARY FILL, SPLIT BODY, COMBINE, MOVE

Related commands:

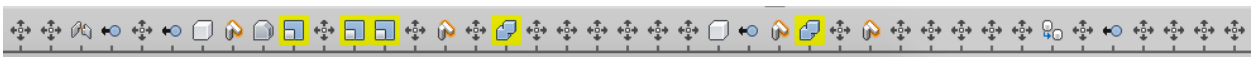
- **Split Body** – Splits bodies using planar input, such as construction planes, or body faces.
- **Combine** – Performs Boolean operations such as Cut or Join using bodies as input.
- **Boundary Fill** – Performs Boolean operations using regions defined by any geometry, including bodies.
- **Move/Align** – Move or align bodies inside a component within the relative framework of the component.
- **Remove** – Right click on a body to remove it from the design. Compared to Delete, Remove retains the parametric history of the body in the timeline, while Delete destroys the parametric history of the body and all objects associated with it. Currently Remove is only available in a right click menu, when selecting Bodies and Components.



REMOVE IS CURRENTLY FOUND WHEN USING THE RIGHT CLICK MENU

Cautions:

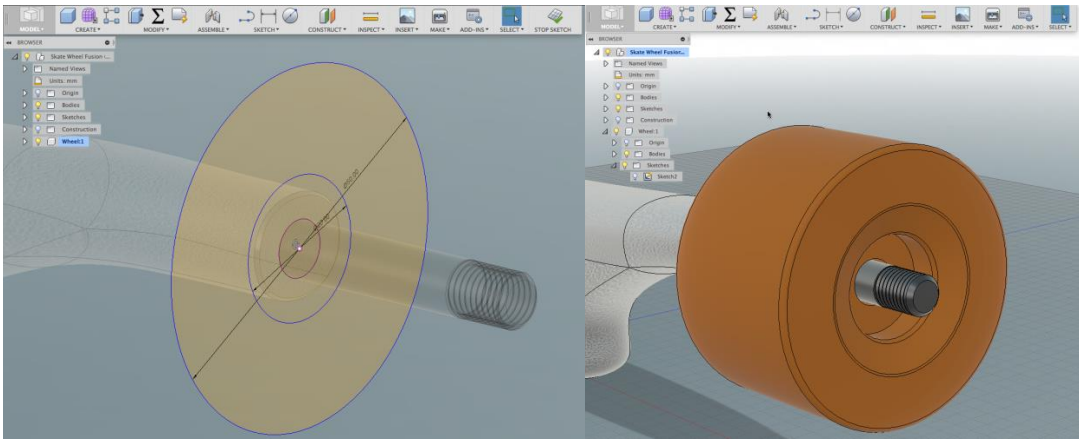
Avoid moving bodies around a lot using Move (physical location) or Cut/Paste (component ownership in browser). Try to design them in place by sketching in place in Active components.
Avoid unnecessary relationships between bodies that reside in different components, such as split body references. Instead use construction planes defined by the origin if possible.
Avoid leaving yellow and red errors in the timeline. Edit to fix errors as soon and as much as possible.



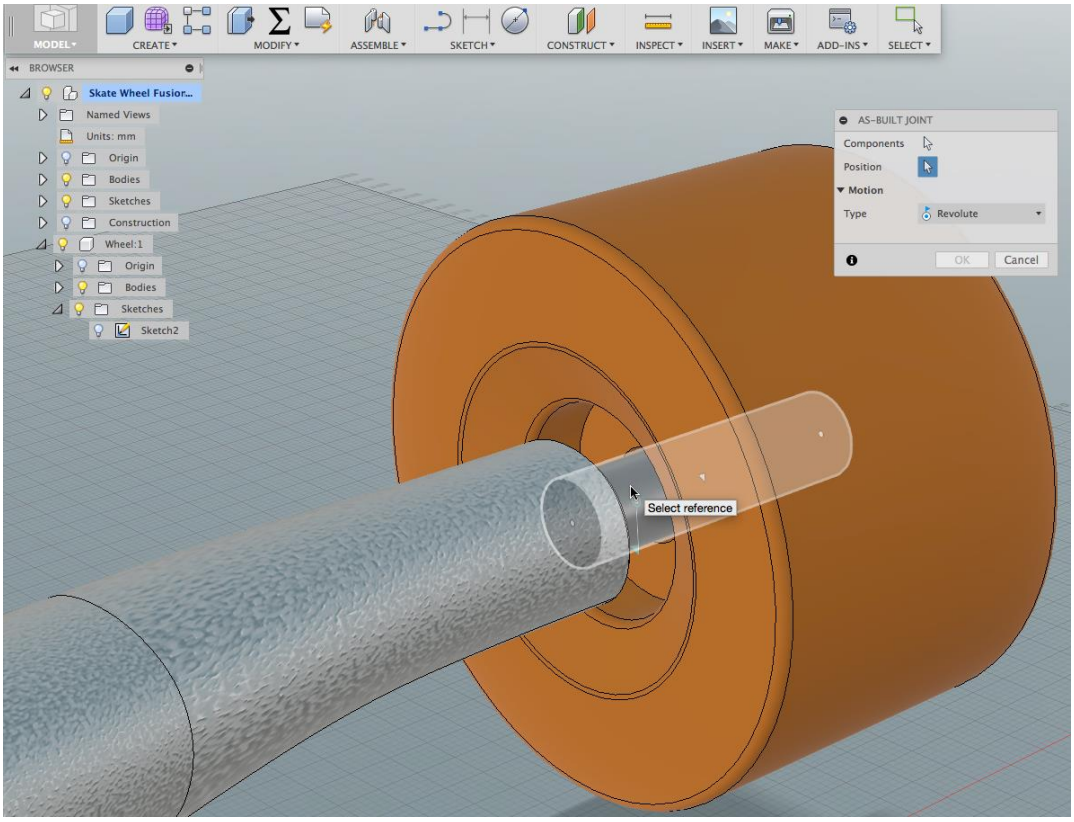
EXAMPLE: TIMELINE WITH ERRORS AND TOO MANY BODY MOVE COMMANDS

Creating components in-place for top-down design

This simple workflow illustrates the essence of top-down design. First a new component is created and then activated. Next comes the sketch, the body, and an As-Built Joint is added to give the wheel motion. The part is created in context of the assembly, with all relevant references immediately available. Moving from concept to details is top-down design.



A: THE WHEEL COMPONENT IS CREATED IN PLACE



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

Options for creating components

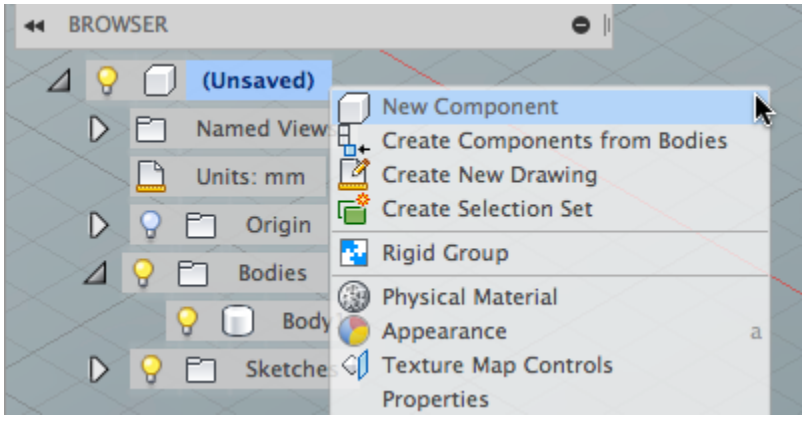
What is a Component? It is a container for bodies, sketches, construction objects, etc. that defines a single part. By default the Fusion 360 browser represents a single component, or part. The top browser node is always considered and referred to as the "root" of the design whether or not it has bodies in its own body folder.

Create components when your design has these requirements:

- **Motion between parts**. Joints require components.
- **CNC.** Manufacture requires components.
- **Drawing.** Drafting requires components.

Additional components may be created from bodies using **Create Components from Bodies** command, or from the new component option in commands like Extrude, from the New Component command, or from inserting linked designs.

The commands Create Components from Bodies and New Component are currently found via right click menu when selecting the root. Individual bodies can be used to Create Components.

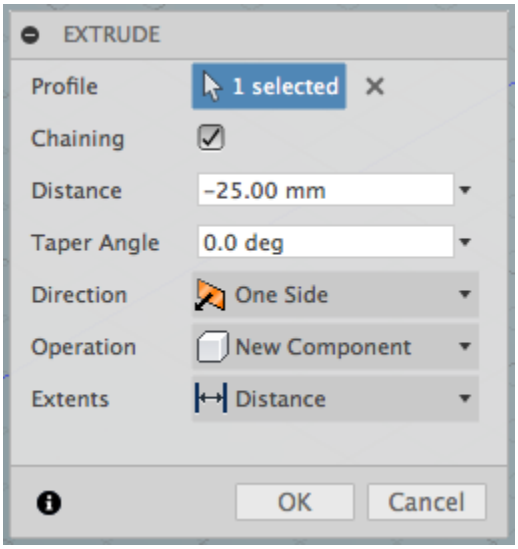


RIGHT CLICK ON THE ROOT TO FIND THE NEW COMPONENT COMMAND

Active Component Workflow:

- **IMPORTANT:** This workflow ensures the parametric history of the component will be completely stored inside the component. Any copy of the component will be parametrically complete, excluding references to other components.
  - Right click the top browser node, pick New Component.
  - Activate the new component. VIDEO
  - Create the sketches and bodies of the component while it's active.
  - Always make the component active when adding more geometry to it.

Caution:
Avoid using New Component from within command dialogs *when your intent is to make copies of this component!* This option is great for a closed design that has motion, hence the need for new components, but the sketch and body remain at the root, which impacts Save Copy As.



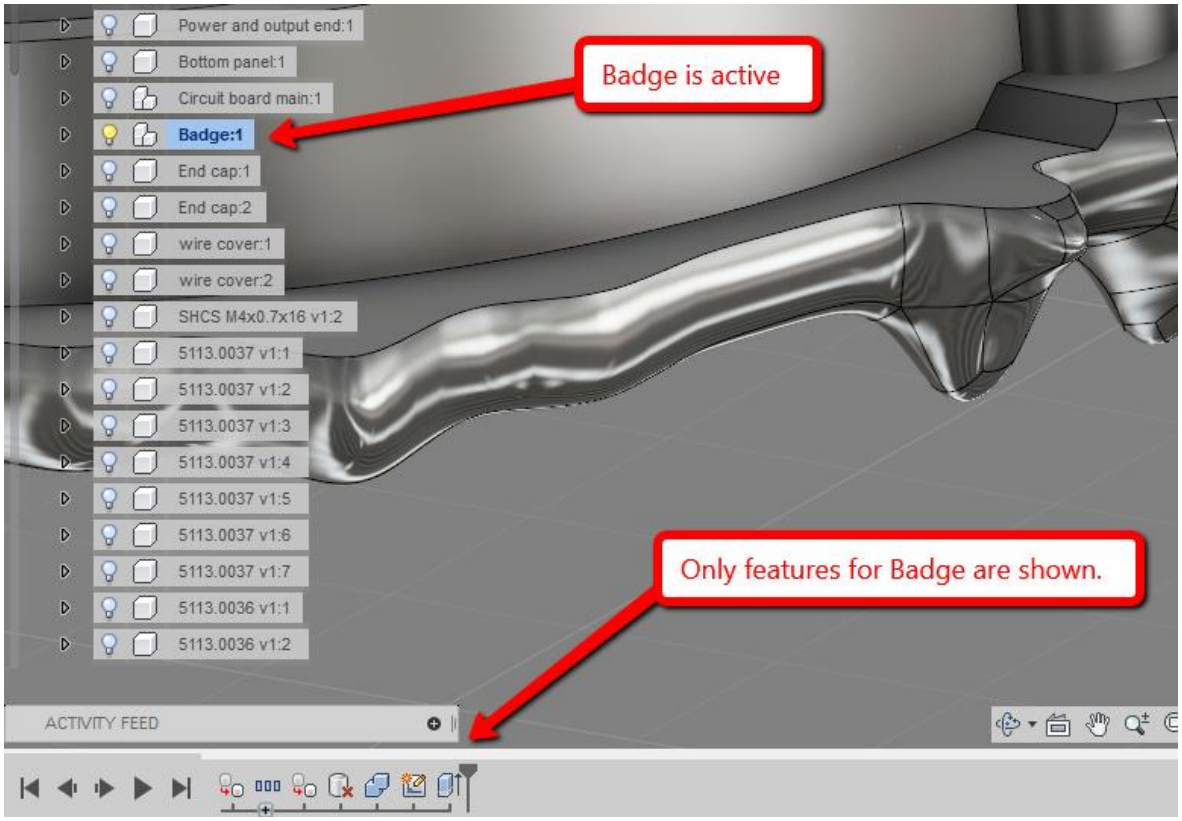
THE NEW COMPONENT OPTION IS FINE, IF YOU DON'T PLAN TO COPY THE DESIGN

Creating Assemblies
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. Components that themselves have sub-components are considered to be assemblies.

When you first make components your single component design becomes an Assembly.

Workflow for assemblies:

- If your intent is to create an assembly of components, it is best to create the components early using New Component, and then create sketches and bodies inside the components they belong to from the start. Use Activate Component before creating more sketches and bodies.
    - ***TIP!*** If your intent is to re-use the components in another design, you must use the Activate Component workflow.
- If you have no need to re-use design, simply using New Component options inside dialogs like Extrude will create a component, and thus your design becomes an Assembly.
- Watch a video demonstration.
    - Activate Component is essential to copy a component!
    - What happens when you don't activate a component? (that is later re-used)



THE TIMELINE ONLY SHOWS THE ACTIVE COMPONENT FEATURES

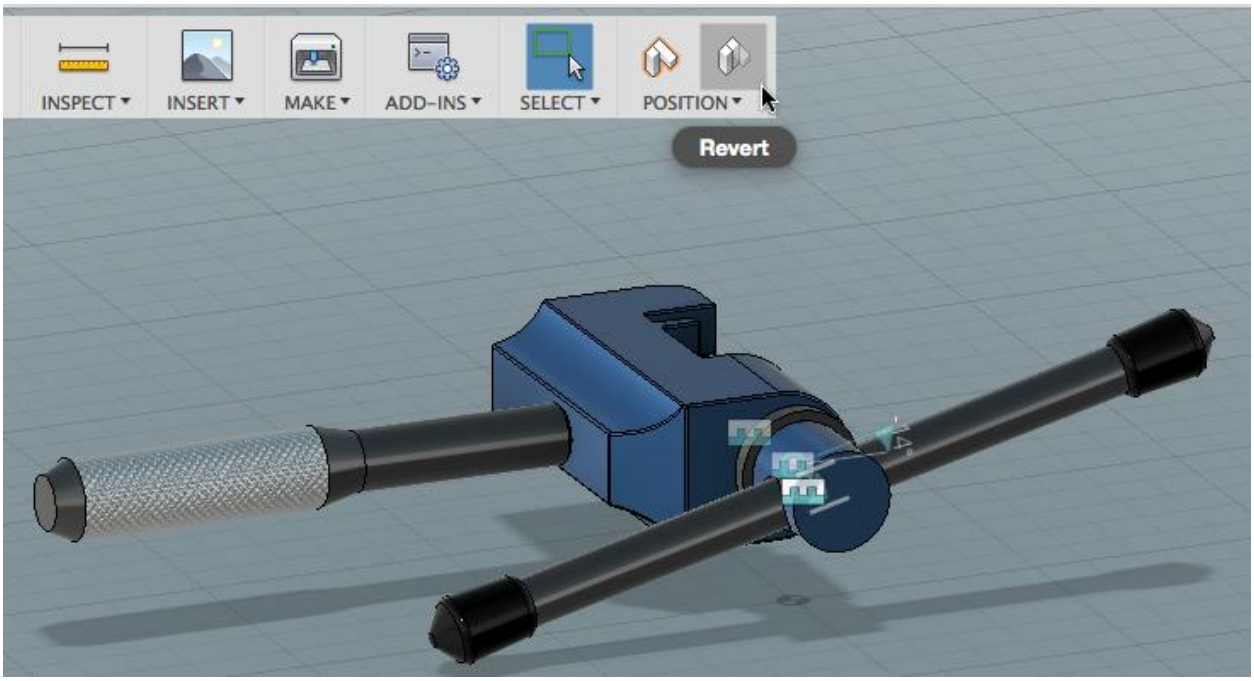
Related commands:

- **Activate Component** – When a component is active, most* new objects such as sketches, bodies, construction, joints, etc. are made inside and relative to the active component. VIDEO
    - ***TIP!*** Only the timeline entries for the active component are displayed while it's active.
- **Create Components from Bodies** – Right click on a component with one or more bodies to convert them all to components, or right click on a single body to convert just it. This instantly turns your single component design into an assembly.
- **New Component** – Right click on the root (top) of the browser to access this.
- **Joint** – Defines motion and alignment between components.
- **Move/Align** – Move or align components to create a Snapshot.
- **Snapshot** - Recorded positions of components at points in the Timeline history of the assembly. Only appears in toolbar when a component has been moved.
    - Example: when the position is required to perform modeling or analysis.
    - Use Revert to discard an unwanted Snapshot. See illustration below.
- **Remove** – Right click on a component to remove it from the design and also retain the parametric history of it.

*Analysis and Contact sets go to the root. Canvas goes to the component it sits on.

Cautions:
Avoid moving components around a lot. You really shouldn't unless the alternate position is required by the design! Otherwise you get extra and unnecessary Snapshots that add to compute time and clutter the timeline.

TIP! Try to design components in place by sketching in place, avoid unnecessary Snapshots.
TIP! When a component has several snapshots between two distinct positions, you can delete all but the last one to prune the timeline.



AFTER DYNAMIC ANALYSIS, USE REVERT TO DISCARD THE UNNECESSARY POSITION

Editing bodies
It's important to realize that any edit of the shape of a Component is a body edit. Components are just the containers for bodies, bodies are the containers for shapes. For this example, two distinct workflows are described, in-place edit and Derived Part workflow.

In-place edit workflow

- If your intent is to edit the original definition of the body, find the sketch or feature that created it and either double click or right click to edit. The feature or sketch will be in the midst of the timeline. In this case the sketch is found in the browser: VIDEO

TIP! **T**he objects will be edited at the time and location they was created. So if you have used Move or Joints you may need to use Find in Canvas to locate your object.

Derived part workflow:

- If your intent is to create variations of components based on a shared starting point, use the **Derived Part** workflow.
    - This is achieved by copying a body that is partially developed into assembly sub-components.
    - More work is then done on the assembly components, making them unique.
    - Changes to the original body are made using Timeline. Go back in time and edit the original body. The downstream components will all update. Thus, you have full control over the original body and the variant bodies using the Timeline.
    - VIDEO

Editing components

The key to working with components is realizing they are parametric containers. You need to preserve the parametric integrity of the component by making it active when you edit it. This forces all the changes to happen inside the component, any new geometry is created inside the component.

Why? Components are parts. Parts get re-used. Parts are re-purposed. You need the components to carry all the critical parametric information when you save copies of them for use outside the current design. Parametrically incomplete exported components carry only cache data and may not perform the same outside their home assembly as they do inside it.

Another reason is ease of edit:

- When Activate Component is used, the timeline only shows the portion required to edit the active component. This is a huge benefit when your design is large. Don't spend time sifting through the timeline! Activate component does it for you.
- Sketches that are created inside components stay with the component when it moves. When you edit the sketch, you don't have to go looking for its original location.
- VIDEO

Workflows:

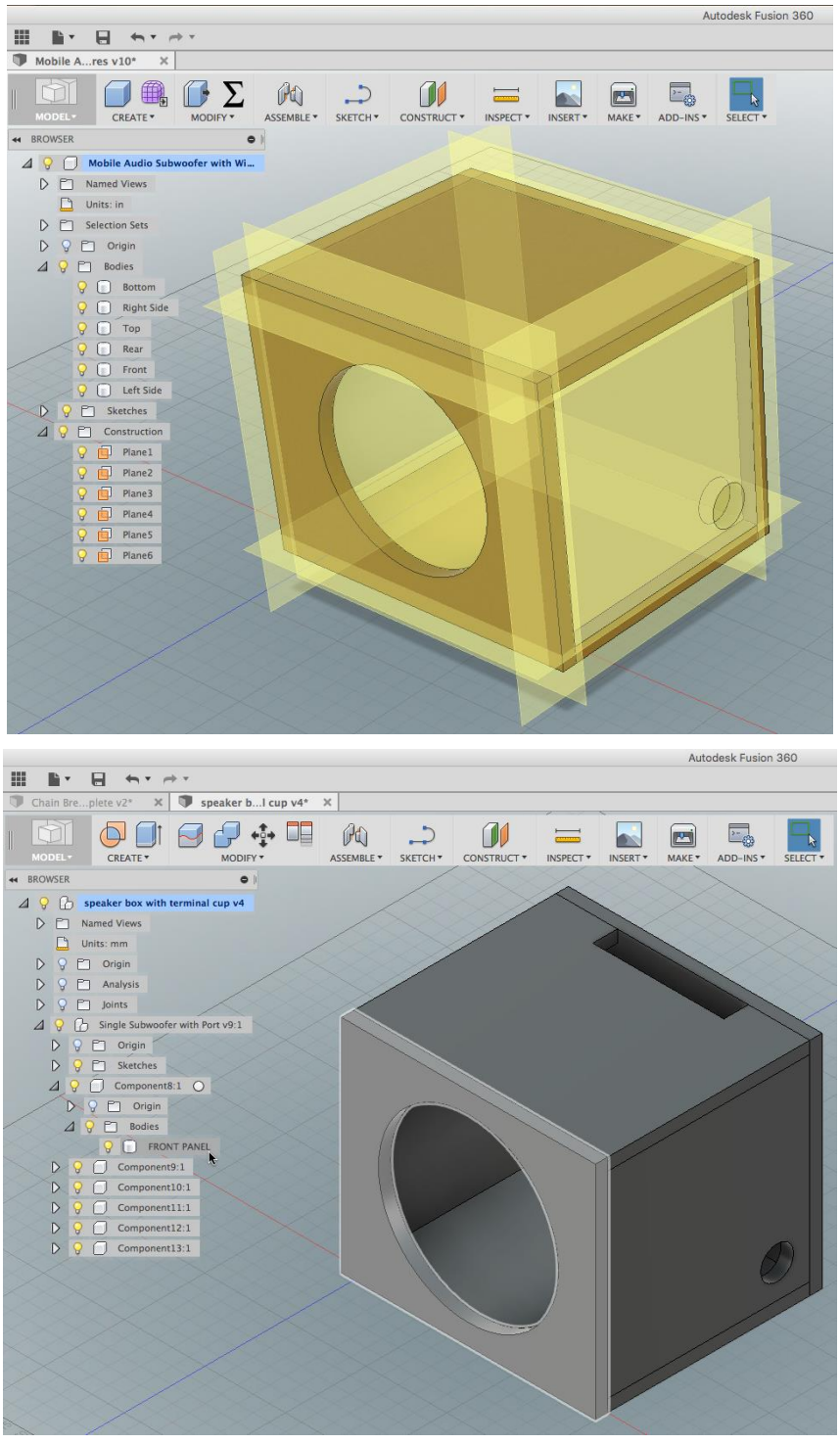
- Activate the component to filter the timeline for that component history.
- Edit the features or add new features.

Multi-body workflow

Multi-body modeling refers to the practice of creating assemblies first as a collection of bodies; with the intent to turn the bodies into components perform assembly tasks once the bodies are fully defined.

The benefit is that sketches and bodies are easier to work with early in the design, prior to the disruptive assembly work of adding Joints, creating motion and moving components in the assembly. This approach also plays into a Skeletal workflow, where bodies are created and driven by a central sketch, only later to become components.

Use the timeline to develop your design in two stages: prior and after the change to components, this is your dividing line.



THE MULTI-BODY ASSEMBLY FIRST AS BODIES, SECOND AS COMPONENTS

TIP! *The dividing line between multi-body part, and assembly, should be the completion of the design before requiring Joints, CNC, or drawings, which all need components.*
TIP! *Move the timeline marker back in time before this divide to edit the multi-body part.*
TIP! *Move the marker to the end to update the downstream assembly components.*
TIP! *Pay close attention to any failures, they need to be fixed immediately.*

Workflows:

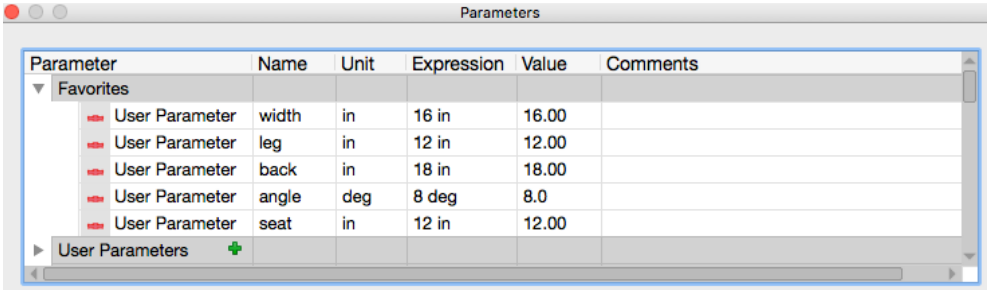
- Create sketches and bodies in a single component.
  - If starting with a new design, Model bodies and sketches at the root.
  - With an existing design, create and activate a component. Model bodies and sketches inside that component.
- Use **Create Components from Bodies** command to turn the bodies into components. This is done when the following requirements arise and you can't go further without:
  - Motion. Joints require components.
  - CNC. Manufacture requires components.
  - Drawing. Drafting requires components.
- **IMPORTANT!** Remember to use the Timeline marker to roll back time for body edits, and to pay attention to downstream failures after rebuilding the remaining model when you move the marker back to the end.

Skeletal workflow

A commonly discussed top-down workflow is skeletal modeling. This term refers to the creation of a skeleton, often based on sketches, that is given the task of positioning the bodies. Edits to the skeleton drive downstream features. The skeleton does not have to be entirely sketch based, for instance construction objects can also participate. Skeletal can also be Multi-Body, but it's not required.

Workflow: Create Skeletal

- Create user parameters that drive your design.
- Create sketches and construction objects that are driven by the parameters.
- Create bodies that are made from the sketches. Add features.
- Only when absolutely necessary, convert the bodies to components.
  - See Multi-Body workflow instructions for "when" and "how".
- Create the remaining assembly details.

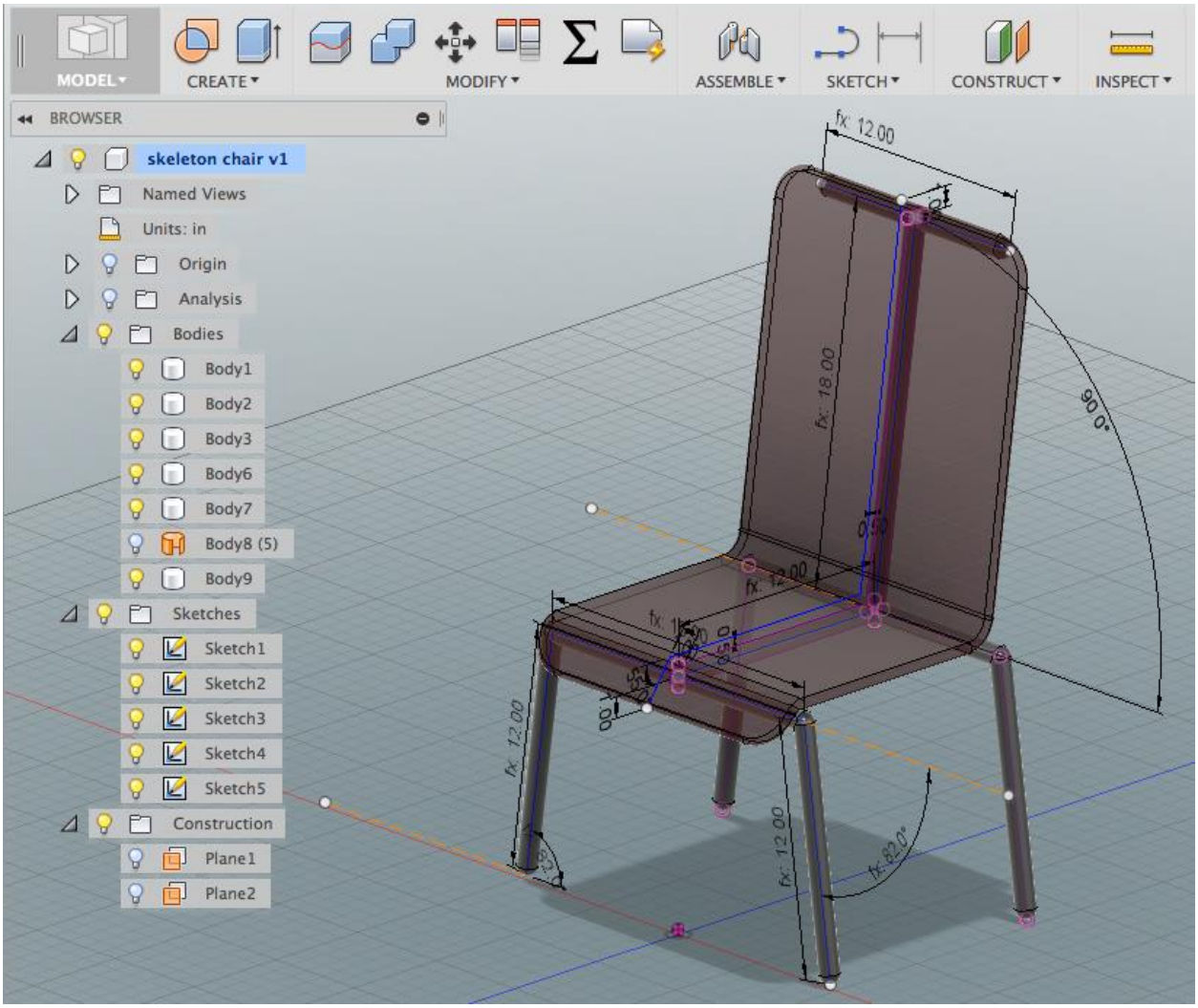


THE USER PARAMETERS OF THE CHAIR MODEL DRIVE THE SKELETON

Example:
In the example of the chair design these parameters are the core of the design. They determine everything about the size and shape of the chair, where bodies are, etc. This is a skeletal design because these requirements are independent of the remaining details. The remaining details rely on and are driven by the skeleton definition.

Workflow: Edit Skeletal

- Use the Parameters dialog to change the parameters. The compute is automatic.
- Or use the visible skeleton sketch to change the parameters. Use Compute All to see the result.
- Fix any errors that occur. Seek to create downstream sketches and features that are not brittle under editing. Work to edit and update all features cleanly, in reasonable limits.
    - Fully constrained sketches are a great place to start.



THE SKETCHES DRIVE THE DOWNSTREAM FEATURES

Related commands:
Change Parameters – This is the parameters table. Found in the Modify menu.
Show Dimensions – A right click option for sketches in the browser.
Compute All – Use to see changes when visible sketch dimensions are edited.

TIP! When Sketches are used exclusively they offer a visual "time tunnel" or wormhole to the earlier history in the timeline. Why? Because they can be made visible, the dimensions made visible, and edited *without going back in the timeline*. You get to reach back in history to make changes in real-time! Use **Show Dimensions** when right clicking on a sketch.

Bottom-up workflows in Fusion 360 modeling
We defined Bottom Up design as starting from "known", then moving on to create the "unknown" to complete the design. Modular cubicles or retail racking is just one example. The shapes are all known parts. Electronic components that populate circuit boards are another example. These come from a library of components you will likely use over and over. Arranging them inside Fusion and then creating a model or drawing of the layout is your job.

Fusion is a synthesizer and aggregator of CAD data, so getting design data into Fusion for re-use, or re-purposing is a core strength. Here are some of the ways to use existing designs in Fusion 360.

Uploading CAD data for reuse
Your existing designs are valuable. You have already paid for them. Why not use them again? If it makes sense to reuse designs, Fusion 360 can translate 2D and 3D CAD data from most formats, all you need to do is upload it. The next section "***How to upload..."*** goes into detail about the workflow.

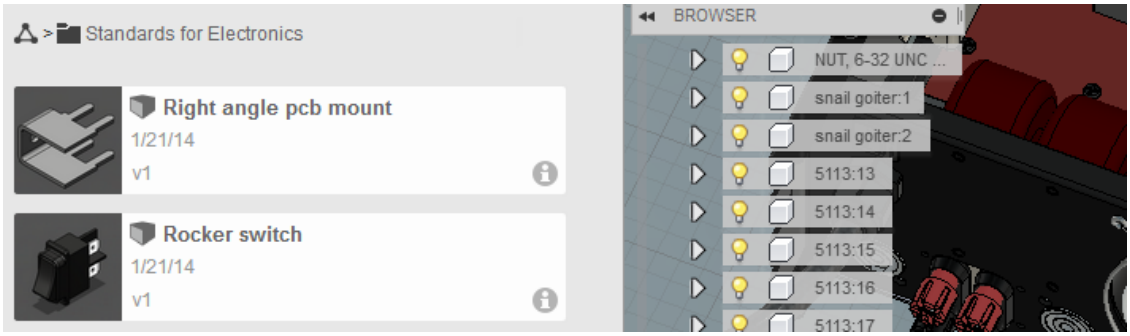
Distributed design
Like most CAD systems, Fusion 360 allows you to insert components into your designs. These components can be used in multiple designs as referenced or linked parts. Linked parts reduce the CPU load when modeling compute happens. The linked files are not computed!

If you ever make a change to a linked file, all assemblies that reference it will indicate they are out of date and an update is needed when you open them.

If you prefer to keep the old definition, or pick any version from the linked design history, right click on the browser node and pick Break Link or Choose Version.

Basic workflow for Bottom-Up Distributed design:

- Using our example of 'retail racking', you would upload or design your standards and keep them as separate designs in your project.
- Insert these files and assemble them with Joints or Patterns.
- **Risks:** it's easy to insert too much information into a design. Performance can suffer.
- ***TIP!*** Use visibility or the Selectable/Unselectable toggle to reduce CPU and GPU load.
- ***TIP!*** Right click on components in the browser to access these commands.



CREATE A LIBRARY OF STANDARD PARTS TO INSERT

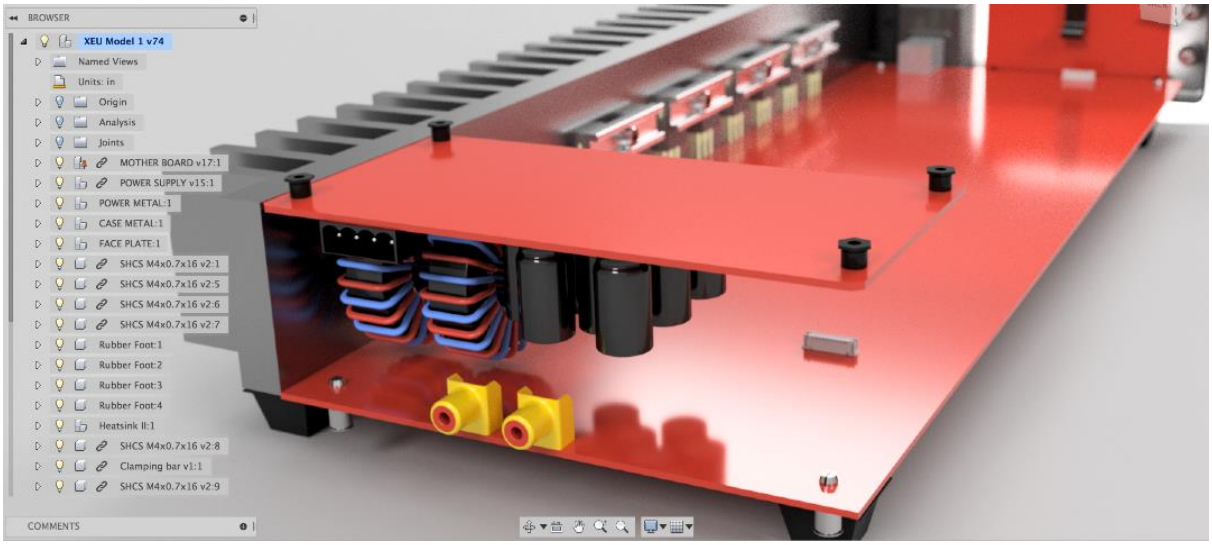
Using Distributed design versions as configurations

Since Fusion 360 keeps a version history of every component, and inserted linked files have instant access to all the version history, you could use design versions as configurations!

There is only one catch, the result will be disconnected to the original file. If you want a different configuration you will have to insert it again and pick the version again. For bottom-up workflows using standard parts this should not be a problem. These are library parts, you probably don't need to edit them at all.

Using Bottom-Up workflow for configurations

- Create your configurable part. Save the version with a tag. You'll need this later.
- Edit to define a new configuration. Save the new configuration with another tag.
- Insert the configuration design into your assembly.
- Right click on it and use Choose Version. The tags you made will help you pick.
- ***KEY STEP!*** Right click on the result and **Break Link**. You don't want to accidentally update it later when using Update All for other inserted design changes that you need.
- VIDEO



MOSTLY INSERTED COMPONENTS FROM A LIBRARY OF STANDARDS

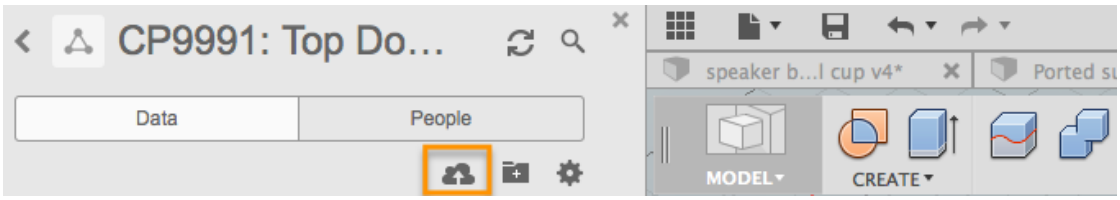
How to upload and use various design data: parts, assemblies, dxf, and images

Get your data into your designs

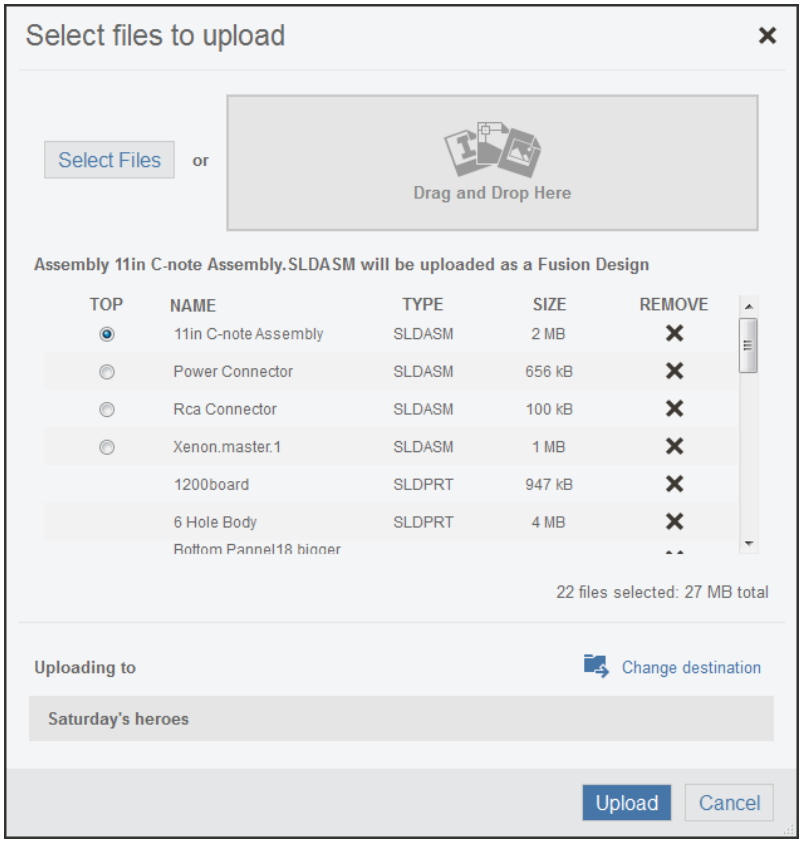
Fusion 360 can consume almost any 3D CAD file including assemblies and parts, 2D CAD, images, and mesh. Each kind of data has a unique purpose in a Fusion design, with unique access points. The intent is to put the access as close to your design workflow as possible. The methods for various data types are described here.

Uploaded Assemblies

Uploading your assemblies inside Fusion is easy.



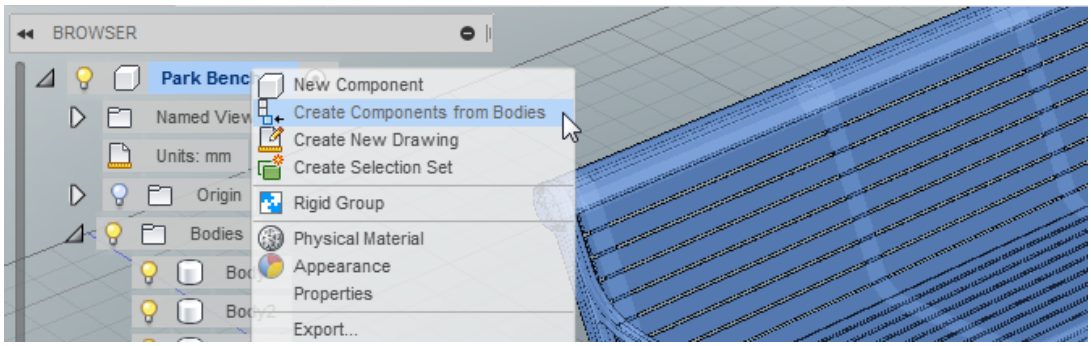
For Autodesk Inventor or SolidWorks assemblies it's helpful to start with a flattened data view. You need to select all the parts and assembly files at once. Fusion will prompt you to select the top level assembly.



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.

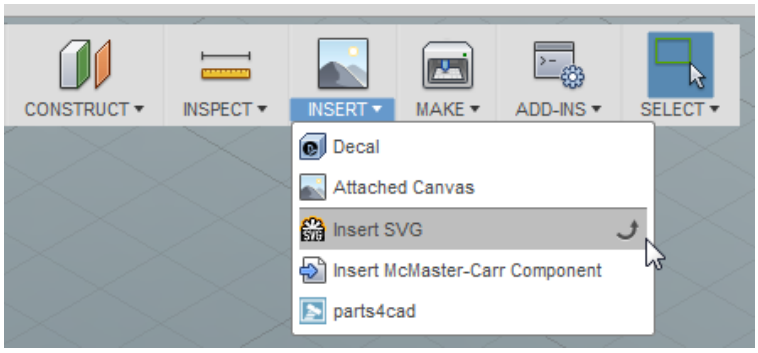


AFTER INSERT OF SAT ASSEMBLIES, USE CREATE COMPONENTS FROM BODIES

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.

DXF and SVG input

2D designs that exist as SVG can be inserted into Fusion 360 designs as sketches using the Insert commands.



INSERT SVG IS ON THE INSERT MENU

Workflow for SVG insert:

- Use Insert menu, pick Insert SVG
- Select the plane you want to place the design on.
- Use the folder icon in the dialog to browse for your SVG file.
- You can scale the preview box or use Modify > Scale to scale the sketch afterwards.
- VIDEO

Caution:

Some SVG designs can be extremely detailed. Currently Fusion 360 may have a difficult time with designs that have many line segments and end points.

Workflow for DXF insert:

- Upload the DXF file to your project folder.
- Use Insert from the data panel icon.
- The DXF becomes a sketch in your design.
- Use Modify > Scale to scale the sketch if needed. The SVG workflow video shows the Scale > sketch workflow.

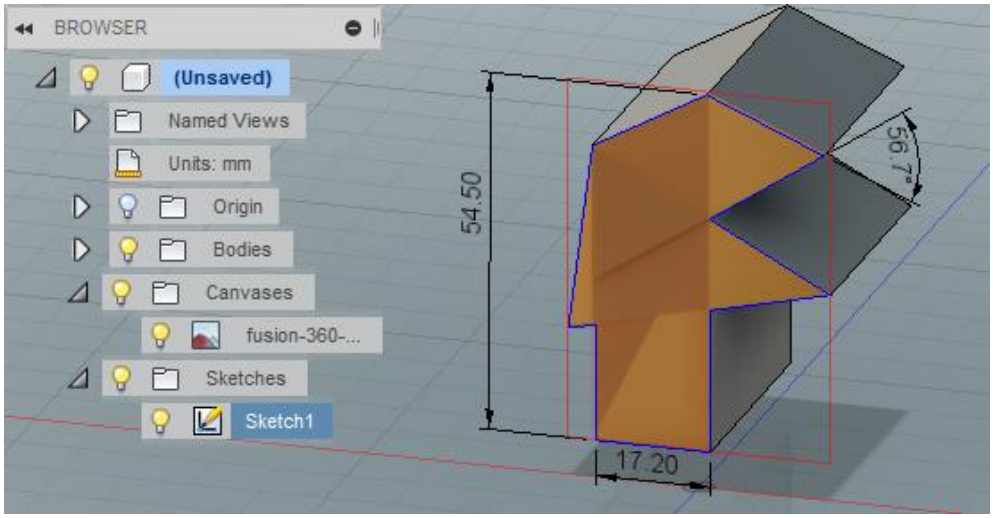
Caution:

- Some DXF designs can be extremely detailed. Currently Fusion 360 may have a difficult time with designs that have many dimensions line segments and end points. Try to use DXF files with no dimensions showing.
- 3D DXF is not supported.

Canvas workflow

The Insert Canvas workflow can be used in top down or bottom up strategy. A sketch of a company logo is a good example of bottom up design with Canvas. Strictly speaking, the logo already exists and may not be changed. It is a bottom up building block, if only for a short while until you create new geometry around it.

VIDEO



THE CANVAS TO SKETCH TO 3D DESIGN PATH

Mesh to solid

There are a couple ways to use Mesh.

- Insert a mesh and model around it using Sculpt or Model tools.
- Insert a mesh, and then convert it to solid.
    - Start a DM (history free) model, or use Create Base Feature
    - Insert the mesh
    - Use Modify > Mesh > Mesh to Brep.
    - VIDEO

How to prepare and use components for distributed design

Linked files are read-only

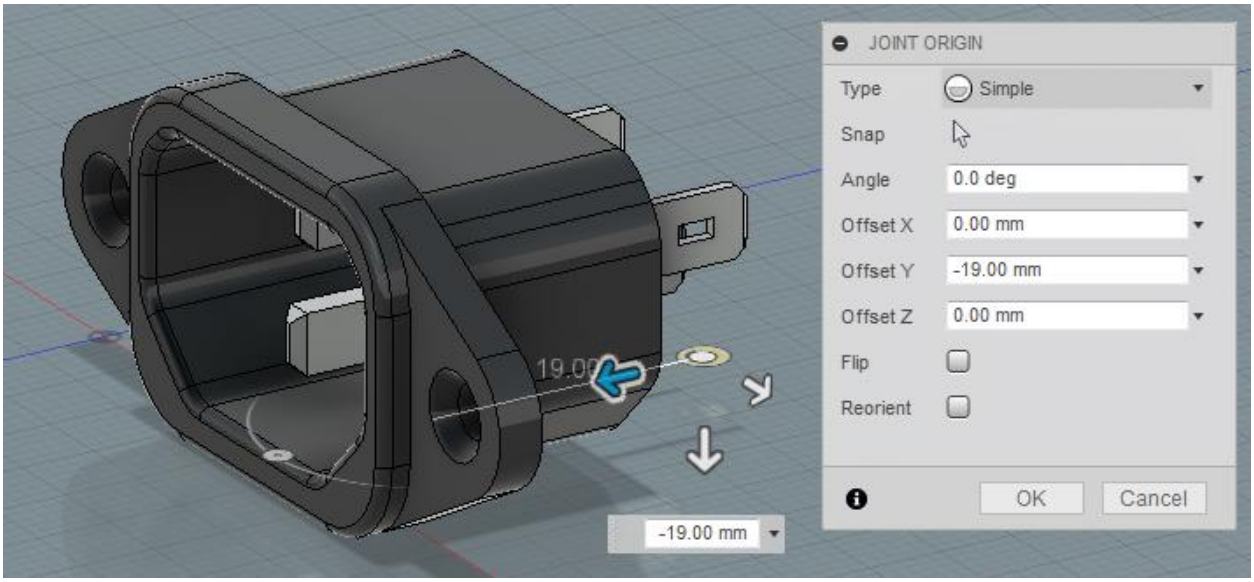
Because linked files are read-only, and because Fusion records versions based on actions like changes in visibility, when you are preparing designs to be inserted as linked designs you should be careful to ensure they are ready for use. Adding joint origins, which aid in assembly, or turning off sketches and other undesirable visual aspects should be done ahead of time.

Without this preparation you would be forced to a) update the linked file, causing another version to be created and b) update the referencing assembly, causing a useless version to be created. Keep the version history clutter free by preparing your designs for insert as linked distributed designs.

Standard Electronic Components

In the case of electronics standards, which are inserted and placed on circuit boards, having a pre-defined attachment point is often handy. It's not required but can be very helpful. The use of joint origins facilitates the assembly process. If you must assemble the part using a joint origin, place the origin in the file before insertion. That way it's available every time you use the library of components.

Joint Origins are particularly useful with circuit board components because attachment points are frequently not on the model body itself. You could use a regular joint, but in this case, Joint Origins just make it all easier.



THE JOINT ORIGIN FOR THIS PART IS ALWAYS IN SPACE BEHIND IT

Jointed Components

- If your component has motion, ensure the joints are all working as expected prior to insertion. Similar to joint origins, you don't want to "re-joint" the mechanism each time you insert it. Define the motion prior to re-use.
- When linked files have objects in the browser such as canvas, sketches, or construction planes that should remain in the same relative positions after insertion, you must use

Joints to attach the child components to the top level origin. This is because the top level origin keeps the canvas and sketch objects in position relative to the child components, so you must attach your mechanism to it via joints in the linked file prior to insertion.

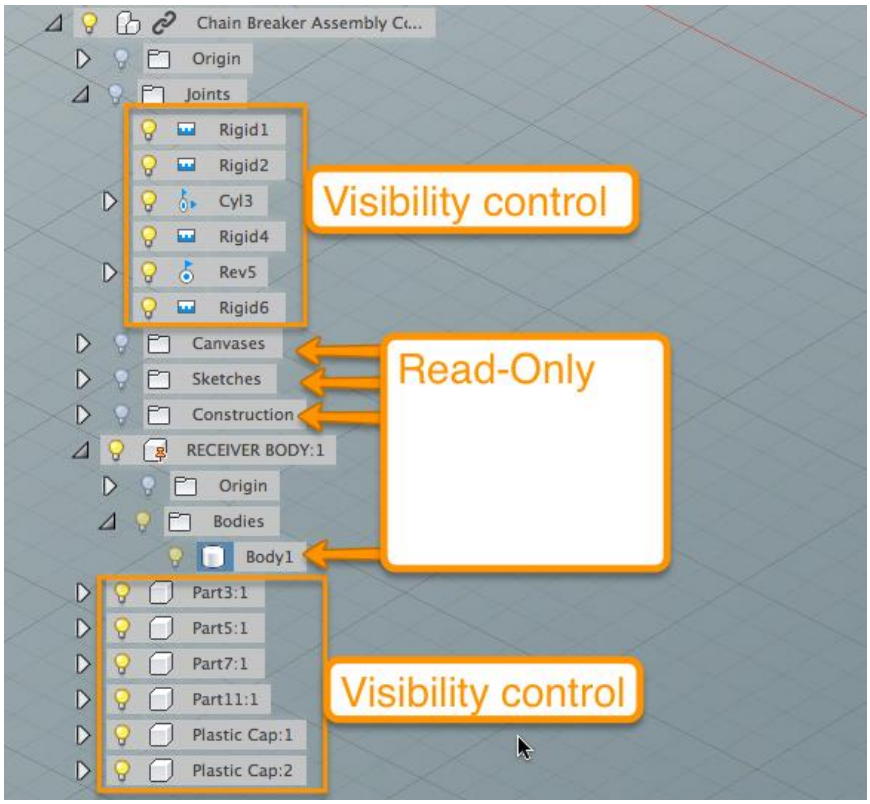
- Grounded child components are not grounded after insertion. The whole design is free floating in the destination assembly.

Workflow for preparing and inserting jointed mechanisms

- If you don't care about the origin or any origin oriented objects in the design to be inserted, then these steps are not for you.
- Rather than grounding a component in the linked file, use a rigid joint to lock it on the top browser origin center point.
- After insertion the linked file can be moved or aligned and then grounded, or jointed into position.
- TIP! Panel mounted components need to bring their own sketches. See video link.
    - VIDEO showing this workflow.

Visibility of key objects

Because inserted designs are read-only, certain objects inside them have visibility controls that are inactive. To keep version bloat to a minimum, consider the following when preparing your designs for insertion into other designs.



JOINTS AND COMPONENTS CAN CONTROL VISIBILITY AFTER INSERTION

Objects that have visibility control after insertion:

- Child components
- Joints
- Top level assembly

Objects that must have visibility defined prior to insertion:

- Origin
- Bodies
- Canvas
- Sketches
- Construction

Objects that do not appear after insertion:

- Analysis

Appearances and Physical Material

Physical material cannot be applied to a linked design. You must use the physical material defined in the design, you cannot change it after insertion.

Appearances can only be changed in linked designs that have appearances applied to components. Unfortunately the default workflow for Appearances, which is to drag them onto the model objects, applies appearances at the body level.

The linked component must have its appearance material applied at the Component level.

Workflow:

- Apply appearance material by dragging the swatch to the component node in the browser. This is a key step.

TIP! Unless you drag the material/appearance swatch to the browser component, it will always be applied at the body level.

Using body vs. component in Fusion 360 design

What, when?

Most of this is covered in the Component and Body defining pages at the beginning of this document. Here are the basics, restated for clarity.

Bodies:

A body is a single container for a contiguous 3D shape in your design. Bodies are also used as modeling tools for Boolean operations like Combine command workflows.

Components:

A component is a container for bodies, sketches, construction objects, etc. that defines a single part. By default the Fusion 360 browser represents a single component, or part. Create components when your design has these requirements:

- **Motion between parts**. Joints require components.
- **CNC.** Manufacture requires components.
- **Drawing.** Drafting requires components.

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