IM462100

Applying Parametric Studies in Inventor Stress Analysis

Eric Schubert
D3 Technologies

Learning Objectives

- Discuss where and when parametric studies can be employed in simulations.

- Investigate how to enable parametric studies in Inventor Stress Analysis.

- Learn how to explore design ideas and changes by varying parameter values.

- Learn how to compare simulation results for different variations and choose the best design concepts.

Description

Designing new products or components often involves some uncertainty about strength or stiffness. By incorporating Inventor Professional software's ability to run simulations with varying parameter values, we can investigate what will happen when we adjust placement, thickness, and even the presence of various design features on our models.

In this class, we'll look into how we can employ parameters in the simulation process, and how to analyze and compare the performance of the values we choose to simulate in Inventor Stress Analysis.

Speaker

Eric Schubert is a Mechanical Engineer, Autodesk Certified Instructor, and Simulation Specialist for Autodesk Platinum Partner D3 Technologies.

With over 13 years of experience utilizing, supporting, and training users on Autodesk products, Eric has applied his background in design, engineering, project management, thixomolding and plastic injection molding to assist customers with analyzing a wide array of products.

Currently, he serves as a subject matter expert for D3 Technologies and their customers around numerous Autodesk simulation products, including: Inventor Stress Analysis, Inventor Dynamic Simulation, Inventor Nastran, Autodesk CFD, Fusion 360, and Generative Design.

This is Eric's first presentation at Autodesk University!

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NOTE: Feel free to adjust and tinker with included class files. Completed results are also included, and should have the word "completed" in the file/folder names to indicate this. You can investigate these files to see how they've been set up and run, then compare them with your own results. Additional files have been included as examples and test models to experiment on your own. **You may need to rerun the examples to get results for included analyses.**

Brief Overview of Inventor Stress Analysis

Engineers, designers, and analysts have been utilizing Finite Element Analysis (FEA) for decades to assist in the design and prototyping of new products. This technique uses the mathematical principles of the Finite Element Method (FEM) to approximate the behavior of a design under various operating conditions, ranging from light use to extreme abuse. FEM breaks down a complex problem into smaller, simpler problems that are easier to solve through the use of a mesh, which approximates the general shape of the design being analyzed. The smaller the mesh is, the more-closely (in theory) the simulation model will represent the real-life version of the component being simulated.

The operator then applies various boundary conditions (loads, constraints, temperatures, material properties, etc.) to the model in an attempt to replicate the same conditions the components would experience in a real-life scenario.

The software then calculates various values for stress, strain, temperature, etc. at each node on the mesh. Once the calculations are completed, the results are displayed on the model, helping the analyst inspect the reaction of the model to loading conditions and decide whether or not the design requires changes prior to proceeding into production.

This is, of course, a gross over-simplification of the process. But, it may help attendees who have not utilized FEA before to understand the gist of what the program is doing under the hood.

This type of simulation method is quite commonly employed across many disciplines. The Finite Element Method can be applied to numerous other types of analyses, well beyond those included in Inventor Stress Analysis, including: material fatigue, vibration analysis, fluid flow for liquids and gases, thermal distribution, thermal stress and expansion, electromagnetic fields, and more.

What are Parametric Studies?

The Inventor Stress Analysis environment includes the capability to test different model configurations by varying the parameter values used to design the model. Users are able to choose one or more parameter values they wish to vary, and assign them corresponding values for the software to test as it calculates the simulation results.

Parameter values can be adjusted for both parts and assemblies. (Assembly parameters typically involve constraint offset or angle values, or assembly features like holes and cuts that align between components.

Advantages of Using Parametric Studies

While Inventor could be set up to run separate, individual simulations for each of the possible parameter value combinations, this requires significant amounts of effort. Each simulation would need to be configured and run separately, leaving the user with no connection between each model and the simulation results.

Instead, utilizing Parametric Studies, the engineer can directly tell Inventor to calculate results for multiple configurations of these parameters, running them all as a set and easily comparing them when the simulation run is complete. Inventor takes care of the rest behind the scenes, and uses the same underlying geometry data, material, and mesh size to make sure that each configuration closely matches the others for proper comparison between results.

Parametric Studies also include a place for including desirable outcomes from the comparison of results. Examples: maximizing stiffness, reducing weight, minimizing contact pressure between components, defining a target safety factor.

Essentially, the Parametric Study capability will:

- Compare how parameter changes affect the design
- Reduce setup time for multiple simulation runs
- Assist in choosing the best candidate based on design constraints for the project

Steps to Create a Parametric Study

The following are general steps needed to ensure the creation and analysis of a parametric study are fully set up and run smoothly:

1. Open desired part or assembly for analysis
2. Open Inventor Stress Analysis environment
3. Create a new study, changing the design objective to Parametric Dimension
4. During simulation setup, select which parameters need to vary
5. Assign specific values to these variable parameters
6. Assign design constraints, if any.
7. Generate all configurations of the model
8. When ready to run the simulation, select to run all configurations or a "Smart Set" of configurations
9. Evaluate results by adjusting sliders in the Parametric Table
10. Promote the desired outcome to the model to keep it permanently

Example: Build and Run a Parametric Study (Single Part)

In this section, we'll run through an example of setting up and running a single-part parametric study in Inventor Stress Analysis. The objective here is to create the strongest, lightest version of this part, ensuring safety with minimal material cost.

To begin, open the Inventor part file labeled "Single Part Study – Support Arm.ipt" from the accompanying class files.

Creating a Parametric Study

The creation of parametric studies in Inventor Stress Analysis is nearly identical to standard Single Point studies that don't adjust any dimensions. The primary difference is in the Create New Study window: we need to adjust the Design Objective setting to "Parametric Dimension":

To enter Stress Analysis, click to the Environments ribbon tab, then select Stress Analysis from the ribbon:
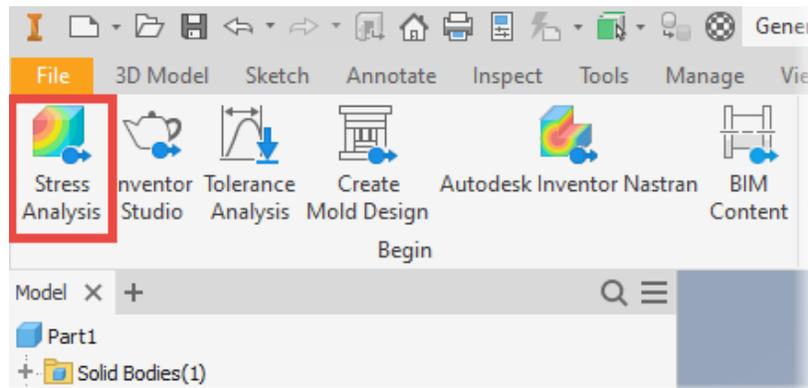
Figure 1: Opening Stress Analysis Environment

Next, create a new study:
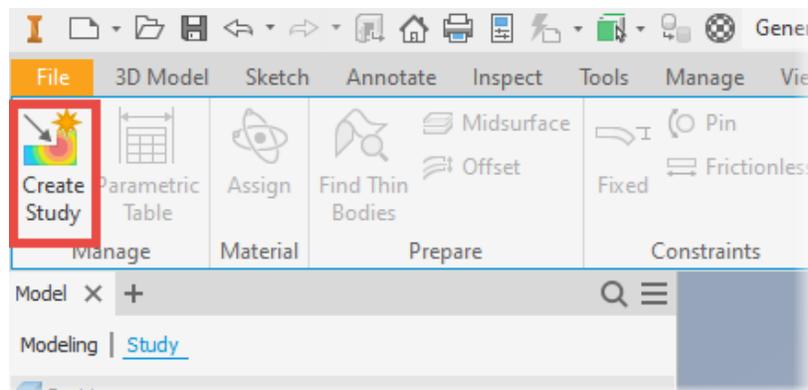
Figure 2: Creating a new study

From the Create New Study window:
1. Enter a study name
2. Select the Parametric Dimension design objective
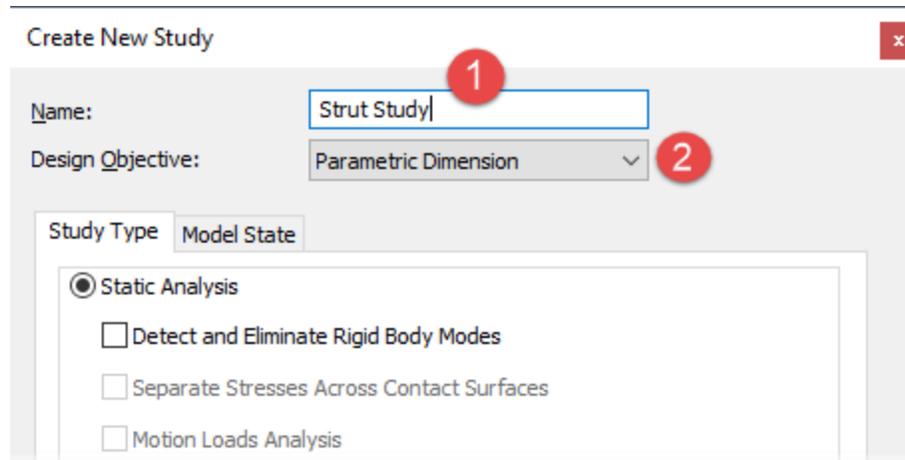3. Click OK to create the study

Figure 3: Creating a Parametric Study

Choosing Parameters for the Study

Once inside the study environment, the next step is to choose the parameters we want to adjust and examine. To do this, right-click on the part and select Show Parameters.
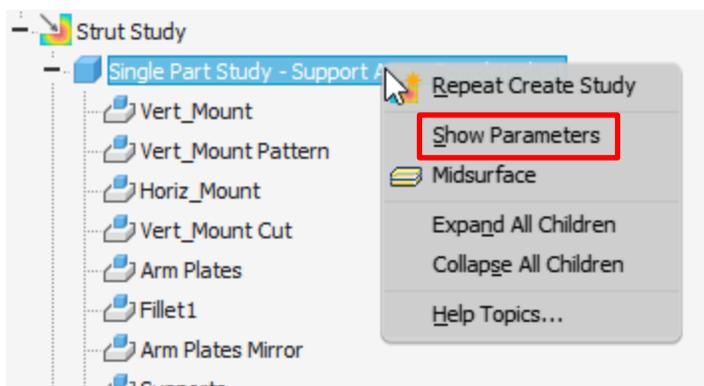

Figure 4: Showing the parameter list

TIP: You can expand the part in the browser and right-click specific features to only see the parameters for that feature. This makes it easier to locate the parameters you need.

For our strut component, we will choose the two parameters labeled:

- Arm_Thickness
- Support_Width

If you go through this example on your own, feel free to try varying other parameters on the model, as well. Some suggested parameters to try:

- Arm_Length
- Arm_Thickness
- Mount_Thickness
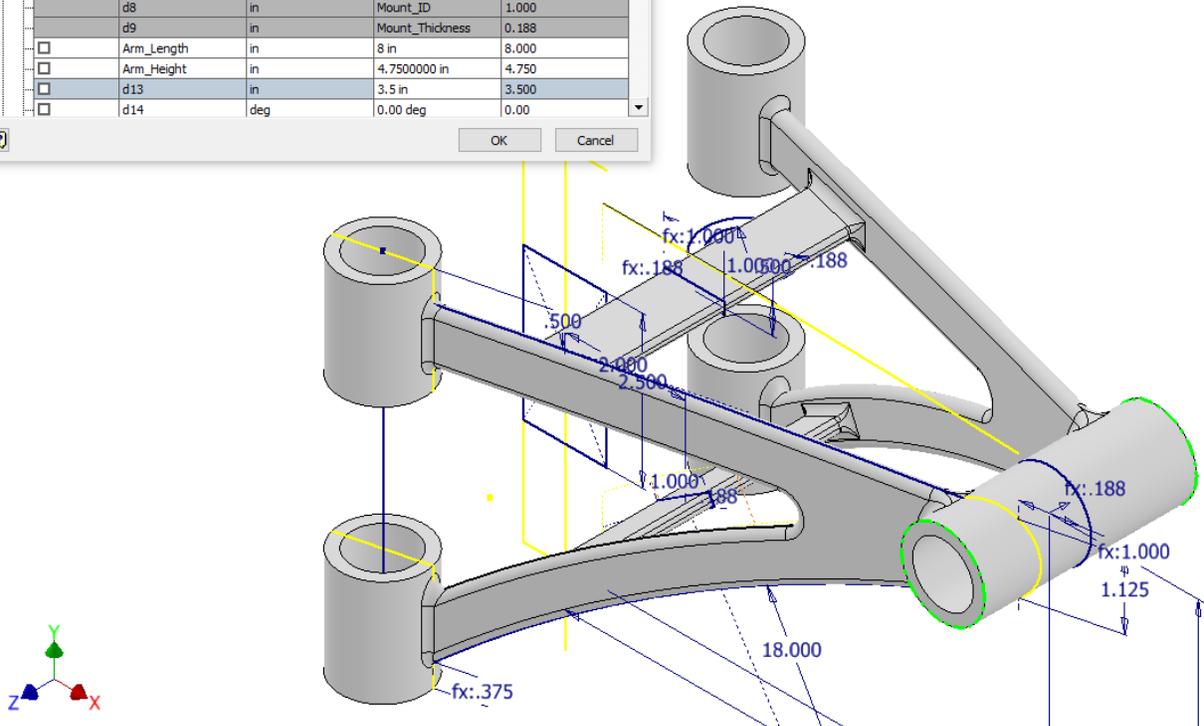- Support_Width
- Support_On_Off

Figure 5: Selecting parameters for the stufy

When choosing your design parameters for the study, Inventor will both list all of the available parameters in a dialog box, as well as show them visibly on the model itself. This will help you determine which parameters are the correct ones, particularly if they have not been named clearly in advance of the study.

You can click on parameters directly in the main canvas highlight them in the parameter list so you can find them and check them.

Once we have our parameters for the study selected, click OK to continue.

Assigning Parameter Values to Test

Now that we have chosen some parameters to examine during the study, we need to tell Inventor what parameter values we should have it simulate. There are multiple options when it comes to doing this:

- Discreet list of individual values (ex: 1, 3, 7, 12, 20)
- Range of values with number of divisions (ex: 2-8:4 produces 4 equally-spaced values from 2 through 8, e.g. 2, 4, 6, 8)
- Range of values with increments (ex: 3-5.5 / 0.5 will produce: 3.0, 3.5, 4.0, 4.5, 5.0, 5.5)
- General range of values, Inventor automatically assumes 3 values (ex: 4-10 produces the same as 4-10:3, or 4, 7, and 10)

To enter parameter values, open the Parametric Table. It will appear on the bottom of the window, but you may drag it out to a floating window or dock it along the sides of the program, if you wish.
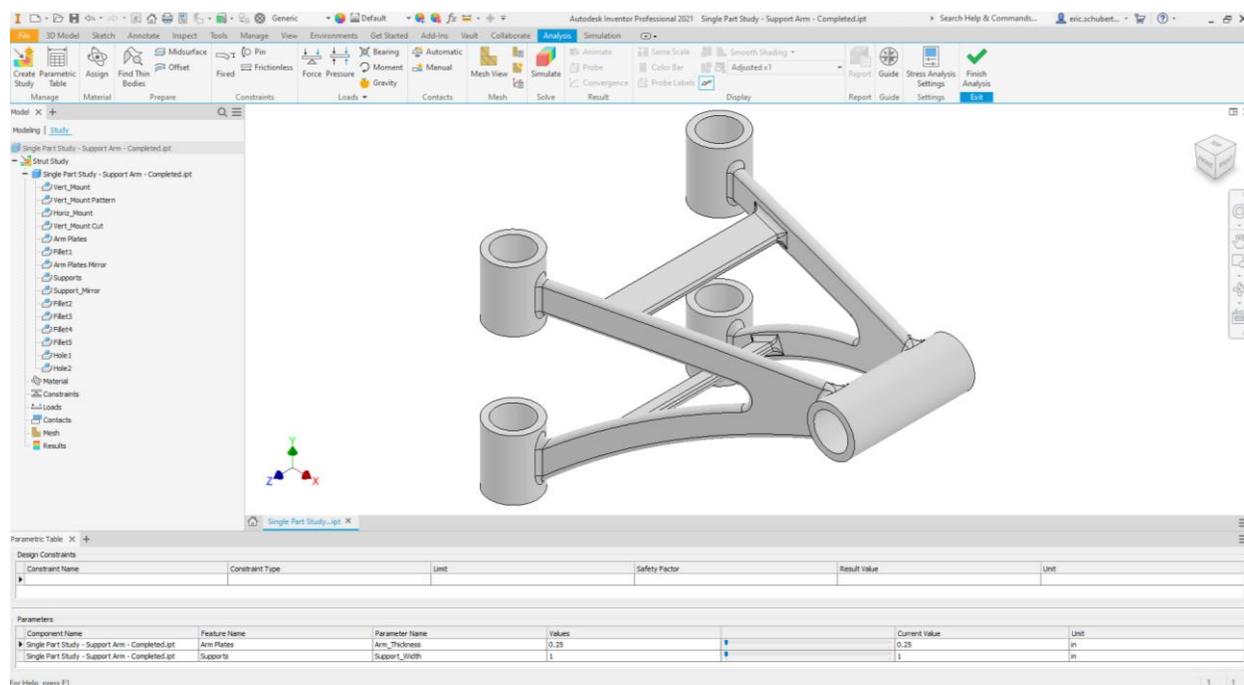

Figure 6: Parametric Table Location

In this case, we want to test a few variations of both parameters. Let's input the following into the Parametric Table:

Parameter Name	Values
Arm_Thickness	0.25 - 0.75:5
Support_Width	0.5 - 1.5:3

Figure 7:Input parameter values

Right-click over the values and select "Generate all configurations". You can then drag the sliders to view each combination of parameter values.

We need to make sure our materials are assigned correctly. Click the Assign Material button show in Figure 8 (right)

Set our part to **Aluminum 6061** material.

Note: You cannot vary the material using a Parametric Study. However, you could copy the existing study and simply change the assigned material to run another set of configurations for that material type.
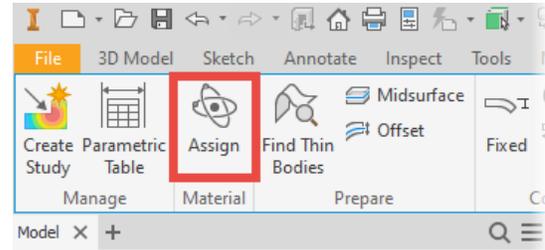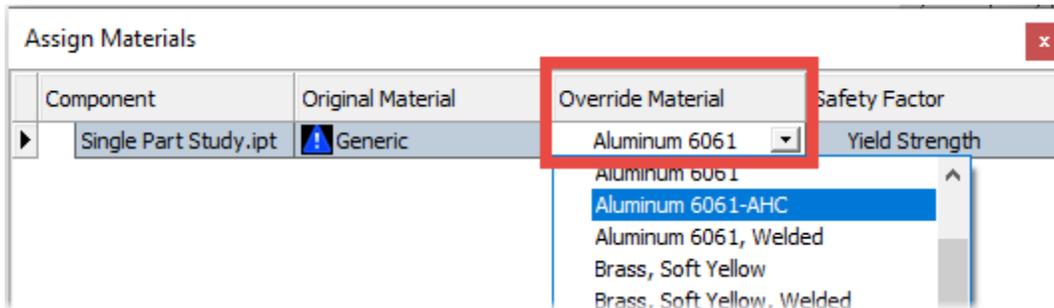

Figure 8: Assign Materials

Figure 9: Assign the material

Constraining the Model

We must apply constraints to the model so that it does not float off into space when pushed by the loads. We will apply fixed constraints to the four vertical mounts as shown here. You can experiment using pin constraints, as well, for some tangential or axial movement:
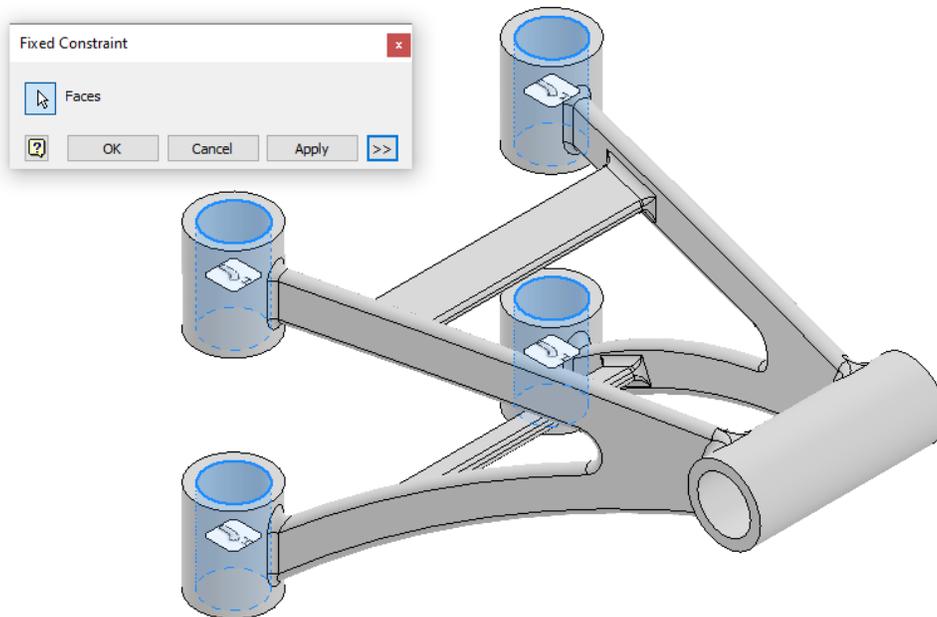

Figure 10: Fixed constraints

Loading the Model

This particular part will undergo both linear forces and moments as it is employed in the design. Apply the following loads:
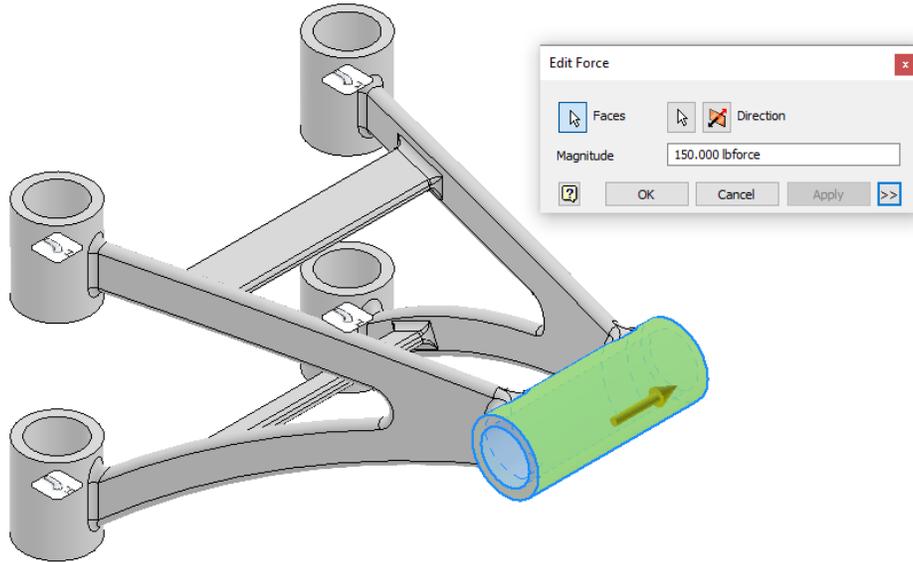
- 150 lbforce load, -Z direction only:

Figure 11: Axial force
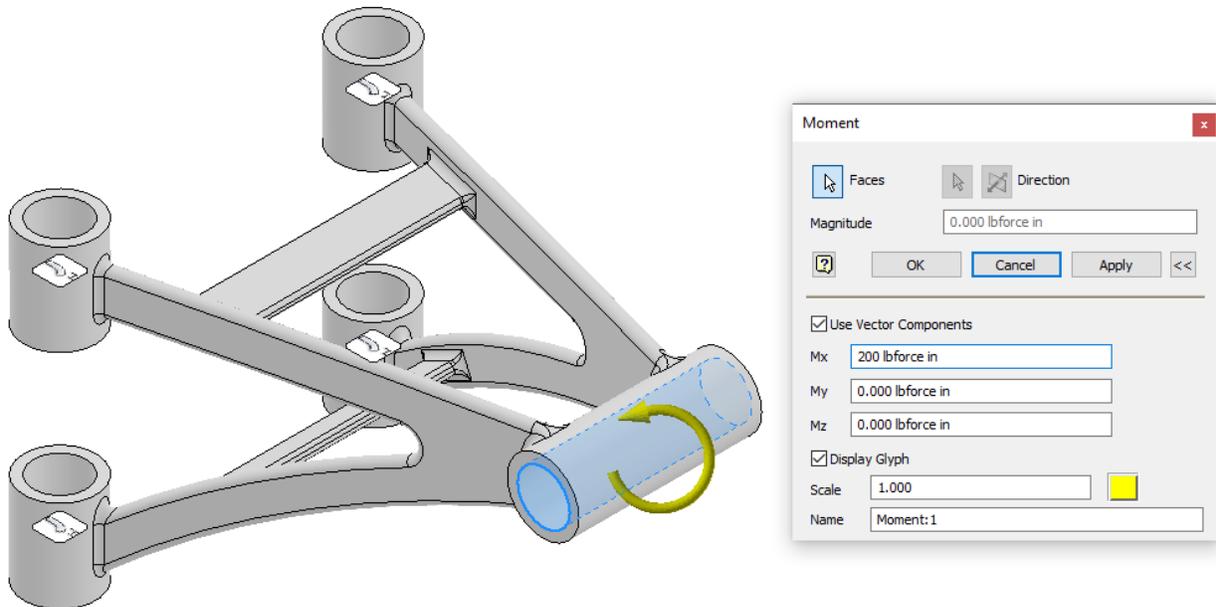
- 200 lbforce-in Moment, around +Mx direction only:

Figure 12: Moment / Torsion Load

Mesh the Model

Open the Mesh Settings by right-clicking the Mesh node in the browser. Set the average mesh size to 0.025. Click OK and then click the Mesh View icon to force Inventor to generate the mesh.

If you've already clicked the Mesh View icon, or the mesh does not appear as below, look in the browser for an Update icon (lightning bolt) next to the Mesh node. If you see this icon, right-click the Mesh browser node and have Inventor update the mesh.

Once the mesh calculates out, you should see at least 2-3 layers of mesh through the thicker sections of the strut. In this case, we're concerned about the connection between mounting points, but not the mounts themselves. If those were concerns, we could add localized mesh controls to reduce the mesh only on those thinner tubular sections for greater accuracy of the results in those areas.
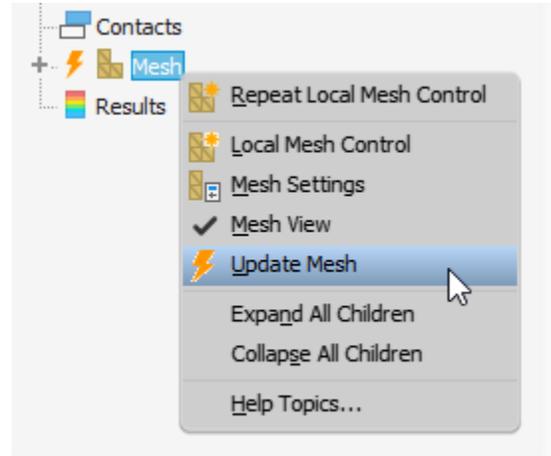
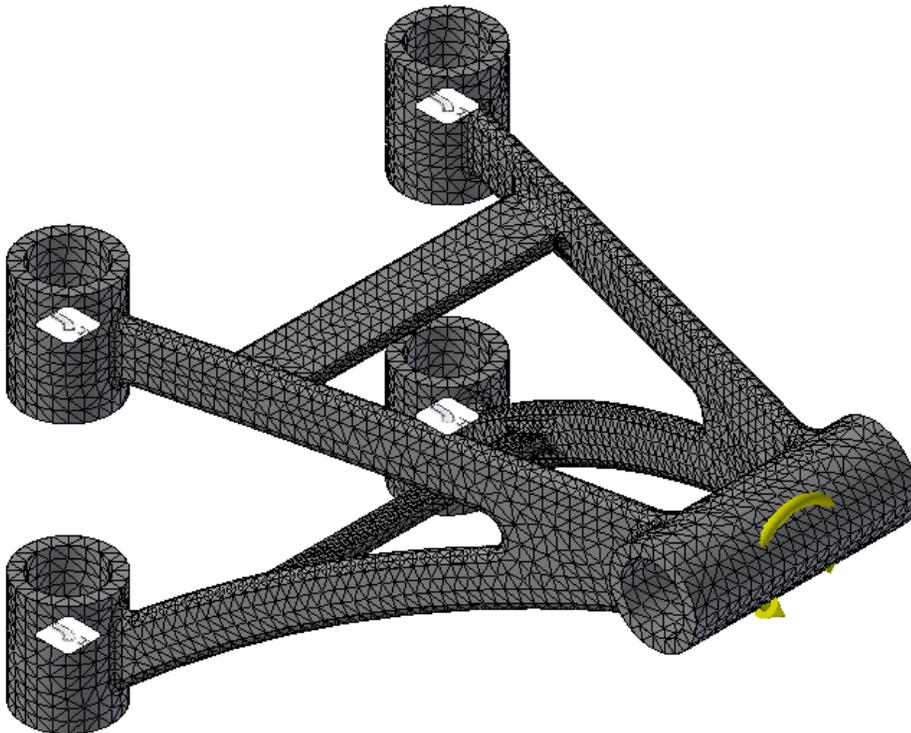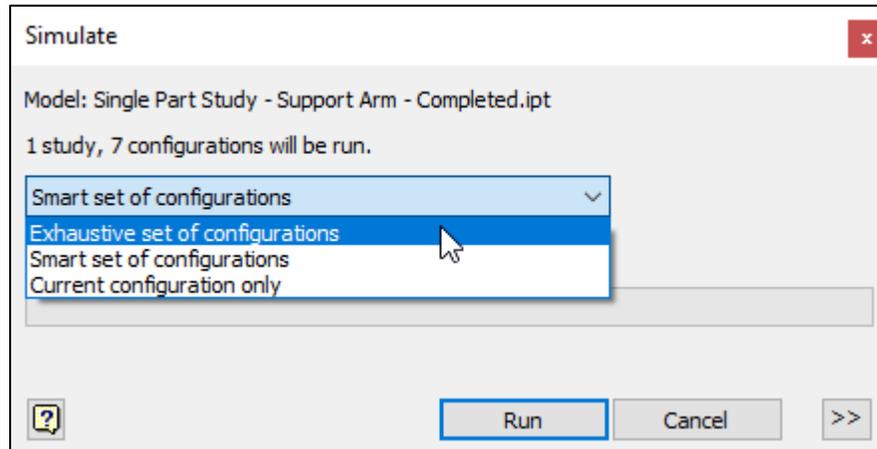*Figure 13: Right-click the Mesh browser node to adjust settings or update the mesh*

Figure 14: Assigned mesh size of 0.025 (fraction of bounding box)

Running the Simulation

It's finally time to run the simulation and view the results of each configuration. When you click on the Simulate button to begin the solving process, Inventor will present you with three options:

These three options provide flexibility on the solving process.

You may want to only run a quick test and verify the setup generates reasonable results. If you run all of the configurations, it may take significantly longer before you catch a mistake in your boundary conditions. In this case, run the "Current configuration only" option.

You may wish to run a full analysis on all of the configurations. In this case, use the "Exhaustive set of configurations" option.

Lastly, Inventor provides a "Smart set of configurations" option. This selection will only solve changes to the primary driving feature being modified by the selected parameters. It then interpolates the results for the other parameter changes with an estimated value. This is useful for a quick check or comparison when time is very limited. However, it will not be as accurate as the "Exhaustive set of configurations" for all parameter combinations.

If time permits, it is always recommended to run the "Exhaustive set of configurations" for the final solution.
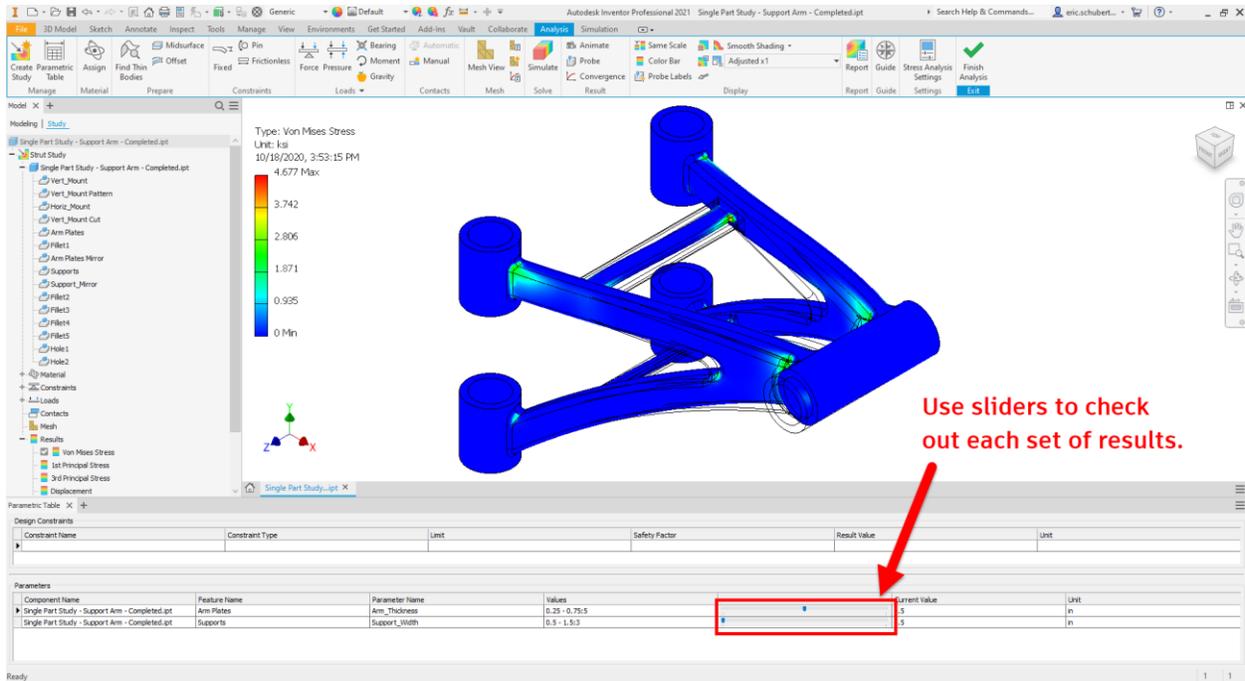
Note: Inventor must individually simulate each and every combination of parameter values. In this case, we have 5 possible values for the Arm_Thickness parameter and 3 possible values for the Support_Width parameter. Therefore, there are 15 different combinations of these two parameters.

This means inventor will mesh and solve 15 separate simulations!

This is not a fast process, particularly with more-complex models. However, the outcomes can be extremely helpful in determining design decisions. This is the big trade-off with Parametric Studies. They take longer to run, but provide an excellent tool for direct comparison of parameter values.

Evaluating the Results

Once the simulations are finished running, you can Activate the result type you wish to view in the browser and use the sliders in the Parametric Table to compare each combination of values and their respective results.

Figure 15: Use the sliders in the Parametric table to view results for each combination

Parametric Studies on Assemblies

Now that we know the basics of running single-part parametric studies, we can easily correlate this workflow with assembly designs (.iam).

Inventor tracks parameters in virtually every type of file, and assemblies are no different in this regard. However, the types of parameters that are tracked can certainly differ. If we consider parameter values that we utilize in assemblies, we can find two primary types:

- Assembly feature parameters (sketches, extrusions, etc.): These parameters only affect features at the assembly level, and will not reflect on changes back at the part level.

- Assembly Constraint and Joint parameters: Offset or angle values assigned to these types of component-to-component relationships can also be adjusted in a parametric study

However, just because we're in an assembly does not mean we can no longer vary part-level parameters. We can **<u>absolutely</u>** reach down into the depths of each individual component and adjust parameter values at every level in the assembly hierarchy!

To access the parameters in each level of the assembly, expand the browser to see the component containing your target parameter, then right-click on it. Simple as that!
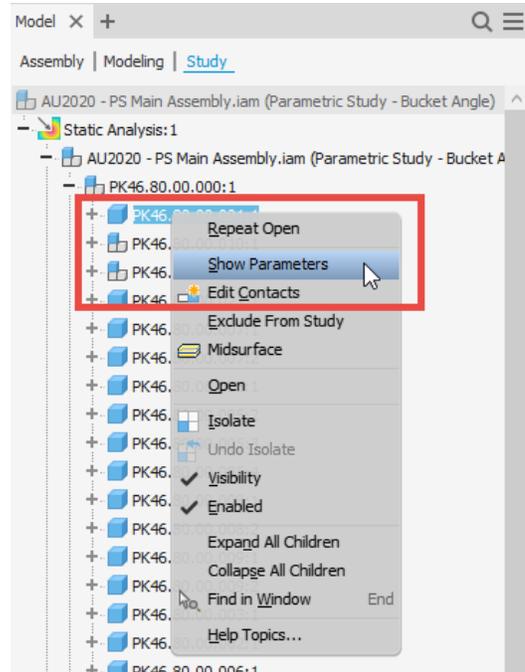

Figure 16: Right-click any component to access its parameters for the study.