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Working in Fusion 360 for Inventor Users – How and Why

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Learning Objectives

- Understand 2 good situations to use Fusion 360 instead of Inventor.
- Understand 2 good situations to use Inventor instead of Fusion 360.
  - Understand basic Fusion 360 design workflow, and how it differs from Inventor.
  - Model, analyze and machine a component.

Description

As a longtime Inventor user I found Fusion 360 interesting, but also challenging. The workflows were close enough to Inventor that I charged right in, but different enough that I went down many dead-ends. And just how does this cloud storage thing work anyway? But I kept at it enough to the point that I am comfortable in either platform. So why did I bother? Besides the obvious answer that learning new software is in my job description I found that it's a great platform for easily doing many simulation analyses. And when I started exploring the 3-D machining tools I found the same thing applied there. Plus it's easy enough to acquire for home use. So if you haven't tried Fusion, but you know Inventor or a similar platform, spend an hour to learn the workflows, and save many more!

Speaker

I am an Applications Consultant with Synergis Technologies LLC, specializing in mechanical design and analysis. I have worked with CAD/CAE tools for over 30 years as a design engineer, CAD manager, trainer and consultant. My primary focus has been in the mechanical design and manufacturing areas. I have taught design classes at Autodesk University, our own Synergis University and at the college level.

Introduction: What is Fusion 360 and why have this class?

Fusion 360 is a full suite of design and manufacturing tools in a single package. It can do many forms of design, including: parametric solids modeling, surface modeling, free-form solid modeling, sheet metal modeling, mesh modeling, digital simulation, rendering, machining, additive manufacturing, and drafting. It has integrated, cloud-based document management that allows team collaboration.

Note: This is being written in September 2020. Fusion 360 goes through frequent updates and additions to its capabilities. Things that I list here as limitations may go away in the near future. The Fusion 360 development roadmap is available here:

https://www.autodesk.com/products/fusion-360/blog/fusion-360-roadmap-update-2020/

Again, Fusion 360 is a suite of tools. So is Inventor, and so is AutoCAD. They each have their uses. When I was first trying to use Fusion 360, I kept trying to use it just like I would Inventor. That caused a lot of frustration. I would dive into a design concept, then hit a dead-end. I had to throw out all my work and start over. I was using the tool wrong.

I started talking with others I knew that were learning Fusion 360. I watched videos and taking notes. And from them came this class.

I also decided I needed a project that would be fun for me to work on. Something where even though I work with different software tools all day I would also enjoy spending time working on at night.

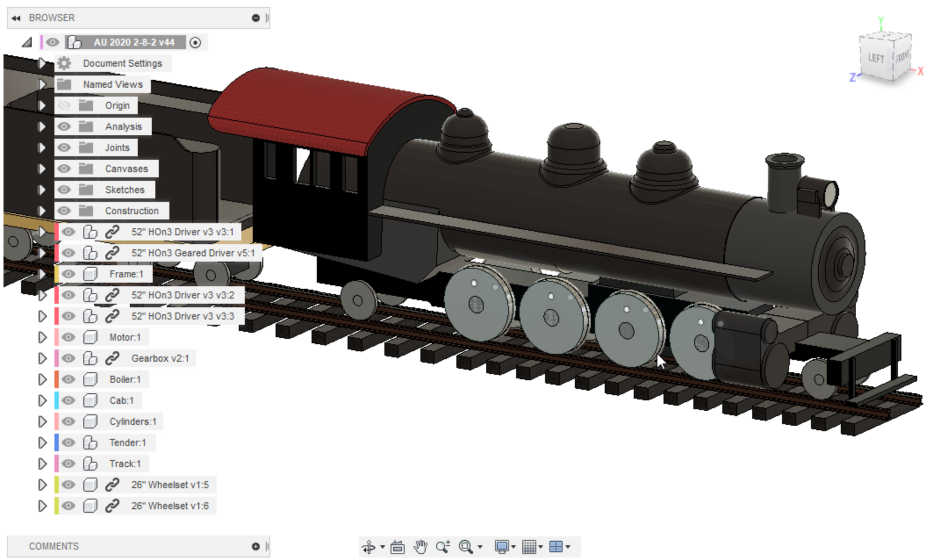
One of my hobbies is model railroading. Coincidentally, one of my favorite railroads is working hard to be operating again in 2021, including restoring a pair of 100-year-old steam locomotives. Since I can't buy models of these engines I decided to design and build my own. I haven't finished my design yet, but I've learned a lot about Fusion 360 as I've worked on it. For the record my model is loosely based on locomotives of the East Broad Top Railroad.

https://www.eastbroadtop.com/

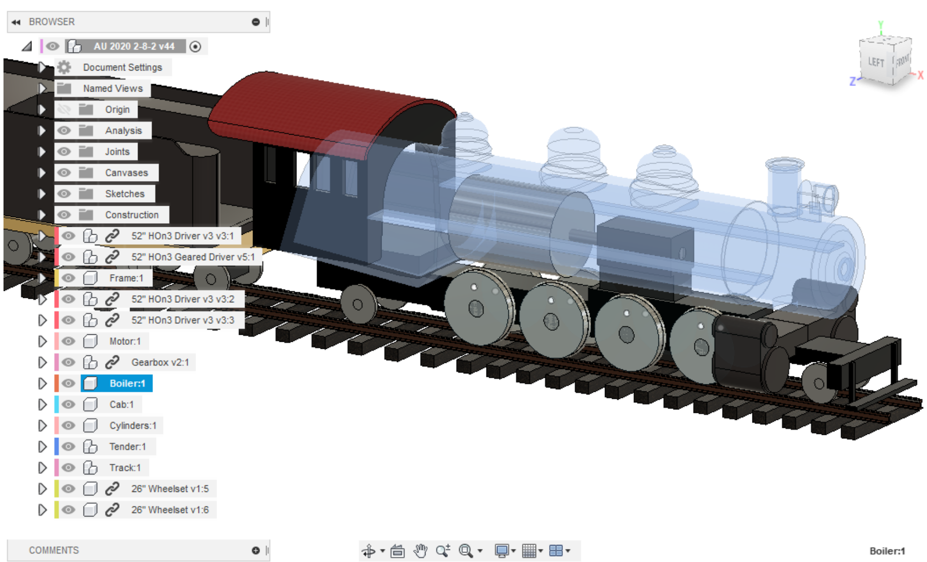
East Broad Top #15

East Broad Top #17

This photograph of #17 was used as the main canvas as I developed my Fusion 360 model.
All photographs by the author, used with the permission of the East Broad Top Foundation.



Three Quarters View of the Steam Engine Model Design



Three Quarters View of the Steam Engine Model Design with the Boiler's Opacity at 40%

Understand 2 good situations to use Fusion 360 instead of Inventor.

Since with the Product Design and Manufacturing Collection I can choose which software to use, when would I choose Fusion 360?

Conceptual Design

I would choose Fusion 360 if I was going to be working on a conceptual design project. I like Fusion 360, combined with Fusion Team, for this fast-paced, small group, interactive scenario. In my case, my team is typically just me, so it is even easier.

Fusion 360 has all the tools I need for my project right there: I can do modeling, simulation, and manufacturing just by choosing different workspaces. For modeling I have similar, though not identical tools, to those in Inventor. If they were identical, I wouldn't be doing this class.

For some design types Inventor has better tools. These currently include tools for designing sheet metal and plastic parts. But to be fair, I probably don't need the more involved tools at this conceptual design stage. I can do my basic design work in Fusion 360 and still have confidence in the viability of the resulting concept. I do have to keep those limitations in mind, but that is true of any conceptual design workflow.

Organic Designs

For projects where the result will be free-flowing shapes or will have a strong human interface Fusion 360 has several advantages. I find the free-form modeling tools In Fusion 360 easier to use than their Inventor counterparts. For concepts that start with scanned information Fusion 360 has more capability to work with the resulting meshes than Inventor at this point. It is also easier to work with concept drawings and digital images in Fusion 360 than Inventor.

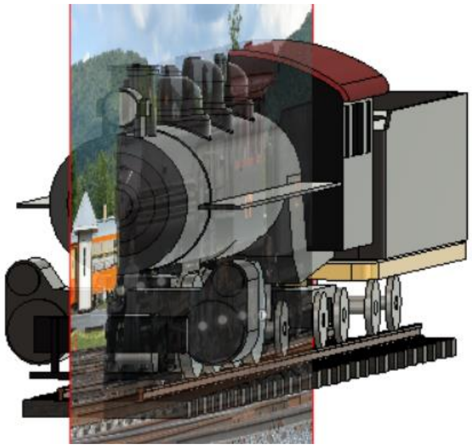
I used a digital photograph "canvas" as the main guide in my design. These canvases can be digital sketches or digital images. Just attach the image, scale and reposition it and start modeling. Here I inserted a digital photo of the steam engine that I am basing my model on. I scaled it and repositioned it to get the origin and size where I needed for the model, then started building the rest of the model around the image.

Here is a link to an example from Autodesk, where a concept of a race car steering wheel is used as the basis for a Fusion 360 model:

https://www.youtube.com/watch?v=thOPmBfAurM

Here is my digital photo, scaled and positioned within the design:

Digital Photo Canvas used as Design Guide



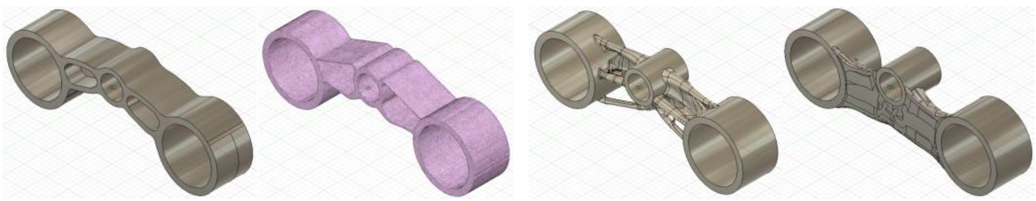
Three Quarters View of the Steam Engine Model Design

Generative Design

There is an entire environment in Fusion 360 that is aimed right at conceptual design, and it doesn't exist in Inventor: Generative Design.

You set up your design constraints, such as loadings and keep-out areas, and desired manufacturing technology. Then the software will develop dozens of potential concepts. From these you pick and choose which concepts you want to investigate further. You can then develop these concepts into your final design.

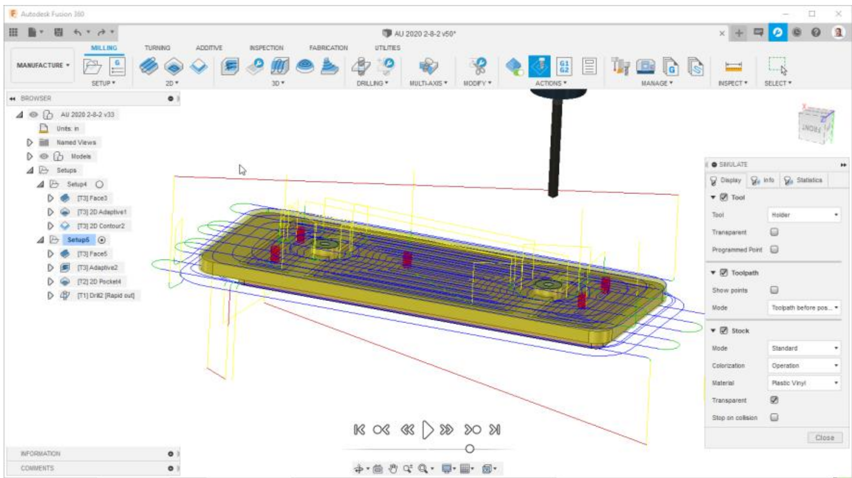
If you haven't seen this in action check out one of the over a dozen sessions on Generative Design in this year's AU alone.



Generative Design Examples

CAM

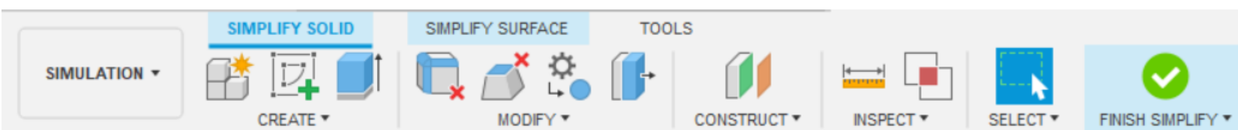
Fusion 360 has a very intuitive Manufacturing environment. It includes both traditional CAM and additive manufacturing capabilities. It is straight forward to learn and operate, but also has quite advanced capabilities. And for conceptual design work the direct link between the model and the CAM package is incredibly convenient. Changes to the design push changes to the tool paths.



Simulated Toolpaths in the Manufacturing Environment

Simulation Work

I would choose Fusion 360 if I was creating or modifying a model for simulation work.
The tools for simplifying geometry prior to starting the simulation are much stronger than their counterparts in Inventor and are built into Fusion 360’s simulation workflow.

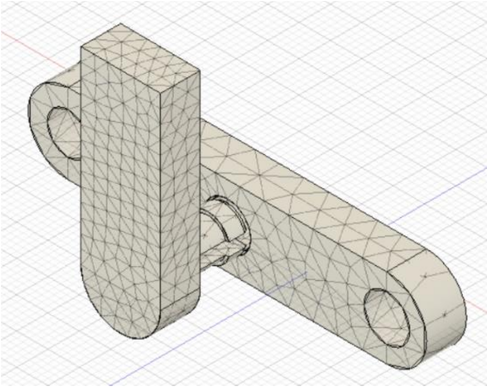


Simplify Solids Environment

These simplification tools include:

- Creating simulation models that include or exclude components.
- Removing features.
- Removing faces.
- Replacing objects with primitive shapes.

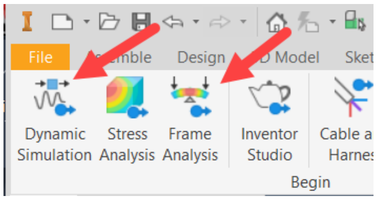
The capabilities of the simulation tools in Fusion 360 are in between those of Inventor and Inventor NASTRAN. It has both heat transfer and non-linear analysis capabilities that Inventor's FEA doesn't have but is much easier to set up than most Inventor NASTRAN studies. Fusion 360 doesn't have all the capabilities that Inventor NASTRAN has, but for most early studies I find I don't miss that extra power.



Fusion 360 Simulation types include:

- Both linear and non-linear static stresses.
- Thermal
- Thermal stress
- Linear Buckling
- Event simulation
- Electronics cooling (preview)

Be aware that there are a couple types of simulation tools in Inventor that don't exist in Fusion 360: Frame Analysis and Dynamic Analysis.



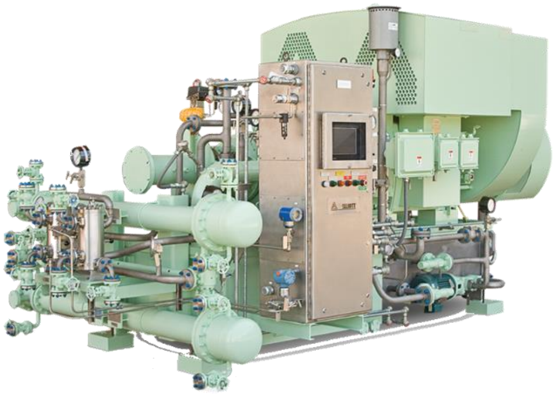
Inventor's Assembly Simulation Tools

Understand 2 good situations to use Inventor instead of Fusion 360.

With all the strengths of Fusion 360 when would I choose Inventor? Here are some keys to when Inventor is a better choice than Fusion 360:

Large Assemblies

Inventor can handle designs with tens of thousands of components. There are established Inventor workflows and techniques for maximizing performance when working in these designs which are outlined in the Inventor Help Topics, under Large Assembly Best Practices.



Specific Design Tools

There are design tools in Inventor that just don't exist in the Fusion 360 world. These include:

- Content Center
- Frame Generator
- Tube and Pipe
- Cable and Wire Harness
- Plastic Part Design
- Injection Mold Filling Analysis
- Dynamic Analysis

There are also tools that are stronger in the Inventor world than they are in the Fusion 360 world:

- Sheet Metal
- Bills of Materials
- Design Assistants
- Drafting tools
- Interacting with Revit
- Families of designs

Autodesk publishes the Fusion 360 Roadmap as a blog. Check it out to see what is coming soon, and what is underway for the not-so-near future:

https://www.autodesk.com/products/fusion-360/blog/fusion-360-roadmap-update-2020/

Understand basic Fusion 360 design workflow, and how it differs from Inventor.

Let's start looking at some of the differences between Fusion 360 and Inventor.

Preferences

- Your preferences are tied to your Autodesk profile.

This means that no matter what device you use to login into Fusion 360 your settings will be consistent.

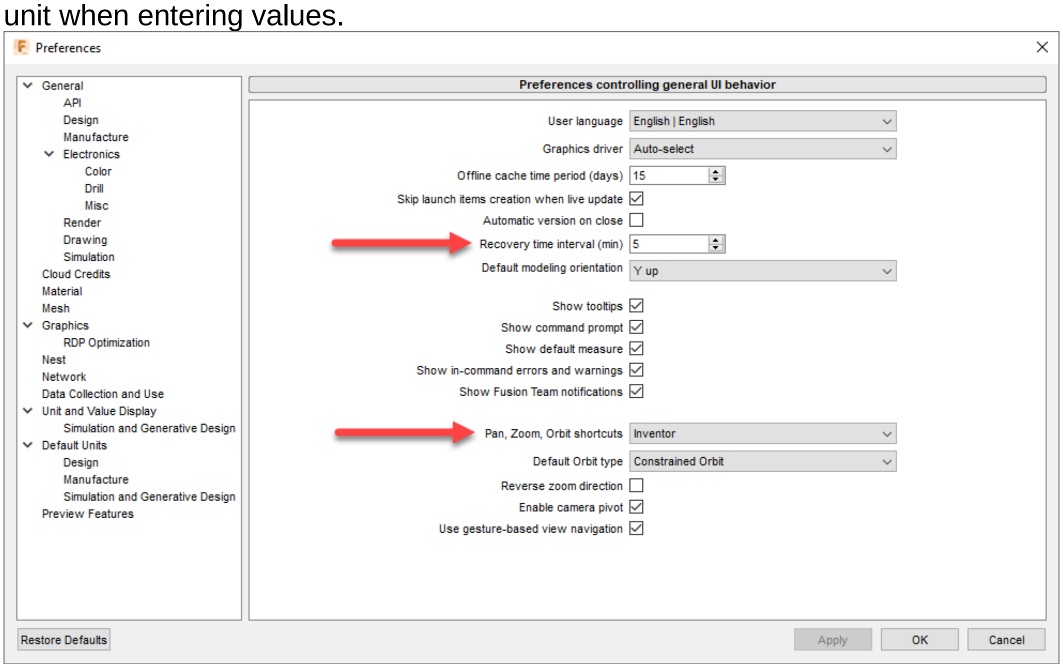
I won't go through all the Preferences screens here, but here are a few that I think are some universal key settings:

General Preferences

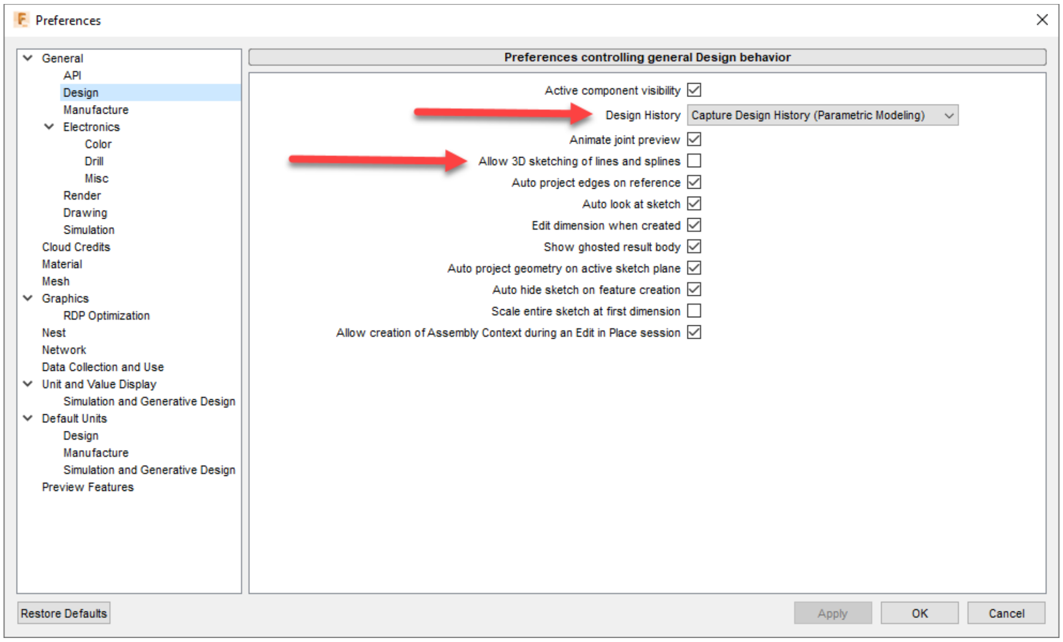
- There is an "autosave" in Fusion 360! It's called Recovery time interval, and the default time is 5 minutes.
- You can set the mouse buttons to behave just like Inventor. You can even swap the zoom direction if you want to.

Design Preferences:

- Turn on History by default. Otherwise you need to remember to do that for each file you start. If you don't have History on you can't readily edit features, just sketches.
  Default Units
- Set this as you wish. You can always change units for any given design and type in the unit when entering values.



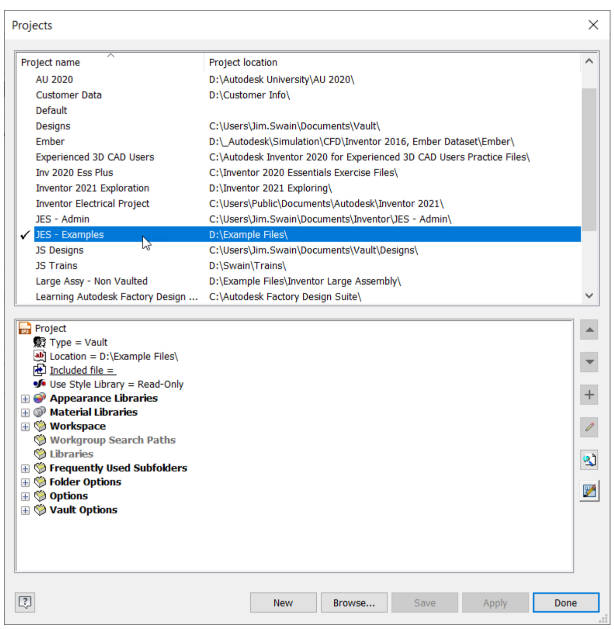
Fusion 360 General Preferences



Fusion 360 Design Preferences

Storage

In Inventor you are always saving to your project locations. Typically, your project's workspace will be somewhere on your physical machine and there may be other network locations set up in your project search paths. If you are using Vault you can check files in and out to have an archived copy saved to the server.

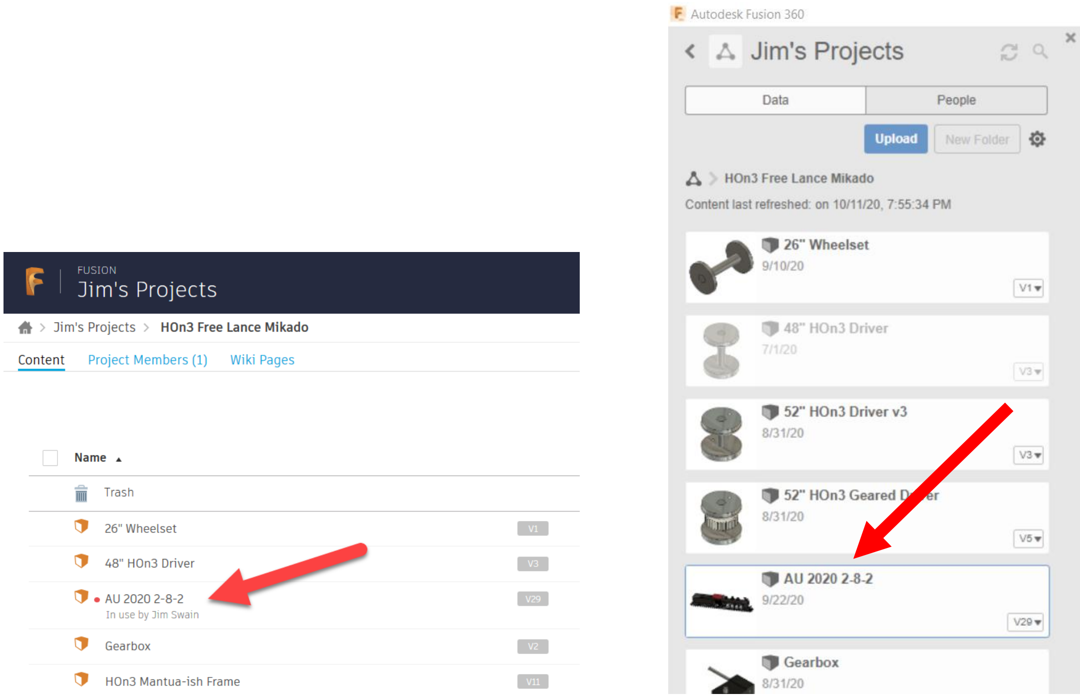


Inventor Project Manager

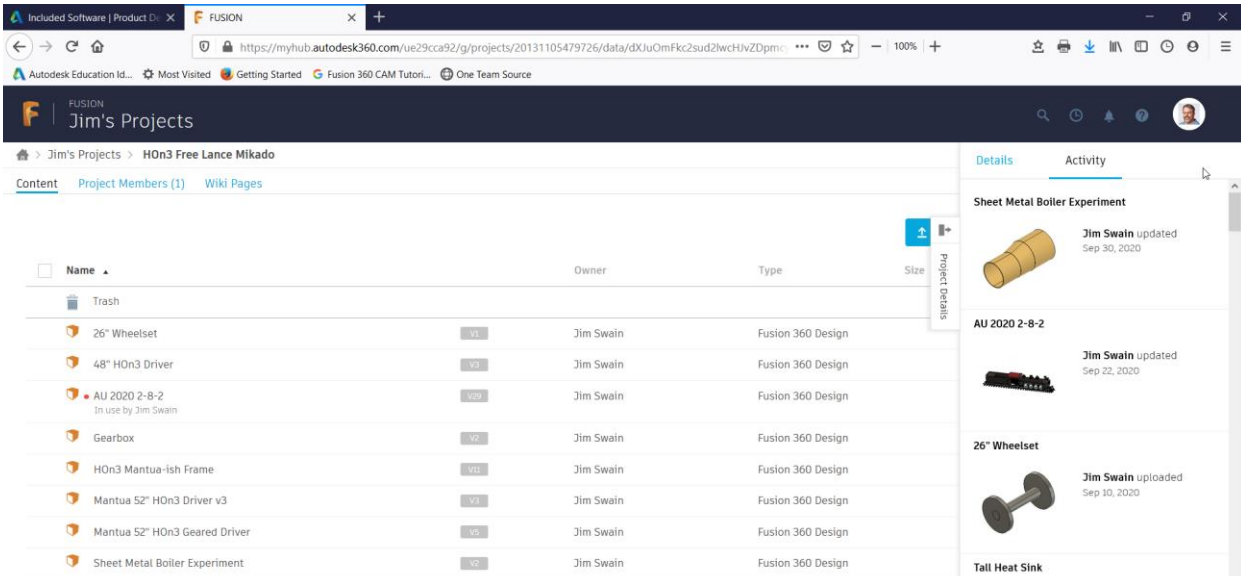
In Fusion 360 a project is a storage location in the cloud. You are always working on a local cached copy of your design, but when you Save you are saving the data to the cloud location. This creates another version of the file in the project's folder. The earlier versions are still in the same Fusion Team folder and could be promoted to the active version if needed.

The design files can be viewed either through Fusion Team via a web browser, or by using the Data Panel in Fusion 360.

If there is no Internet access Fusion 360 can work from the local cached data. When the cloud is available your next Save will push a version to the cloud.



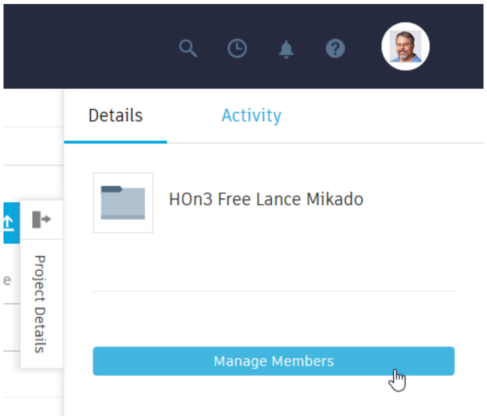
The Design in Fusion Team and the Fusion 360 Data Panel



Overall View of the Fusion Team Project

Additional members can be added to a design project by inviting them, like other cloud storage tools.

Keep in mind that Fusion 360 doesn't allow two team members to edit the same design at the same time. This is the same as Inventor. You will have to divide your design to allow parallel work.



Adding Members to a Project

Saving Locally

One of my favorite tips for Fusion 360 is that you can save to your local cache without pushing a new version to the cloud.

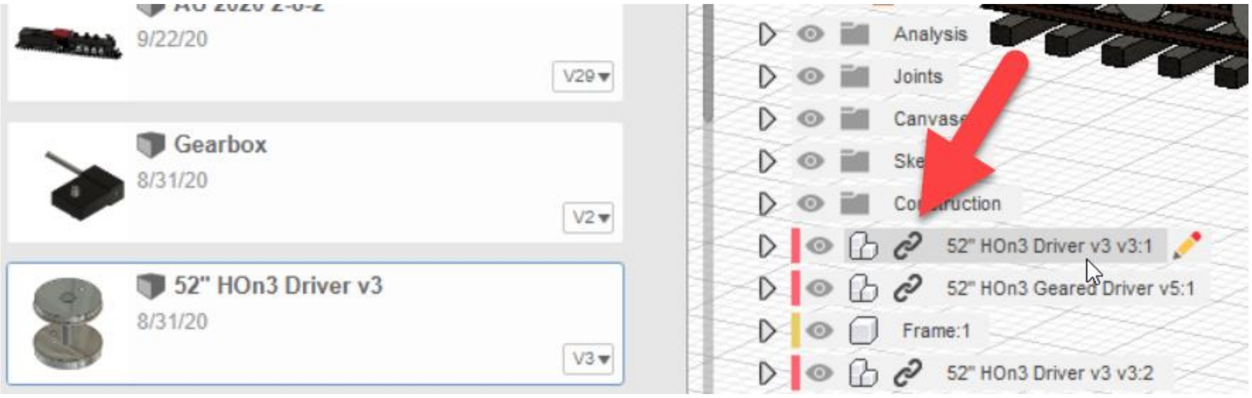
- Use Shift+Ctrl+S instead of the regular Save.

This will save a recovery file to your machine. If your Internet connection is slow, or your design is large, this won't pause everything while the regular Save is writing to the cloud.

Internal components vs. Linked External Components

Typically, in Fusion 360 you are working in a self-contained design, but you can also link in other designs from your project.

- Drag the component from the Data Panel into your design. The component will display a link in the Browser.



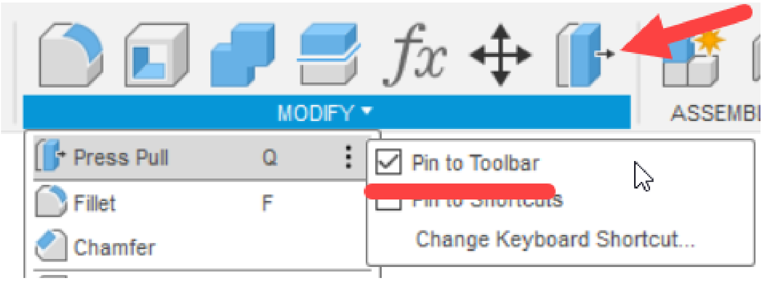
A Linked External Component

Design Workflow Tips:

Editing the Toolbar (Ribbon)

- Adding a toolbar button to by clicking the ellipses.

To add a toolbar button for a given command hover over the command until you see the ellipsis button on the end. Pick the ellipsis, then check the box to Pin to Toolbar.



Adding a Toolbar Button

Starting from an Inventor Design

- Use existing Inventor designs in Fusion 360.

Use File > Upload to upload Inventor files from your machine to the cloud. Individual part files can be directly uploaded, but assemblies must go through Inventor's Pack and Go utility. These articles detail the workflows to bring existing from Inventor or other solids modelers into Fusion 360.

https://knowledge.autodesk.com/support/fusion-360/troubleshooting/caas/sfdcarticles/sfdcarticles/How-to-import-or-open-a-file-in-Autodesk-Fusion-360.html?_ga=2.166420861.298188360.1602890232-1525448009.1516027422

https://knowledge.autodesk.com/support/fusion-360/troubleshooting/caas/sfdcarticles/sfdcarticles/How-to-export-Inventor-assembly-files-and-open-them-in-Fusion-360.html

Starting a New Design from Scratch - Components

- Make a new component right away.

In Inventor terminology, you are effectively doing top-down design in an assembly file right from the start. In some ways the New Component tool is like demoting an Inventor component. By making a new component you can to a different material and to call it out on a parts list.

- Joints connect components, not bodies.

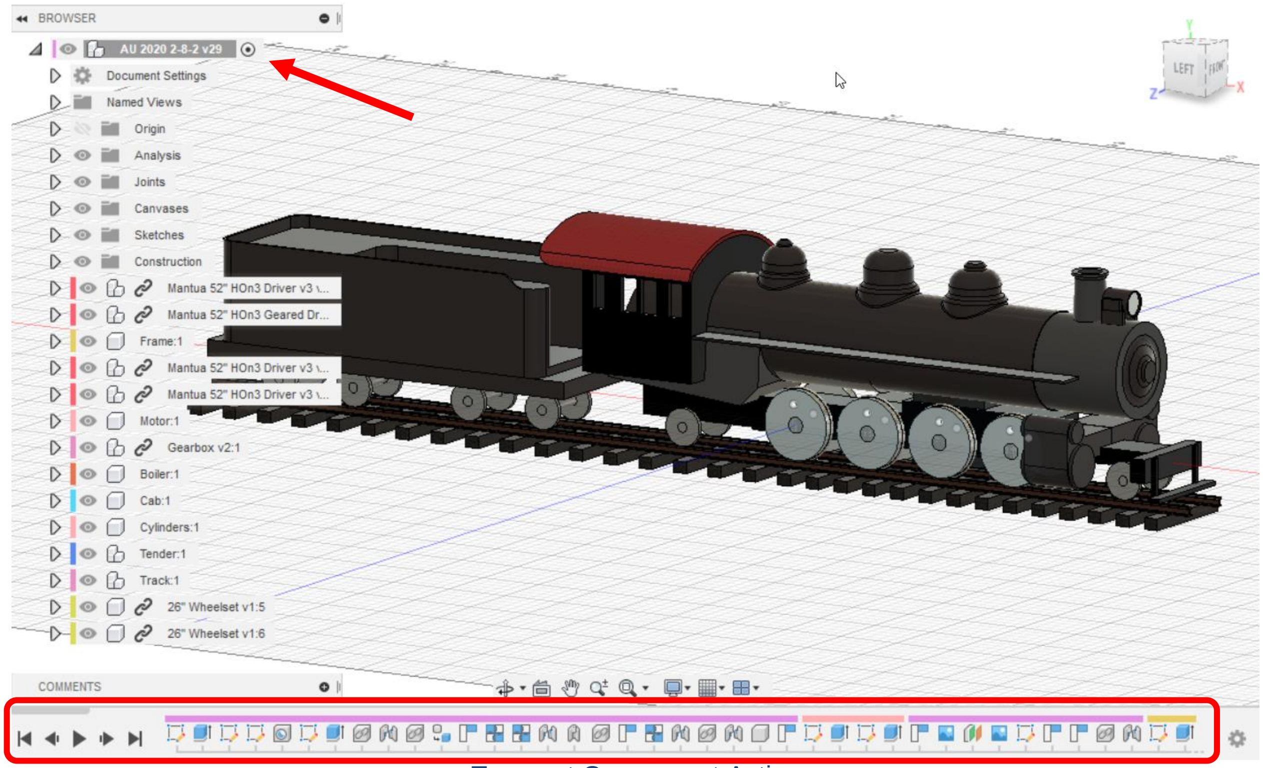
This is critical. If you want to have relative motion between bodies, they must be in separate components. If you want to precisely locate an object with respect to each another object, using the geometry in those objects, they must be separate components.
This key difference between a body and a component probably caused me the most frustration in Fusion 360 until I finally understood it. I tried fixing my mistakes by using the tool to create components from bodies, but the sketches and features were all shown on the overall timeline, not the components timeline. That got very confusing when I tried to go back and edit extrusion lengths and such. The sketches were in the Browser, but the features weren't, so I deleted and recreated objects to force my way through my work. Not the way to run modern software.

- Always keep an eye on which component is activated.

It is very easy to create subassemblies without meaning to. All it takes is to create a new component while an existing component is activated. If you are just trying to make another top-level component make sure the very top node in the Browser is active.

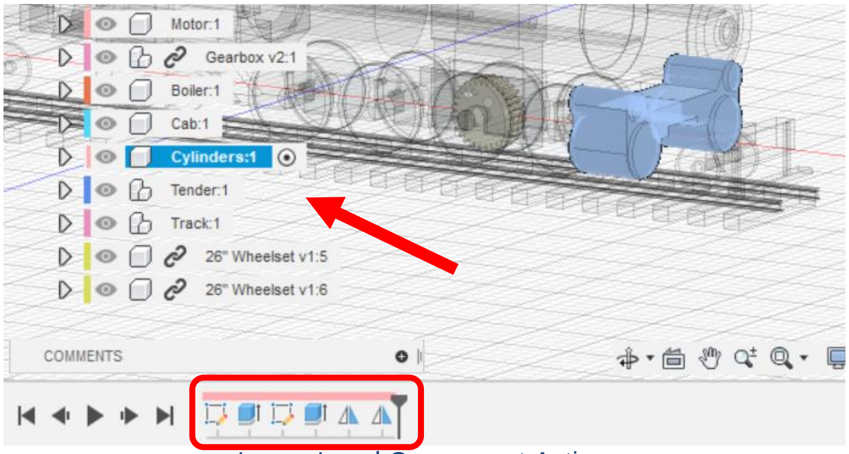
Look at the Design History timelines in the two pictures below:

In the first picture the top-level component in the entire design is active, and the Design History is showing all the features in the entire design.



Topmost Component Active

In the second picture a lower level component is active, and the Design History is showing the features in just that component.



Lower Level Component Active.

Replacing a Component

- There is no “Replace Component” command.

If you want to swap out one part for another you do it by deleting the old one, inserting the new one, then locating the new one as desired.

Locating Components in Space

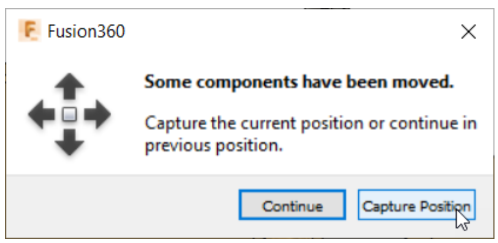
- Joints, instead of Assembly Constraints.

There are no Assembly Constraints, but there are Joints. The Fusion 360 joints work like the ones in Inventor, so if you've been using them for years you may not notice a difference.

If you haven't used them in Inventor, here is a quick guide. A Joint tells a component how it can move with respect to another component in the design. This is pretty much the opposite of Inventor's assembly constraints, which tell a component how it is glued to another component. Note that for a joint you need to pick the component that has to change position first, then pick the one that is staying in position second.

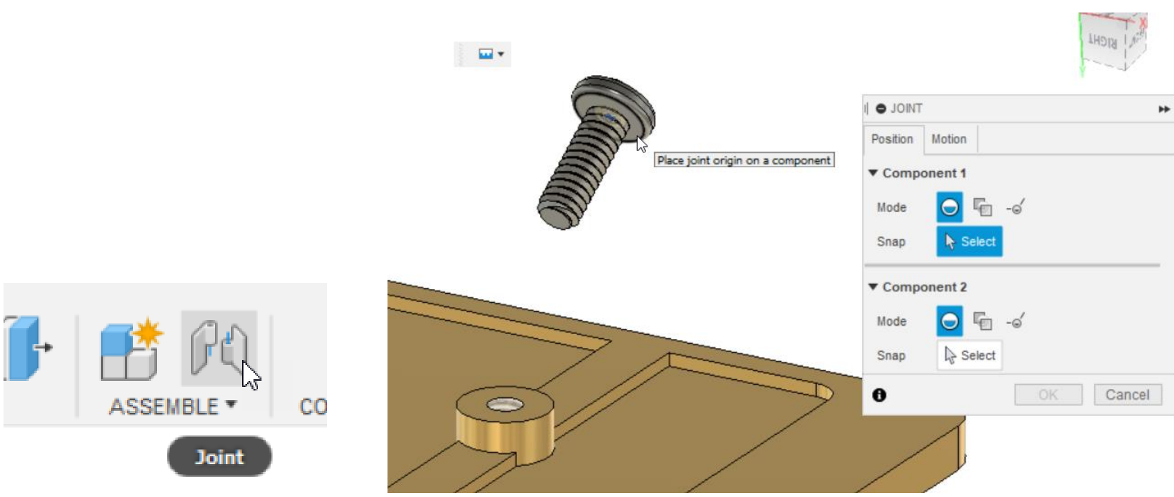
Generally, a single joint can do the job of three assembly constraints. If you keep thinking about that you might find yourself using joints back in Inventor assemblies, and there is little wrong with that.

If you have moved components from their initial placement point before adding the joint, you will get this message:



Capture Position Dialog Box

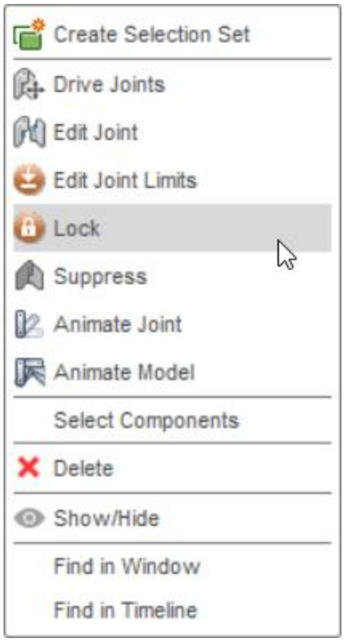
Take the Capture option, otherwise the component will be put back in the original position.



Adding a Joint

- Joints can be locked to freeze components in place.

If a joint normally allows motion between the components, then that joint can be locked. Right click on the joint in the Browser. The joint can be unlocked later, allowing whatever movement that joint permits.



Right Click Options for Joints, Including Lock.

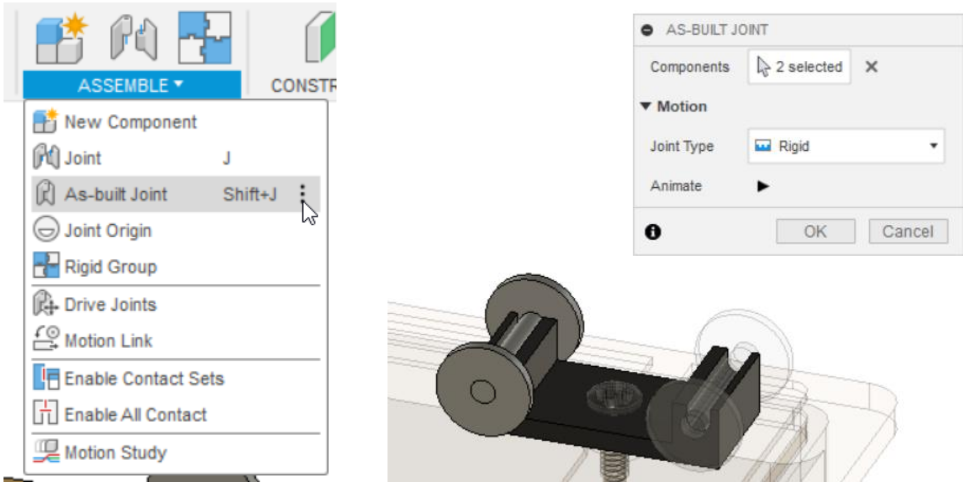
- Use As-built Joints to fix components to each other.

This will quickly fix components in their current position to each other. This is very useful when components have been built in their final position. Any of the joint types are available through the As-built Joint tool.

- Use a rigid As-built Joint instead of Grounding for a Subassembly.

Use the rigid As-built Joint to lock a component to the top level of an assembly design instead of grounding that component. Grounding only works within the context of the current design. If a design is used as a subassembly within another design any grounding in the subassembly is ignored. Instead it acts as if the subassembly was never grounded at all.

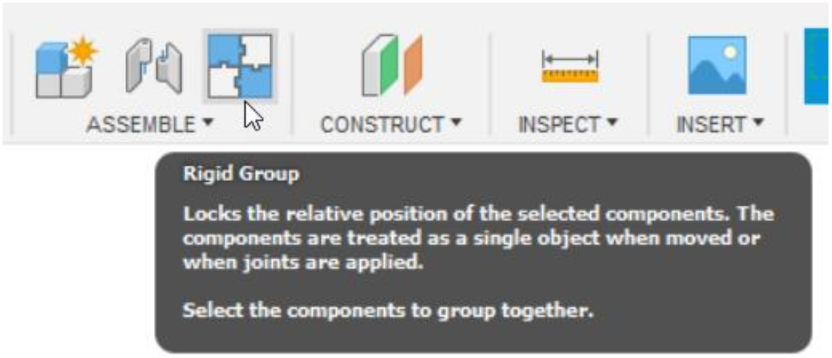
Grounded subassembly vs. As-built Joint: https://www.youtube.com/watch?v=2w8BuJ-3_80



As-built Joint Tool

- Use Rigid Group

A Rigid Group is useful for locking down all the components inside a subassembly.



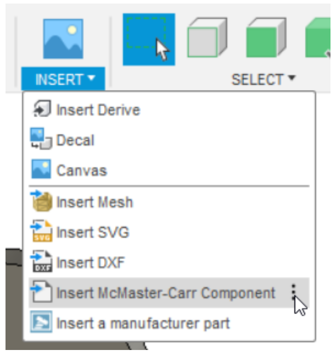
Rigid Group Tool.

In general I find I have to be more careful dragging components in Fusion 360 than I have been in Inventor. This is because subassemblies behave like they are flexible. This means that parts in the subassembly will move if you drag them unless Joints, Rigid Groups, or As-built Joints are used to keep components in position.

Content Center

- Insert McMaster-Carr Component

There isn't a Content Center. Instead you insert components directly from the McMaster-Carr online catalog.



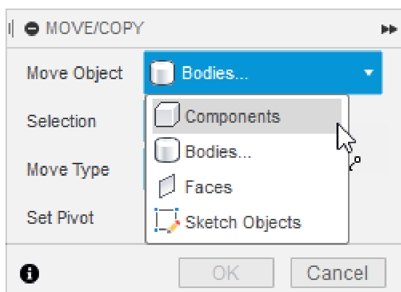
Insert McMaster-Carr Component Tool

You can create a folder in your project for those fasteners that you use frequently. You can treat that like a fastener library. But remember that these are specific size and length fasteners, so you can’t just drag a grip to get a longer bolt.

Selecting Objects

- Be sure of the selected object type during operations.

Many commands can operate on bodies or components. Be sure you have selected the correct type.

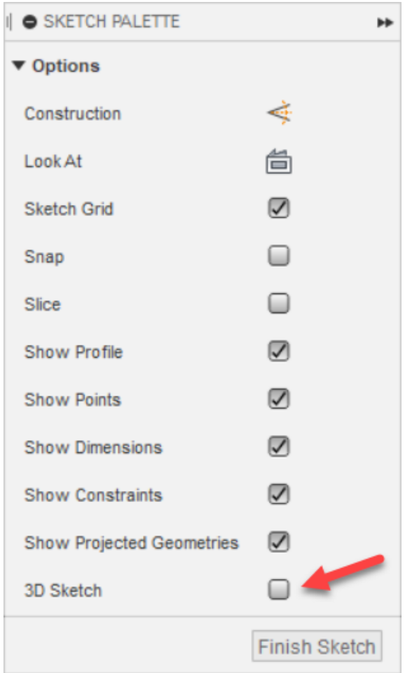


Choosing Object Type (Move/Copy)

Sketching

Sketching is very similar between Inventor and Fusion 360. In Fusion 360 the tools are either in the Ribbon, or in a Sketch palette. Either pick on an Origin plane, a flat face, or on a construction plane (work plane). Unlike Inventor, you can’t directly create an offset construction plane while creating the sketch. The construction plane must already exist.

Be careful that the 3D Sketch option is turned off, otherwise you may end up dragging geometry out of the original sketch plane.



Fusion 360 Sketch Palette

When I add dimensions to my sketches, I find it useful to place the dimension's value farther away from the model geometry than I do in Inventor. This makes it easier to pick the text to edit the value later. If I keep it as close as I do in Inventor, I sometimes have a hard time picking the text as Fusion 360 selects model edges and faces first by default.

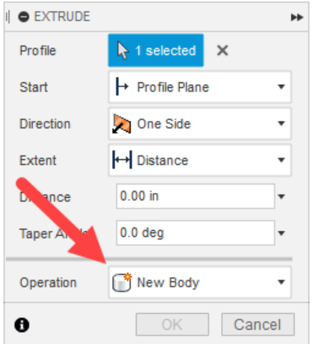
Selecting Objects

Create vs. Modify for Mirror and Pattern

In Inventor tools like Mirror and Pattern are in the Modify panel. In Fusion 360 they are in the Create list of tools.

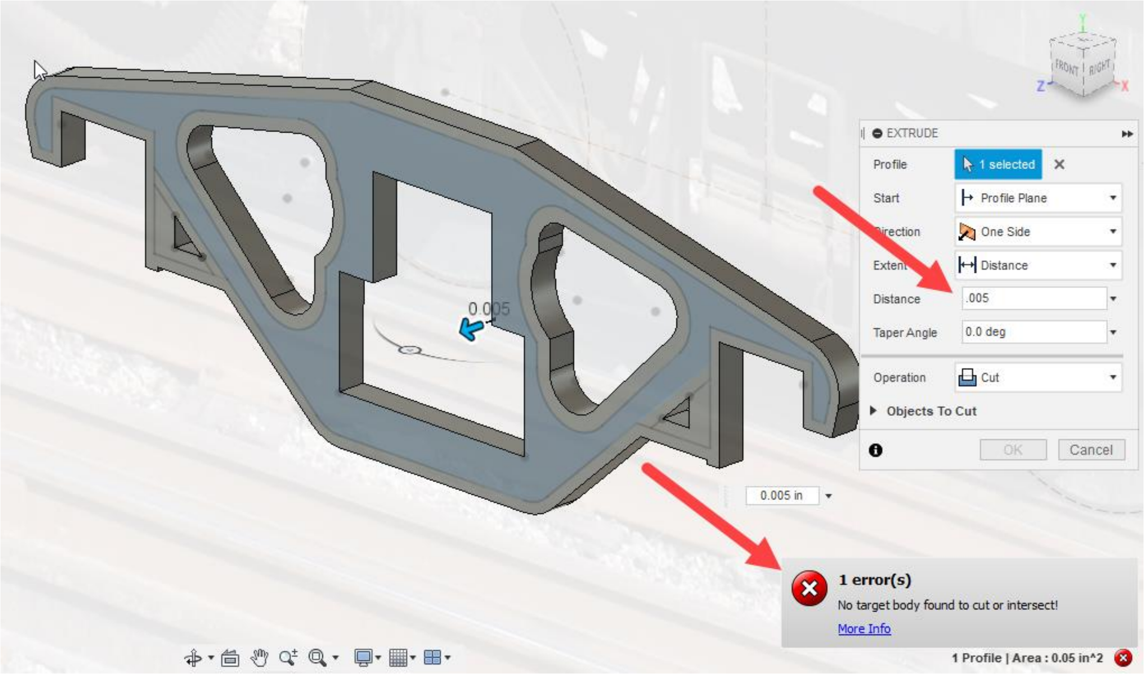
Extruding

The Extrusion tool starts off assuming you are making a new body.

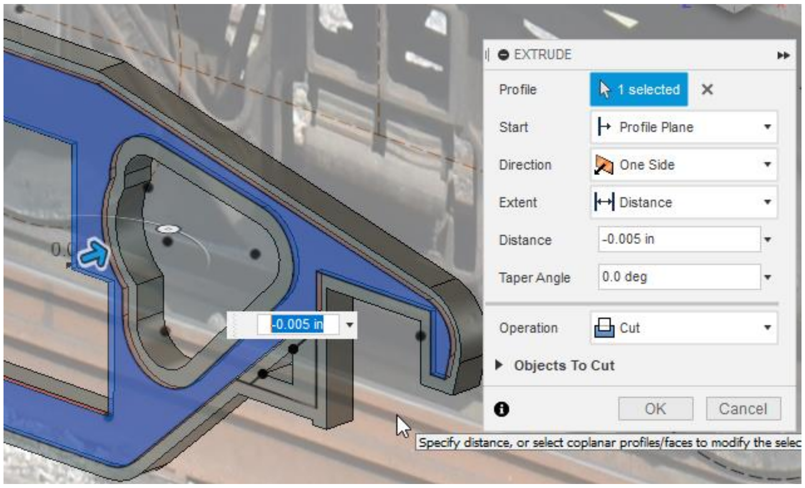


Extrusion Dialog Box

If you are making a cut you must enter the distance as a negative value. Otherwise you will get an error message. You can also drag the distance arrow in the opposite direction to make the value negative. Be careful if you then type a precise value, you will need to keep the negative sign.



Extrusion Error due to Positive Value

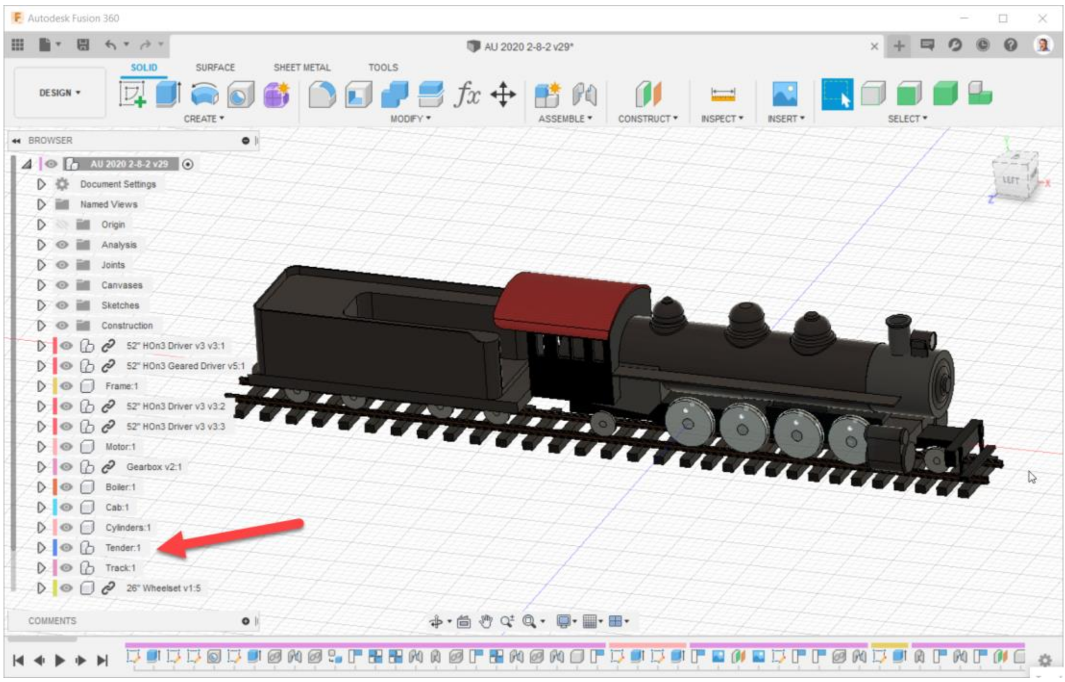


Extrusion Cut, with Correct Negative Value

Model, Analyze, and Machine a Component.

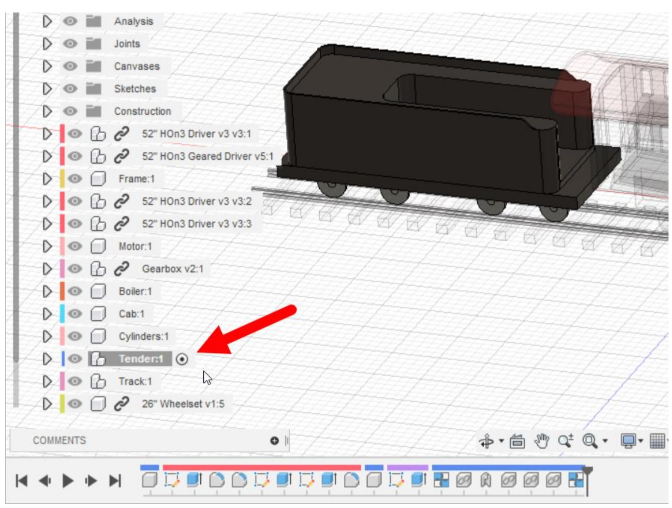
To see how this entire process works let's look at creating the frame for the tender. Here is the workflow I used, broken into three phases: Model, Analyze, and Machine. Note that the Tender Shell has already been created.

Phase 1 - Model the Tender Frame



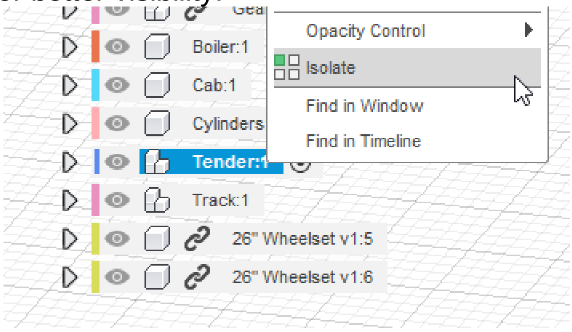
Current Design, Top Level Active

1. Activate the Tender subassembly.



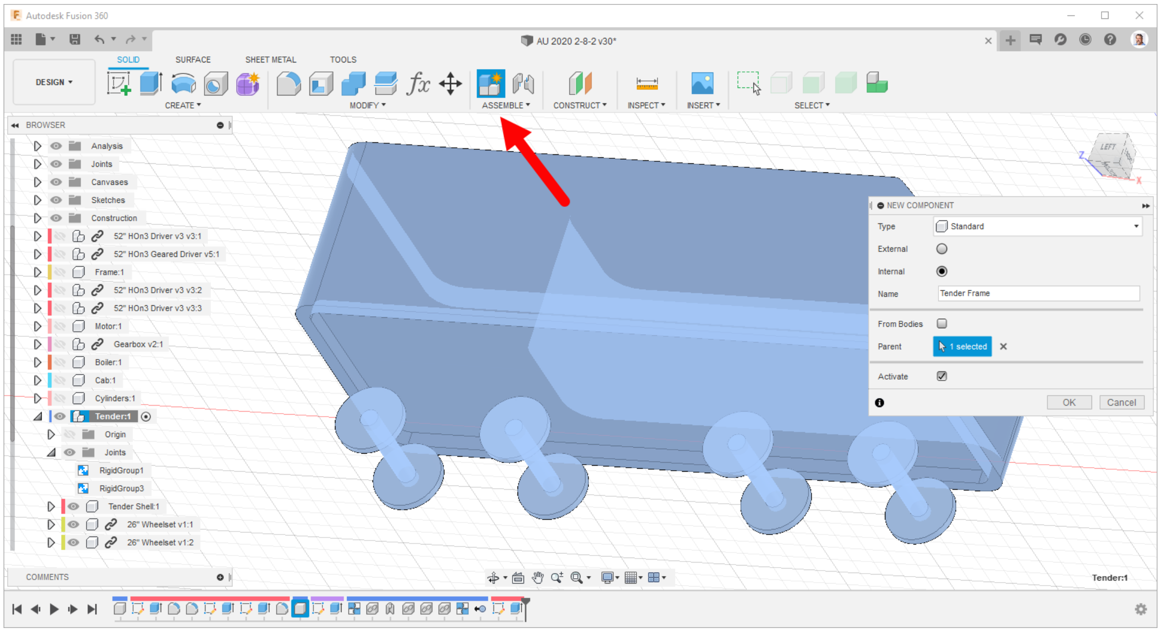
Current Design, Tender Active

2. Isolate the Tender, for better visibility.



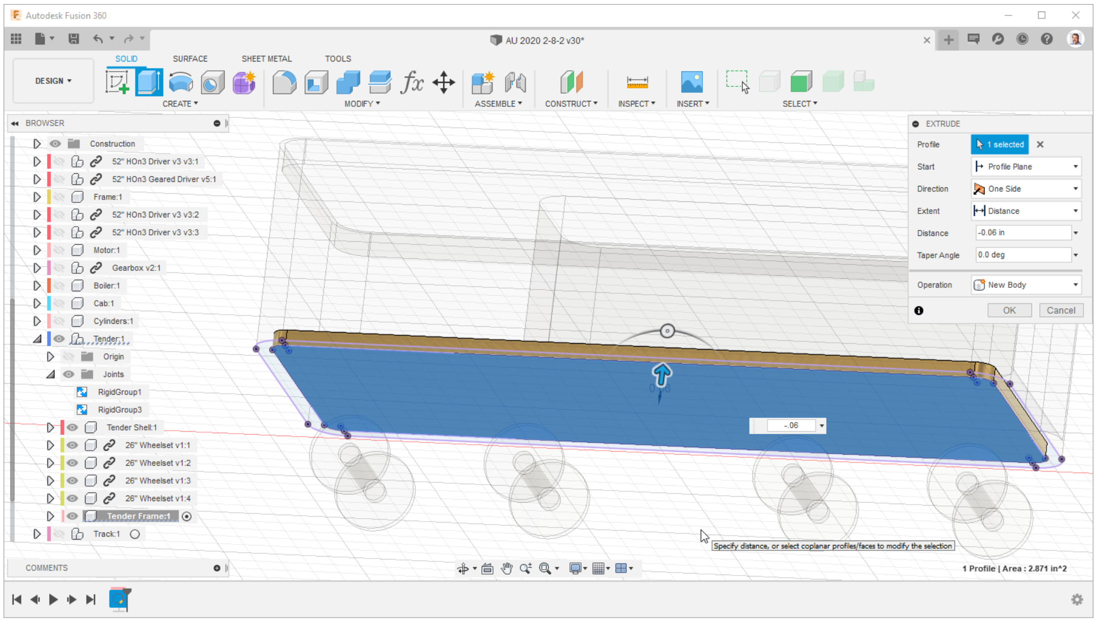
Isolate the Tender

3. Create a new component. I am choosing to create an Internal component, so it will be within this same design file. The new component is also activated right away.



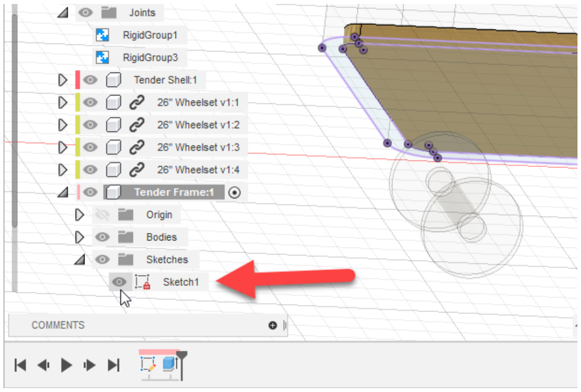
Create the Tender Frame Component

4. Create a sketch on the bottom face of the Tender Shell, then project that bottom face onto the new sketch.
5. Finish the sketch.
6. Extrude the inner profile up into the Tender Shell. The distance is a negative value here due to the normal direction of the sketch plane. This is different from Inventor, as there isn't a Flip Direction option in Fusion 360.



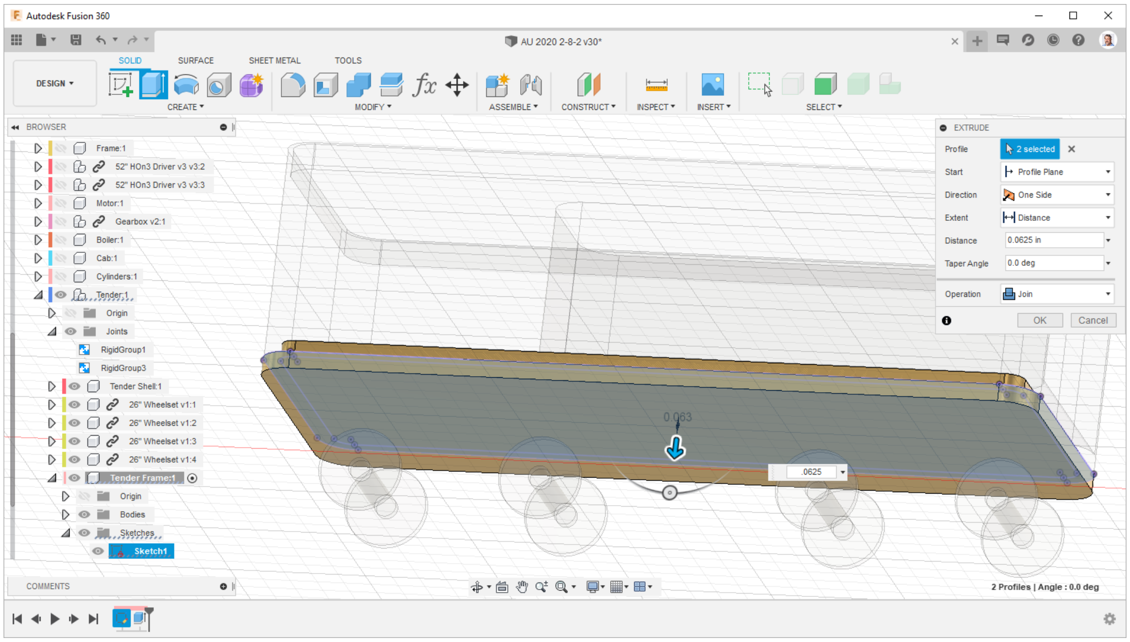
Extrude into the Tender Shell

7. Expand the Sketches node in the Browser and turn the visibility of the existing sketch back on. This is like Inventor workflow for reusing a sketch.



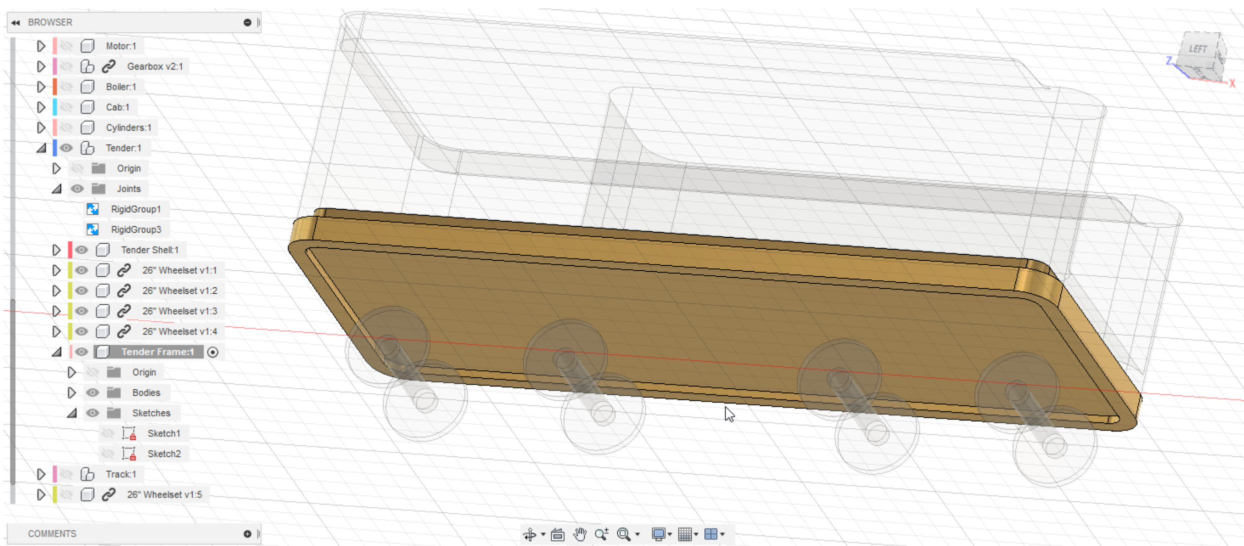
Turning the Sketch Visibility On

8. Extrude both the inner and outer outlines.



Continuing the Frame Body

9. Extrude the outer profile to form a lip around the frame.
10. Turn off the visibility of the sketches.

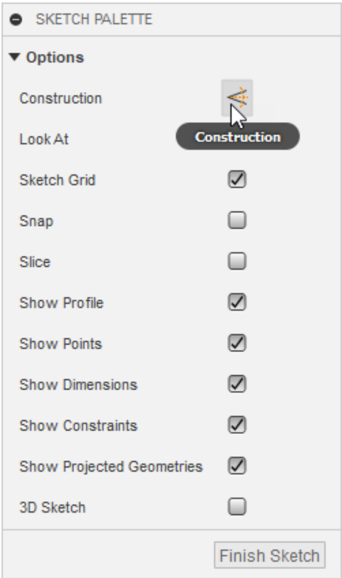


Tender Frame Lip

11. Use Press/Pull to extend the right face towards the right. The offset value can be changed later by editing the feature in the Design History.

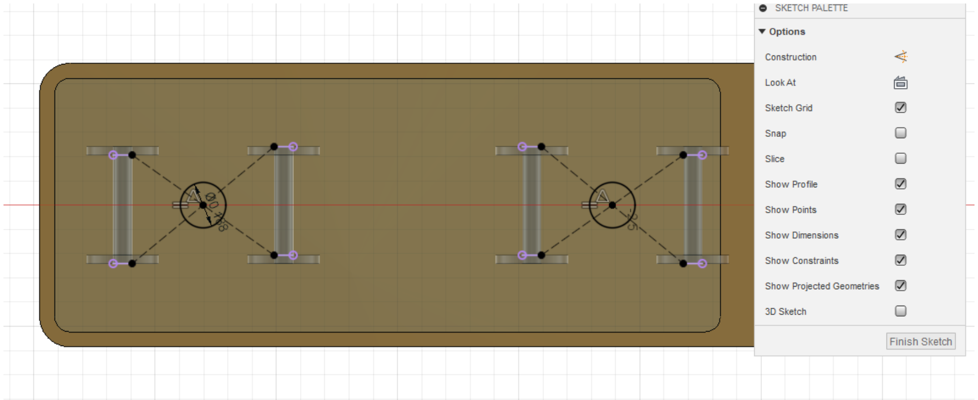
Pull the Frame End

12. Create a new sketch on the underside of the frame.
13. Turn on the Construction option.



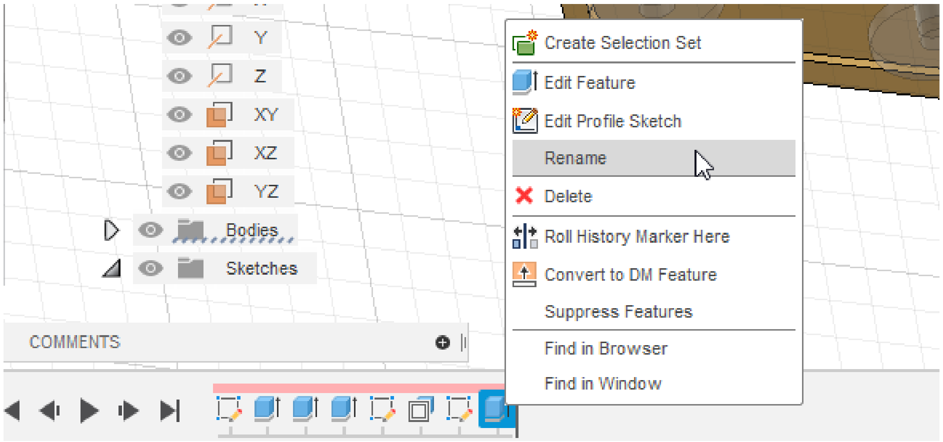
Sketch Palette with Construction Active

14. Project the axles and add additional construction geometry to determine the truck mounting hole locations. Draw circles centered on these points for the truck mounting bosses.



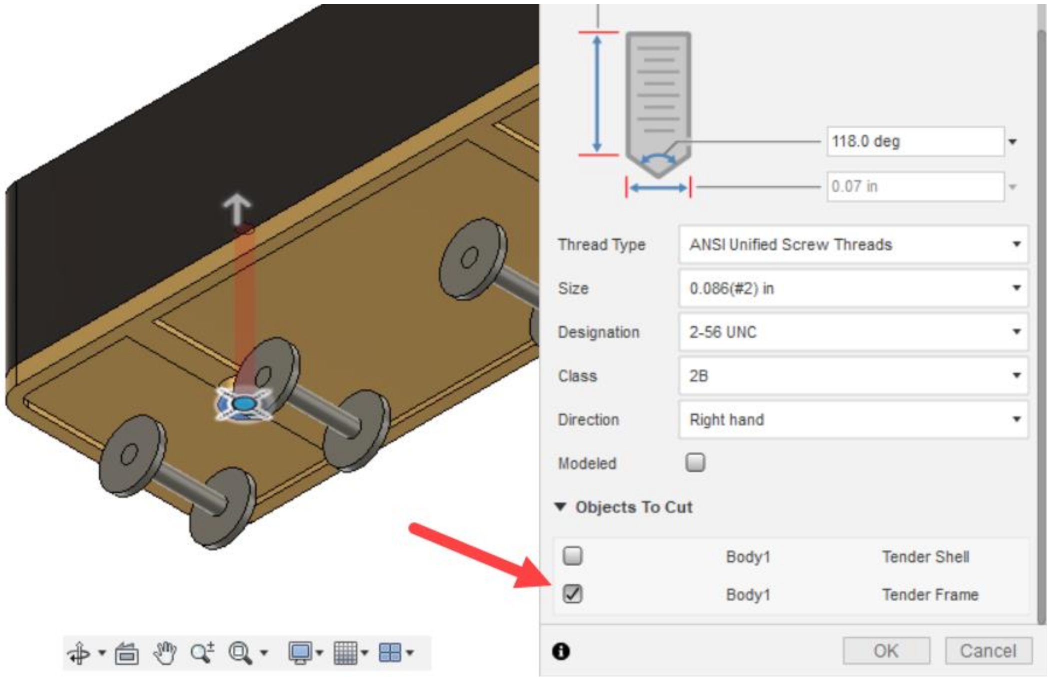
Locating the Truck Mounting Screw Holes

15. Add bosses at these points. Rename these extrusions by right-clicking on the feature in the Design History.



Sketch Palette with Construction Active

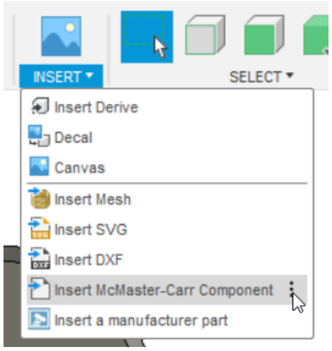
16. Add tapped through holes to those locations. Clear the check mark for drilling through the Tender Shell, so only the Tender Frame is drilled and tapped.



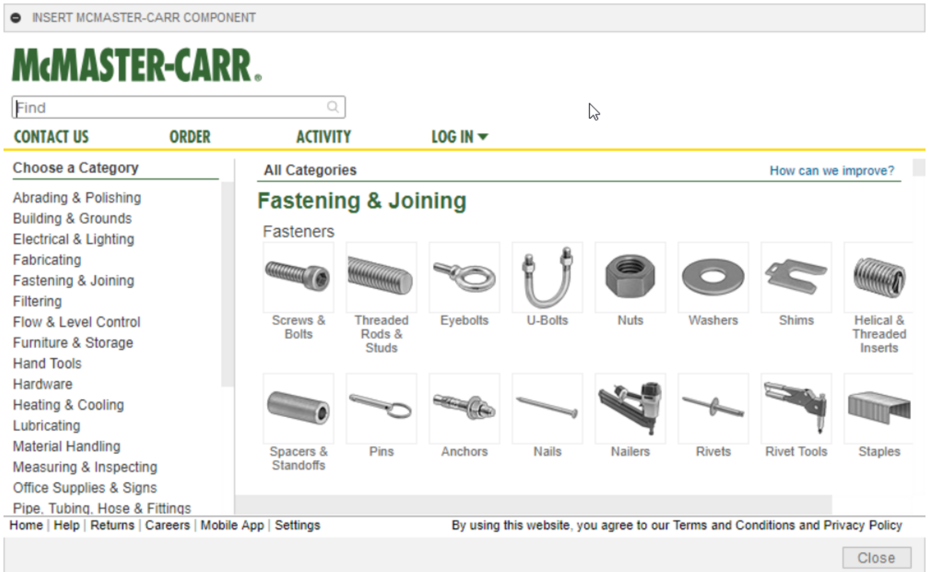
Drilling only the Tender Frame

Add the Truck Mounting Screw

17. Activate the Tender Assembly.
18. Insert a 2-56 UNC x ¼” Phillips Pan Head Screw.

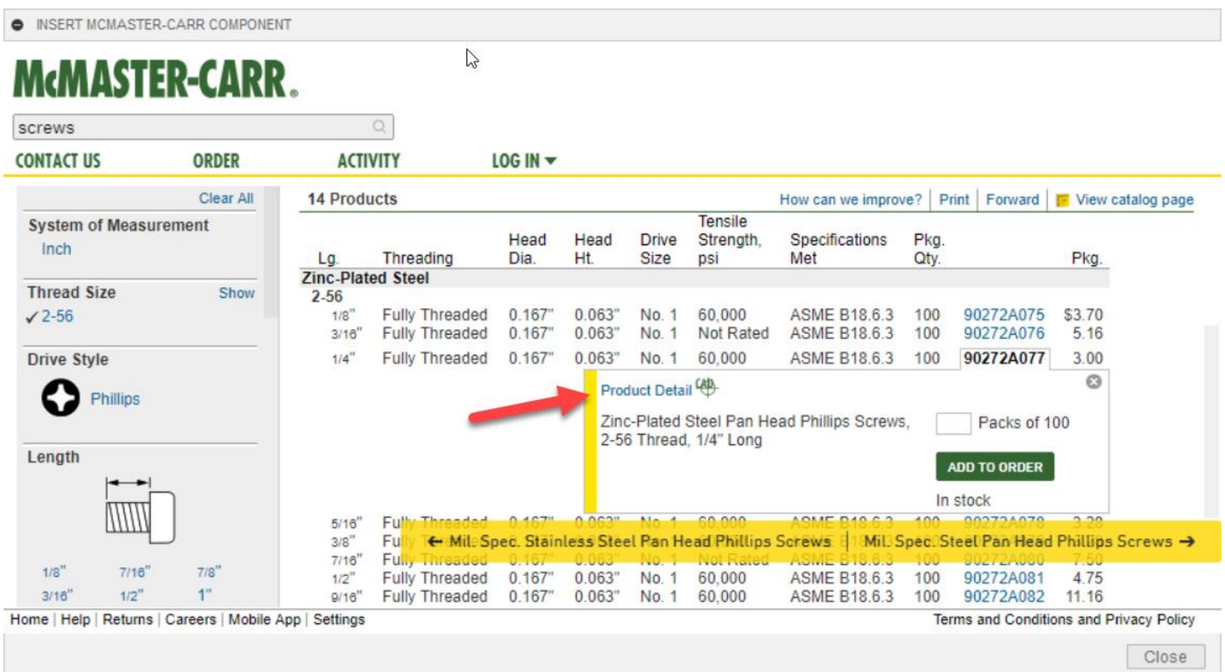


Insert from McMaster-Carr Tool



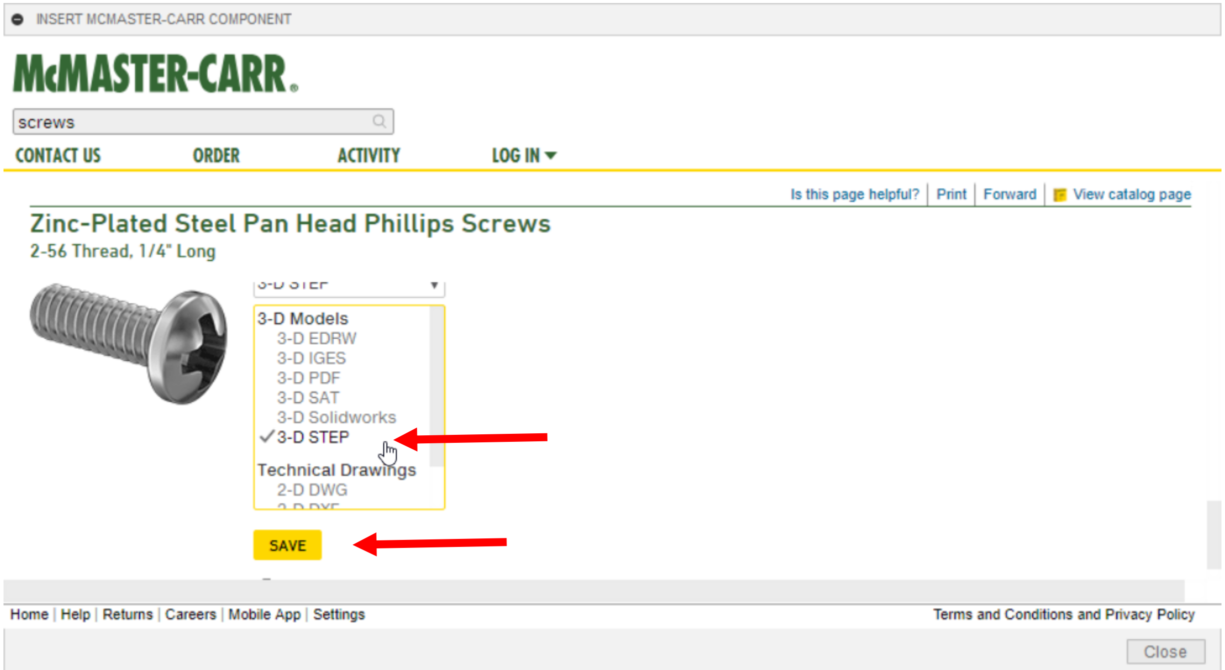
McMaster-Carr Catalog Home Page

19. Browse to the Zinc-Plated Steel Pan Head Phillips Screws page and select the Product Details link.



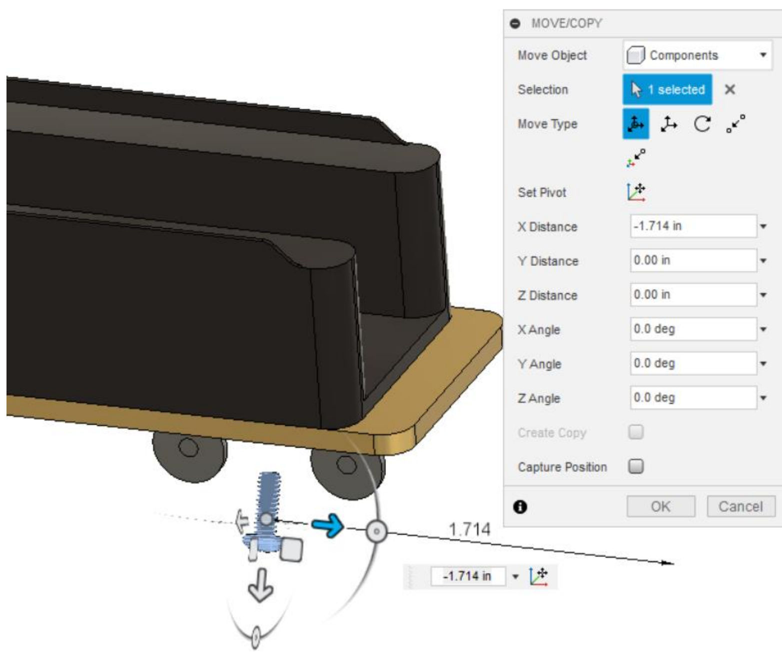
Product Details Link

20. Scroll down the Product Details page and select 3-D STEP format for 3-D Models and SAVE the fastener. After a few moments the fastener will be inserted into the design.



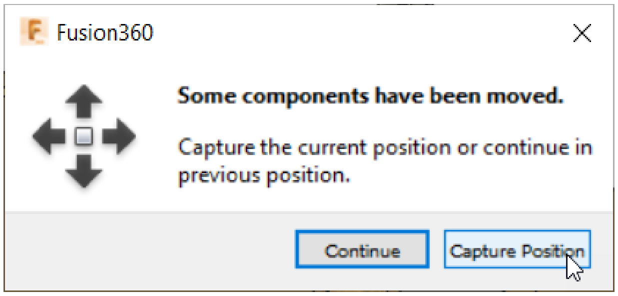
Save the Fastener

21. Move the inserted screw closer to the final location to make selections easier when adding the joint between the screw and the frame.

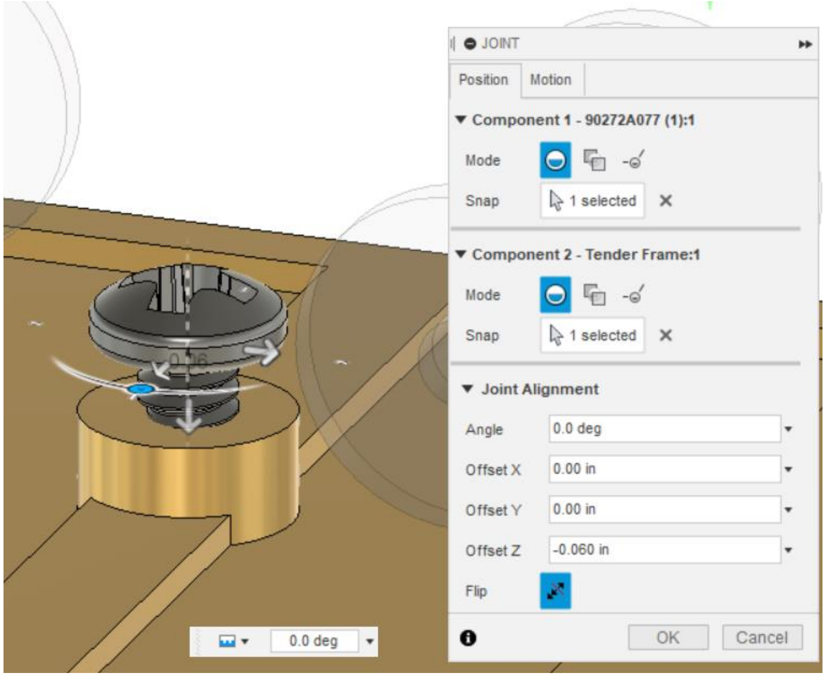


Move the Fastener

22. Add joints for the screws to the Tender Frame. When asked, Capture Position for the screws. Otherwise they move back to their original location.



Capture Position Dialog Box

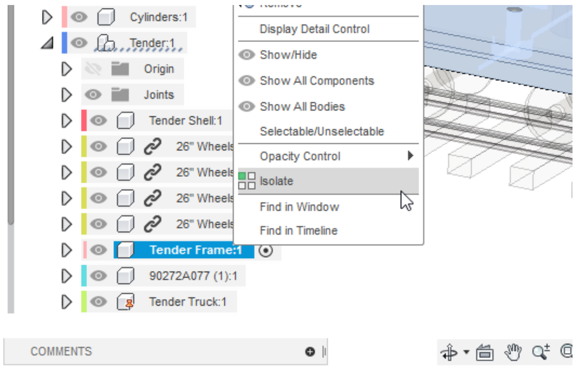


Add a Rigid Joint

Phase 2 – Machining the Tender Frame

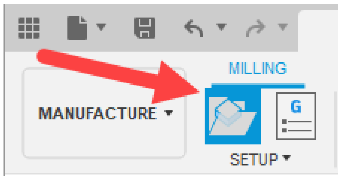
The Tender Frame is designed as a machined piece of brass, both to add weight for better tracking and to act as a heat sink for the model's eventual electronics modules. It is a simple design, intended for 2-1/2 D milling. Please note that I am a hobbyist when it comes to machining. I would have a shop supervisor check my program before running it on a CNC machine.

1. Activate the Tender Frame component and Isolate it for easier visibility.



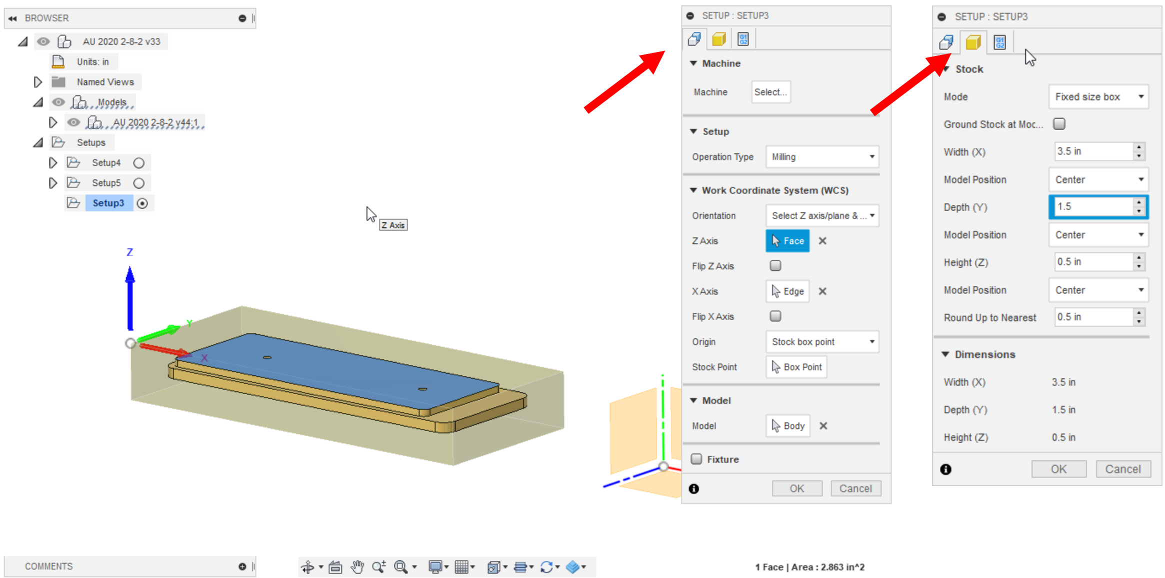
Activate and Isolate the Tender Frame

2. Switch to the Manufacturing environment.
3. Create a setup for milling the top of the part.



Creating a New Setup

4. Have Z pointing up from the part, and X along the part. Pick the corner of the stock box for the origin.
5. Set the stock size to 3.5" x 1.5" x 0.5".

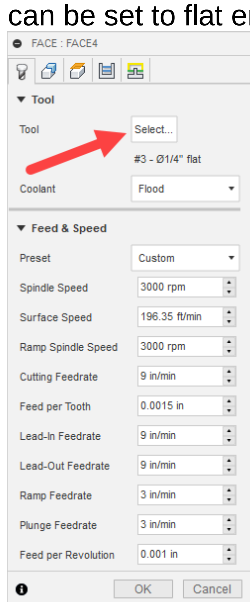


Setup for Top of Tender Frame

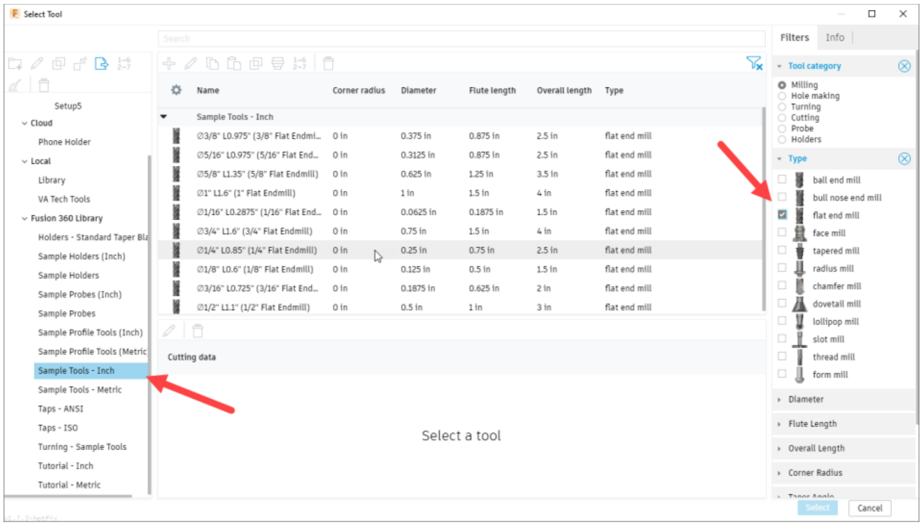
6. Use the 2D Face tool to remove stock to the top of the frame.

2D Face Tool

7. Select a ¼" End Mill from the Tool Library. Initially it can be selected from the Sample Tools – Inch category. The Type can be set to flat end mill to filter the choices.

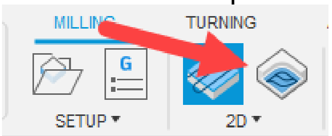


Select Milling Bit



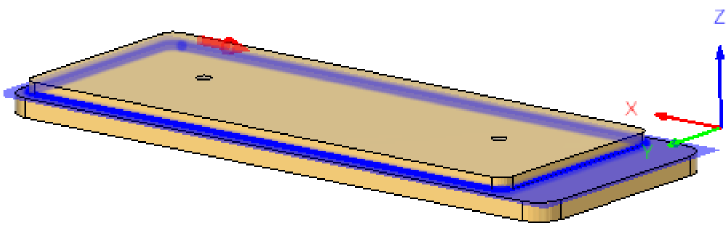
Select ¼” Flat End Mill in Tool Library

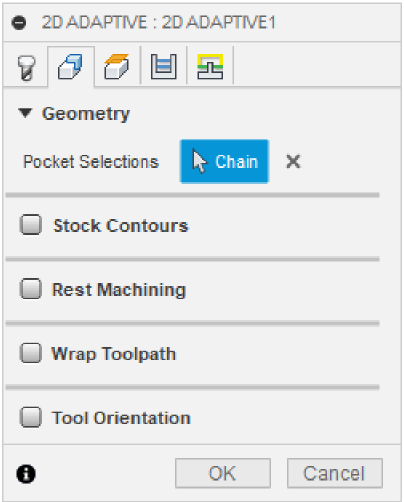
8. Set the Pass Extension to 0.25” choose OK. This will generate the first toolpath.
9. Use the 2D Adaptive Clearing tool to machine the lip for the shell to set on.



2D Adaptive Clearing

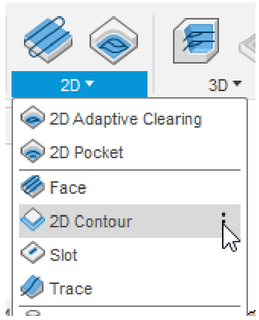
10. Select the inner edges of the highlighted surface for the contour. Note: the same tool will be used here, so no selection is needed on the Tool tab.





Selected Edges

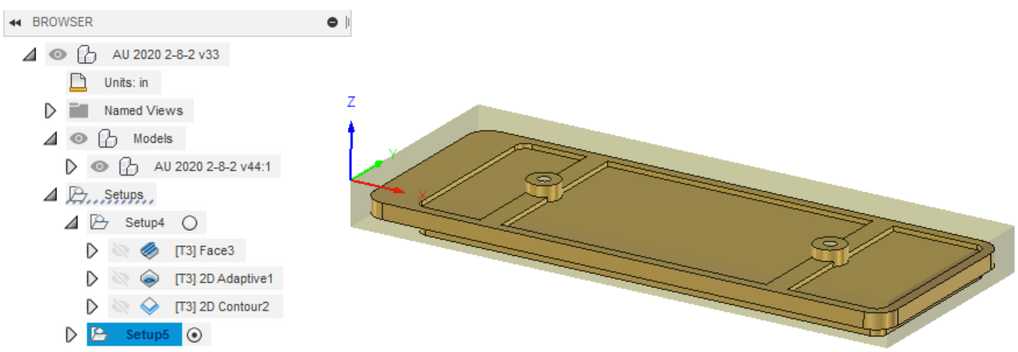
11. Use the 2D Contour tool to complete the topside machining. It uses the same tool and selected edges.



2D Contour

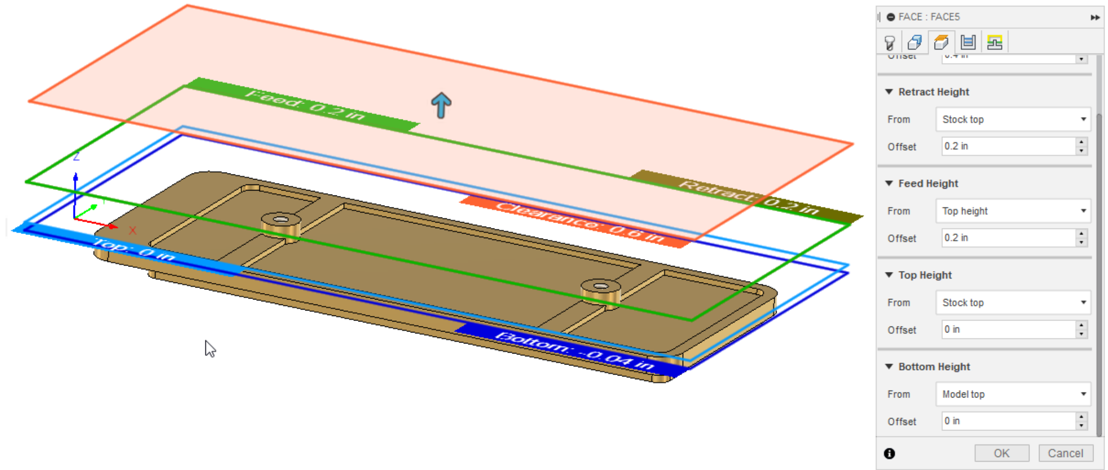
12. A second setup is needed, with the Z axis going the opposite direction from the first setup.

New Setup



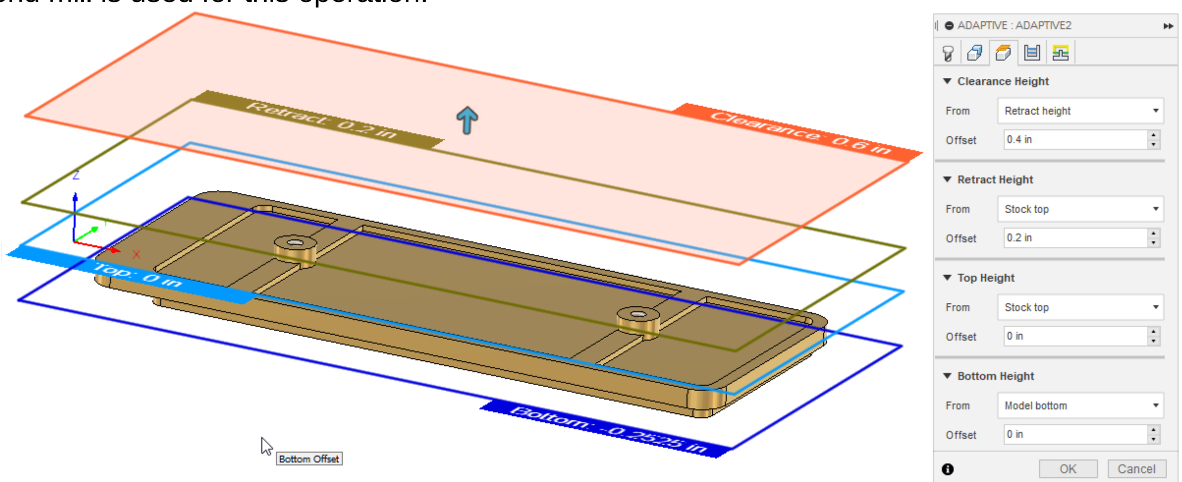
New Setup Orientation

13. The 2D Face tool is used to take the overall stock height down to the top of the two bosses. The ¼” end mill is used for this operation.

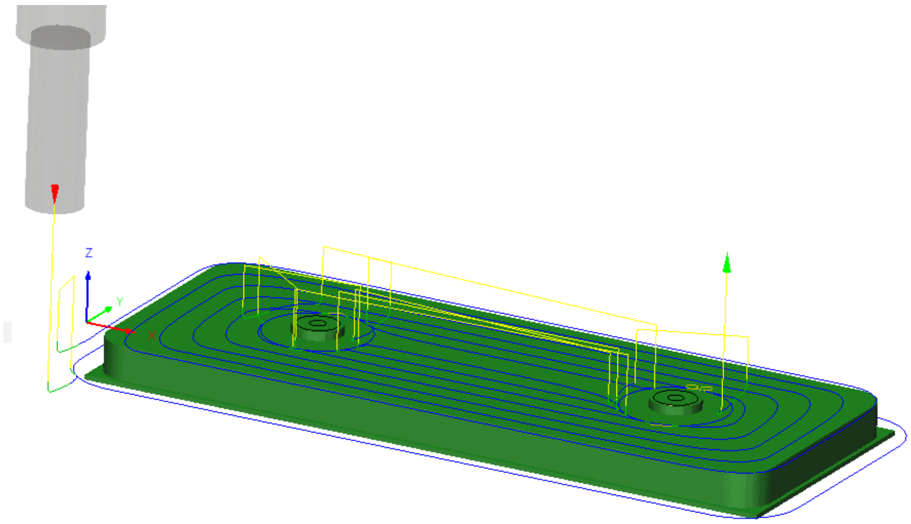


2D Face Dialog Box – Heights Tab

14. The 3D Adaptive Clearing tool is used to make the bosses and the frame bottom. The ¼” end mill is used for this operation.

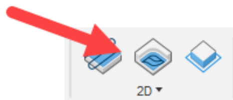


3D Adaptive Clearing Heights

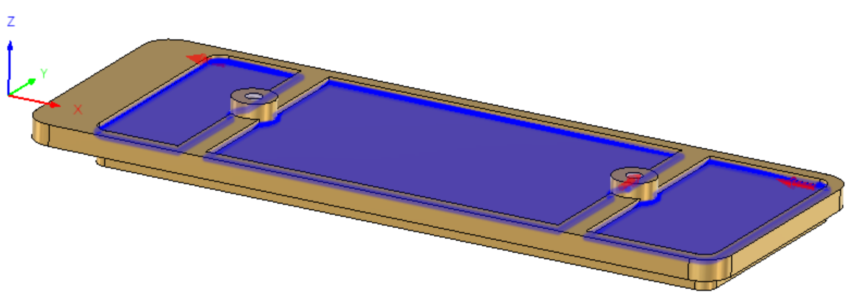


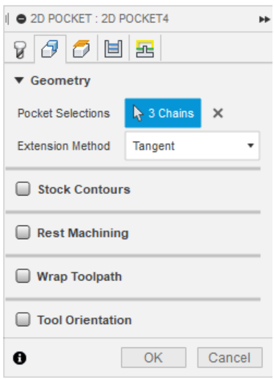
3D Adaptive Clearing Toolpath

15. The 2D Pocket tool is to clear the inner pockets. A 1/16” flat end mill is used, allowing for tighter corners.



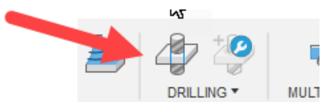
2D Pocket Tool



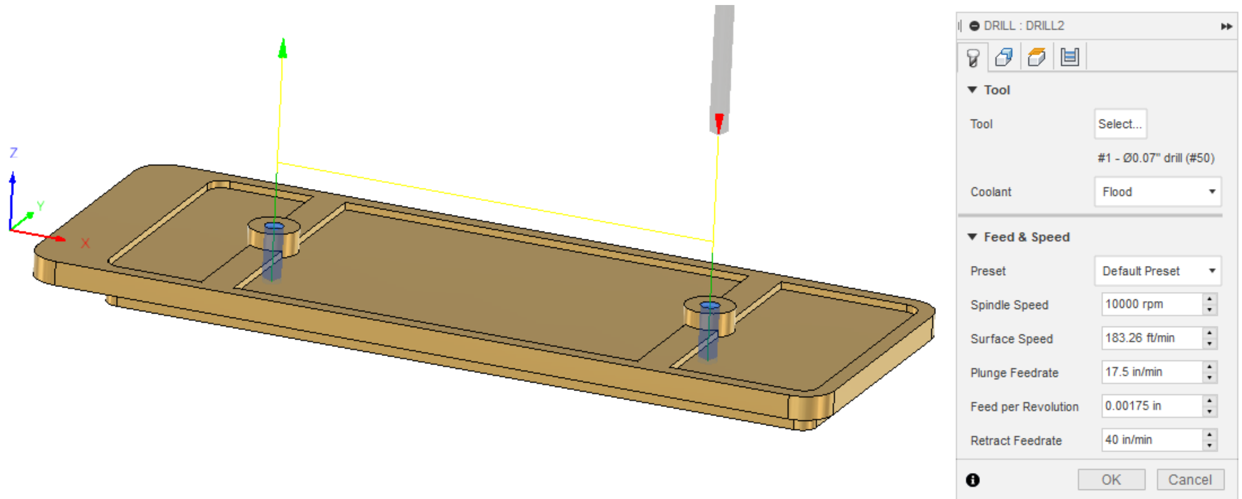


2D Pocket Tool – Selected Edges

16. Finally the Drill tool is used to drill the tap diameter holes for the 2-56 screws that will mount the tender trucks. A #50 drill is selected for the operation. Note that I didn’t tap these holes as I am planning on doing that manually.

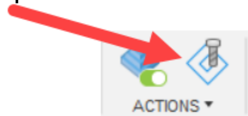


Drill Tool



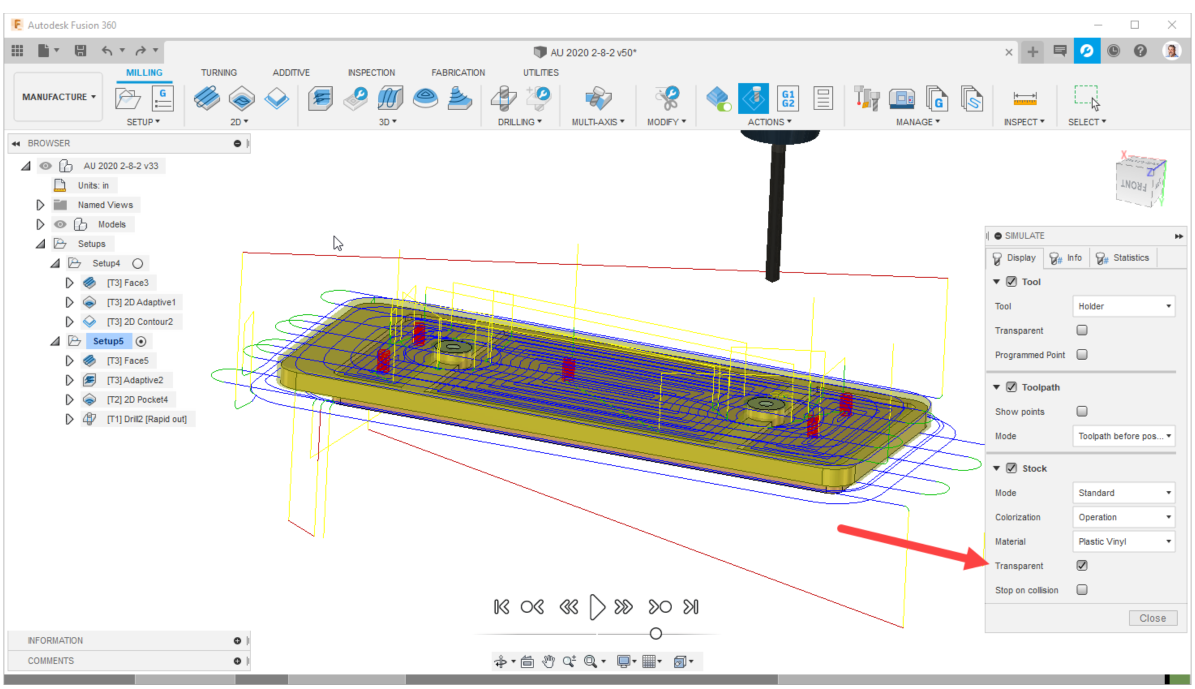
Drilling Operation

17. Simulate the tool paths to verify the operations.



Toolpath Simulation

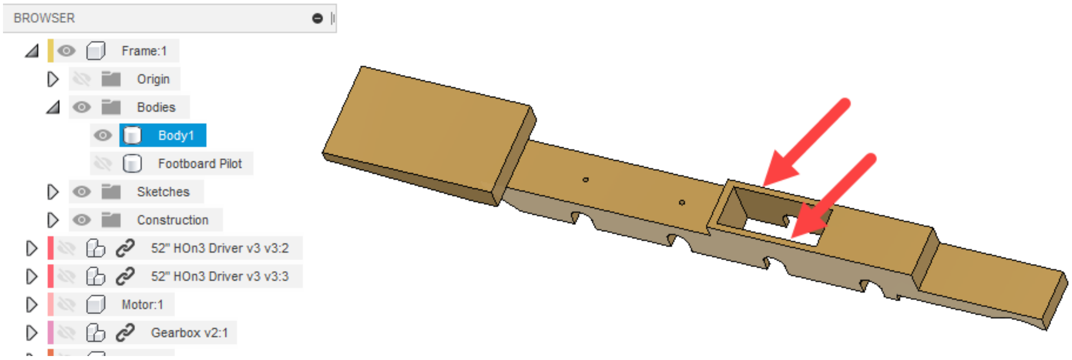
The Simulate tool can show an animation of the tool as it performs its operation. All the operations for a given setup can be animated to show the machining progress. One option in Simulate that I recommend is to show the stock as translucent during the simulations.



Toolpath Simulation

Phase 3 - Check the Design Strength

The frame design has a tall, thin wall around the gear box. If the model locomotive were dropped how likely is this wall to buckle?



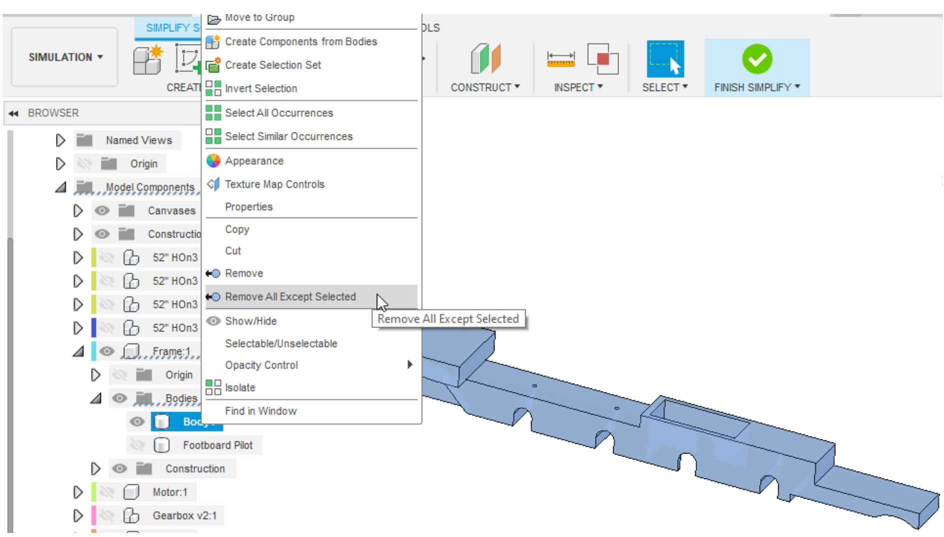
Thin Wall Areas

1. Enter the Simulation environment.

Typically, the Simulation workflow is to start with the left most tool in the ribbon, and work towards the right. In this model the material is already assigned in the design, so the Material tool can be skipped.

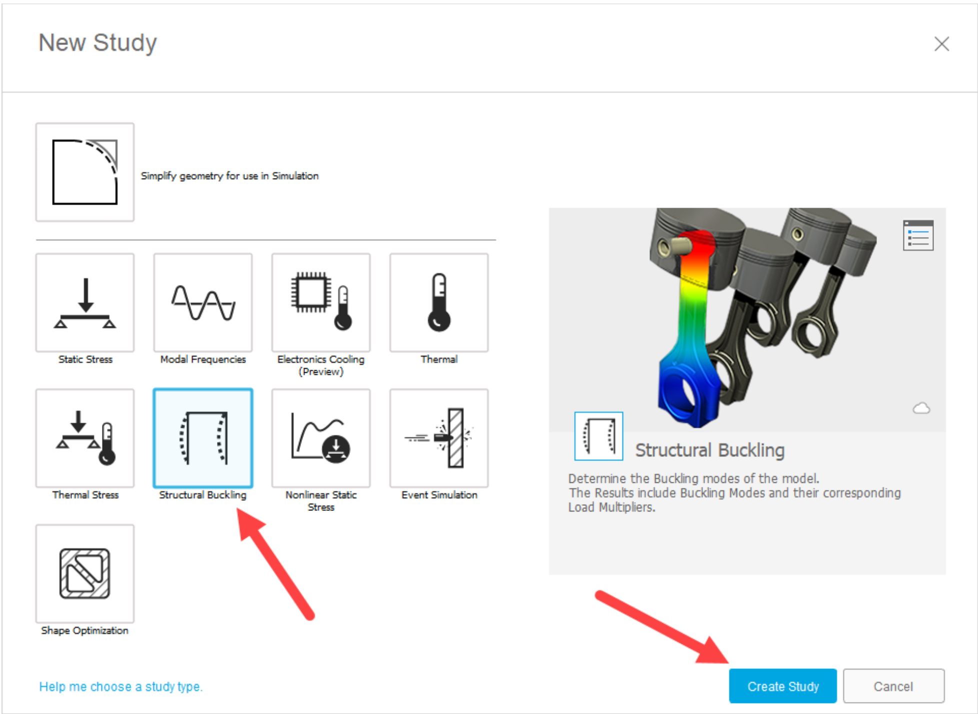
Simulation Environment Ribbon

2. Choose the Simplify option.
3. In the Simplify tool select the Frame in the Browser, right-click, and choose Remove All Except Selected. This will limit the analysis to this single body.



Limiting the Analysis to the Frame

4. Finish the Simplify.
5. Select Structural Buckling for the analysis type, then Create Study.

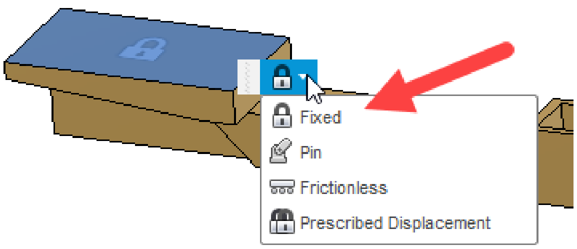


Selecting the Analysis Type

6. Add constraints.

Adding Constraints

7. Select the top rear of the frame. A fixed constraint is added by default.

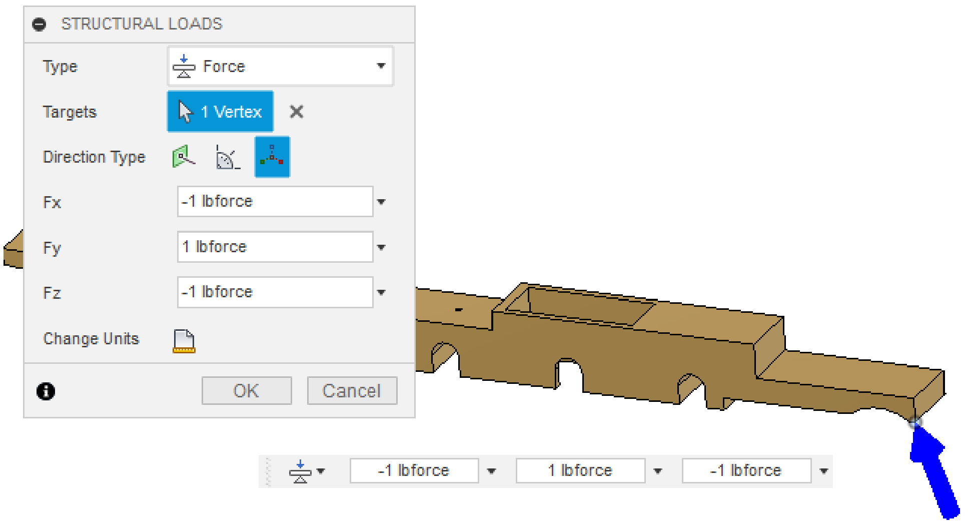


Adding a Fixed Constraint

8. Add a load to the front corner of the frame.

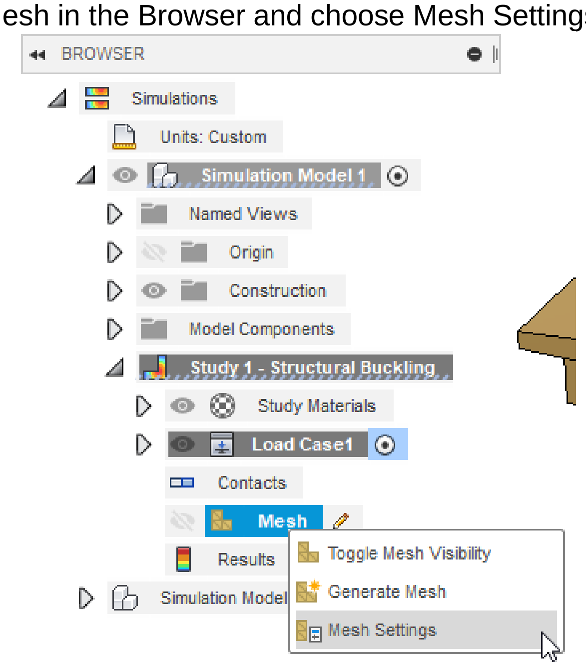
Adding a Load

9. Add the force with values to simulate a hit on the corner. All values are set to plus or minus one.



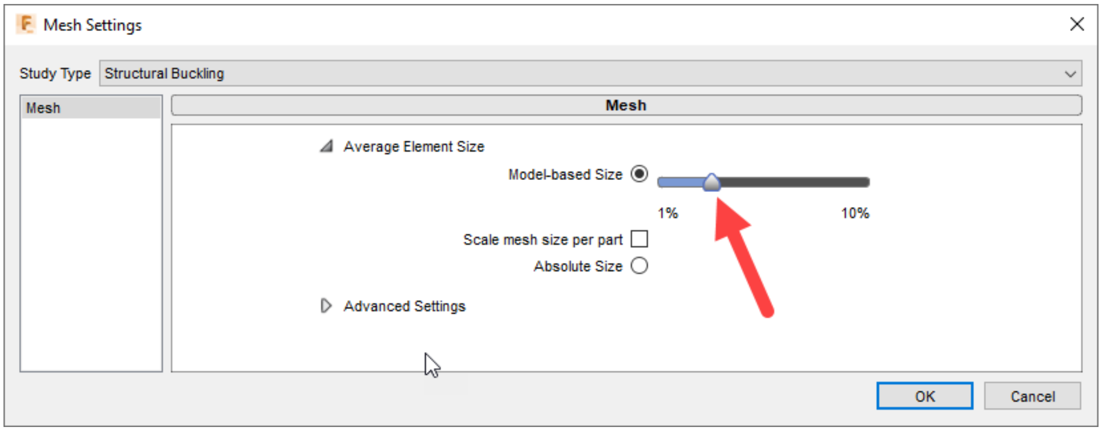
Force Settings

10. Right click on the Mesh in the Browser and choose Mesh Settings.



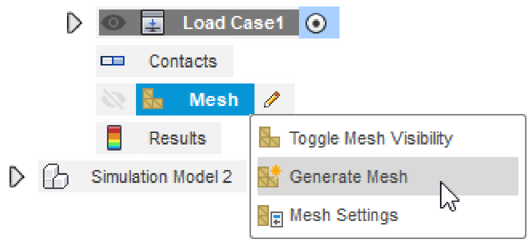
Mesh Settings

11. Move the Average Element Size slider approximately 3%.

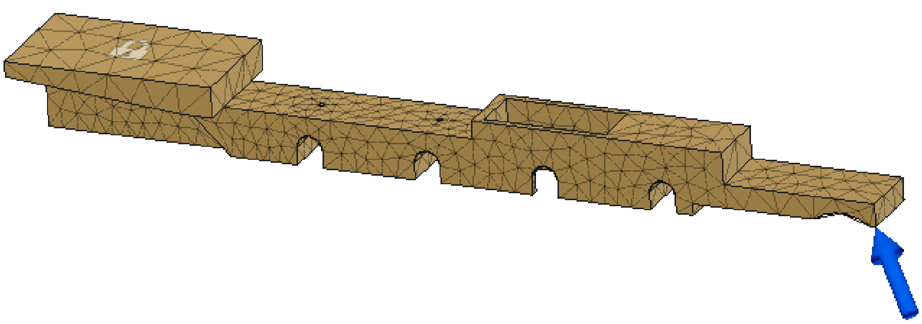


Mesh Settings

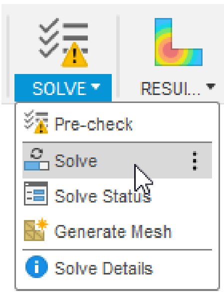
12. Right click on the Mesh in the Browser and choose Generate Mesh.



Generate Mesh

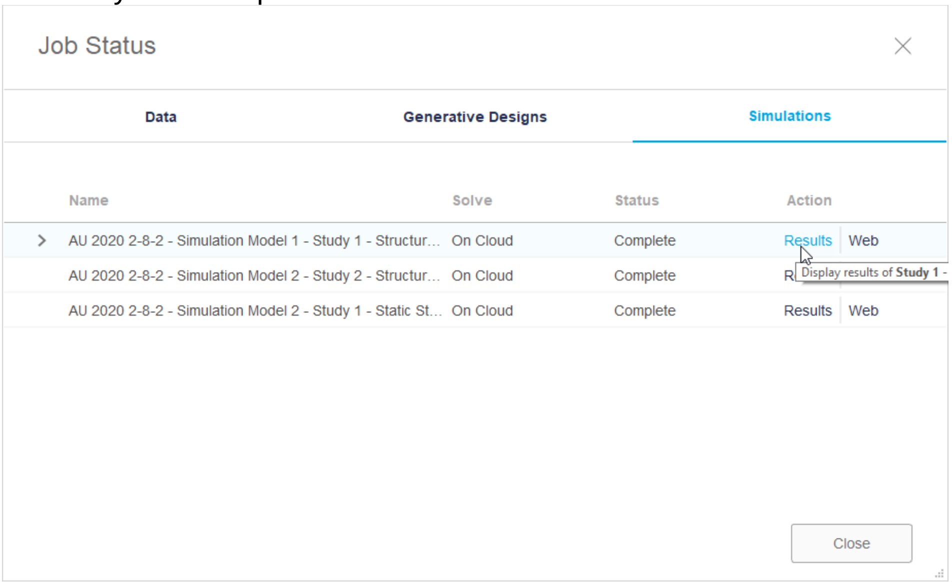

Resulting Mesh

13. Pick Solve to start the calculations.

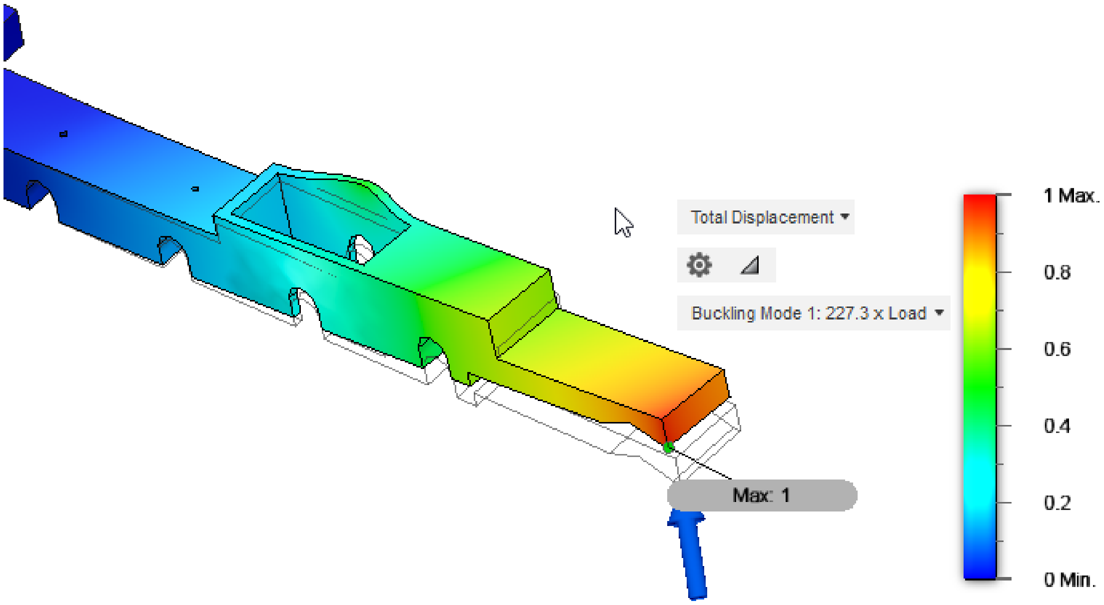


Solve, with Pre-check Warnings

14. When the analysis is complete view the results.



Accessing the Results



Buckling Results

The results indicate that buckling won't occur until the load shown is increased approximately 200 times. So, I feel safe that buckling in the thin section of the frame isn't likely under normal operation and handling of the model.

Conclusion

Fusion 360 has many advantages over traditional modeling tools in a conceptual design situation. Modeling is quicker. Simulations are easy to perform. Machining is straight forward.

At the same time, if you try and run Fusion 360 just like Inventor you will cause trouble for yourself. Remember to create and activate components. Remember to use As-Built Joints.

Now go give Fusion 360 a try on your next project!